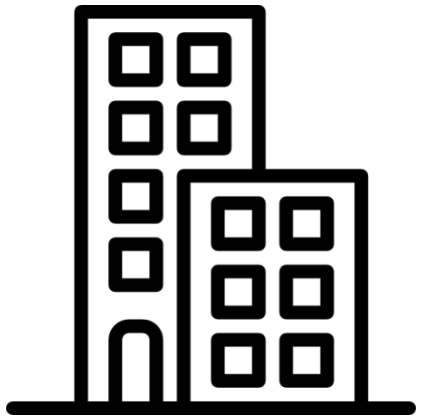
FEA NX INTRODUCTION

(English Course)

沈約翰 (John)
john@midasuser.com.tw

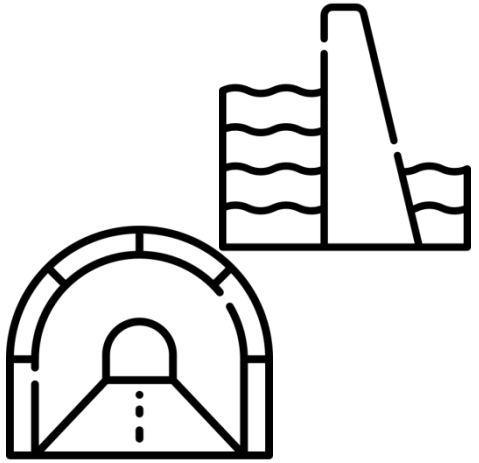
CONTENTS

MIDAS FEA NX is a **Finite Element Analysis software** developed by MIDAS IT, specifically designed for **advanced structural analysis in civil and geotechnical engineering**.

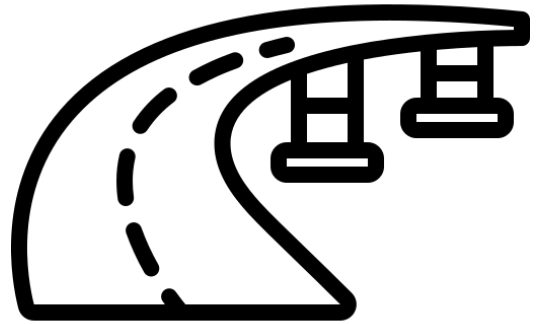


Concrete Behavior

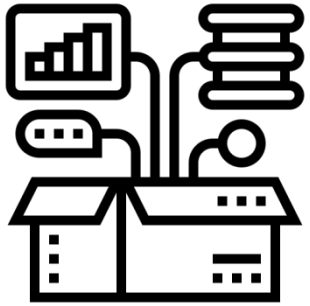
Soil - Structure Interaction



Tunnels, dams, and underground structures

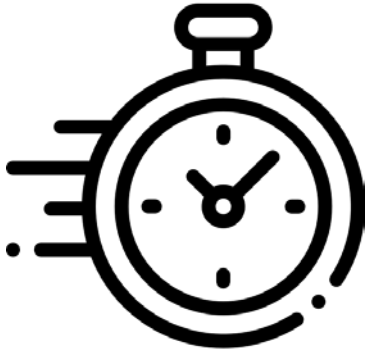


Bridges, large infracstructures and other complex geometries



FEA NX KEY FEATURES

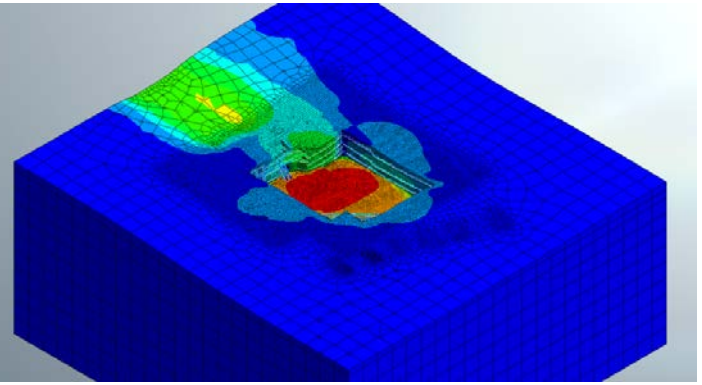
Feature	Description
Advanced Material Models	Includes nonlinear concrete, steel, rock, and soil behaviors
3D solid modeling	Supports solid, shell, and beam elements for detailed modeling
Contact and interface elements	Models interactions between soil and structures, joints, and more
Automatic meshing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

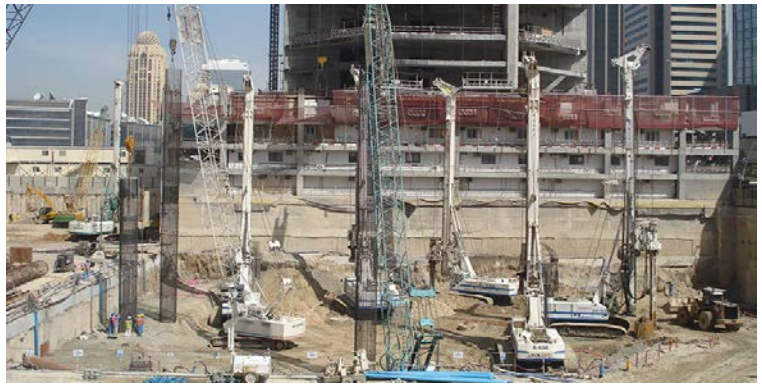


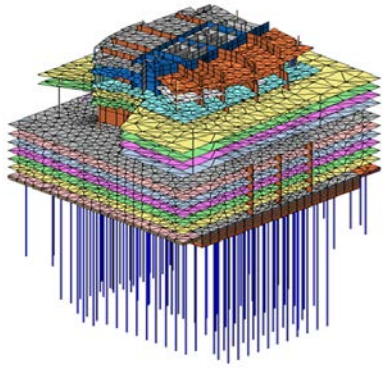
Maximizing Efficiency with MIDAS FEA NX

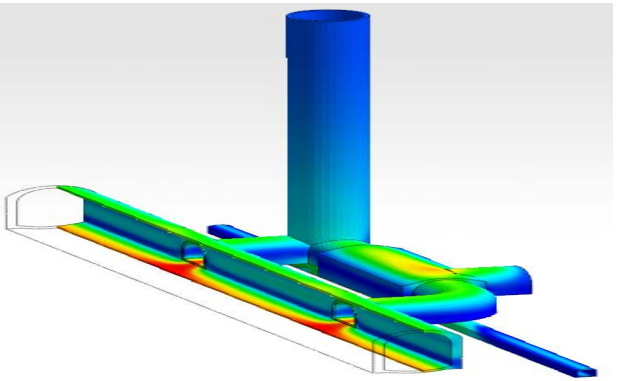
- Automated Meshing and Geometry Handling
- Advanced Nonlinear Solvers
- Integrated Modeling & Post-Processing
- Template-Based Modeling & Reusability
- Accurate Simulation Reduces Rework

Reasons for 3D



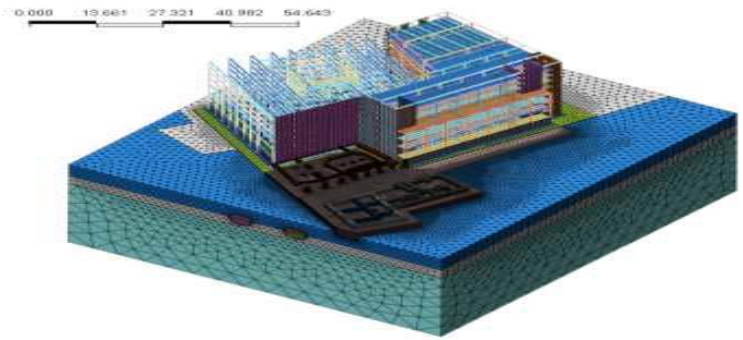


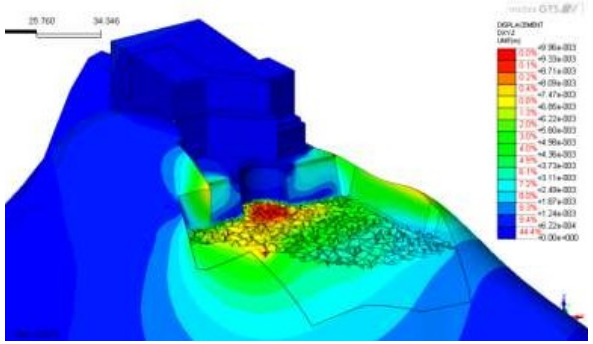


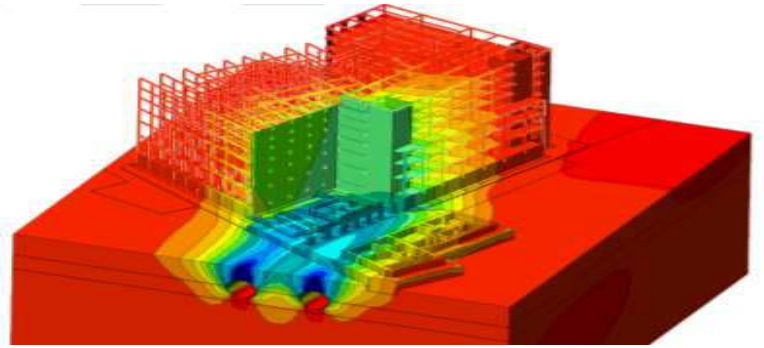


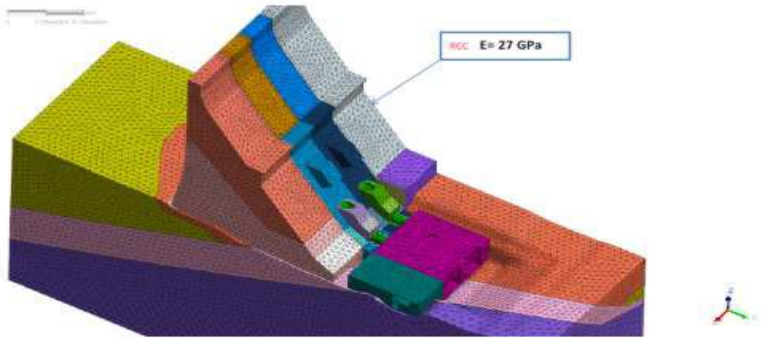
Our MIDAS commitment

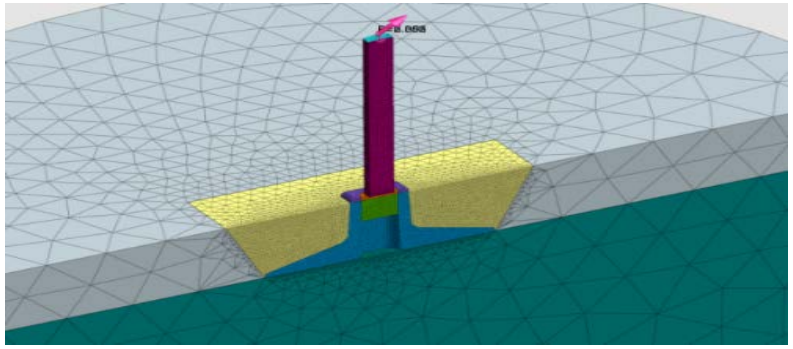
- Experts in 3D/2D specialized engineering
- Constantly providing technical support, materials, and training

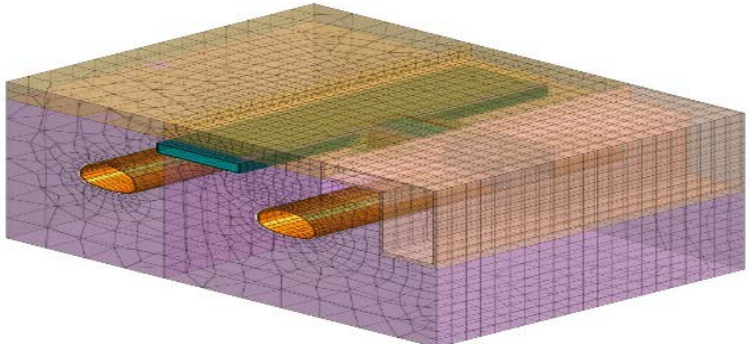








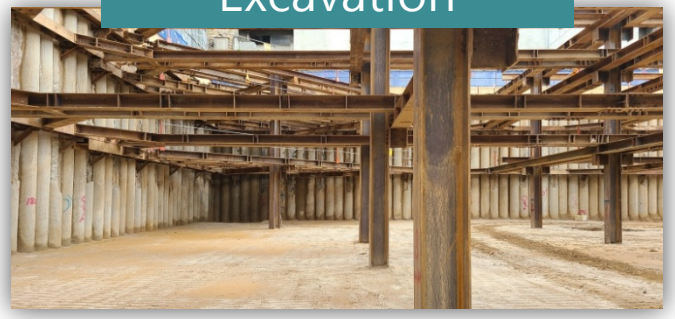




KEY APPLICATIONS

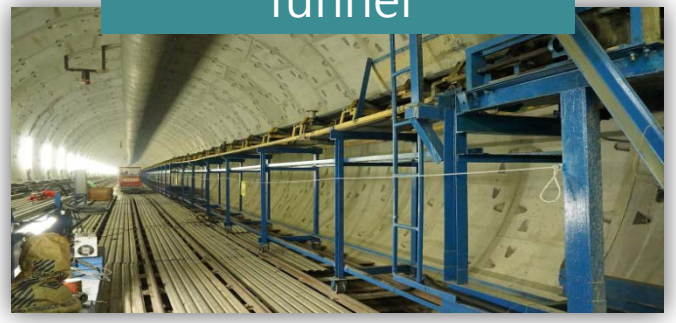
Pile

Excavation

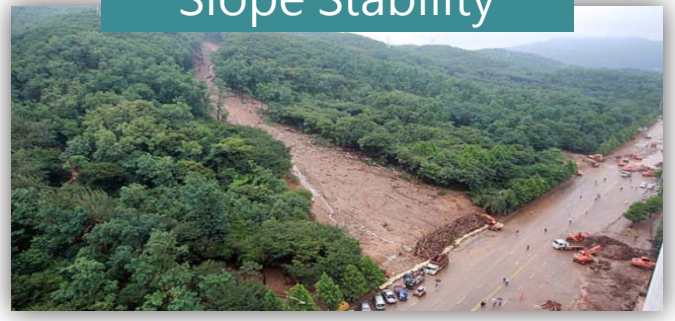


FEA

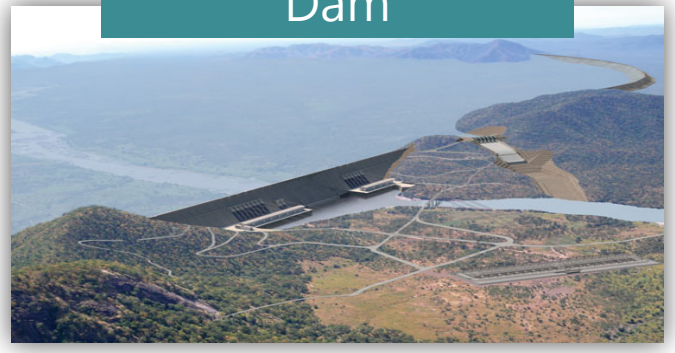
Tunnel



Slope Stability



Dam



Soil-Structure Behavior

MULTIPLE TYPE OF ANALYSES

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

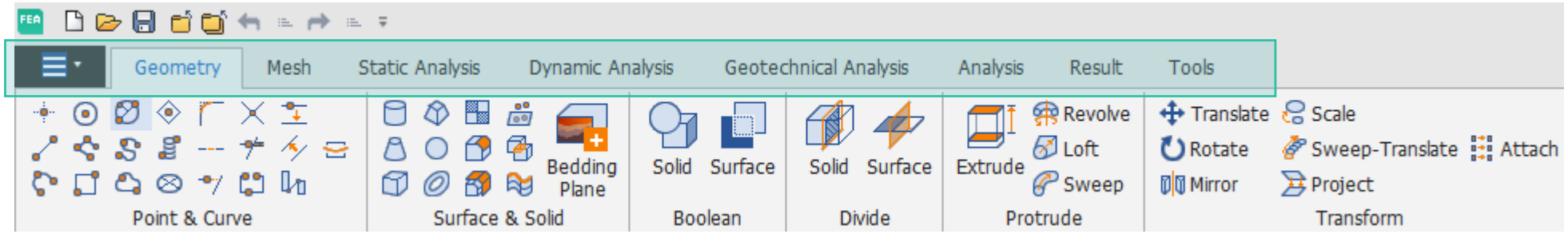
- Eigenvalue / Response Spectrum analysis
- Linear Time History (mode/direct methods)
- Nonlinear Time History analysis
- 1D/2D Equivalency Linear analysis
- Nonlinear time history + SRM Coupled

Slope Stability Analysis

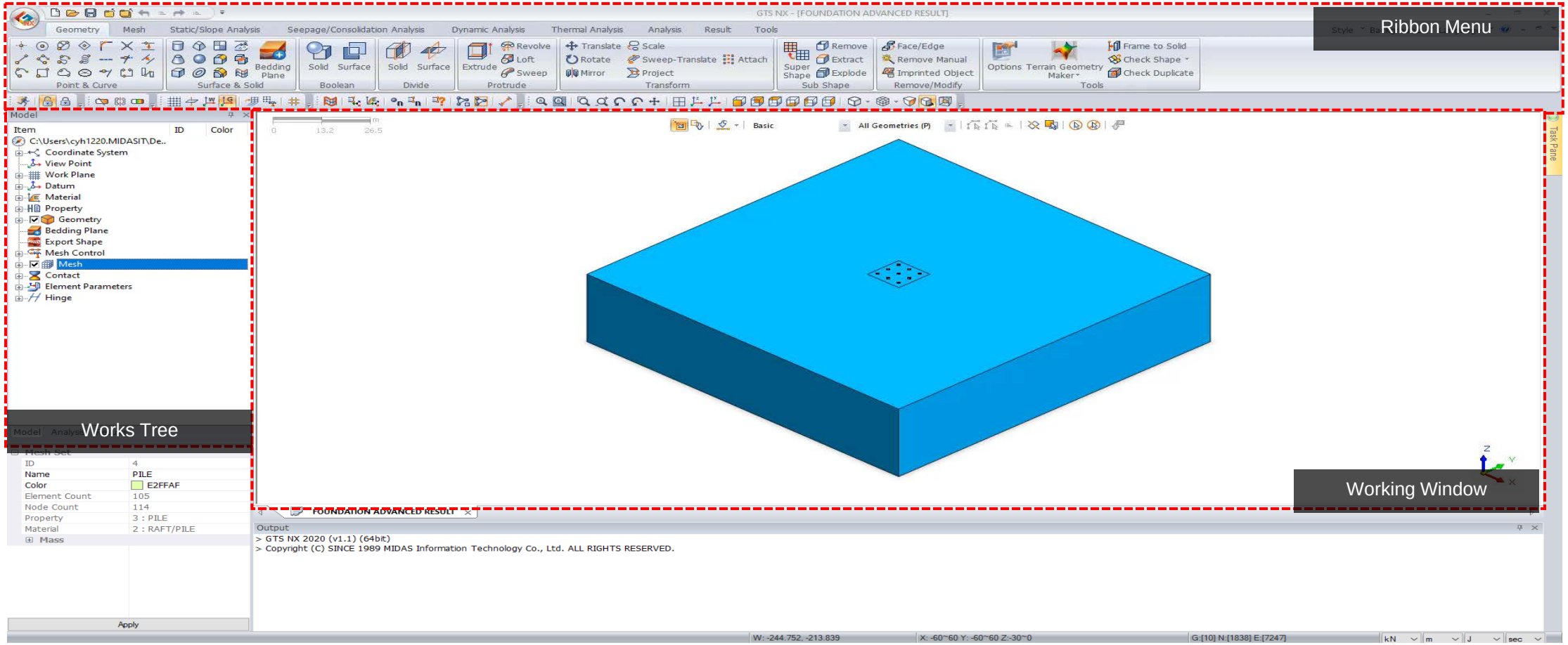
- Strength Reduction Method (SRM)
- Strength Analysis Method (SAM)
- Construction stages Slope stability (SRM/SAM)

DESIGN PROCESS

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



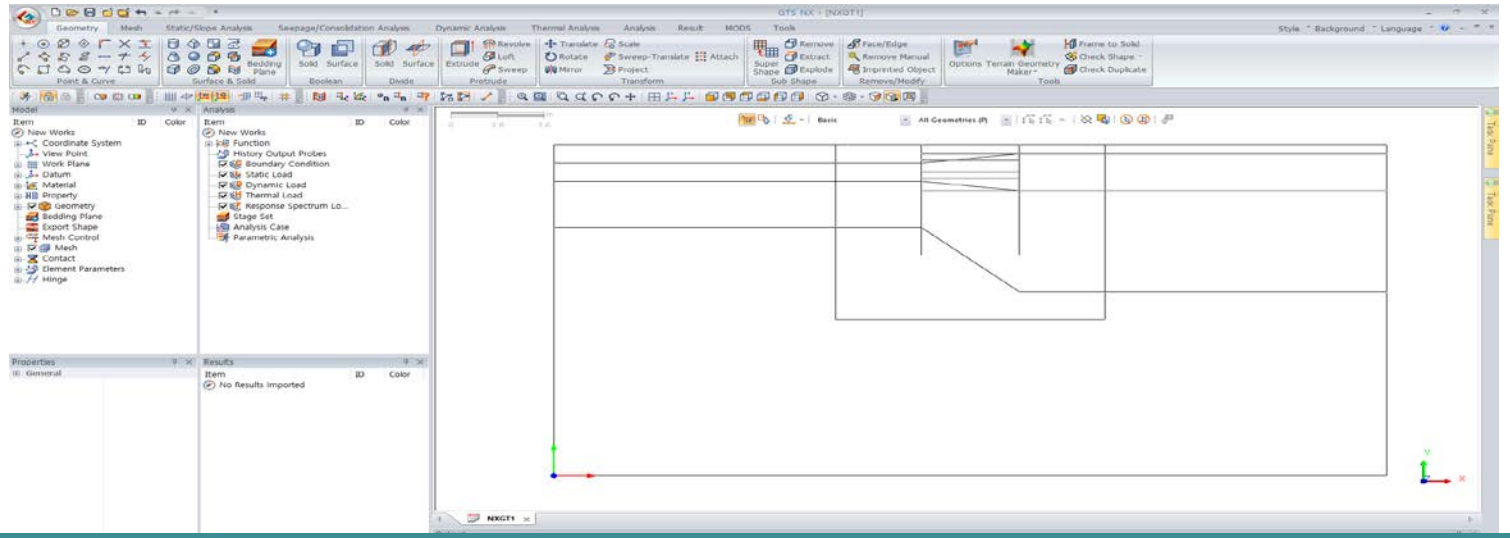
GRAPHICAL USER INTERFACE



COMPATIBILITY

Works with most drawing files / CAD formats

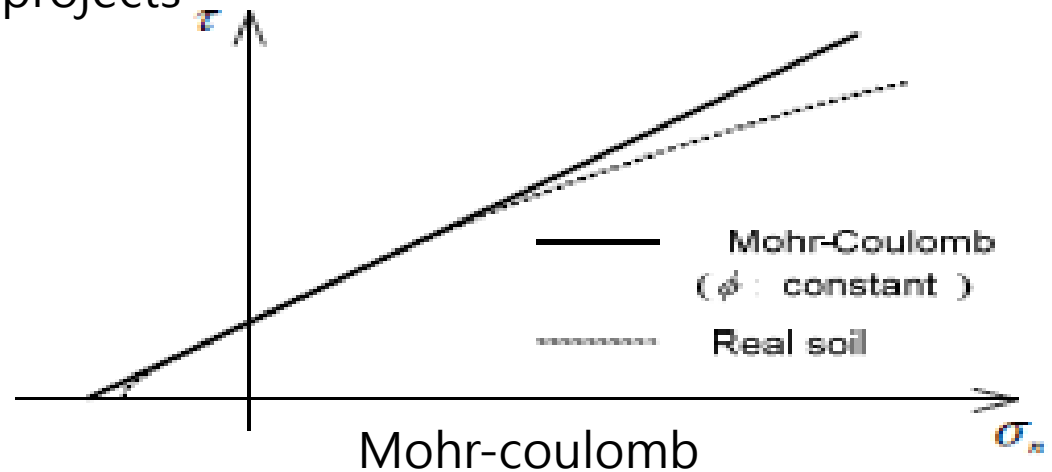
File format	Description
*.dwg	AutoCAD drawing files
*.dxf	AutoCAD drawing interchange files
*.x_t; *.xmt_txt; *.x_b; *.xmt_bin	Parasolid (9 to 29) files
*.sat; *.sab; *.asat; *.asab	ACIS (R1 to 2018 1.0) files
*.stp; *.step	STEP (AP203, AP214) files
*.igs; *.iges	IGES (Up to 5.3) files



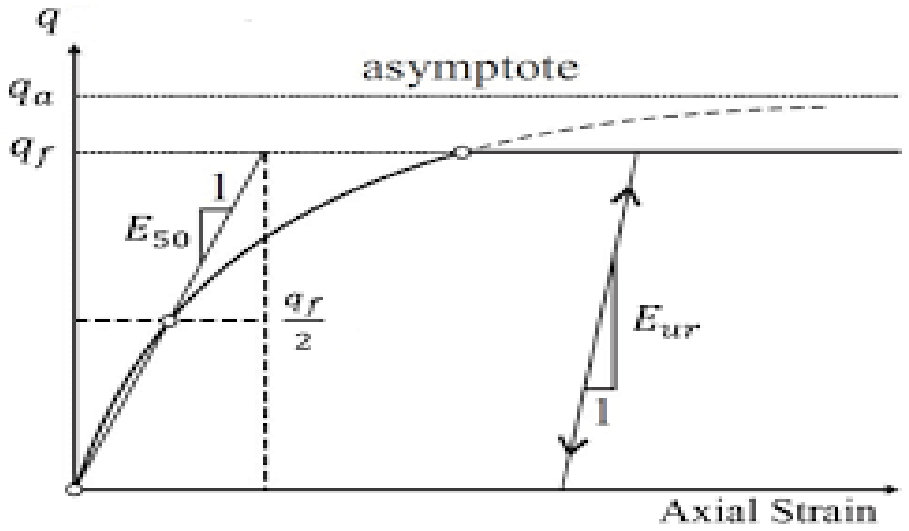
COMPREHENSIVE SOIL MODELS

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

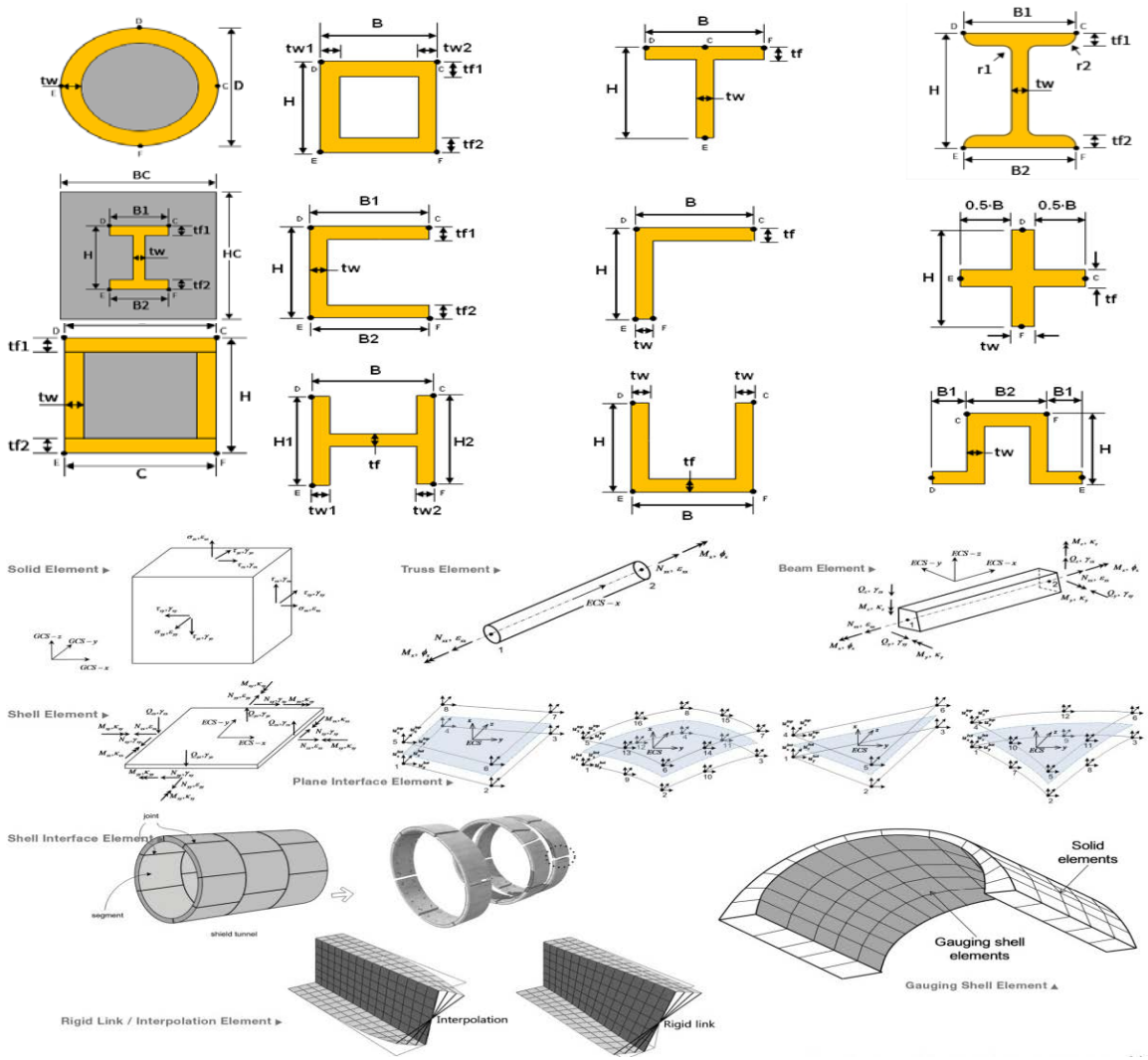
Saves time by just selecting the necessary elements from the database

1D Element
- Geogrid
- Truss
- Beam

2D Element
- Shell
- Gauging shell
- Plane stress
- Plane strain
- Geogrid
- Axisymmetric

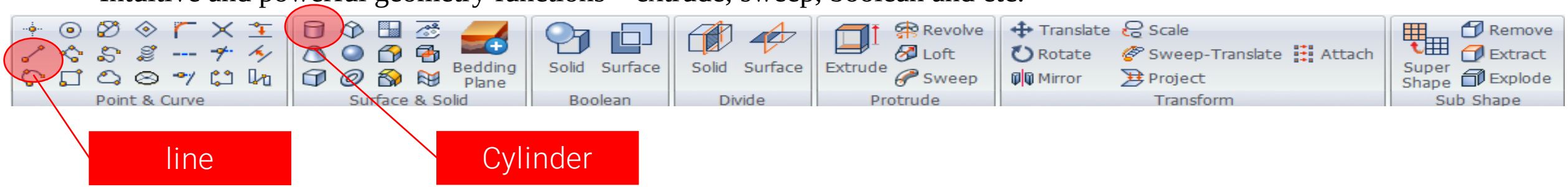
3D Element
- Solid

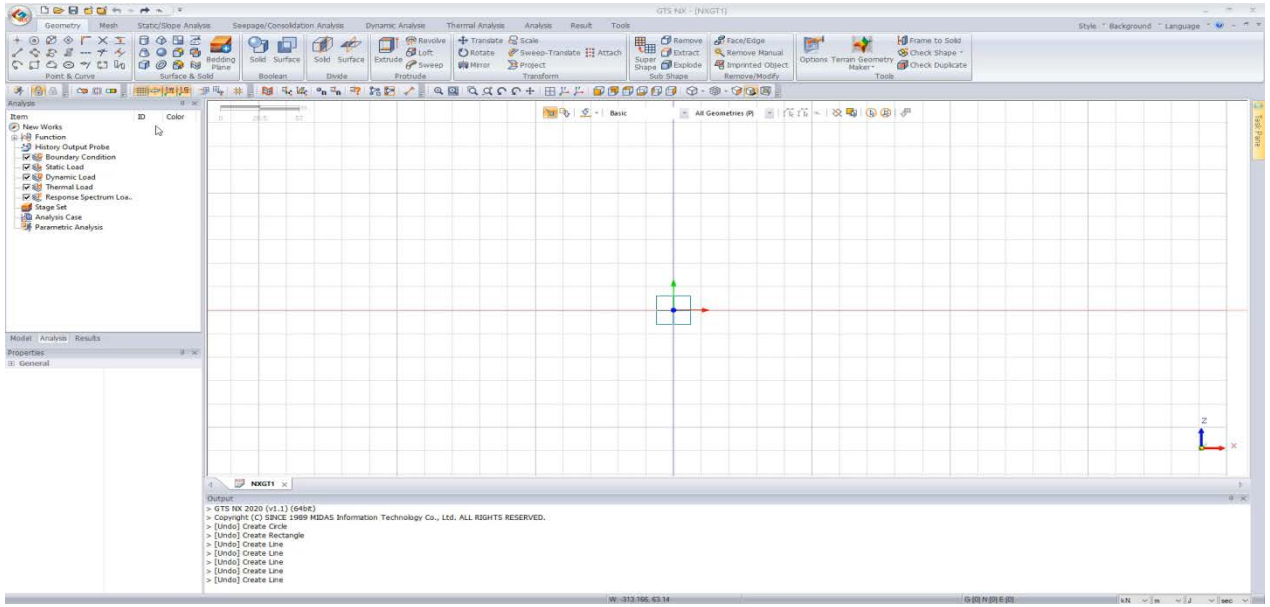
Others
- Interface
- Elastic / Rigid link
- Pile interface / Pile tip
- User specified behavior



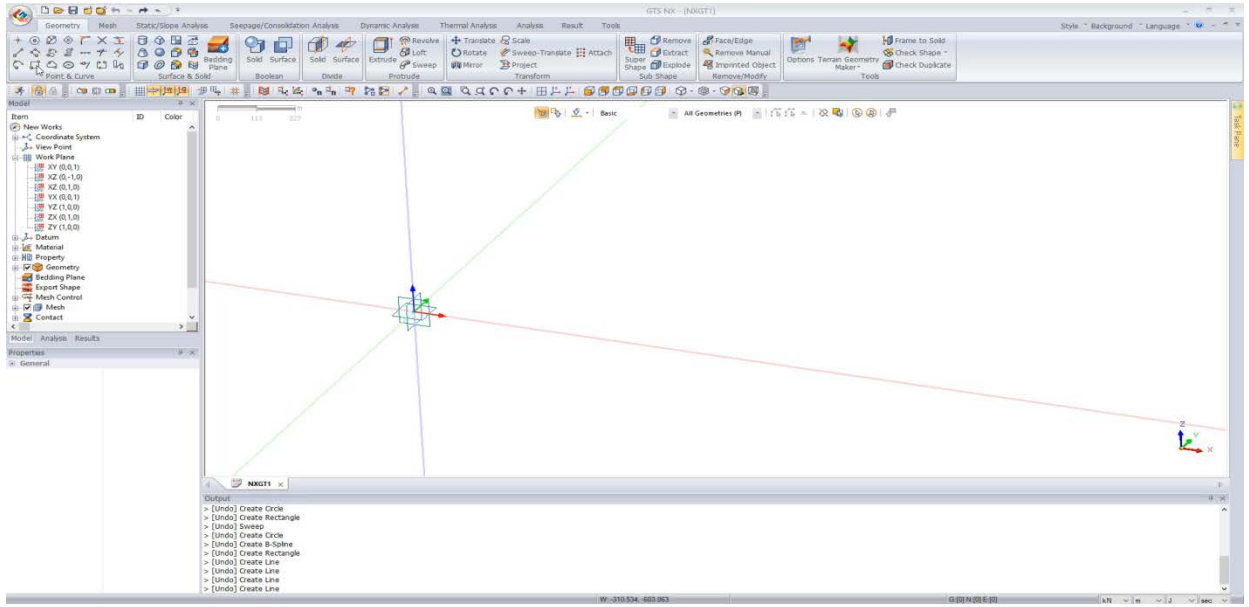
GEOMETRY

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





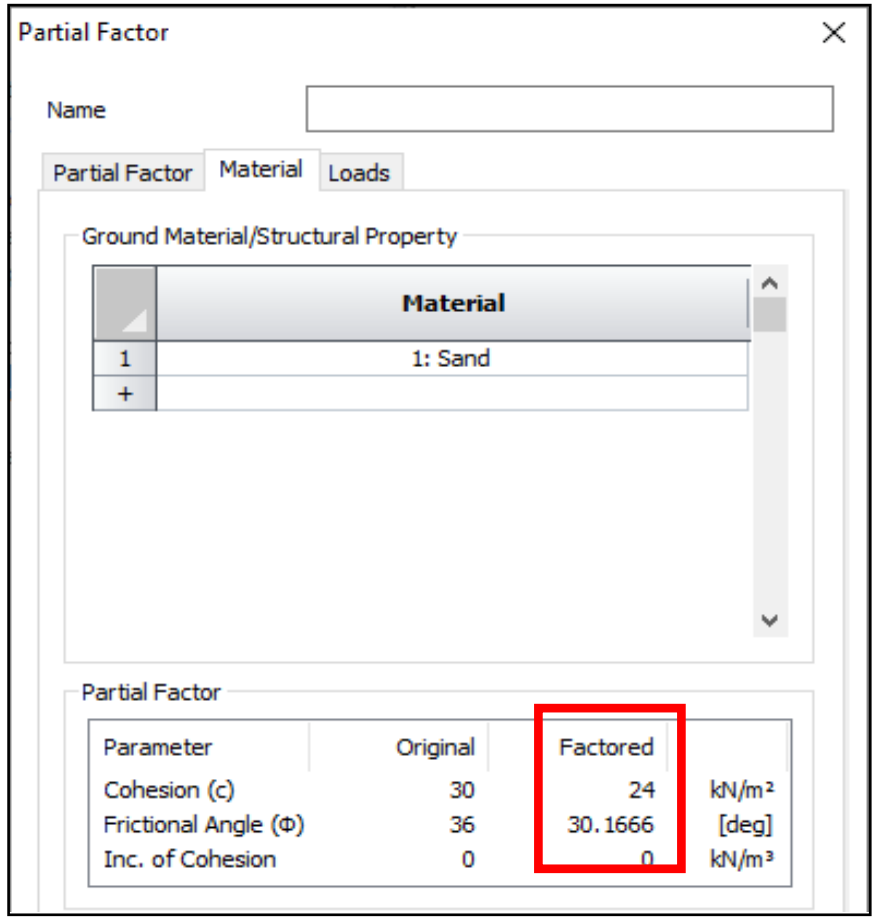
Line / Square / Circle



Extrude / Sweep

PARTIAL FACTORS

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

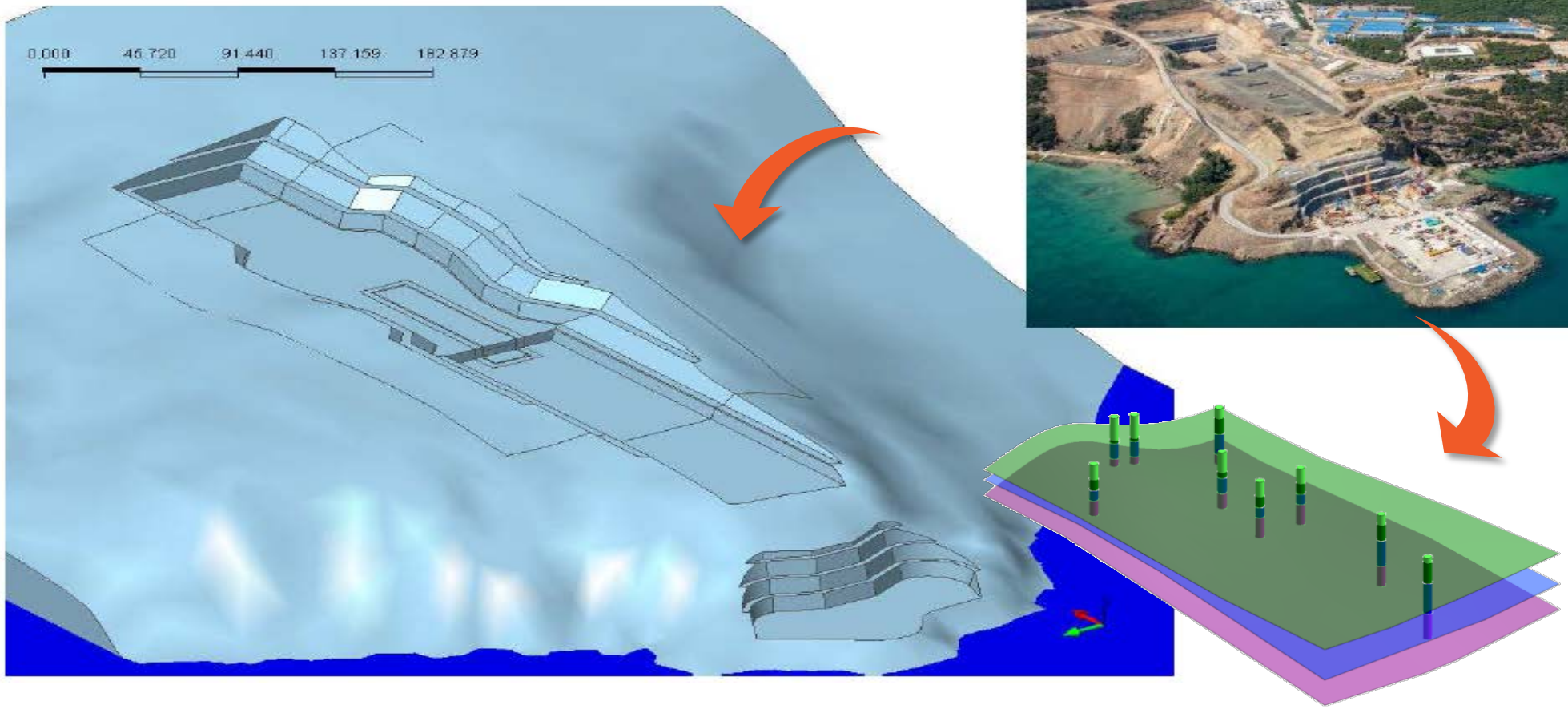


GTS NX is providing the database for this partial factor as below:
Design Approach 1 Combination 1
Design Approach 1 Combination 2
Design Approach 2
Design Approach 3...

Directly checking the original and factored parameters

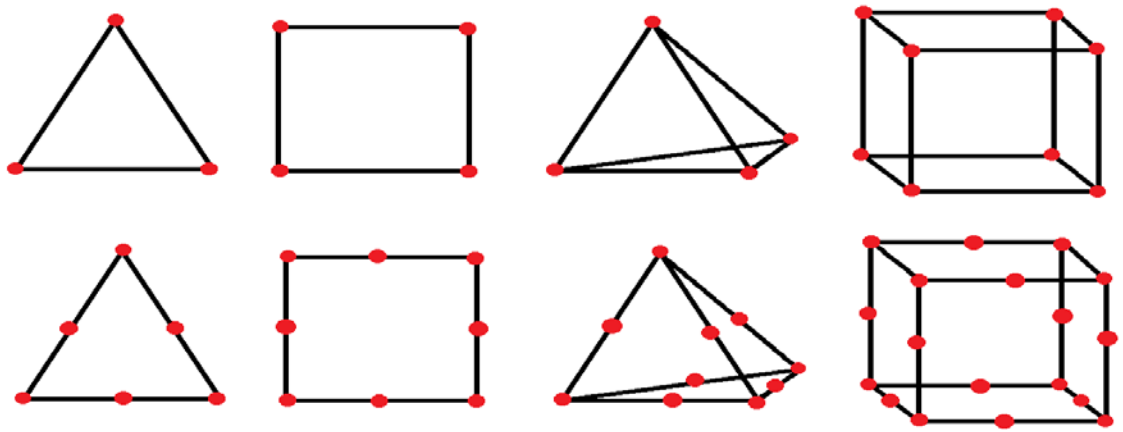
TGM & Bedding plane wizard

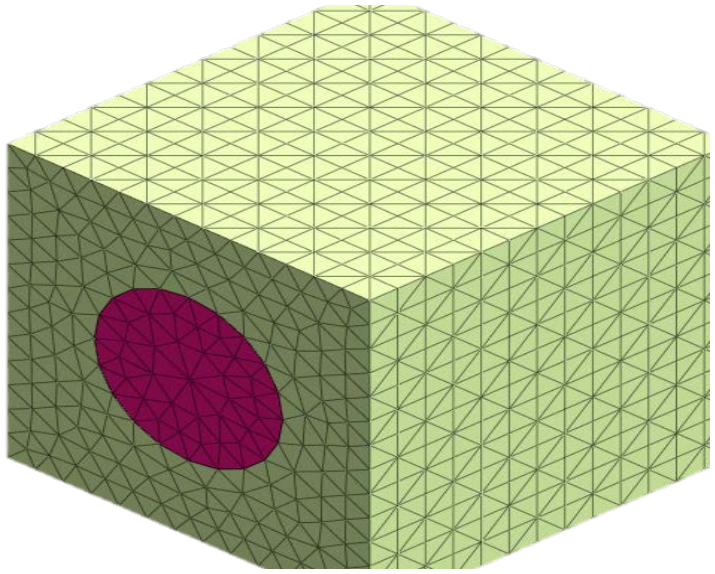
Easily create the surface of the site by simple topography import



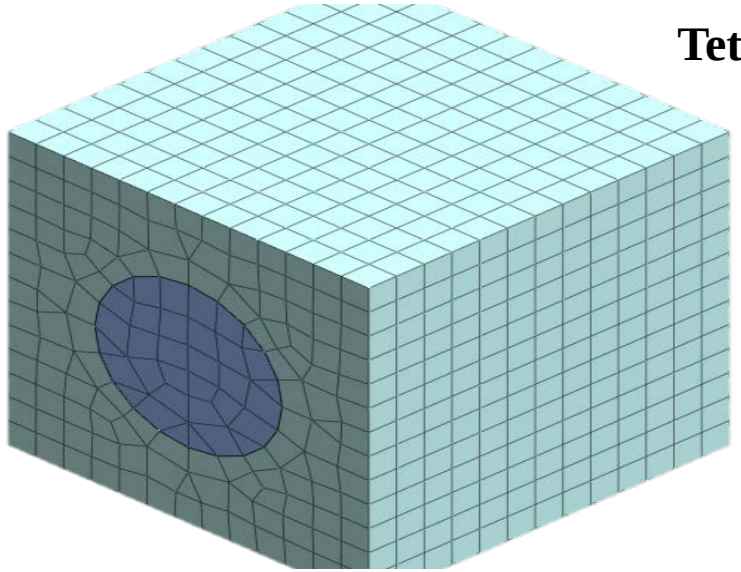
MESH

Powerful meshing algorithm with Hybrid technology

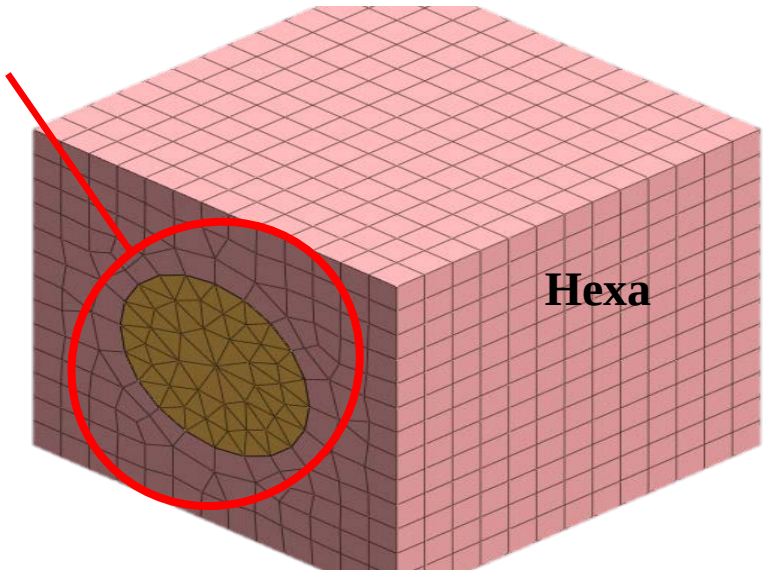




Tetrahedral



Hexahedral



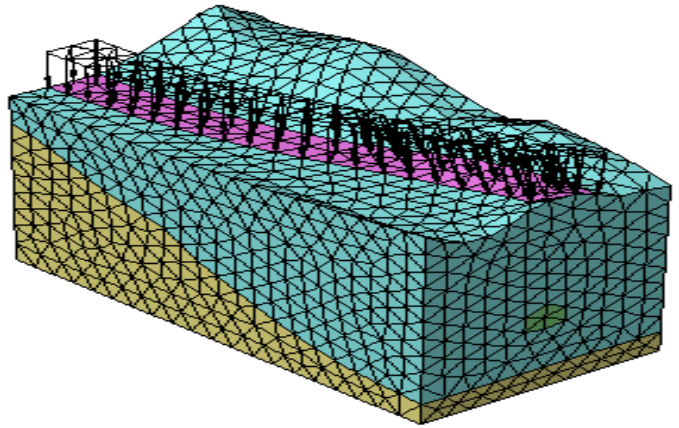
Tetra + Hexa(hybrid)

BOUNDARY CONDITION AND LOADS

Boundary

Constraint
Change Properties
Review
Water level
Nodal Head
Surface Flux

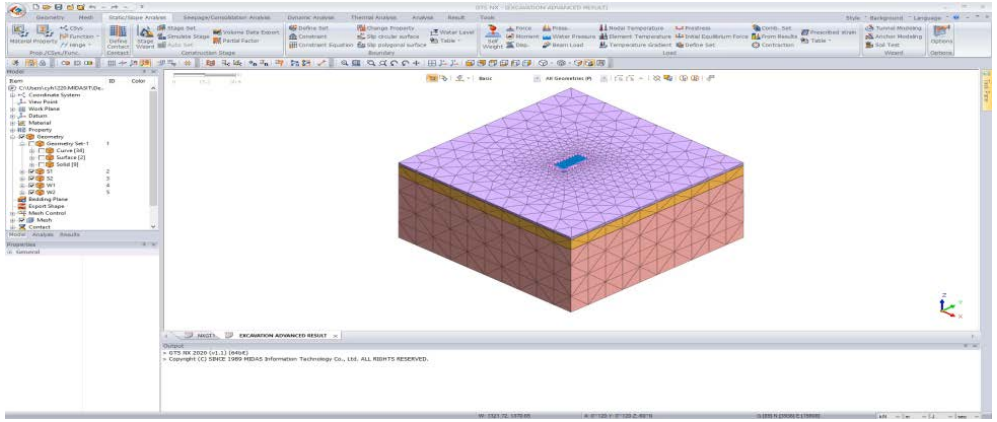
Slip Circle/Polygonal Surface
Draining Condition
Non Consolidation
Transmitting



Loads

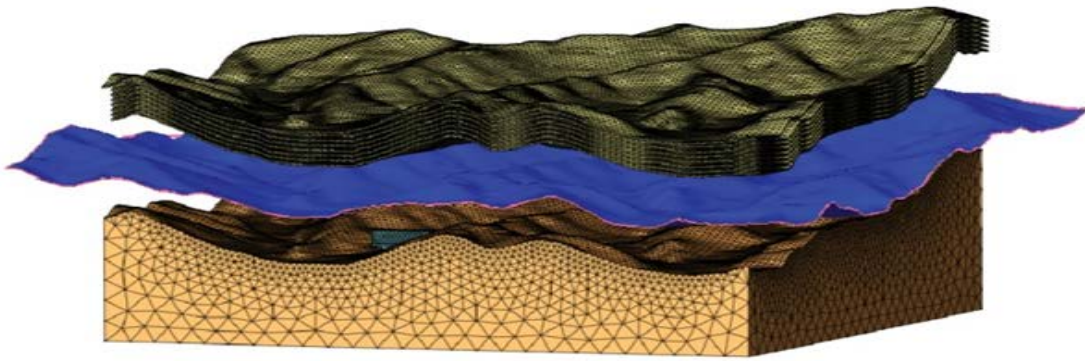
Self Weight
Force
Moment
Displacement
Pressure(Surcharge / Water)
Line Beam Load
Element Beam Load
Temperature
Pre-stress

Contraction
Initial Equilibrium Force
Combined Load
Response Spectrum
Ground Acceleration
Time Varying Static
Dynamic Nodal / Surface
Load to Mass
Train Dynamic Load Table

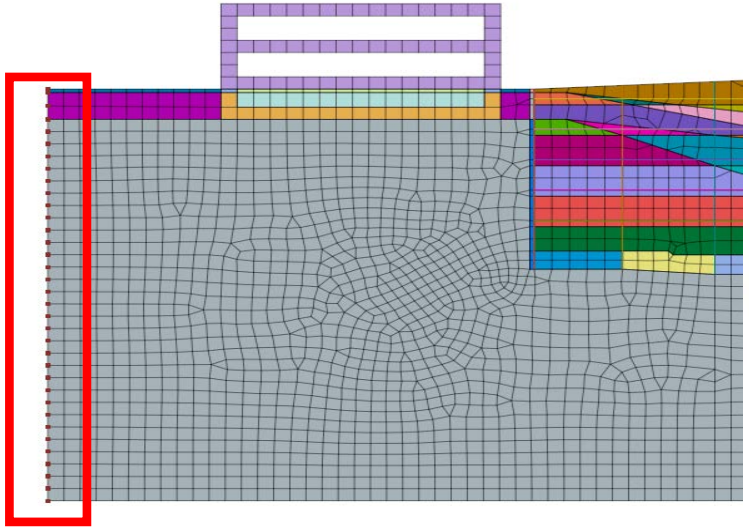


WATER CONDITION

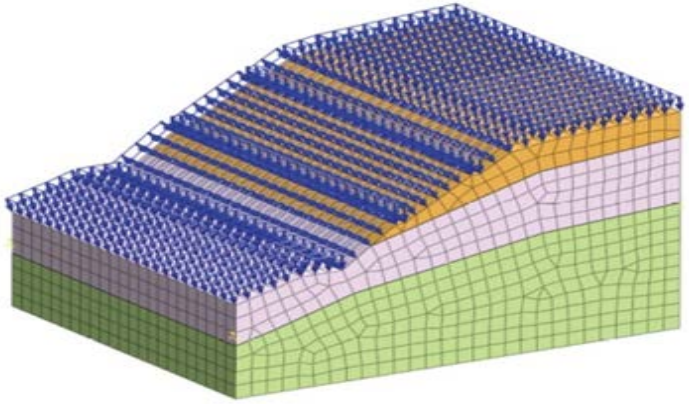
nodal head, line & surface flux, water level



3D water level automatic generation



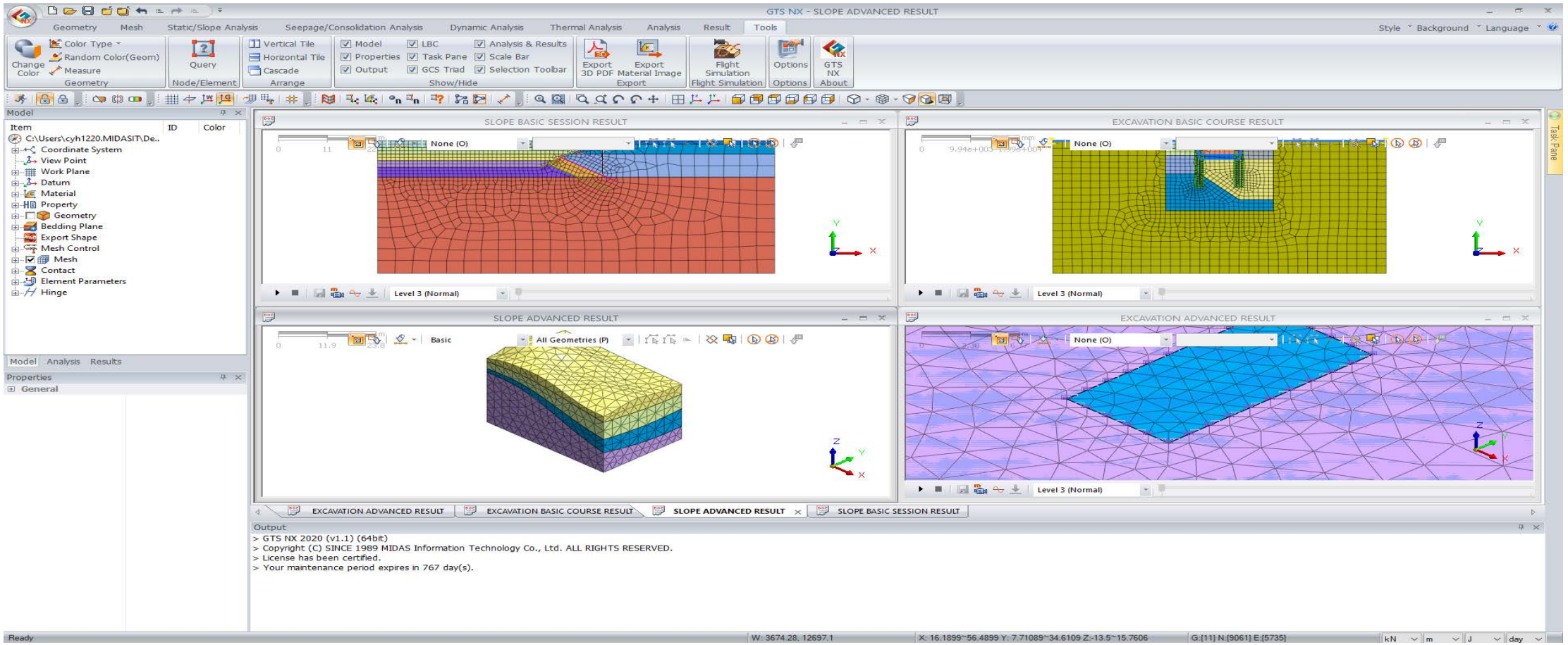
Nodal head for water level



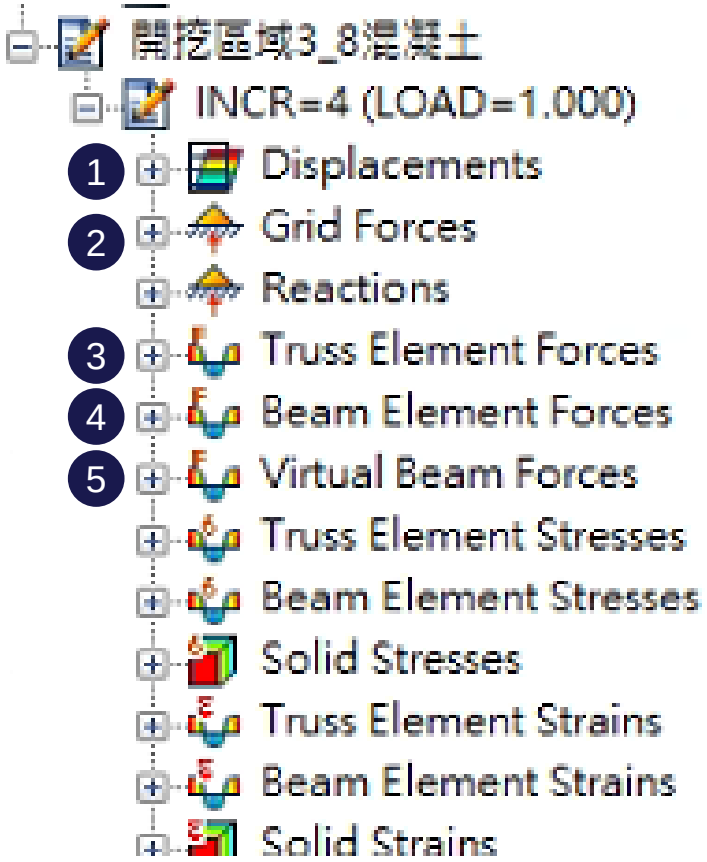
Rainfall intensity input

MULTI WINDOWS

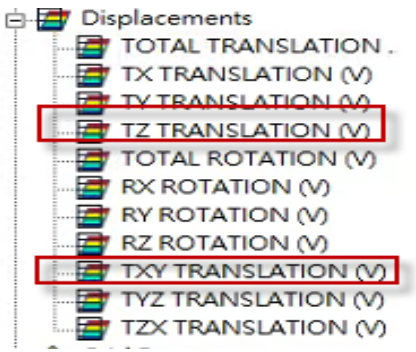
Compare various sections or different analyses in one program window



COMPREHENSIVE BREAKDOWN OF RESULTS

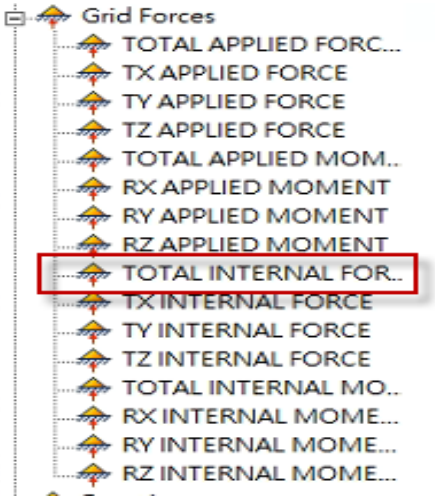


1



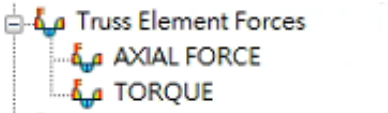
水平變位TXY
垂直變位TZ

2

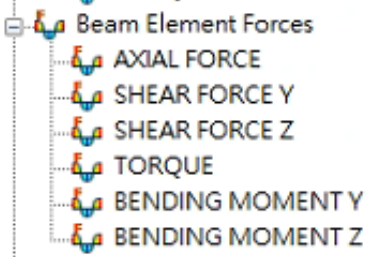


內力
Grid Force

3



4

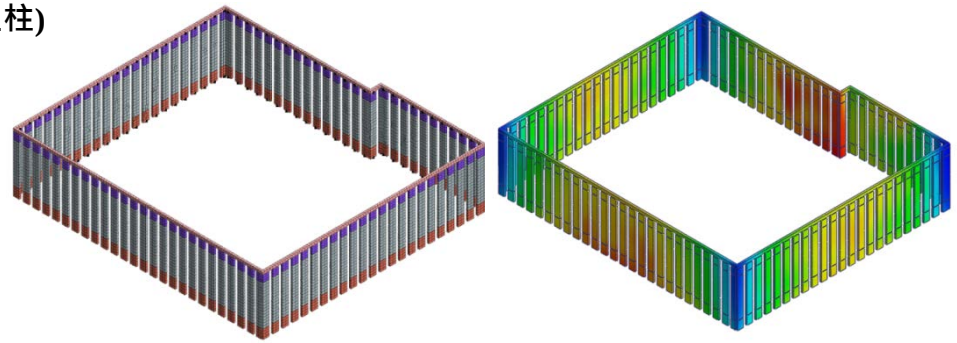


5

軸力
Axial Force

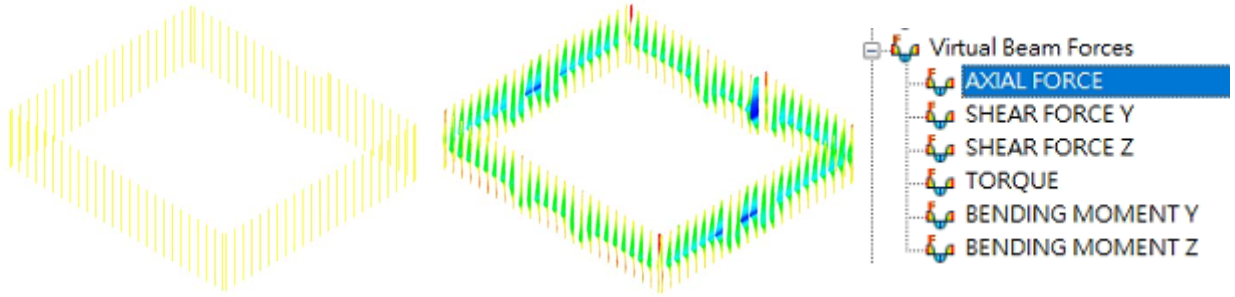
RESULT - VISUALIZATION

實體元素
(擋土柱)

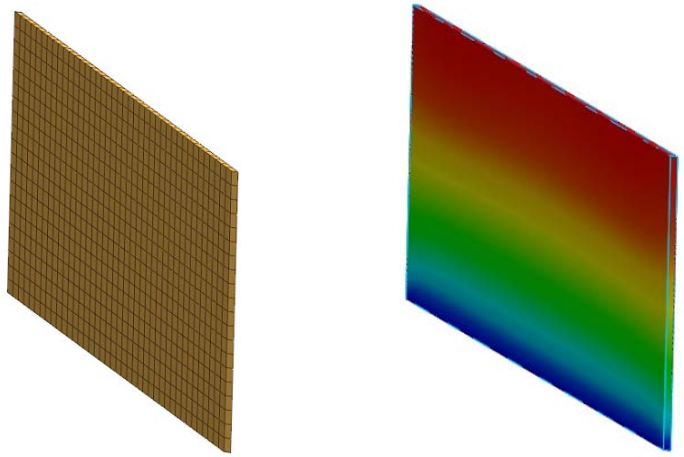


如何計算等效軸力?

虛擬梁(Virtual Beam)

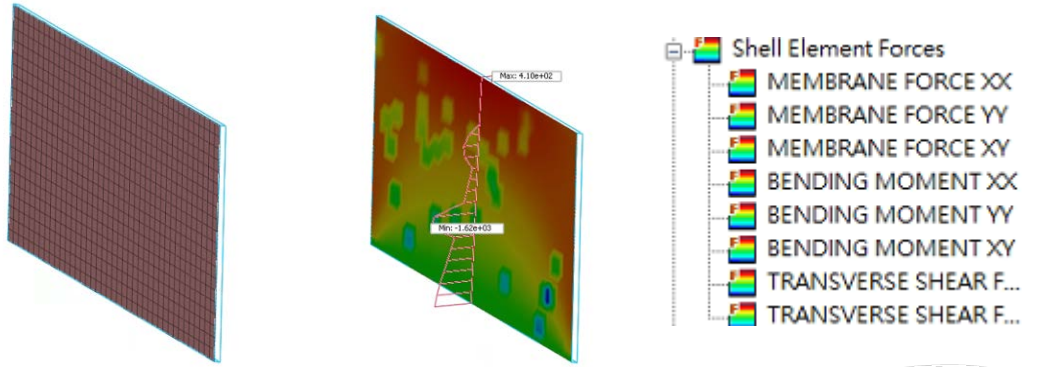


實體元素



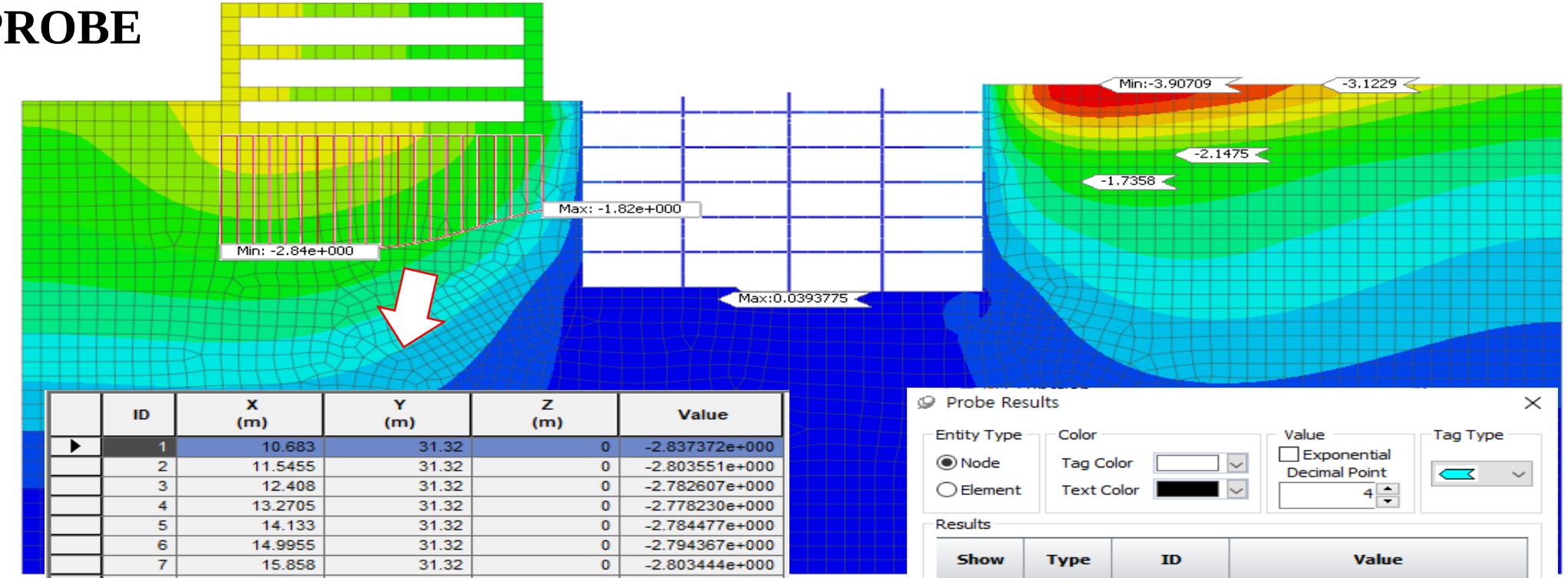
如何得到受力方向大小?

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
☐			

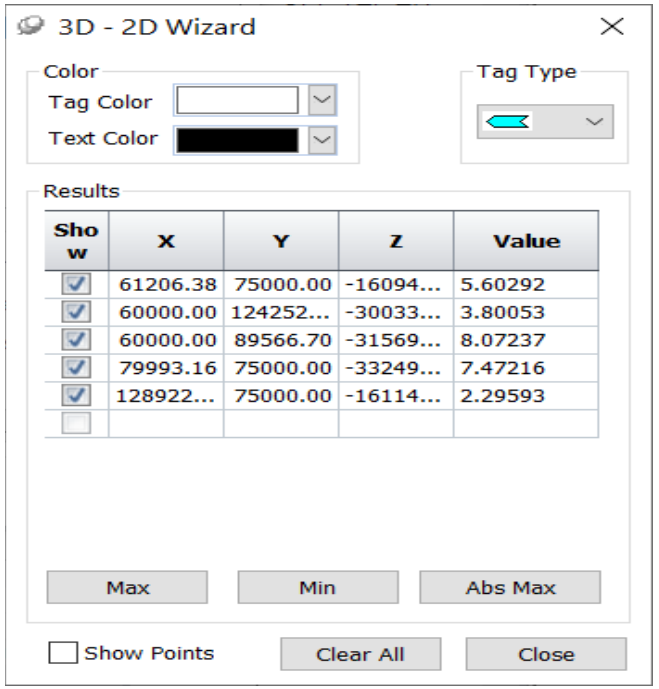
Max | Min | Abs Max | Clear All

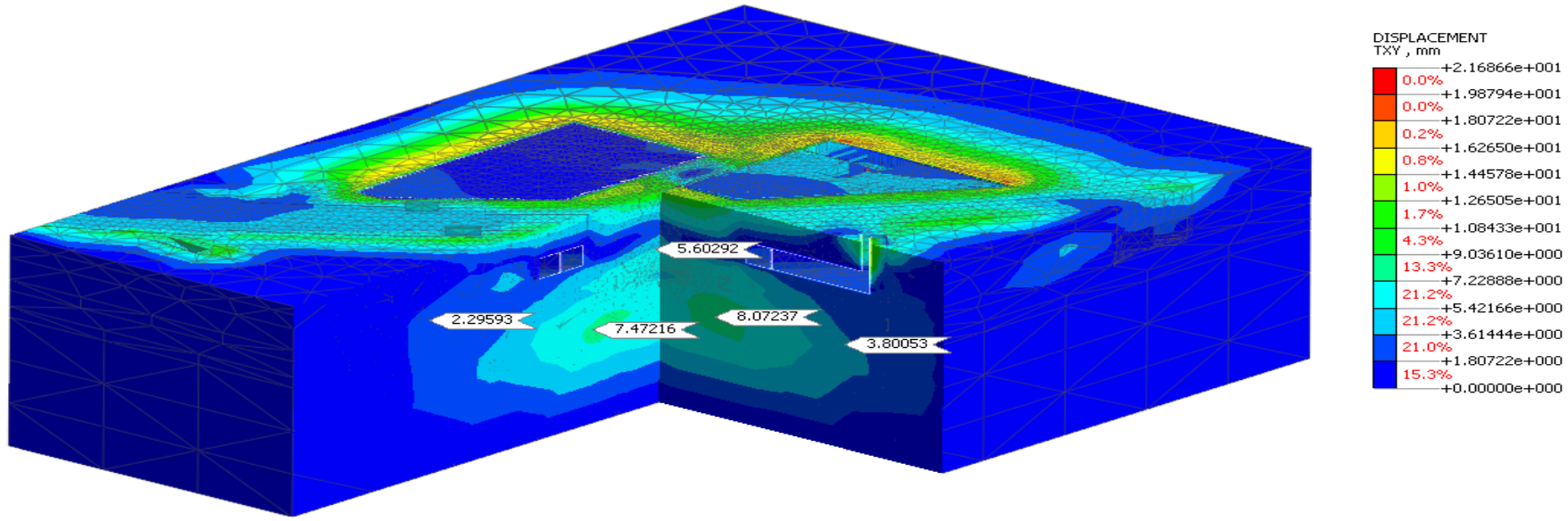
☐ Min/Max Value of Each Part

Close

RESULT - MULTIPLE METHODS FOR VERIFICATION

PLANE CLIPPING and PROBING





RESULT - ANALYSIS

REPORT and EXPORT

materials and properties with “**PDF**” format

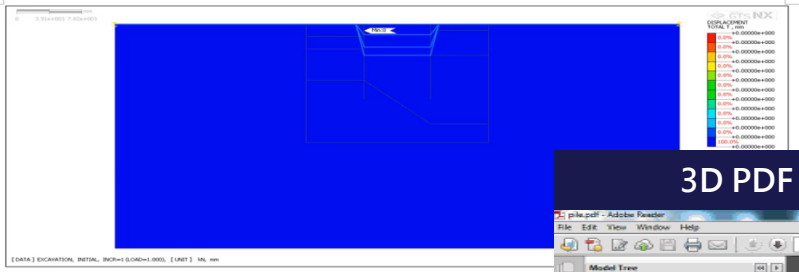
Material

Elastic

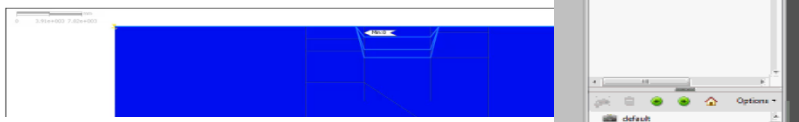
Name	E (kN/mm²)	Inc. of Elastic (kN/mm²)	Inc. of E Ref. Height (mm)	v	γ (kN/mm³)	Ko	Thermal Coeff. (1/[T])	Molecular Vapor Diffusion Coeff. (mm/sec²)	Thermal Diffusion Enhancement	Damping Ratio
	γ_sat (kN/mm³)	e_o	kx (mm/sec)	ky (mm/sec)	kz (mm/sec)	Ss (1/mm)	Conductivity (W/(mm·[T]))	Specific Heat (J/(ton·[T]))	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

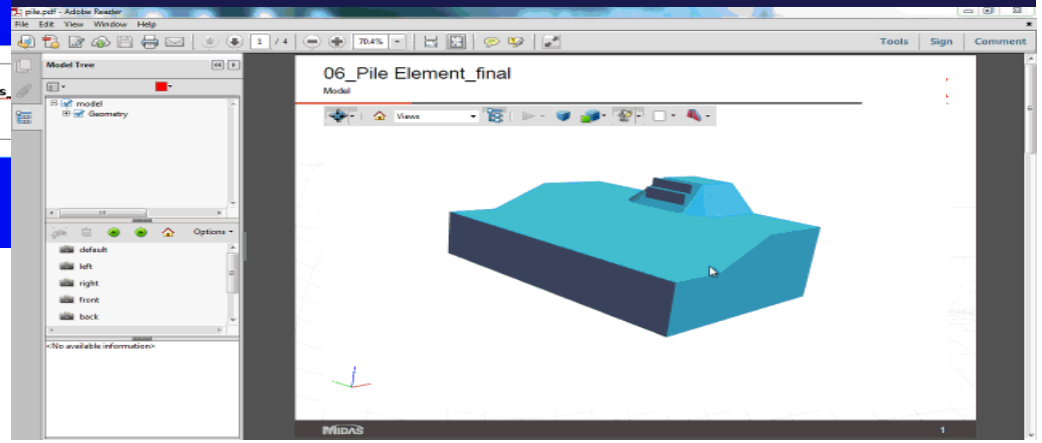
Results export with “**WORD**” format



EXCEPTION_INITIAL_INCR=1 (LOAD=1.000)_Displacements (V)_Current.png

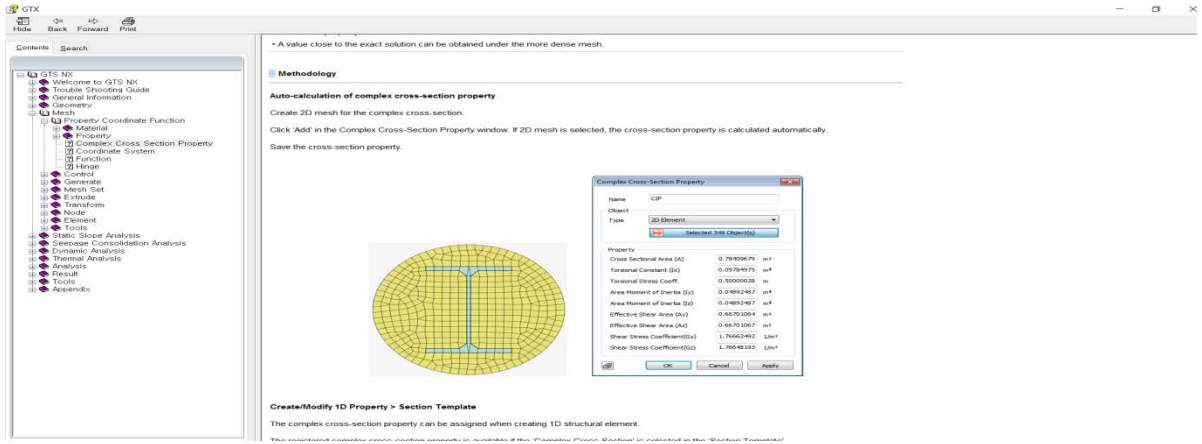


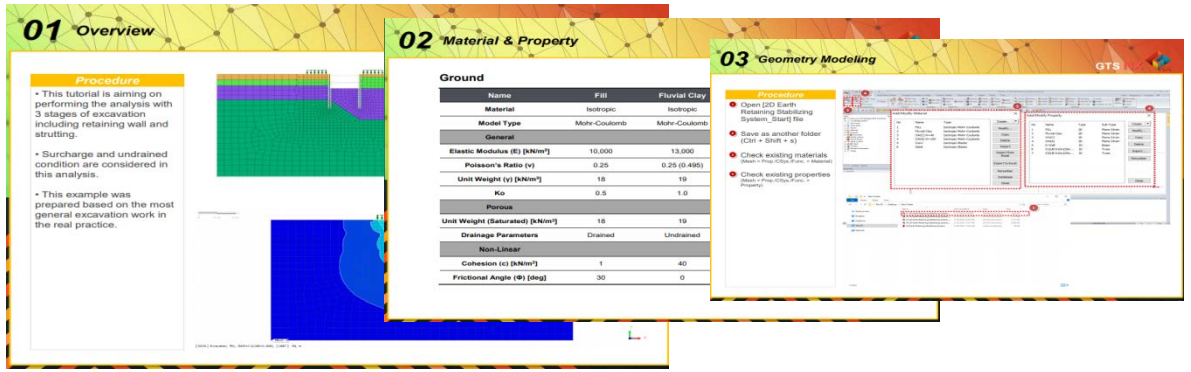
3D PDF Directly checking every results from 3D PDF



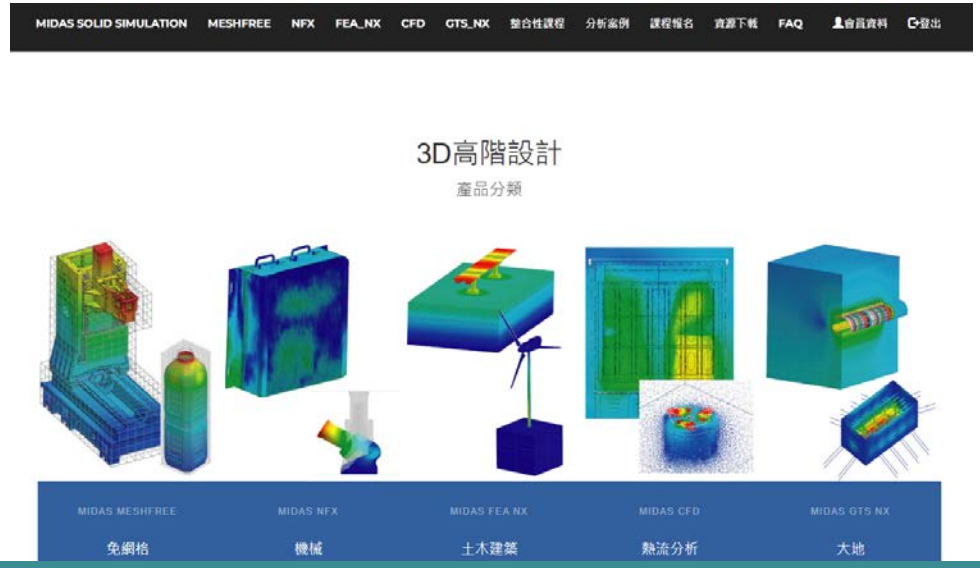
TECHNICAL SUPPORTS

Help manuals / tutorials with various topics





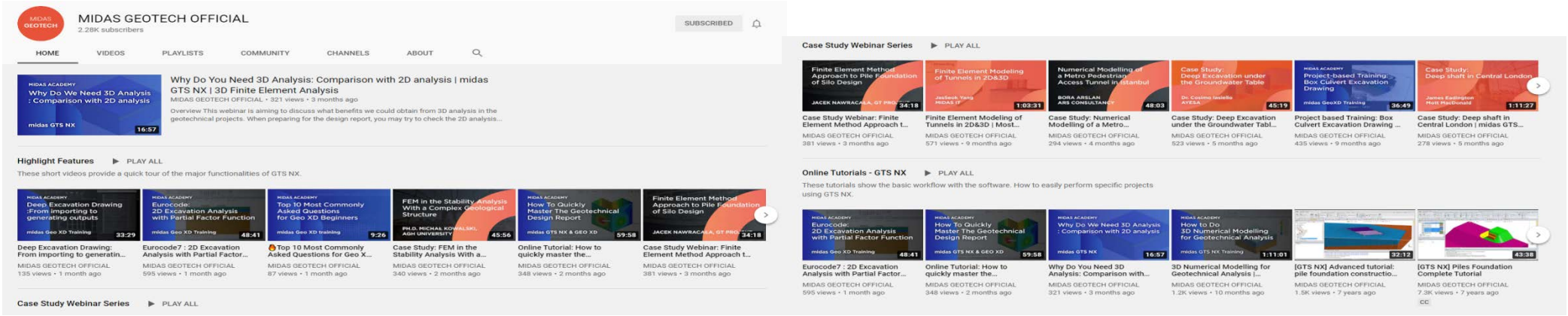
Taiwan MIDAS Solid-Simulation website



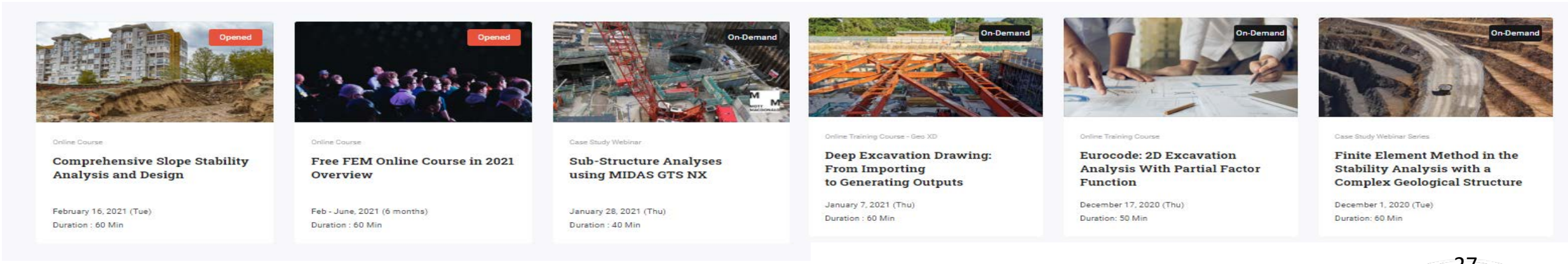
https://www.midasuser.com.tw/SolidSimulation/

TECHNICAL SUPPORTS

Online training videos



Online Courses / Case studies



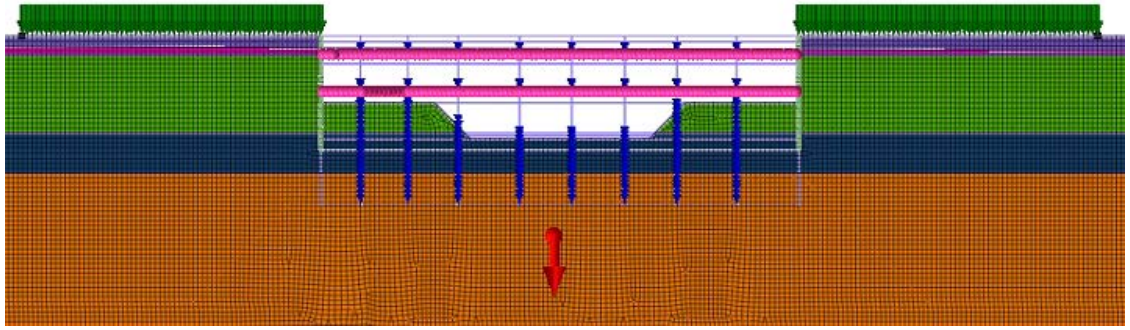
MIDAS Geotech has rich experience and know-how to grow the practical engineers

MIDAS

FEA

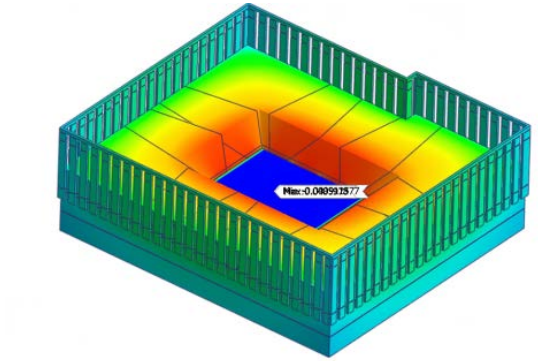
CASE STUDIES

2D分析

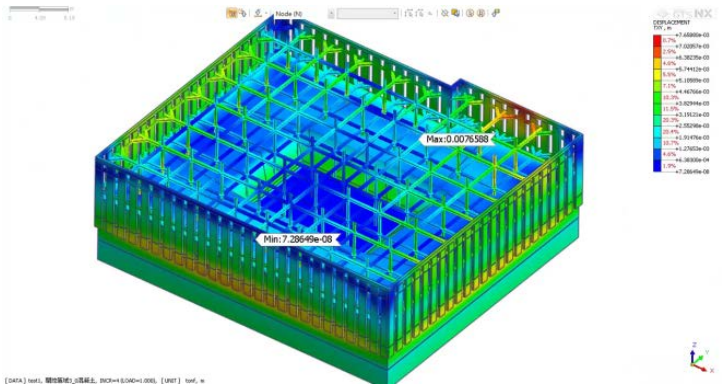


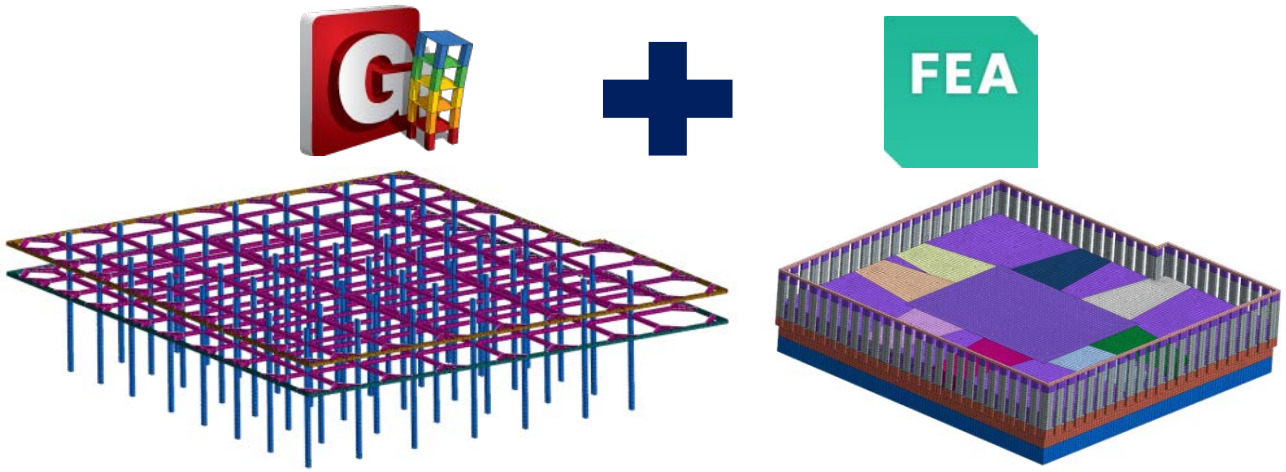
2D 建模只考慮模型的一個平面，縱向支撐、水平支撐和傾斜梁無法建模。

開挖面最大垂直位移 (m)

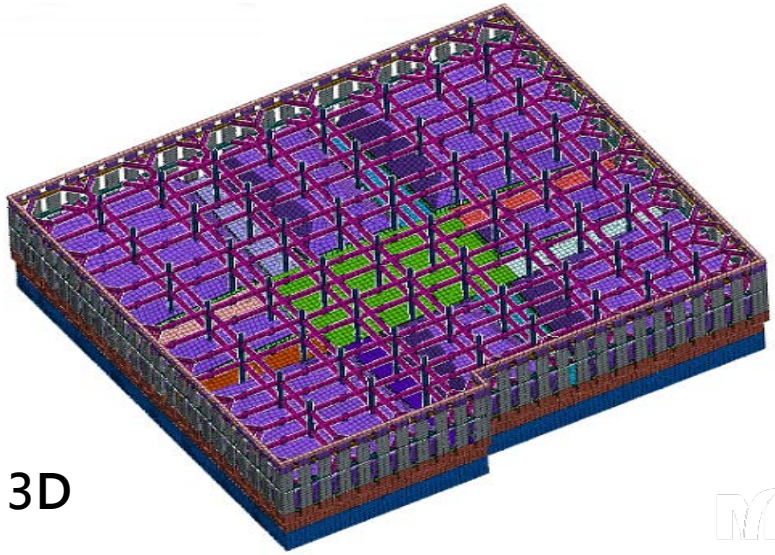


開挖面最大水平位移 (m)



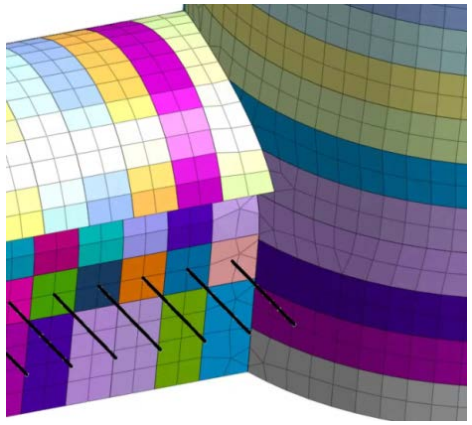


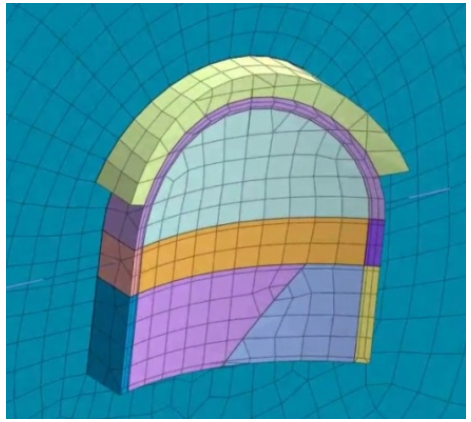
GEN+FEA 3D



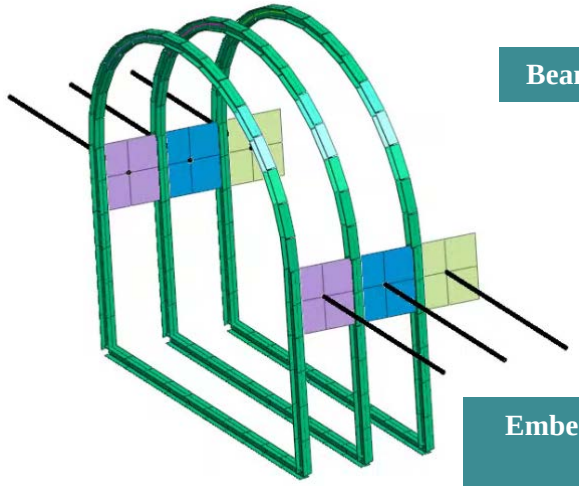
FEA NX NATM實例

Mixed Mesh with Fully Compatible Nodes





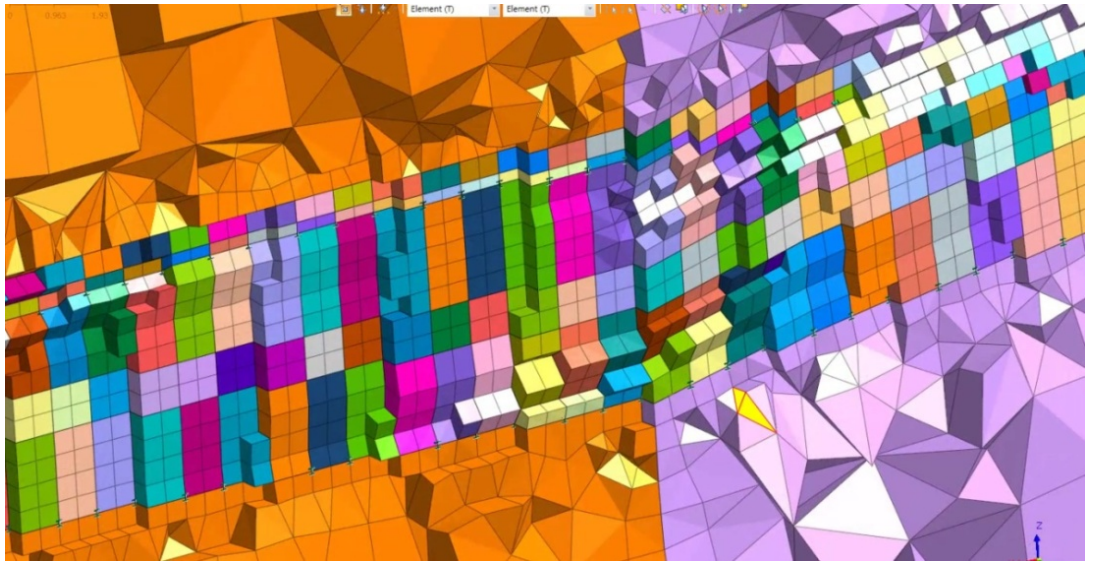
Shell Element(傳力裝置)



Beam Element(H型鋼)

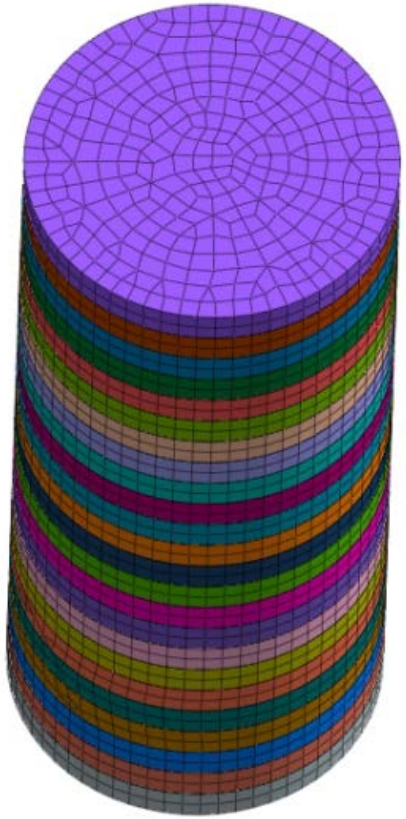
Embedded Truss Element (Rock Bolt)

FEA NX 混合網格/全共點建模

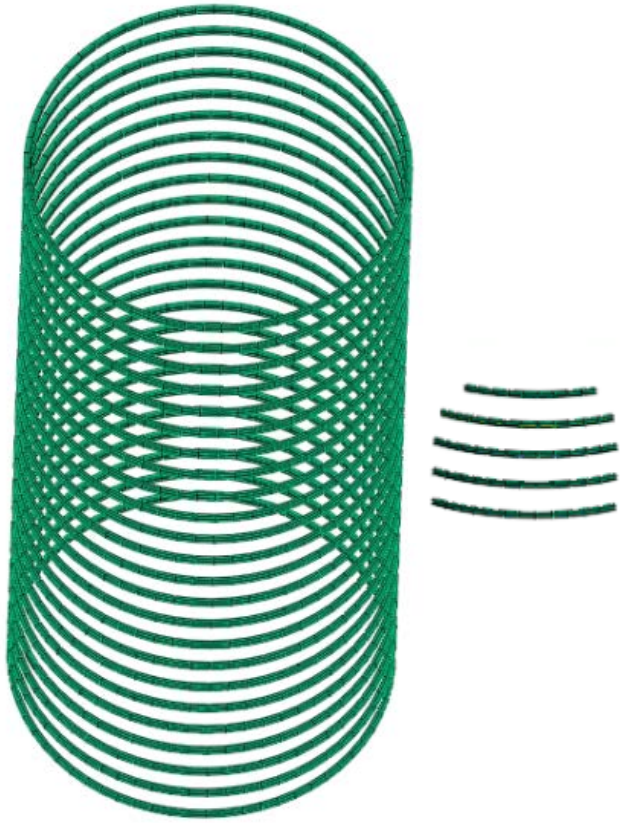


FEA NX Case Study: NATM Tunnel Analysis

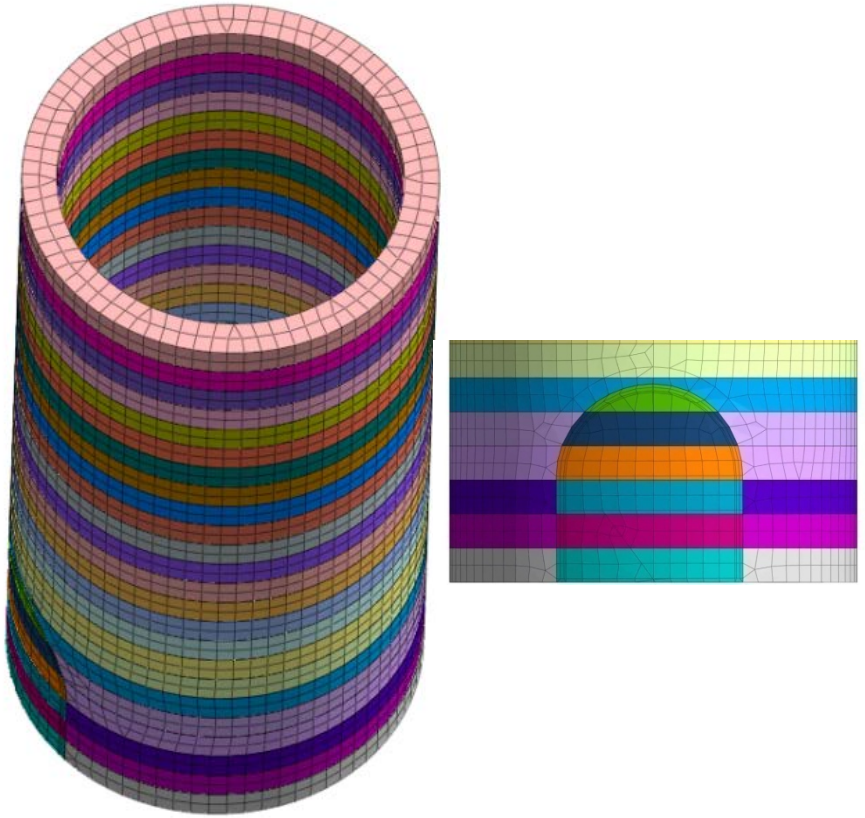
VERTICAL SHAFT



Vertical shaft excavation



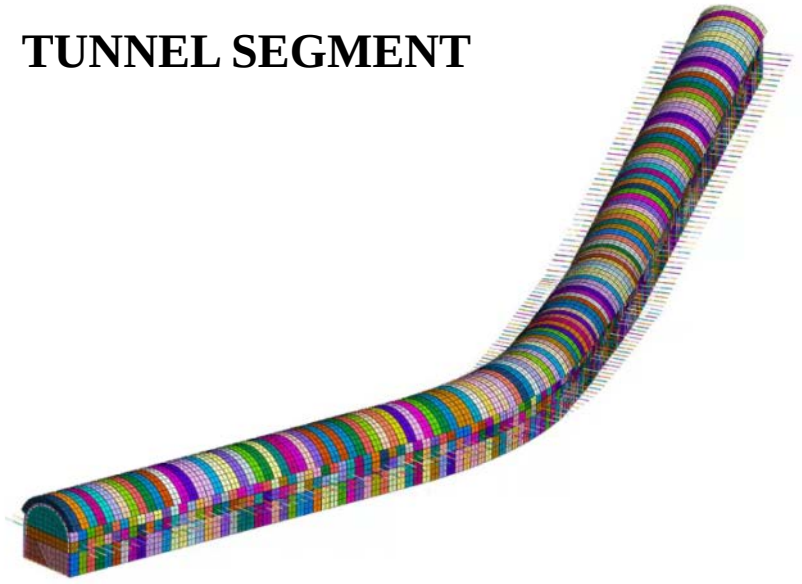
Steel Bracing



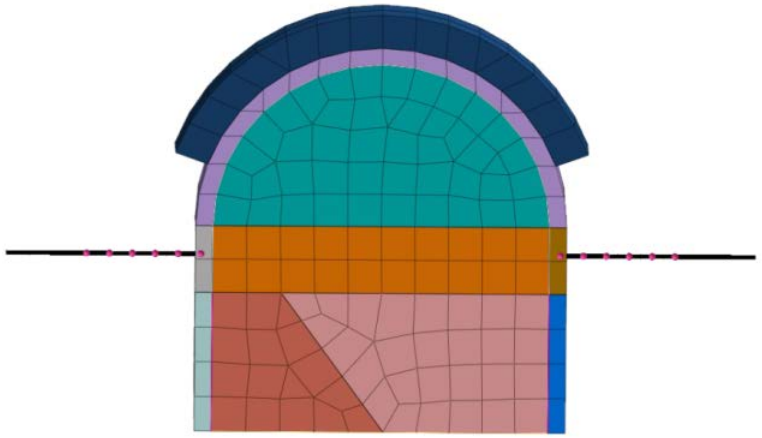
Backfill

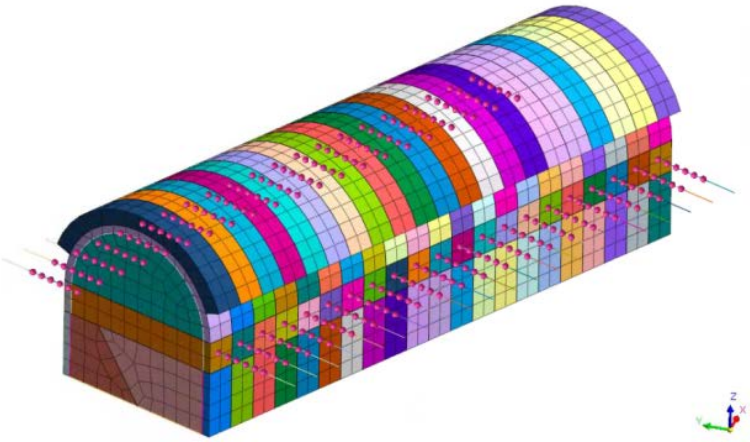
FEA NX Case Study: NATM Tunnel Analysis

TUNNEL SEGMENT

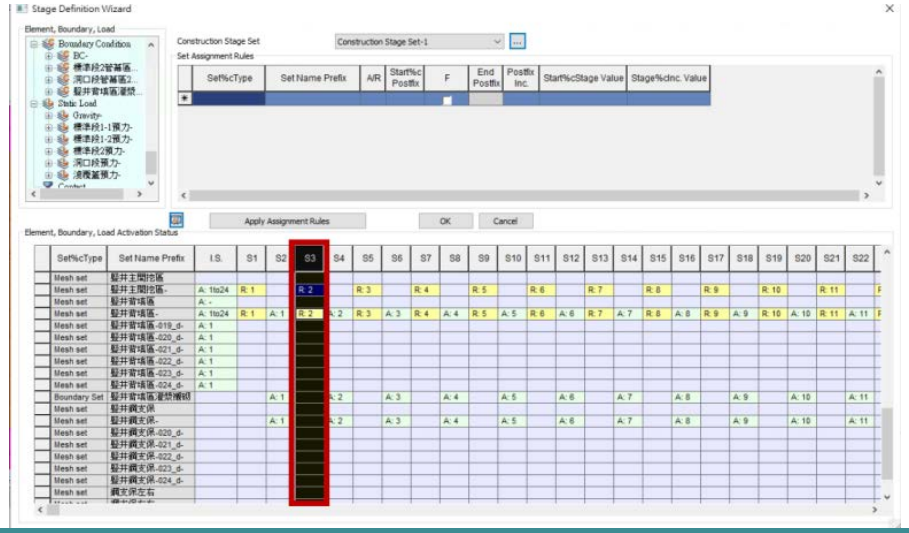


Anchors (Prestressed Application)

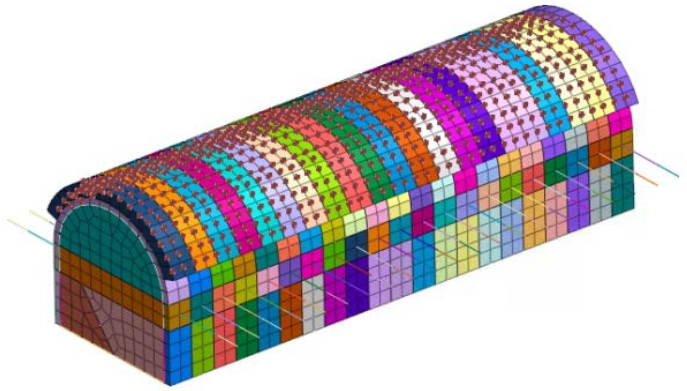




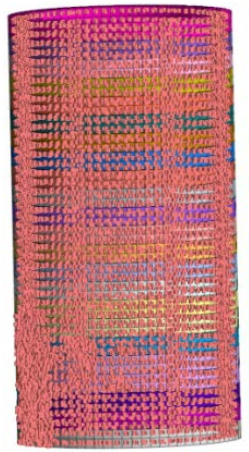
Construction Stage



管幕區變更材質



Backfill Grouting and Lining



FEA NX SLOPE STABILITY ANALYSIS

2D分析 - 方法 1

Limit Equilibrium Method(LEM)

2D分析 - 方法 2

Stress Analysis Method (SAM)

2D分析 - 方法 3

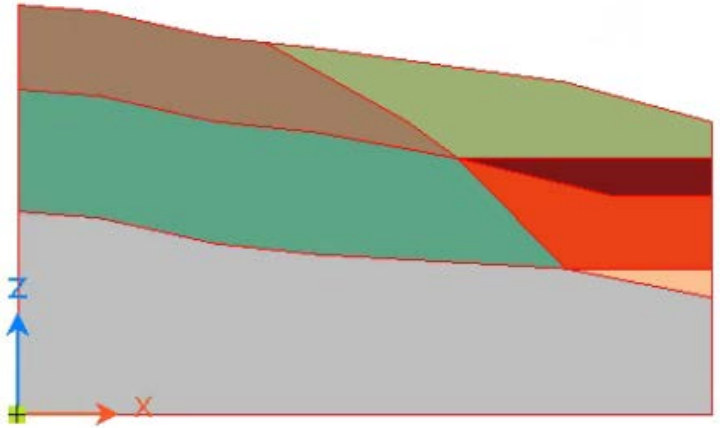
Strength Reduction Method (SRM)

3D分析

Strength Reduction Method (SRM)

FEA NX 2D SLOPE STABILITY ANALYSIS

Post-Excavation Slope Stability Calculation

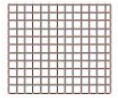


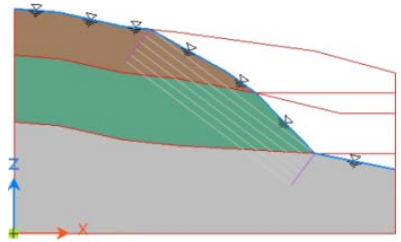
Weathered Soil

Weathered Rock

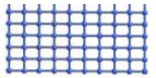
Soft Rock

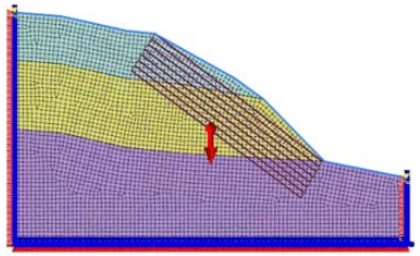
2D分析-方式1.LEM



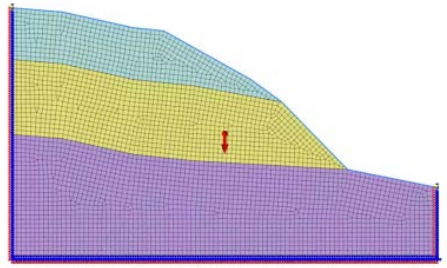


2D分析-方式2.SAM



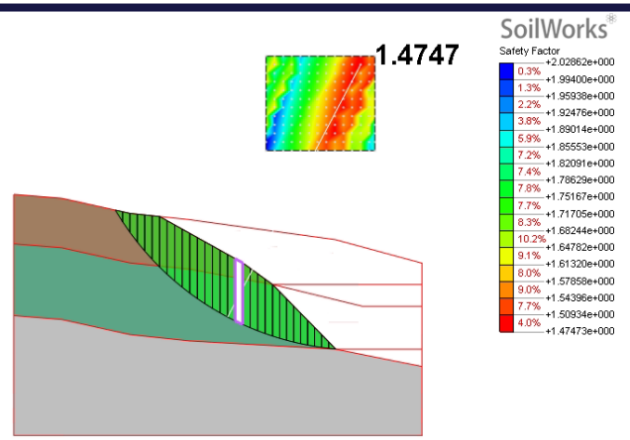


2D分析-方式3.SRM

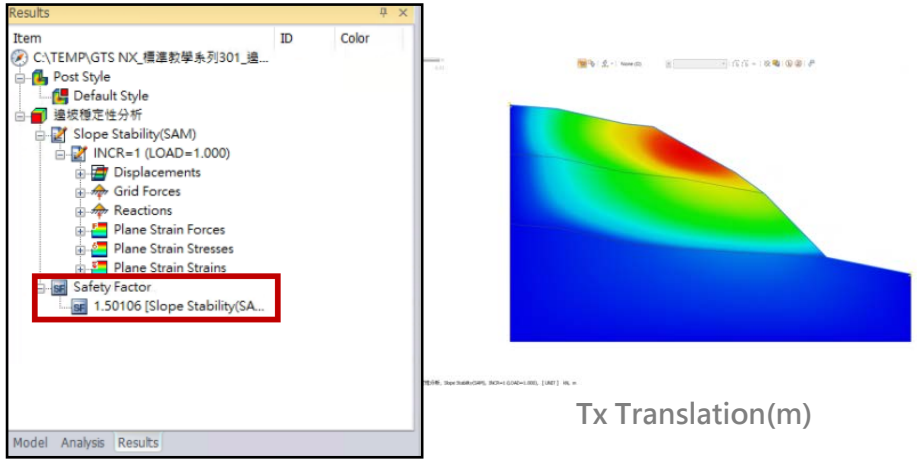


SRM gradually reduces the shear strength parameters (cohesion c and friction angle φ) of the slope foundation material until the analysis becomes unstable (diverges). At this point, slope failure is assumed to occur. The maximum strength reduction factor at this divergence point is taken as the minimum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

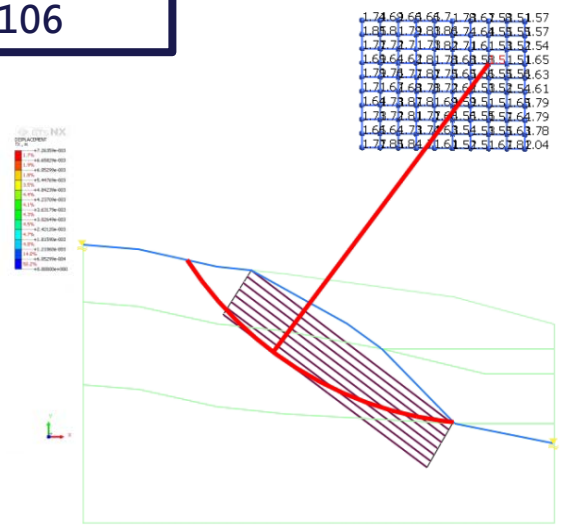
2D分析-方式1
LEM計算之安全係數1.4747



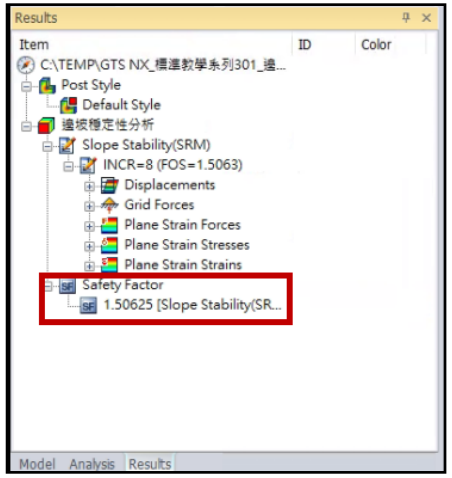
2D分析-方式2
SAM計算之安全係數1.50106



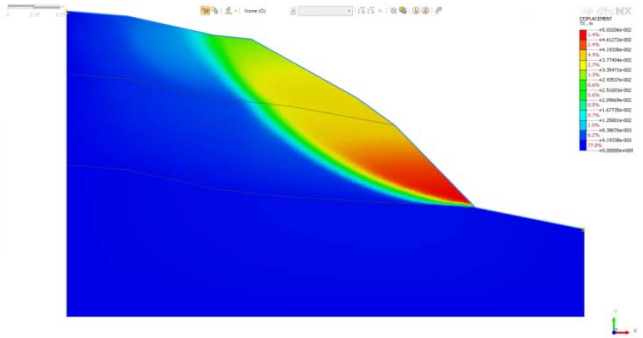
Tx Translation(m)



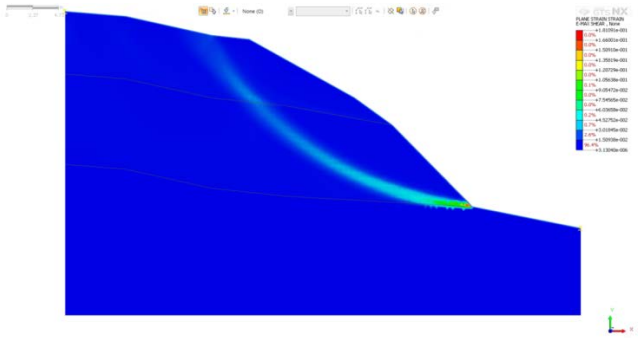
2D分析-方式3
SRM計算之安全係數1.50625



2D分析-方式3
SRM透過水平變形和最大剪切應變判斷破壞面



Tx Translation(m)

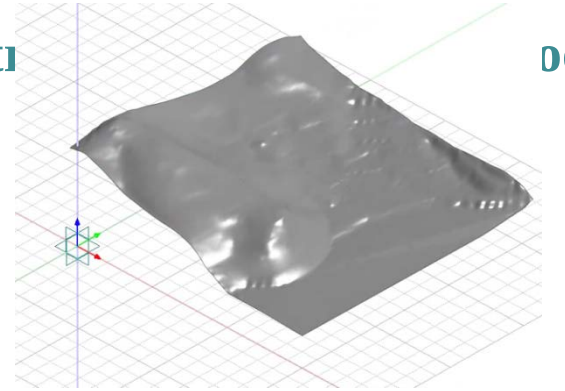


Maximum Shear Strain

GTS NX 3DSlope Stability Analysis

3DTerrain Surface Features

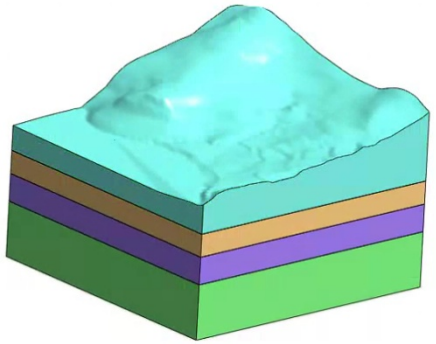
St



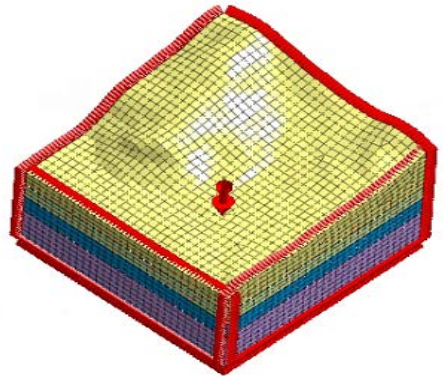
od (SRM)

300m ×300m

3DTerrain Solid Features

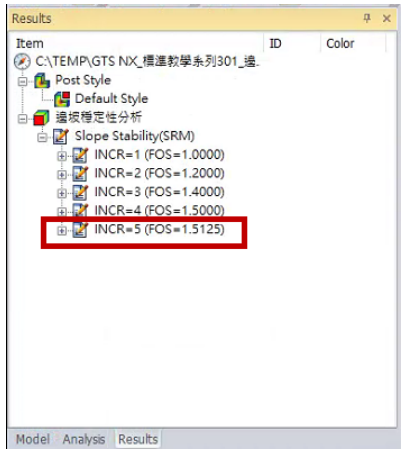


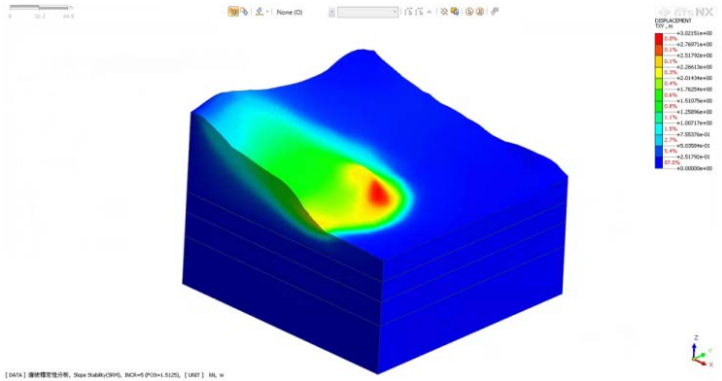
Self-Weight



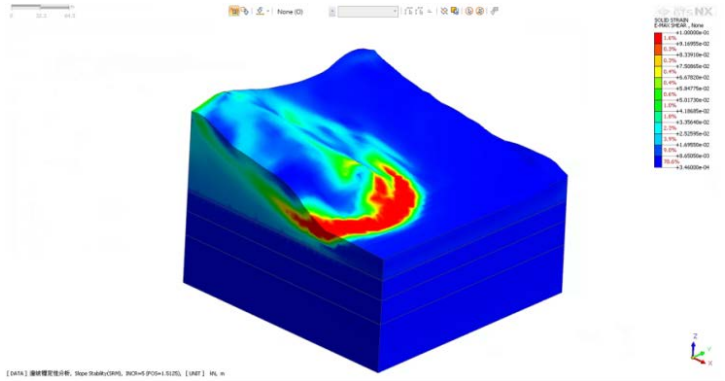
3D分析

SRM計算之安全係數1.5125





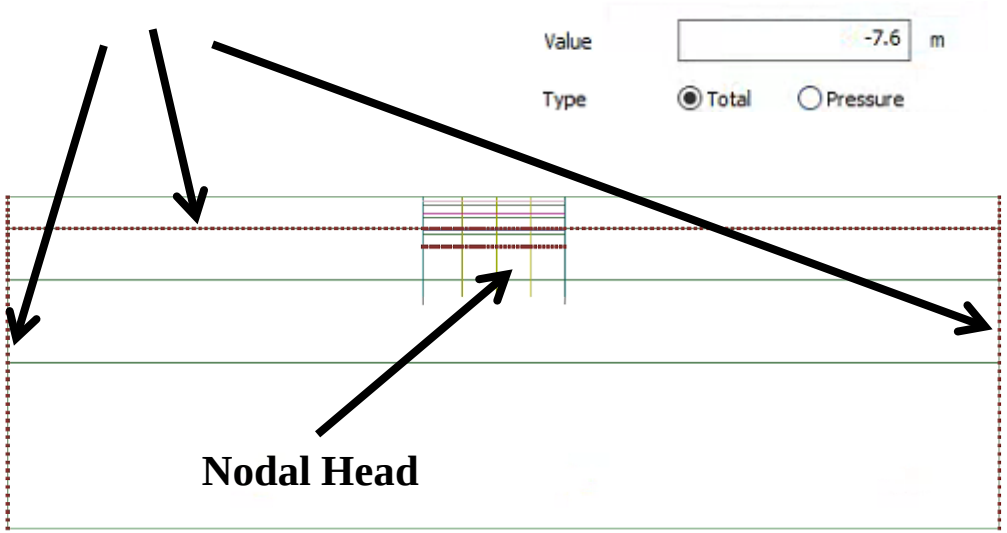
Tx Translation(m)

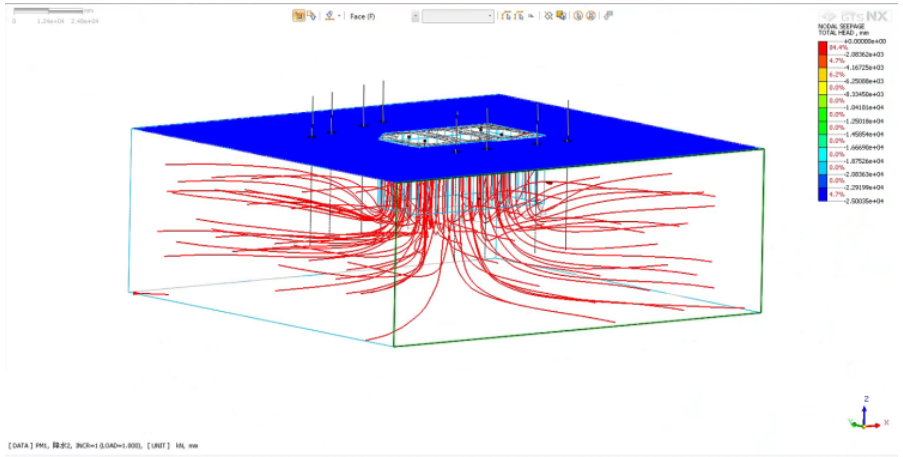


Maximum Shear Strain

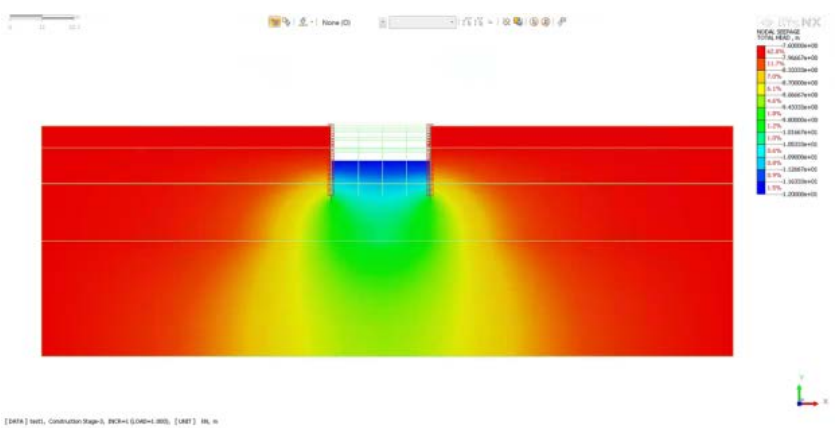
FEA NX GROUNDWATER SIMULATION

Nodal Head

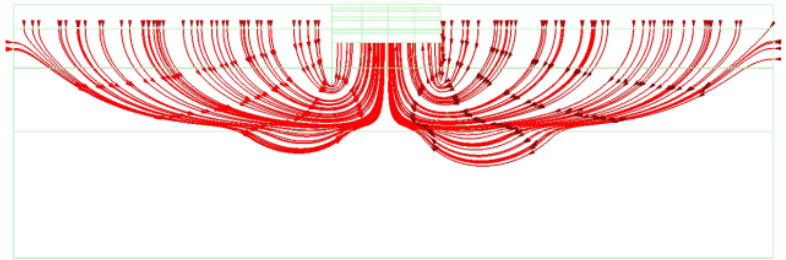




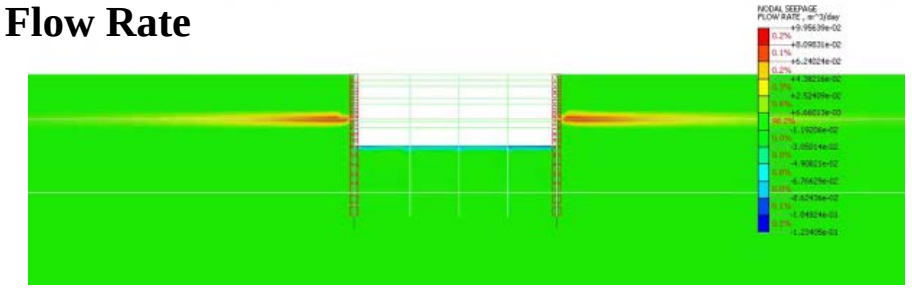
Total Head



Flow Path

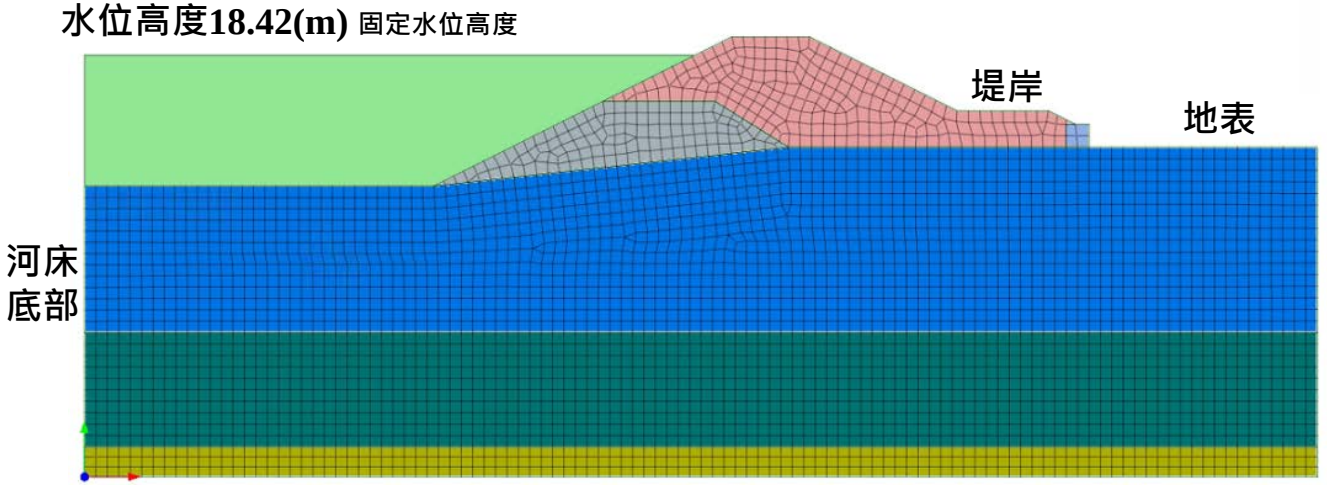


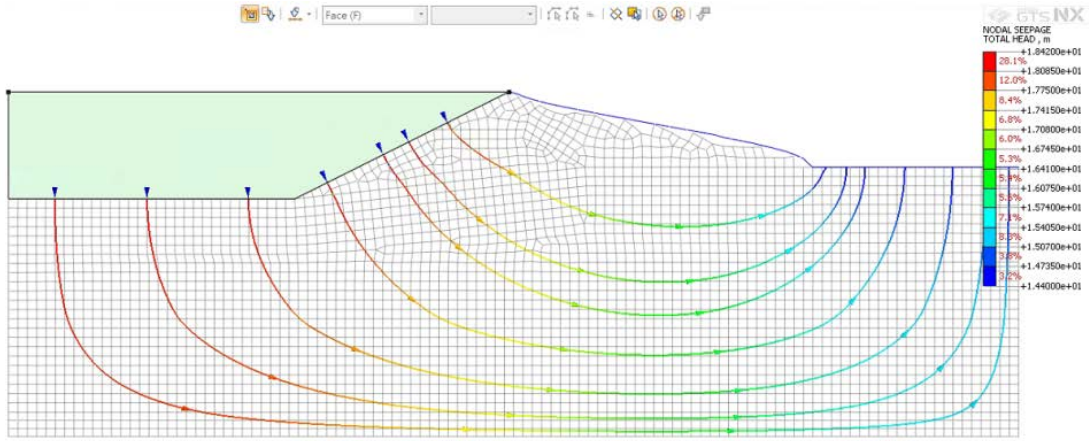
Seepage Flow Rate



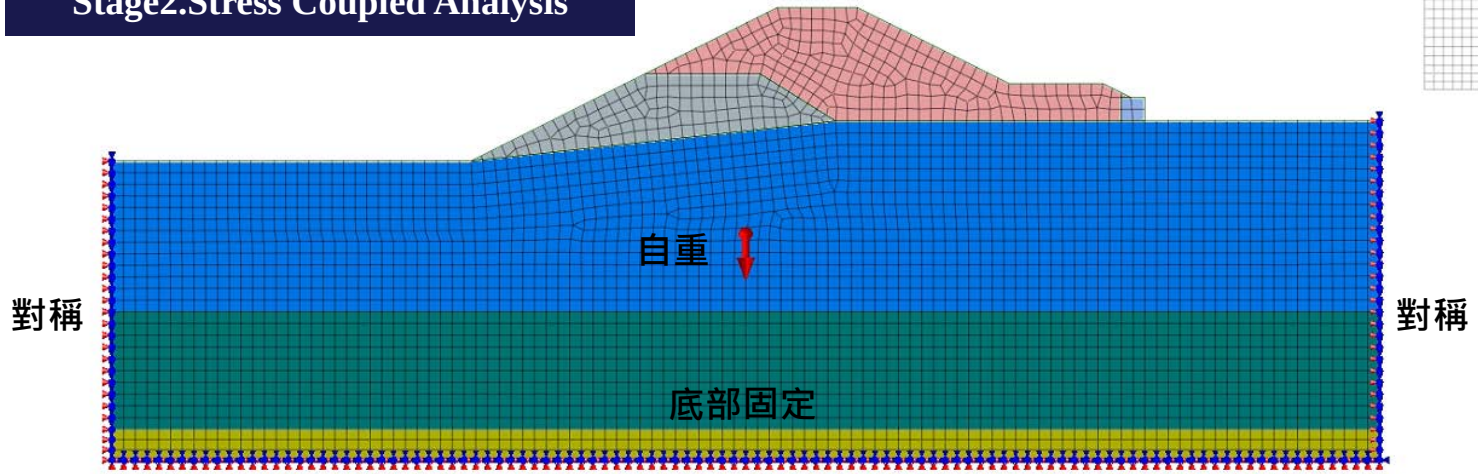
FEA NX Seepage-Stress Coupled Analysis

Stage1.Seepage Analysis



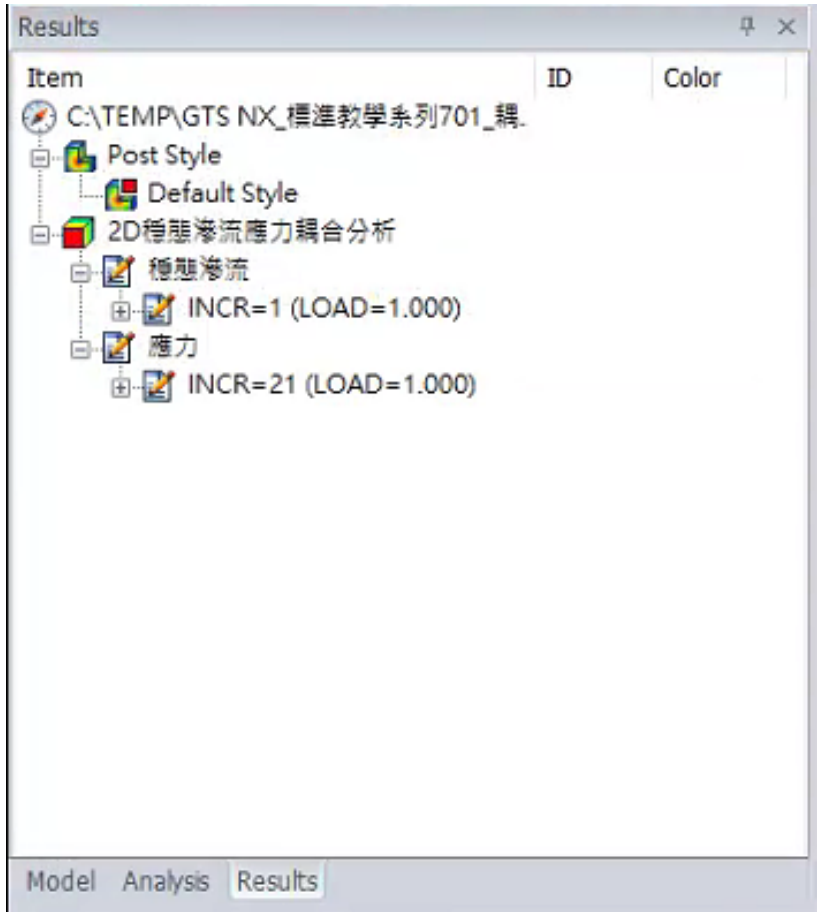


Stage2.Stress Coupled Analysis



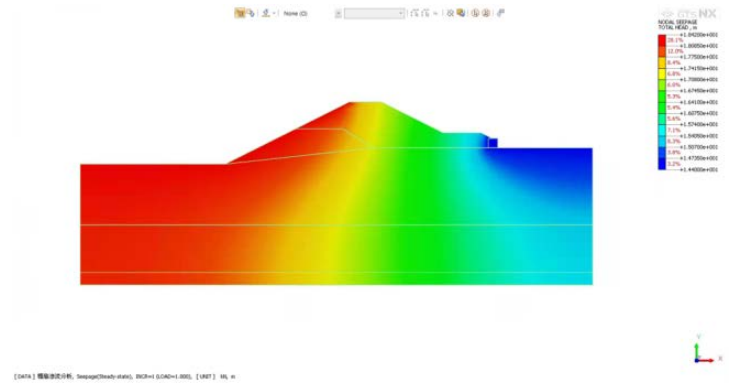
FEA NX Seepage-Stress Coupled Analysis

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

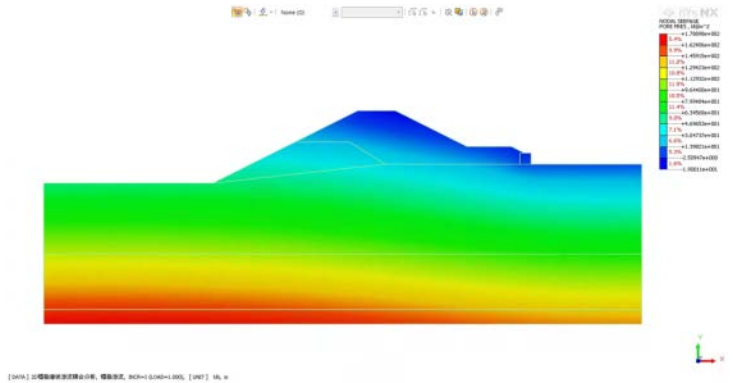


滲流結果

Total Head(m)

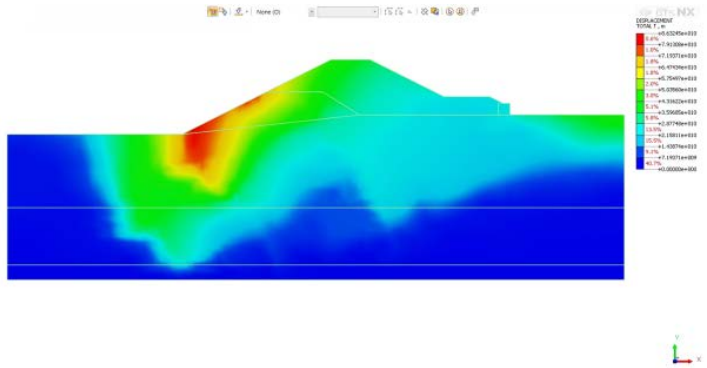


Pore Pressure(KN/m²)

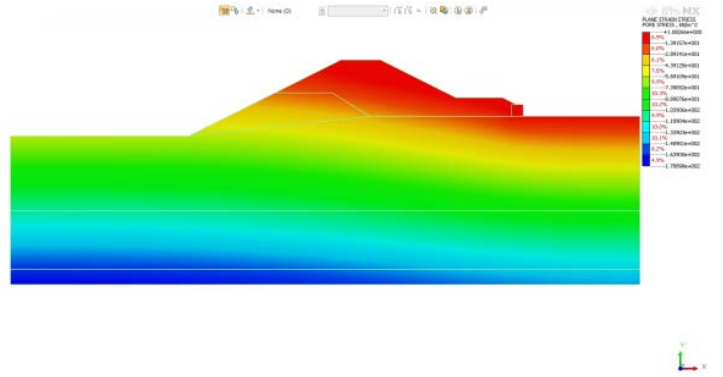


應力結果

Displacement Total(m)

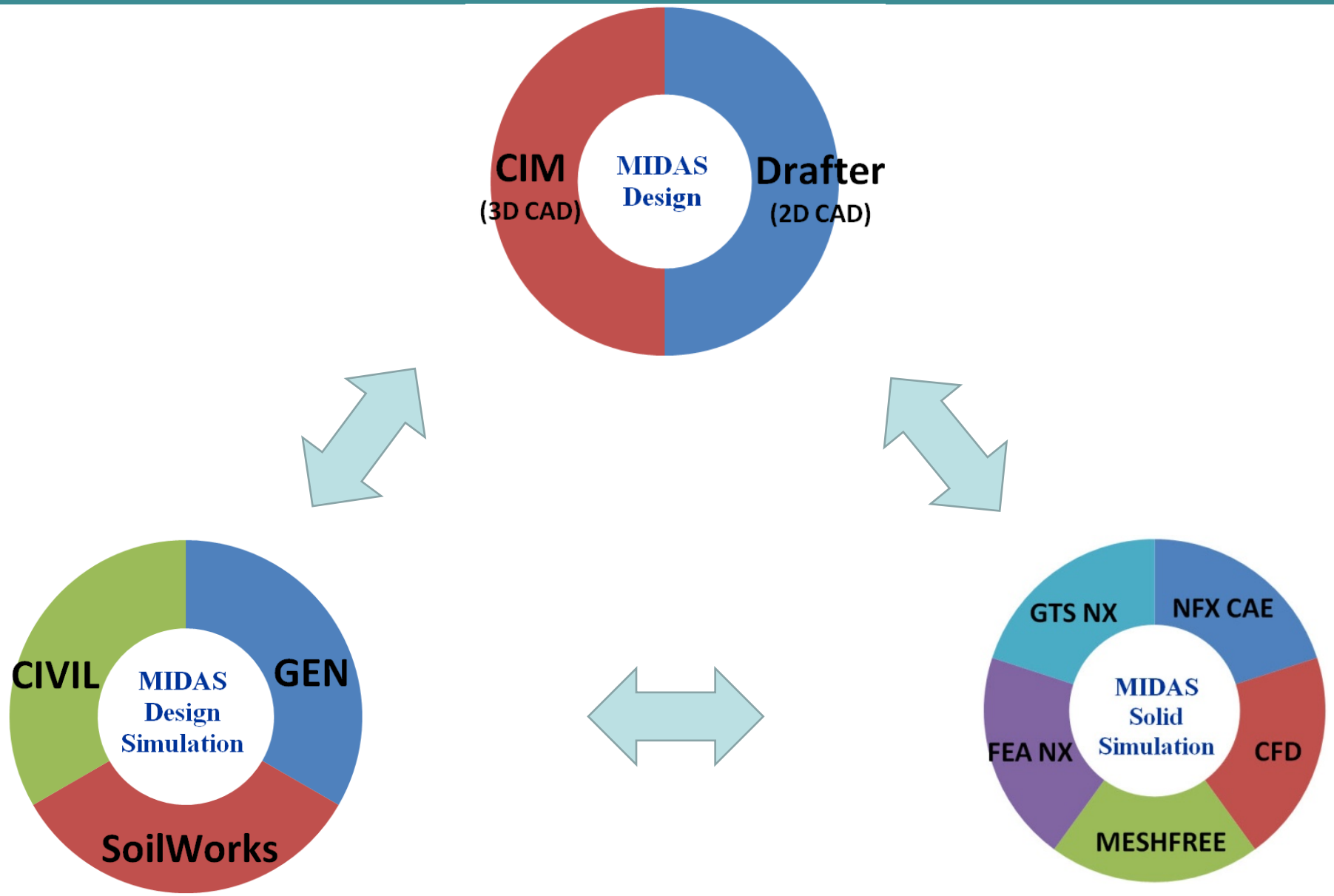


Pore Stress(KN/m²)



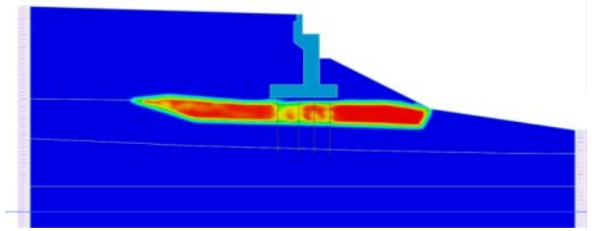
SYSTEM INTEGRATION

MIDAS INTEGRATION

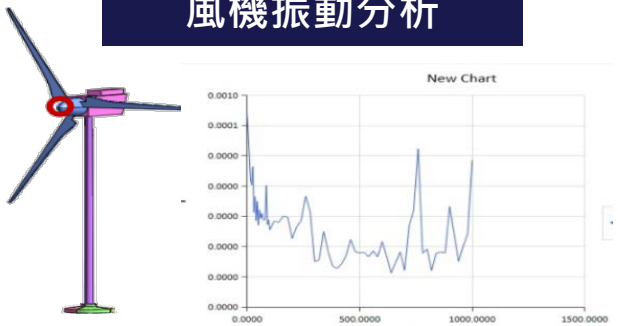


Solid Total Solution

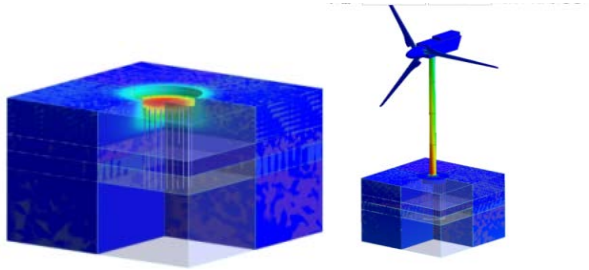
土壤液化分析



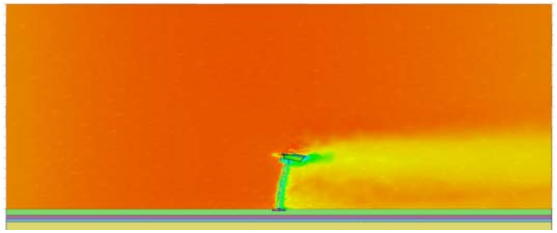
風機振動分析



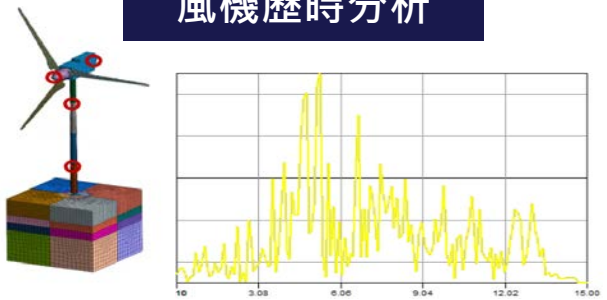
風機安裝施工階段分析

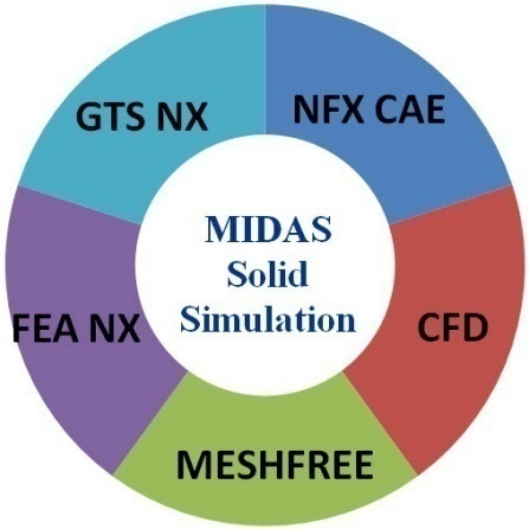


風機流固耦合分析

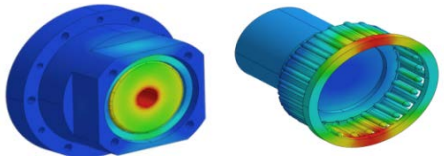


風機歷時分析

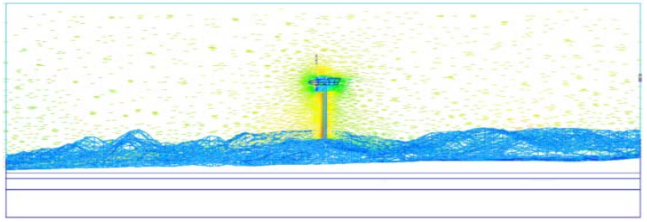




齒輪組分析

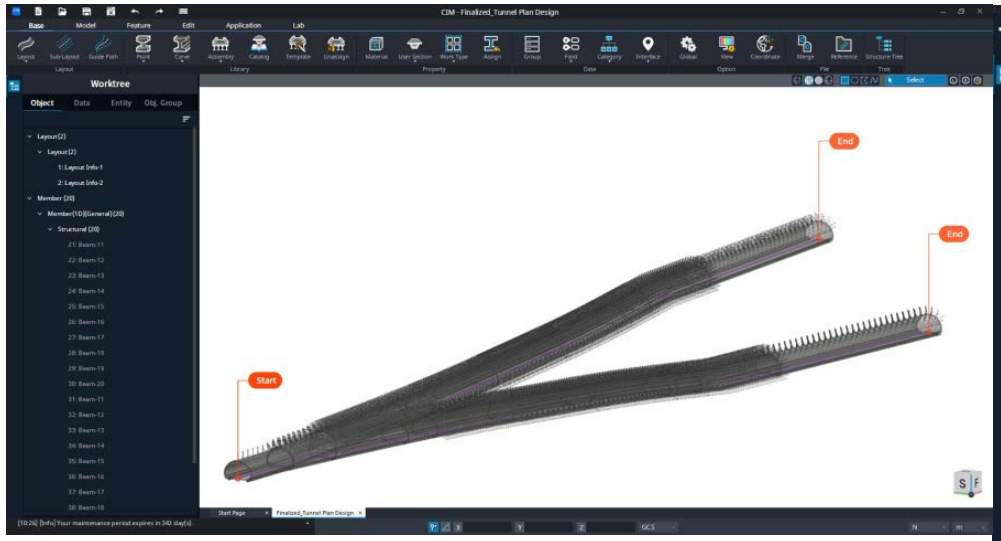


風機地形風場分析

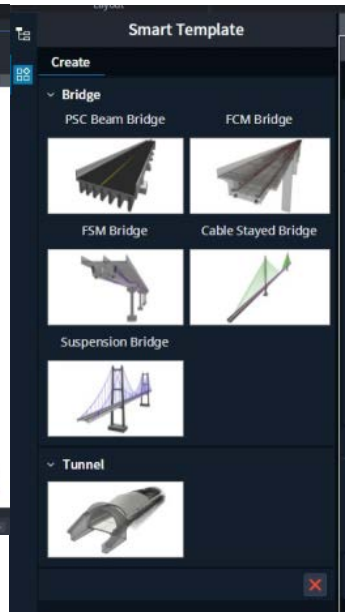


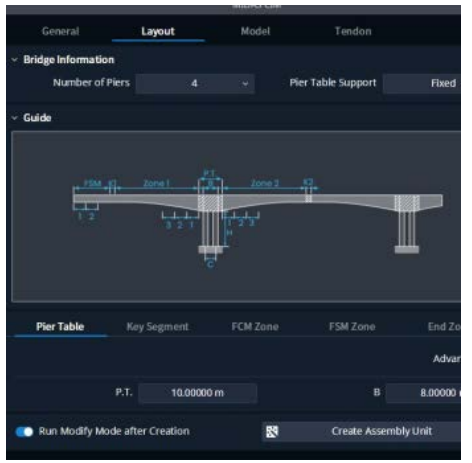
CIM + FEA/GTS 3D Model Integration

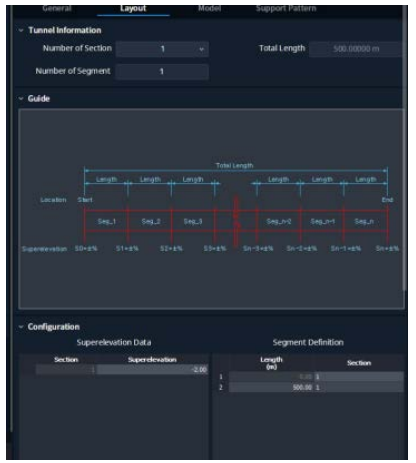
CIM-3DAutomatic Model Adjustment Along Alignment



Bridge & Tunnel Wizard

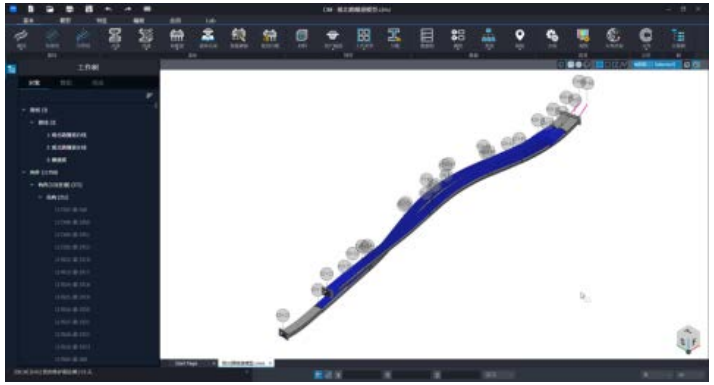


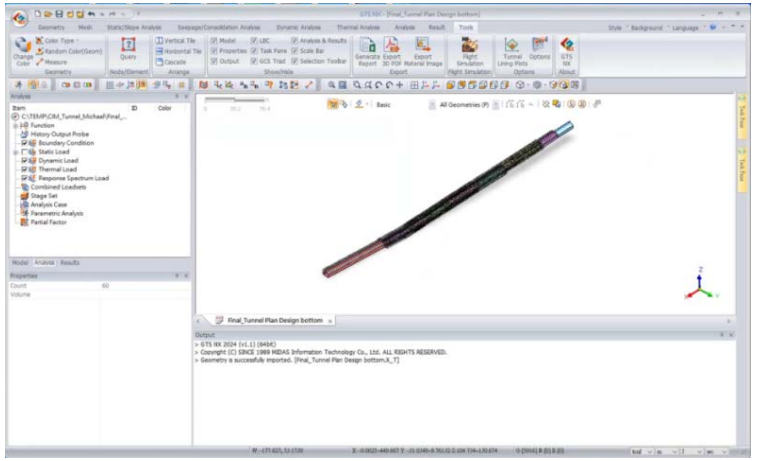


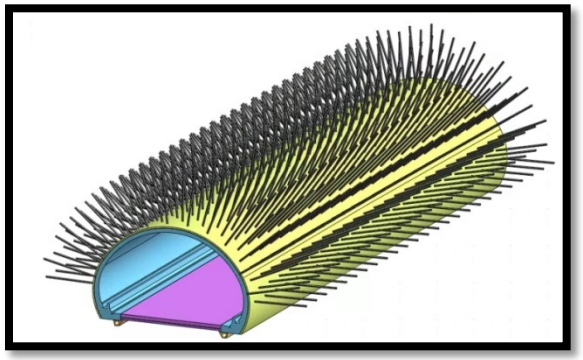


CIM+GTS 3D Model Integration

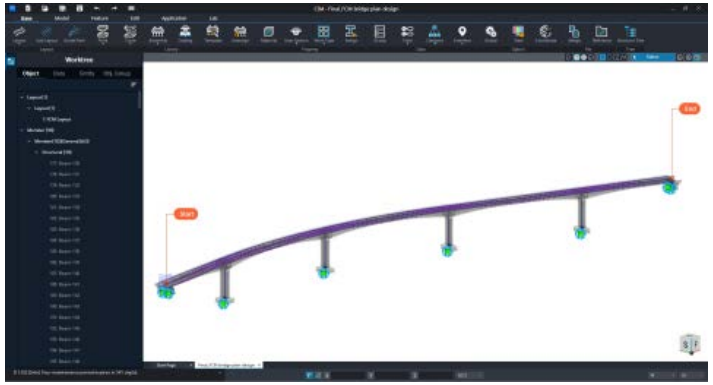
Tunnel

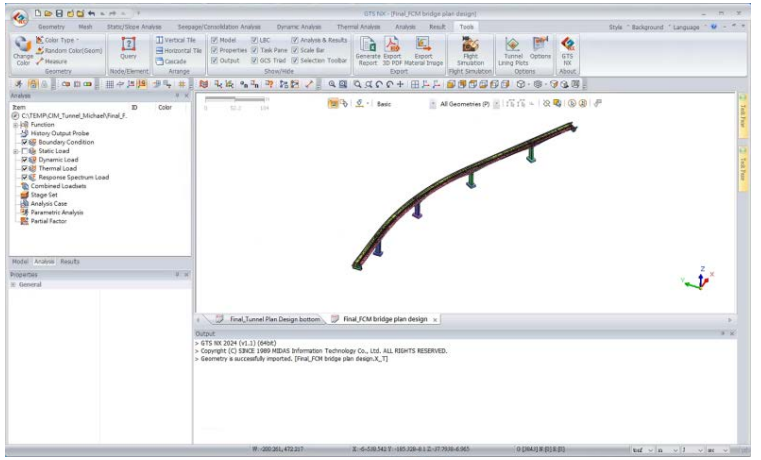


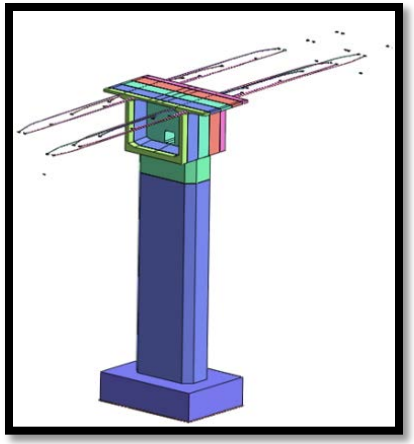




Bridge

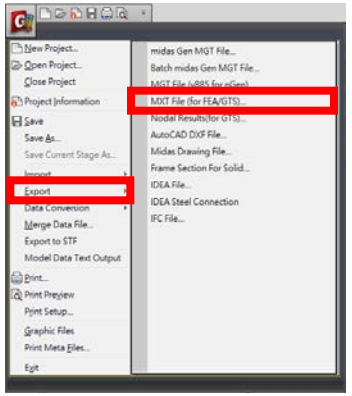


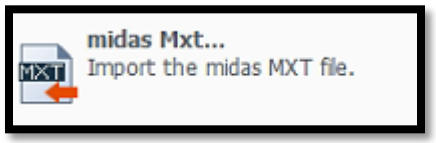


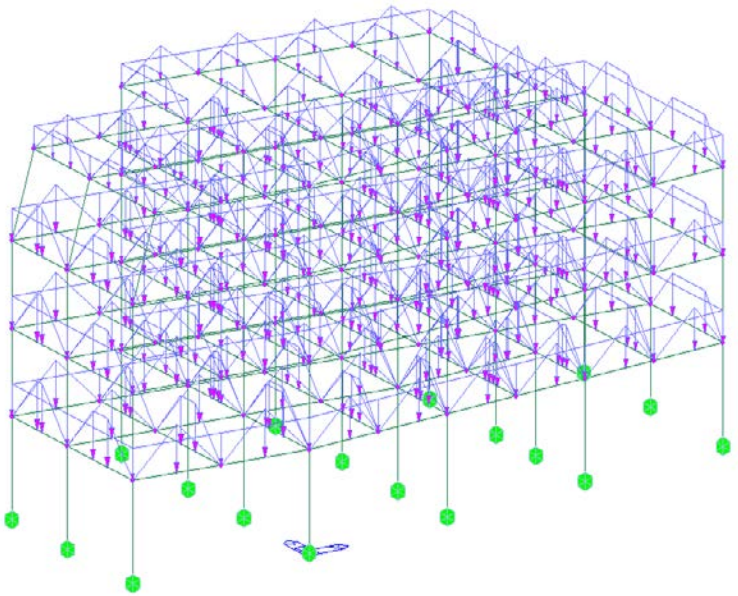


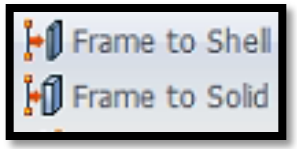
CIM&GTS NX
Direct Conversion of Solid Features

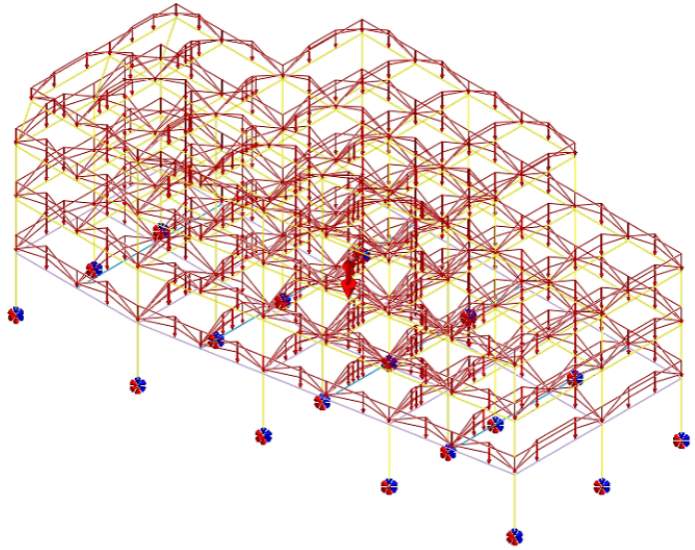
FEA or GTS NX & Gen | Structural Interaction Analysis



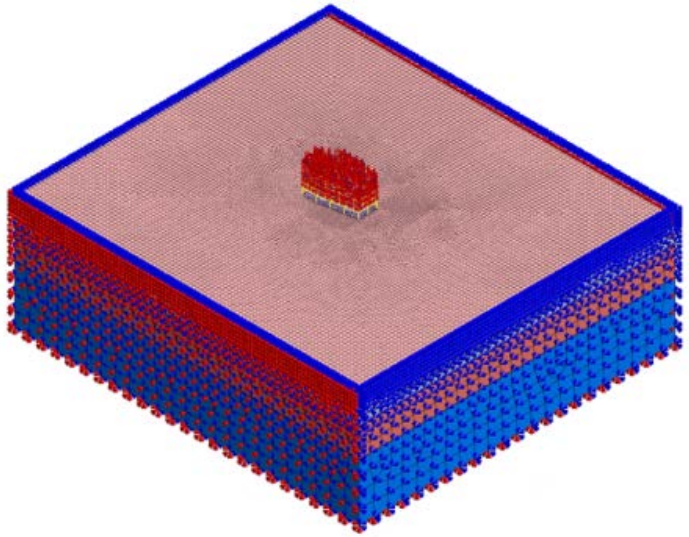


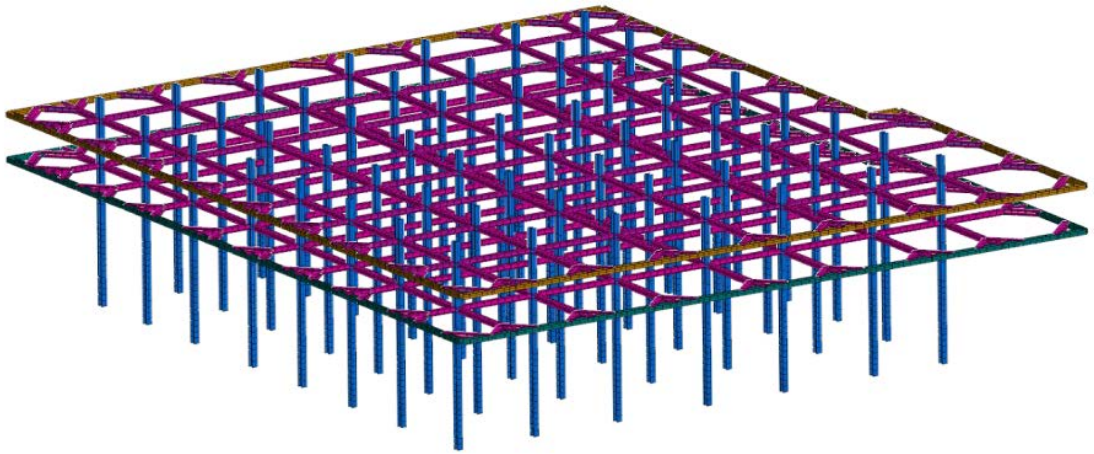


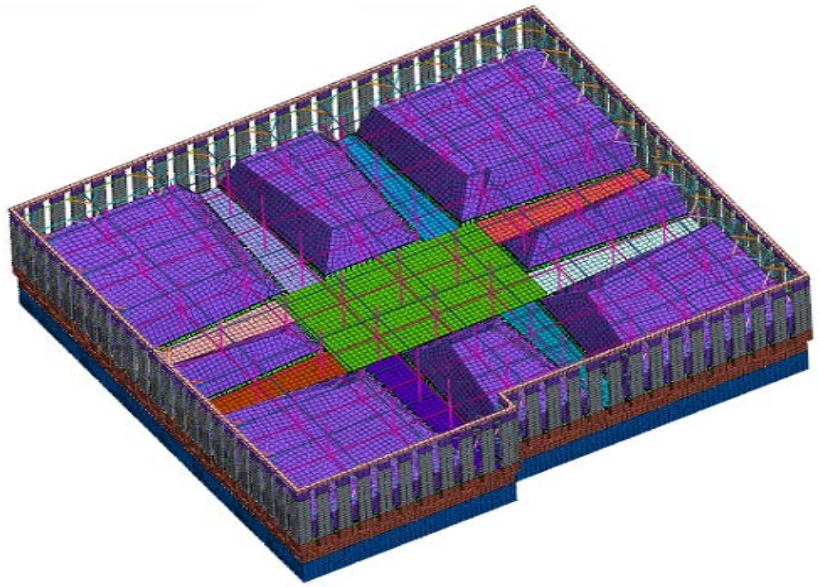


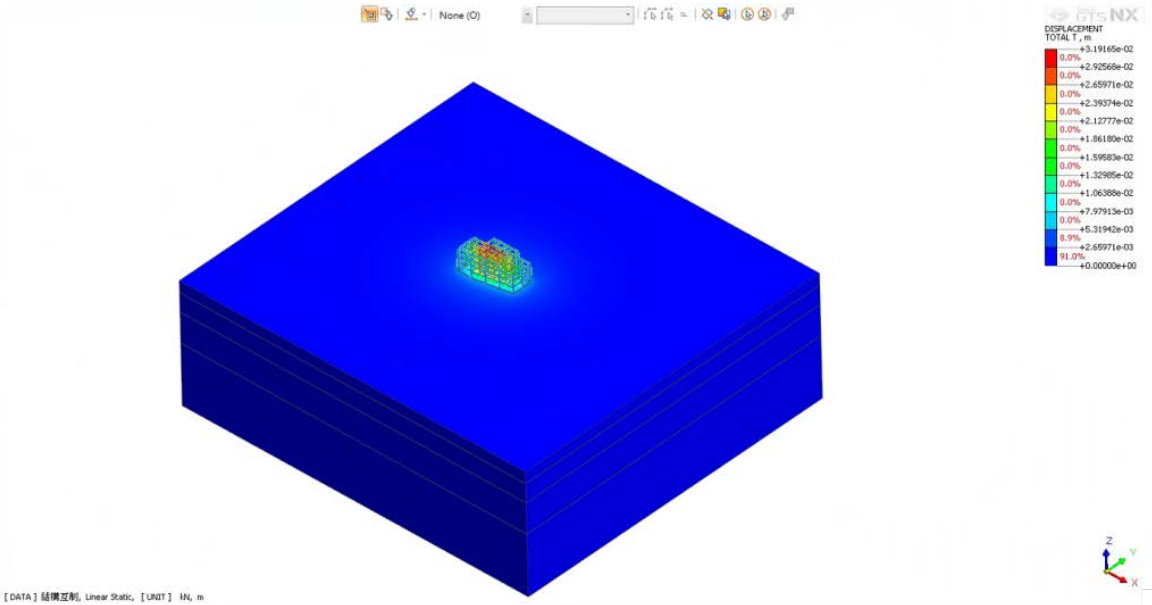


FEA or GTS NX & Gen | Structural Interaction Analysis



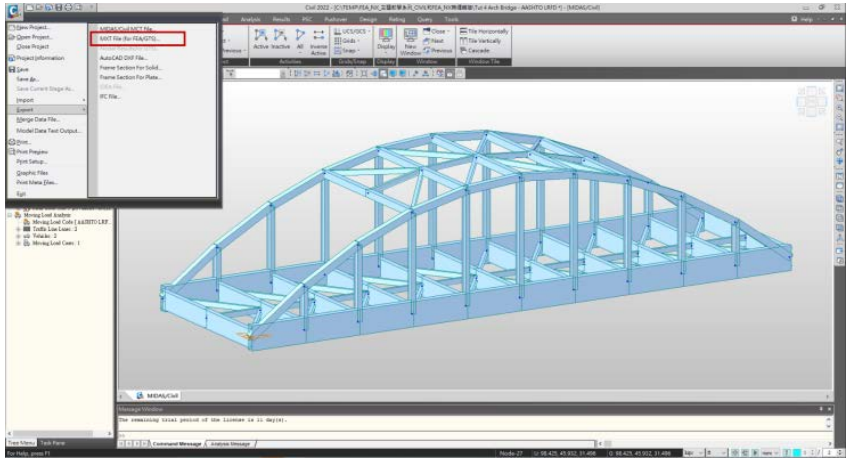


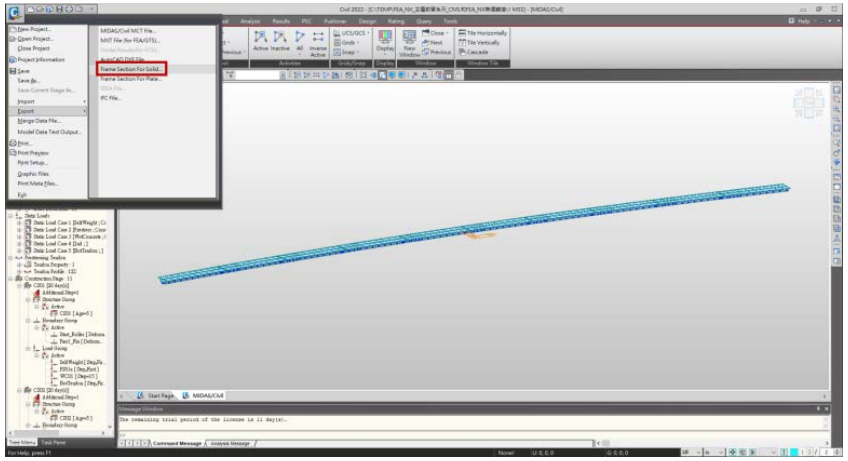




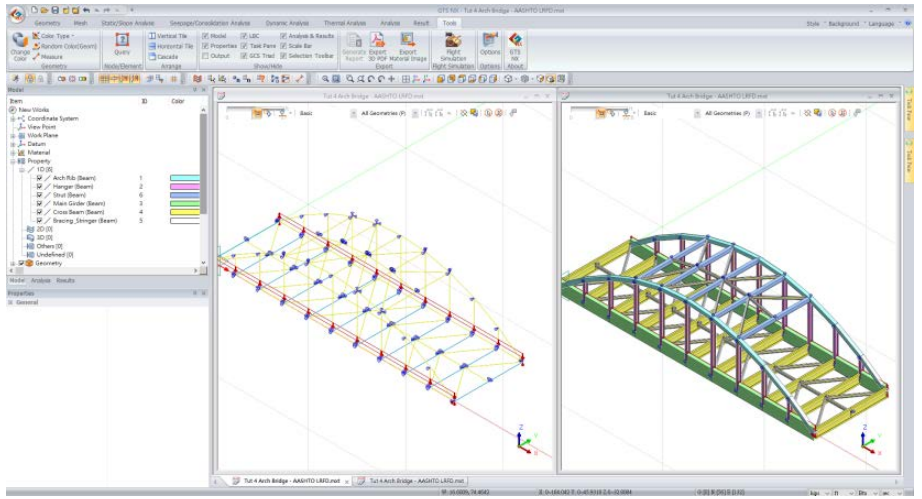
FEA / GTS NX+CIVIL Smooth Conversion

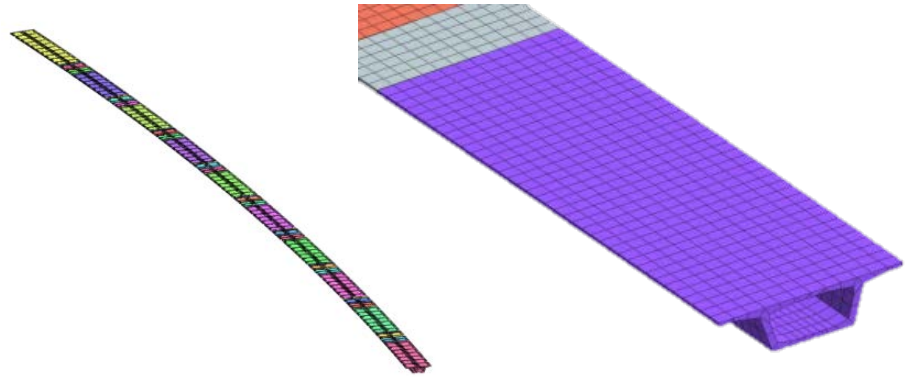
匯出MXT Files(*.mxt)檔案格式





元素&特徵無縫轉換





Thank you.

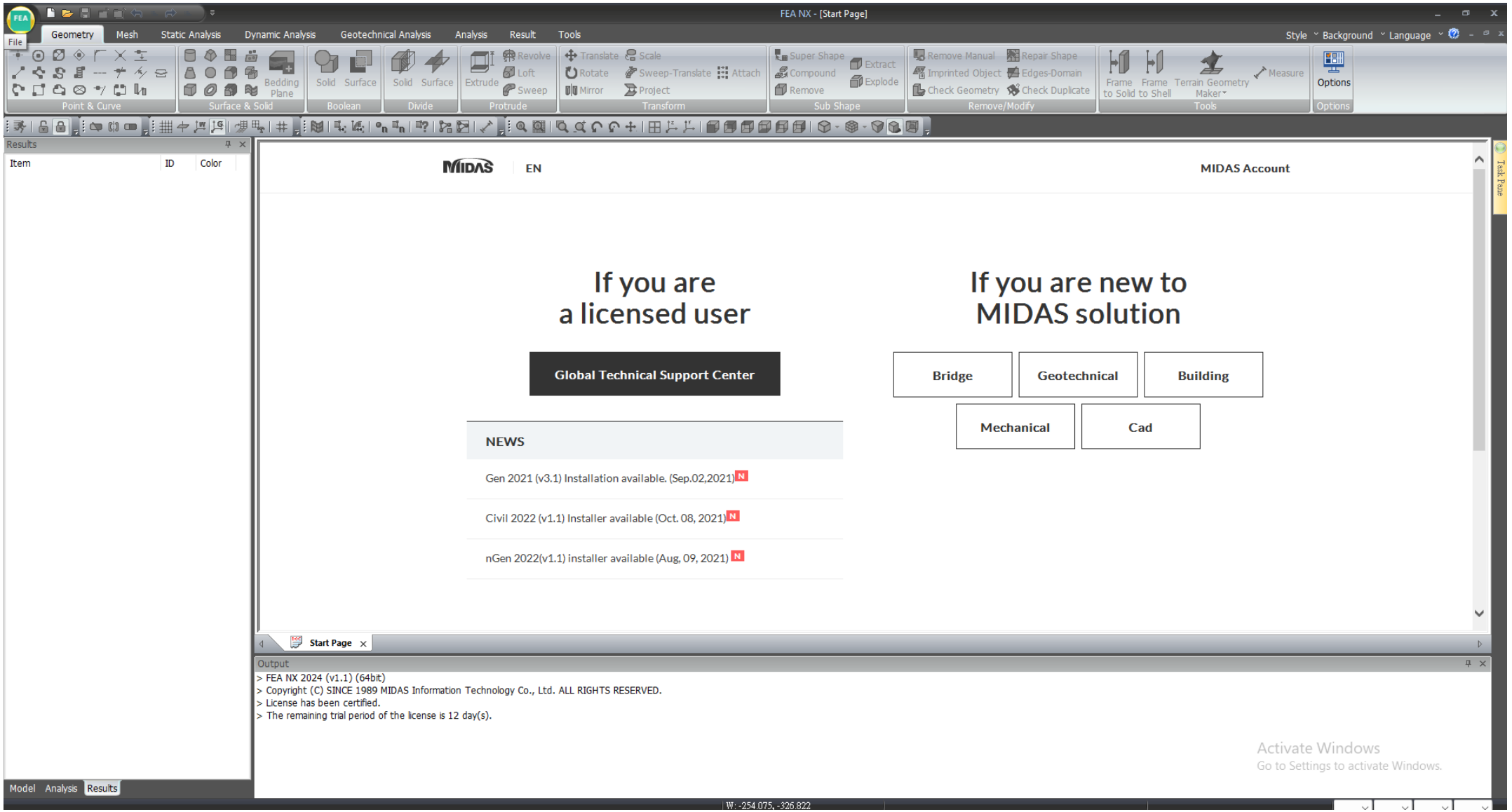
MIDAS

FEA NX

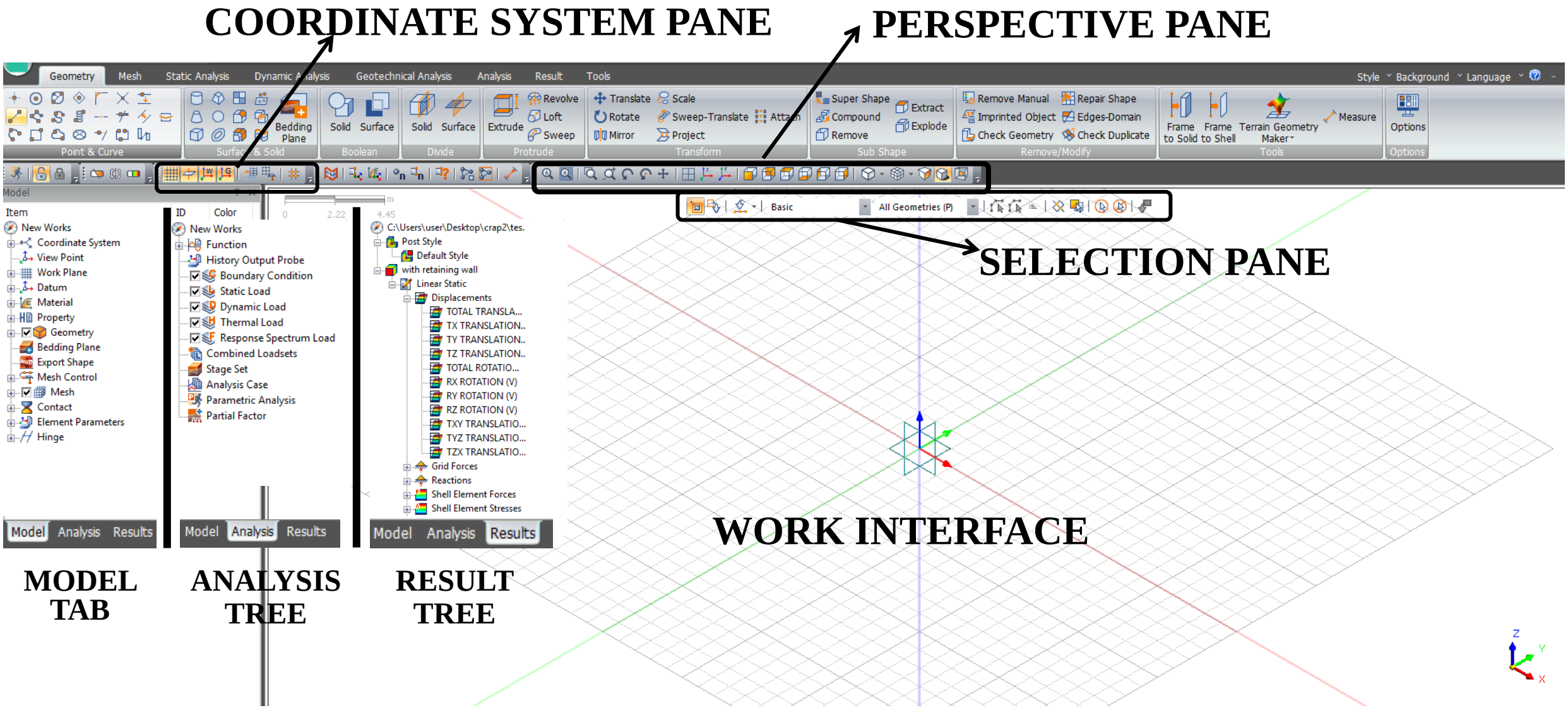
FEA NX GUI

Version: FEA NX 2025 V1.1 | APRIL 14 2025

MAIN INTERFACE



WORKSPACE


COORDINATE SYSTEM PANE
PERSPECTIVE PANE
SELECTION PANE
WORK INTERFACE
MODEL TAB
ANALYSIS TREE
RESULT TREE
Geometry
Mesh
Static Analysis
Dynamic Analysis
Geotechnical Analysis
Analysis
Result
Tools
Style
Background
Language
Point & Curve
Surface & Solid
Boolean
Divide
Protrude
Transform
Sub Shape
Remove/Modify
Tools
Options
Bedding Plane
Solid
Surface
Extrude
Revolve
Loft
Sweep
Translate
Rotate
Mirror
Scale
Sweep-Translate
Project
Attach
Super Shape
Compound
Remove
Extract
Explode
Remove Manual
Imprinted Object
Check Geometry
Repair Shape
Edges-Domain
Check Duplicate
Frame to Solid
Frame to Shell
Terrain Geometry Maker
Measure
Options
Model
Item
New Works
Coordinate System
View Point
Work Plane
Datum
Material
Property
Geometry
Bedding Plane
Export Shape
Mesh Control
Mesh
Contact
Element Parameters
Hinge
ID
Color
New Works
Function
History Output Probe
Boundary Condition
Static Load
Dynamic Load
Thermal Load
Response Spectrum Load
Combined Loadsets
Stage Set
Analysis Case
Parametric Analysis
Partial Factor
C:\Users\user\Desktop\crap2\tes...
Post Style
Default Style
with retaining wall
Linear Static
Displacements
TOTAL TRANSLA...
TX TRANSLATION..
TY TRANSLATION..
TZ TRANSLATION..
TOTAL ROTATIO...
RX ROTATION (V)
RY ROTATION (V)
RZ ROTATION (V)
TXY TRANSLATIO...
TYZ TRANSLATIO...
TZX TRANSLATIO...
Grid Forces
Reactions
Shell Element Forces
Shell Element Stresses
Model
Analysis
Results
Basic
All Geometries (P)

SHORTCUT and MOVEMENT

LEFT CLICK
-Select

RIGHT CLICK
-Additional options

MOUSE WHEEL UP/DOWN
-Zoom in or Zoom out

MOUSE 3/ MOUSE WHEEL PRESS
-Rotate or Translate

CTRL + MOUSE 3/ MOUSE WHEEL PRESS
-Move or Drag

CTRL + S
-Save

CTRL + Z
-Undo

CTRL + Y
-Redo

F3
-Measure

F7/F8
-2D or 3D Generate Mesh

MIDAS

FEA NX

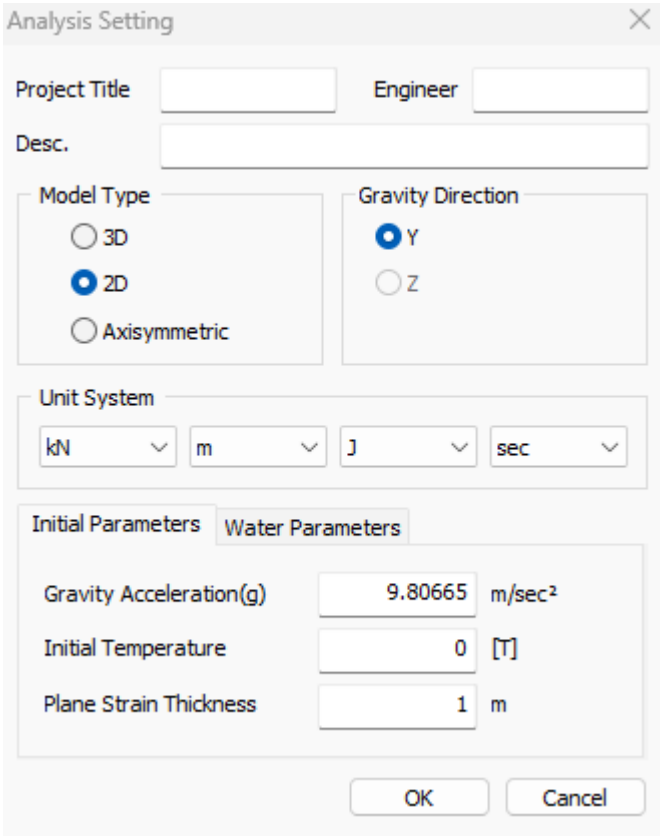
2D/3D Excavation with Retaining Wall Tutorial

MIDAS

FEA NX

2D TUTORIAL

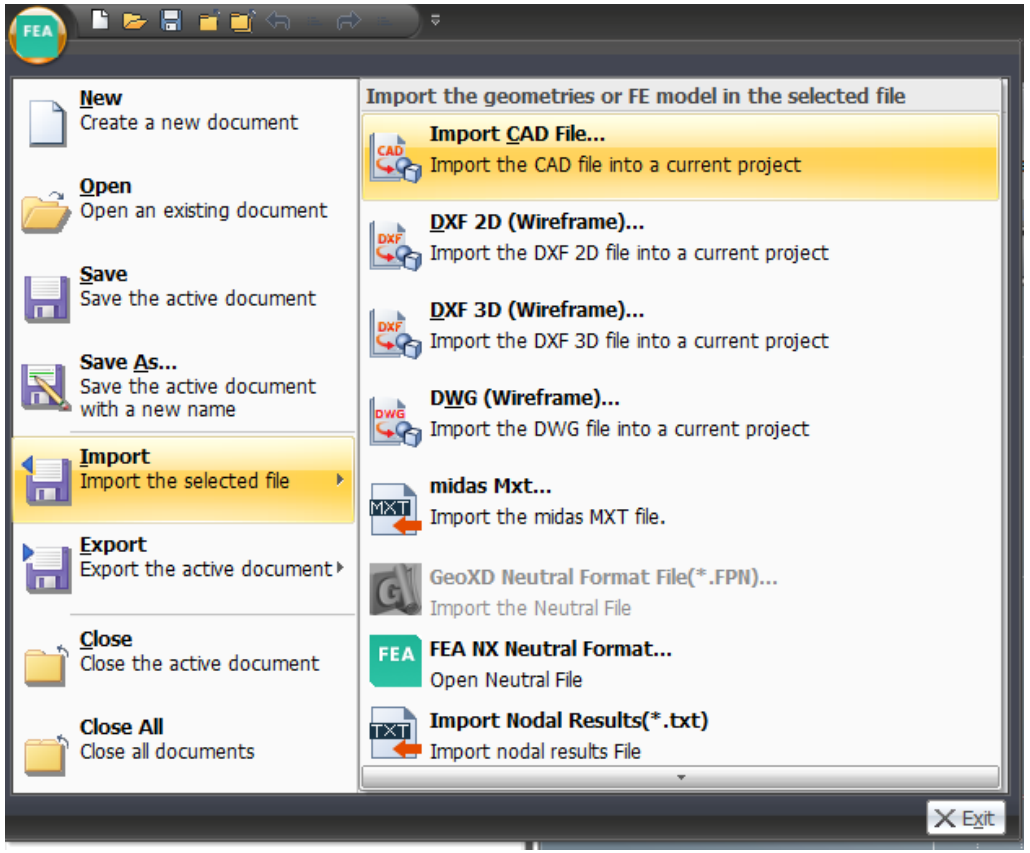
GEOMETRY SET-UP



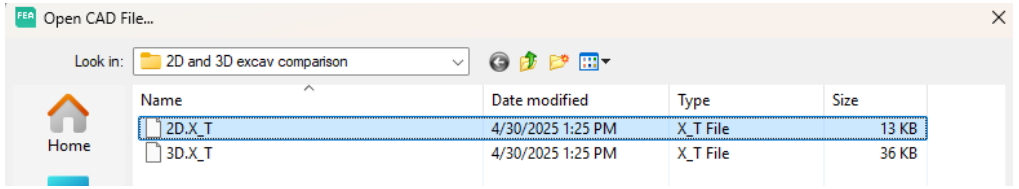
Analysis setting:

- Model Type : 2D
- Choose the preferred unit system

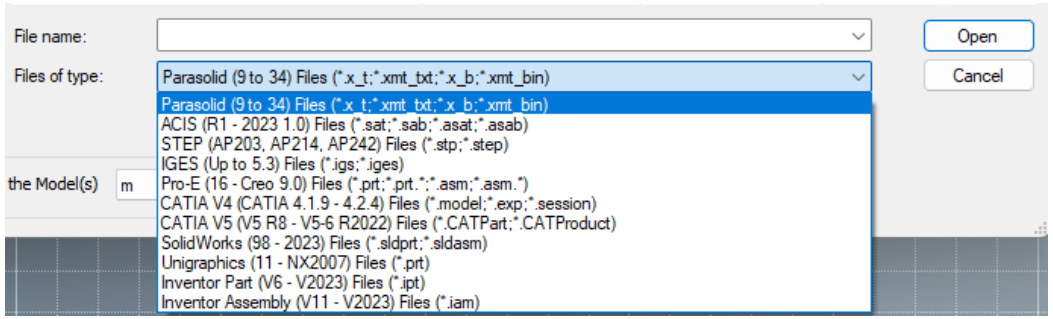
GEOMETRY SET-UP



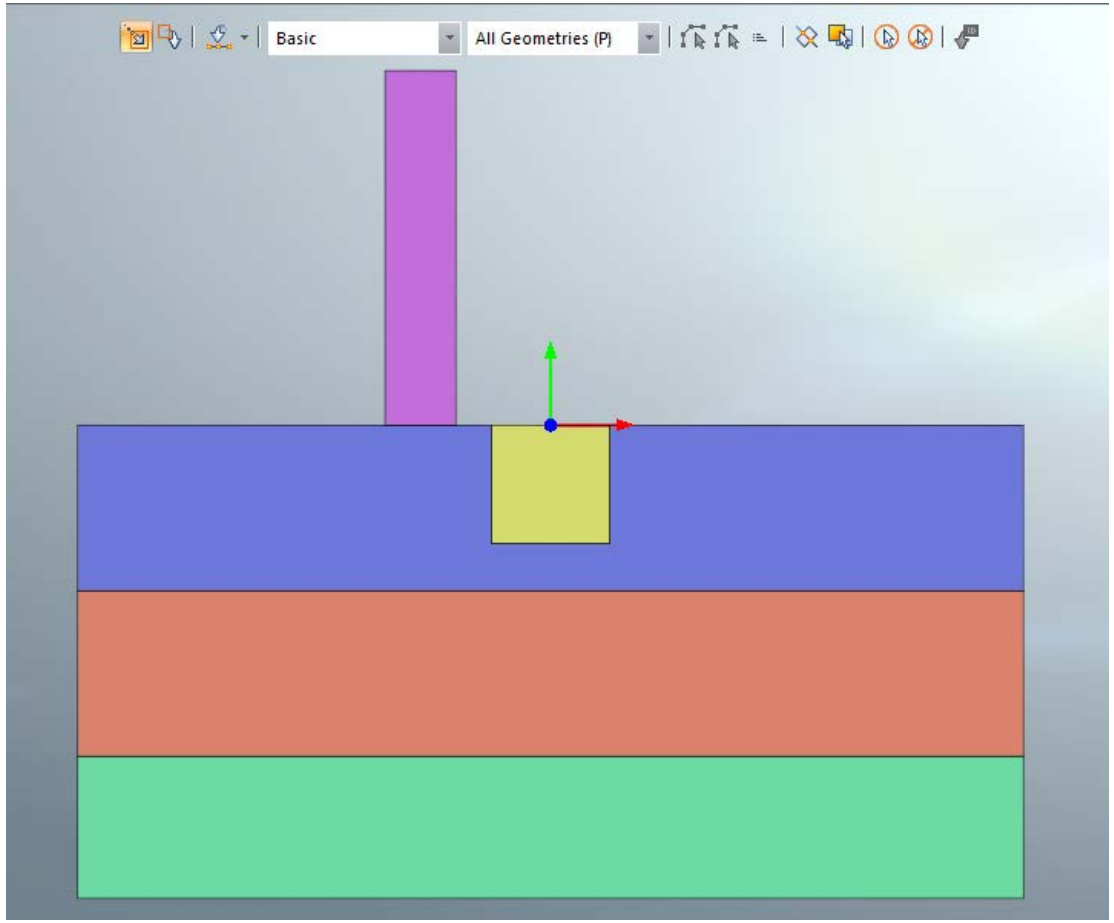
IMPORTING CAD FILE



COMPATIBLE FILES

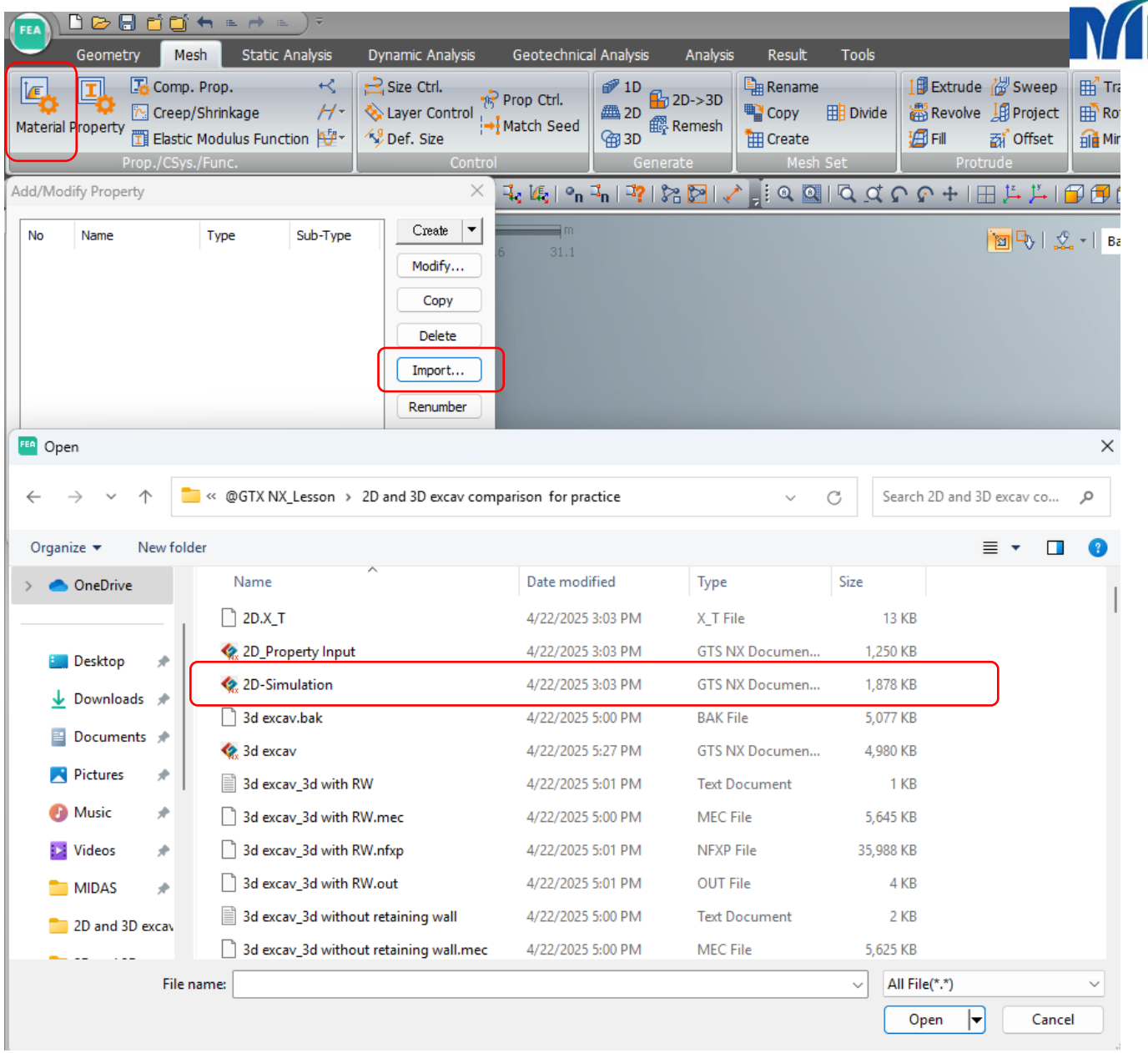


GEOMETRY SET-UP



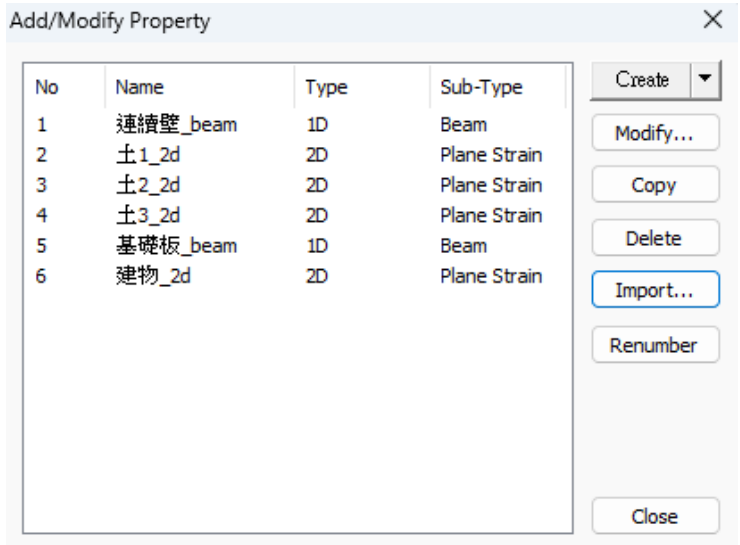
MESHING

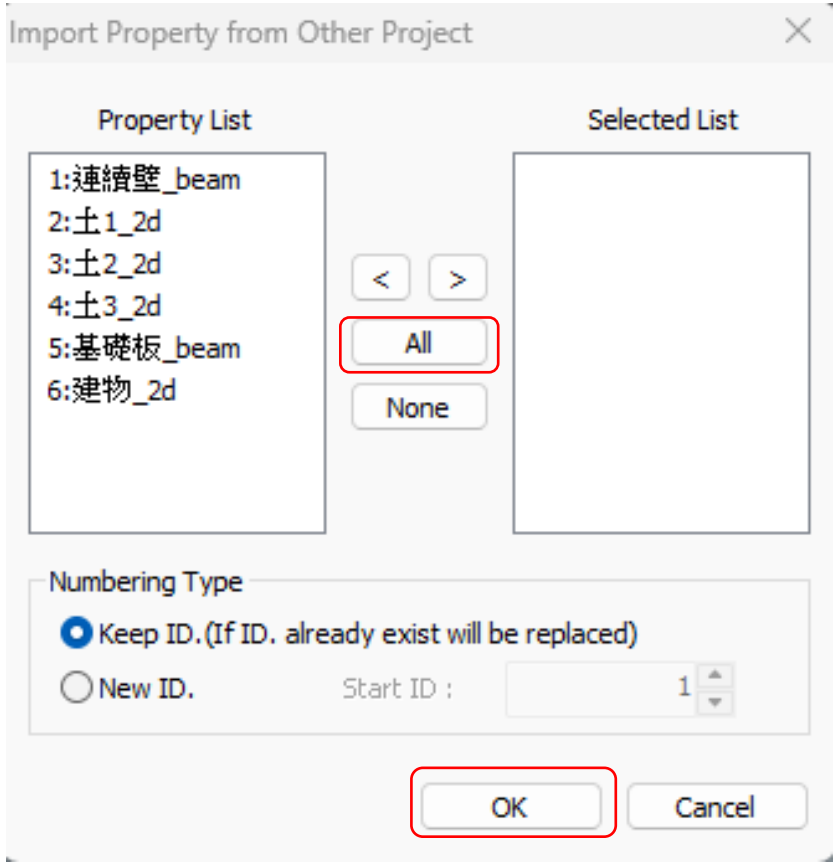
IMPORTING PROPERTY



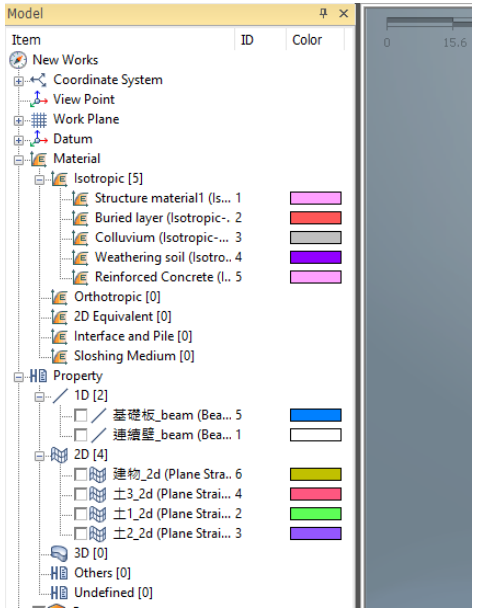
MESHING

IMPORTING PROPERTY





*Properties and materials can be manually added, imported or both.

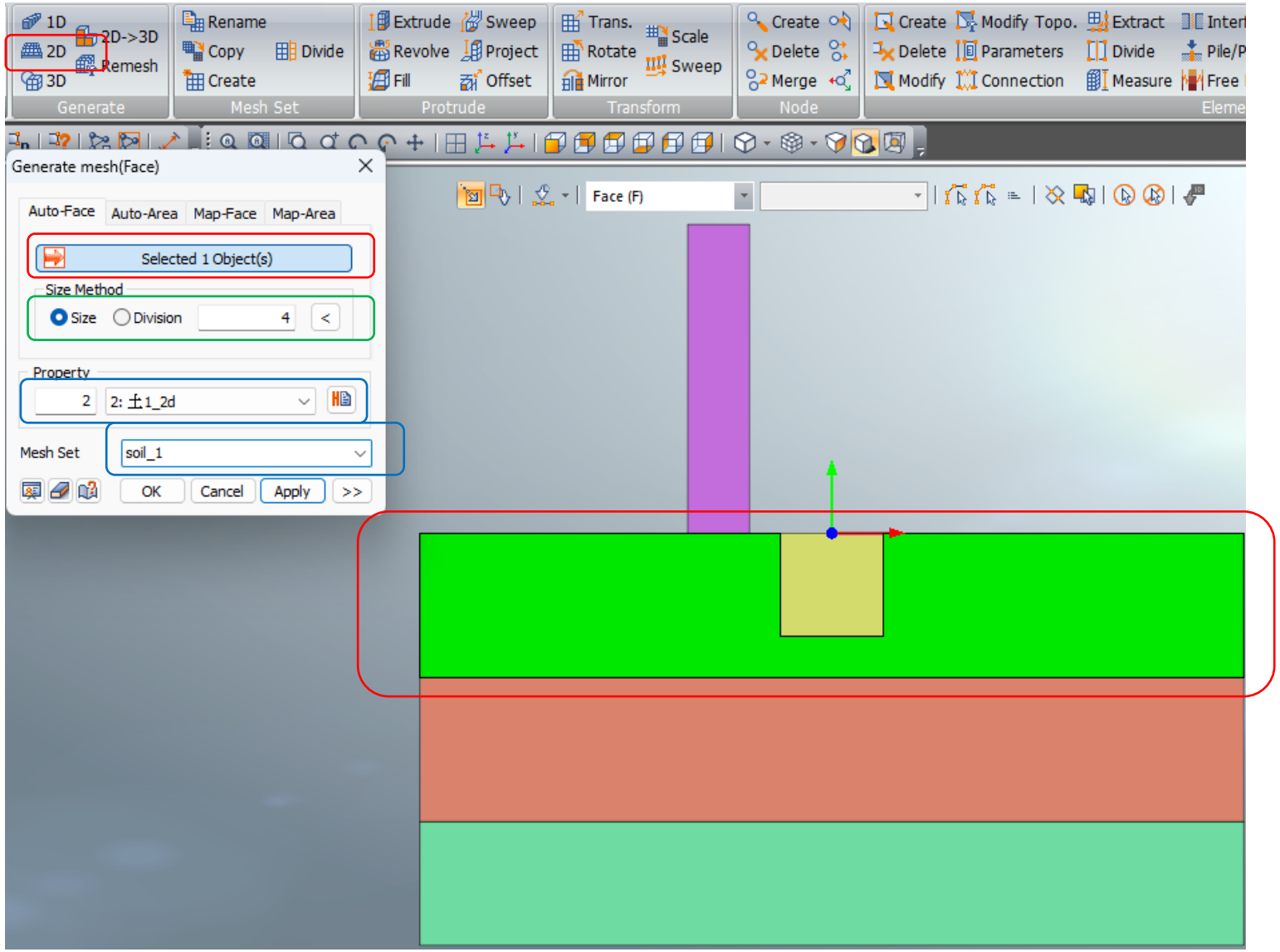


Materials and properties should be reflected in tree model

MESHING

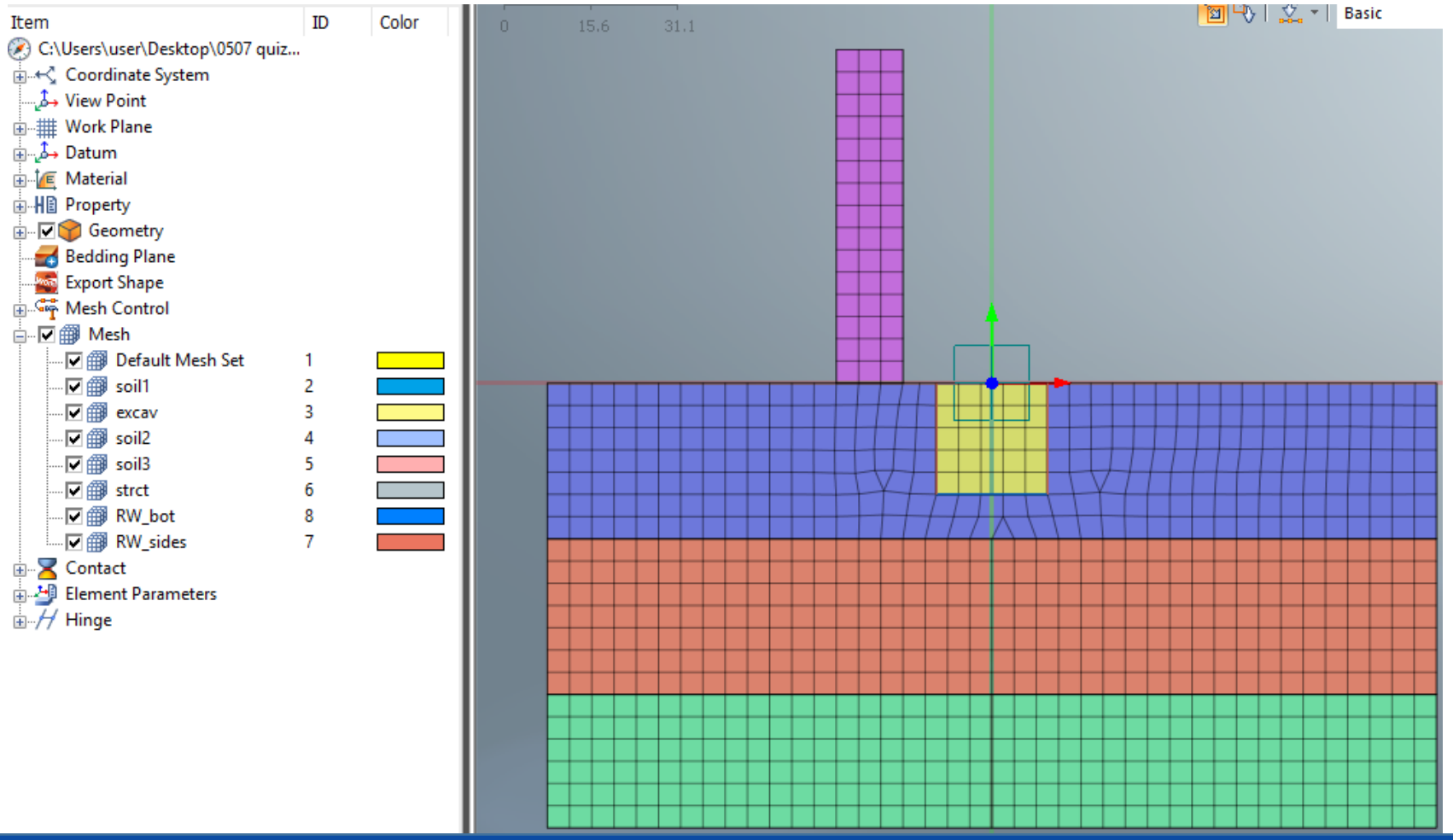
MESHING THE ELEMENTS

1. 2D > Select the object > highlight the element
2. Define the mesh size (The smaller the size, the more accurate the result but also the more difficult it is for the computer to process)
3. Select the appropriate property for the element
4. Rename the mesh
5. Repeat for all elements in the project



MESHING

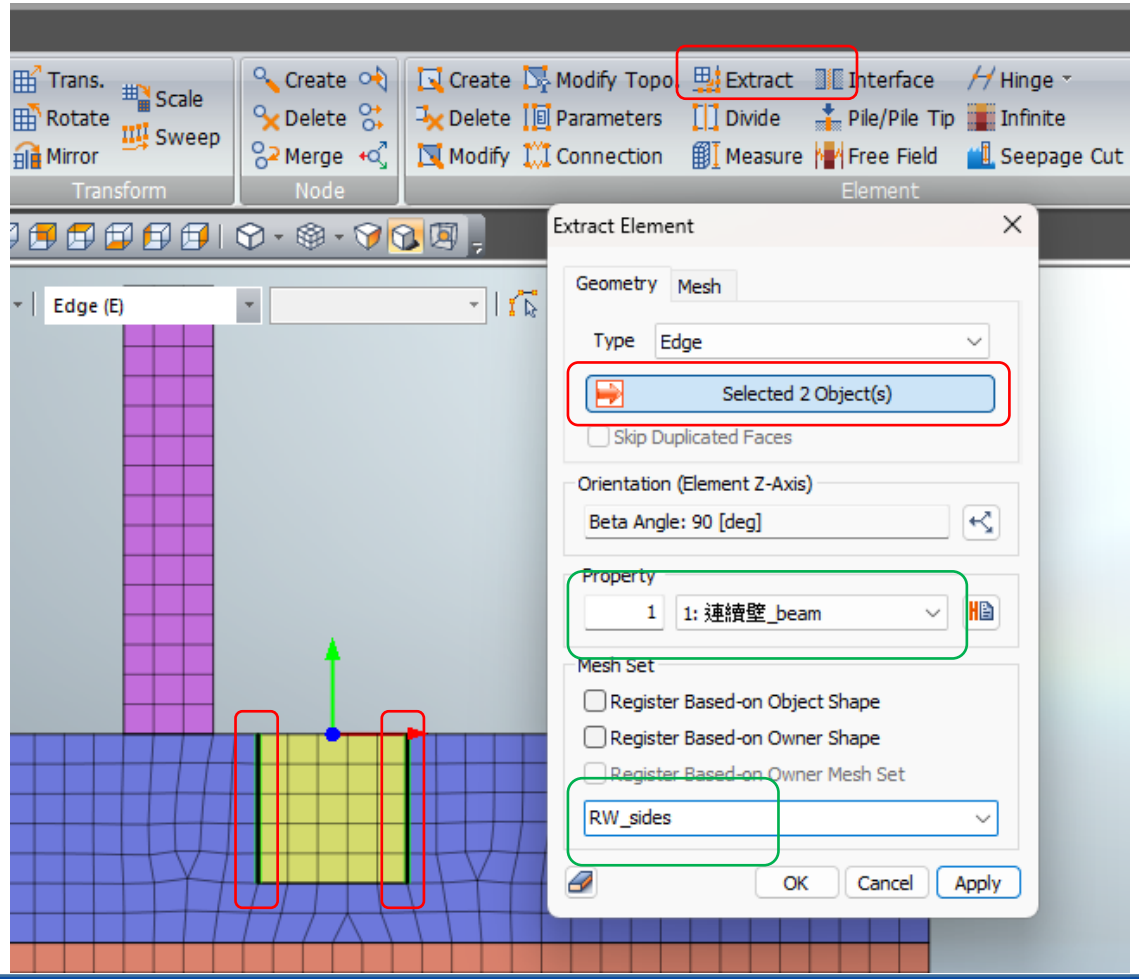
MESHING THE ELEMENTS



NOTE: After meshing all the elements, it should appear in the drawing, also in the model tree

MESHING

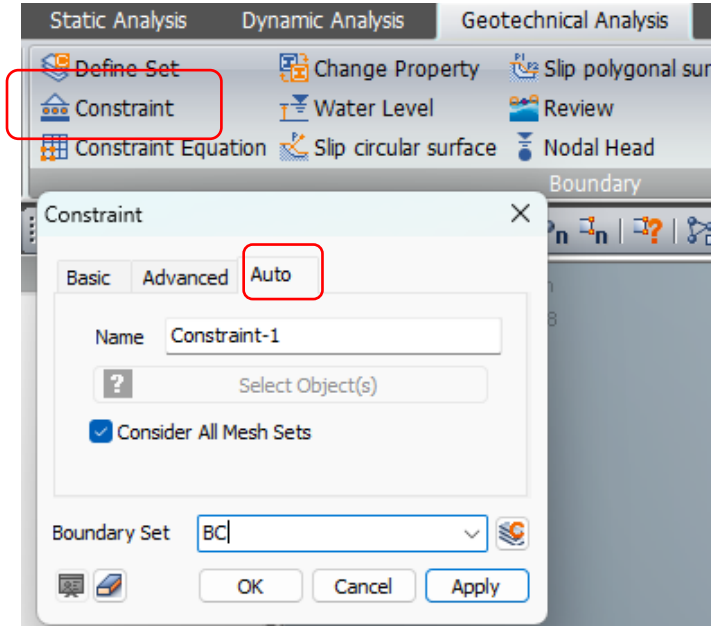
DEFINING THE RETAINING WALL

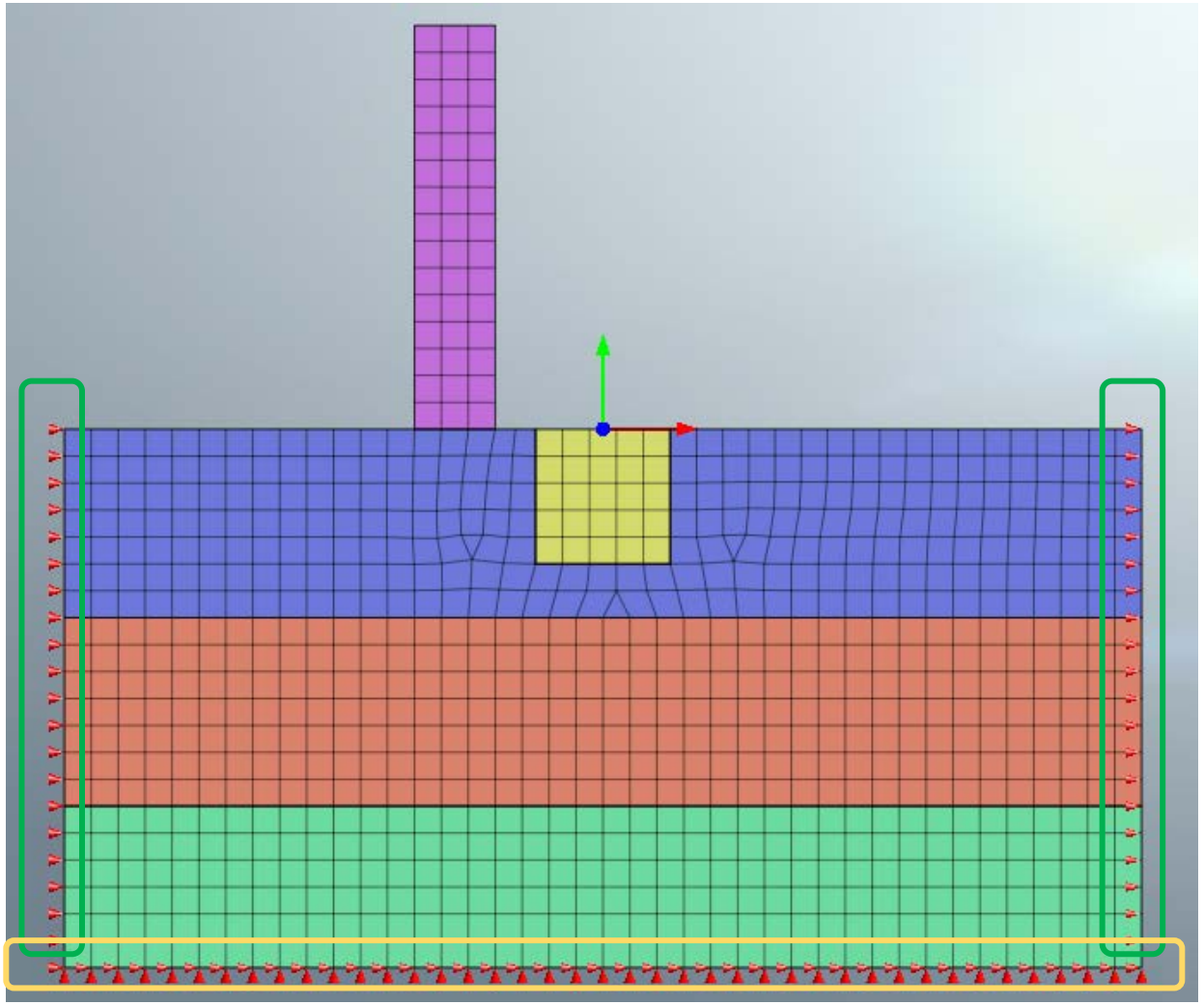


To define a retaining wall

1. Click Extract
2. Select objects/element to become a retaining wall
3. Define the property
4. Rename
5. Apply and repeat to all elements

BOUNDARY CONDITIONS





To set the boundary conditions

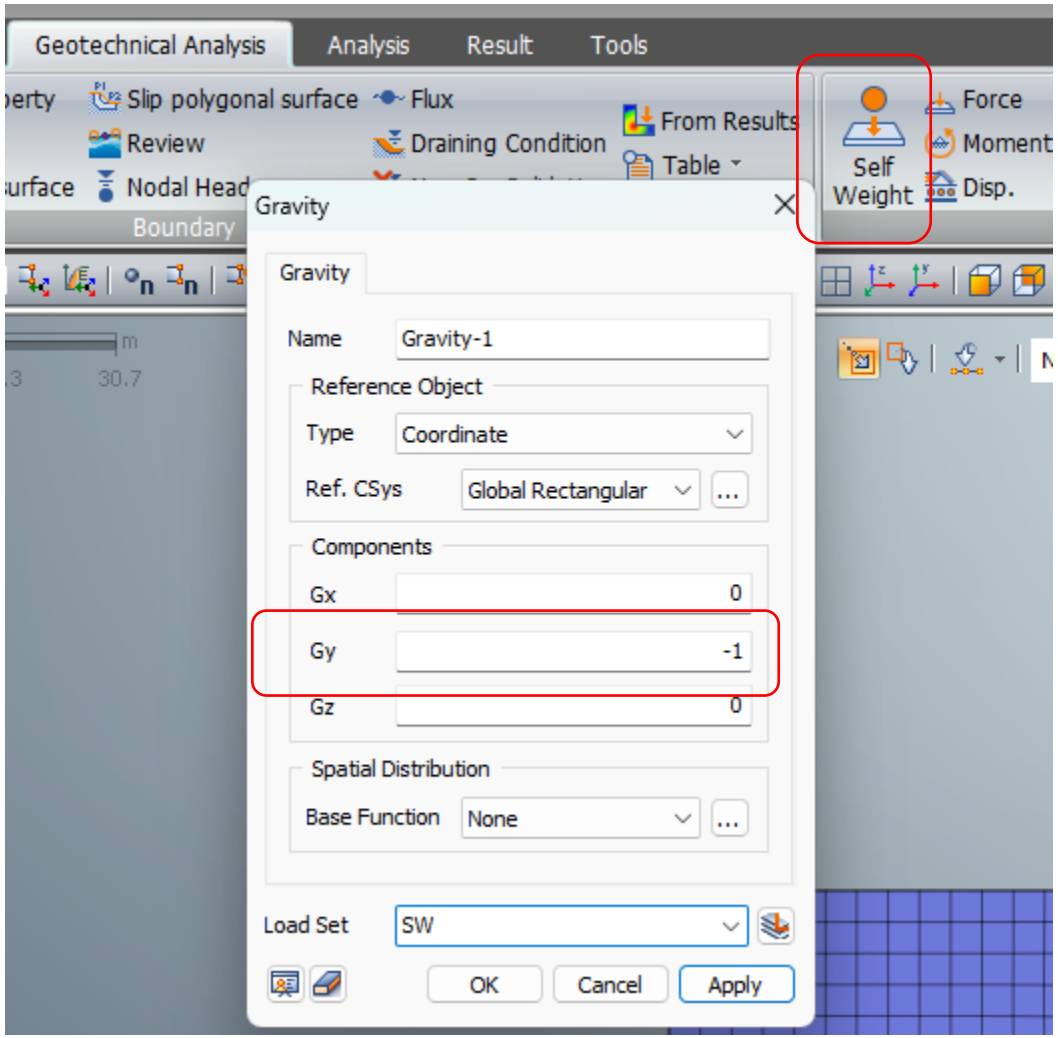
1. Click Constraint

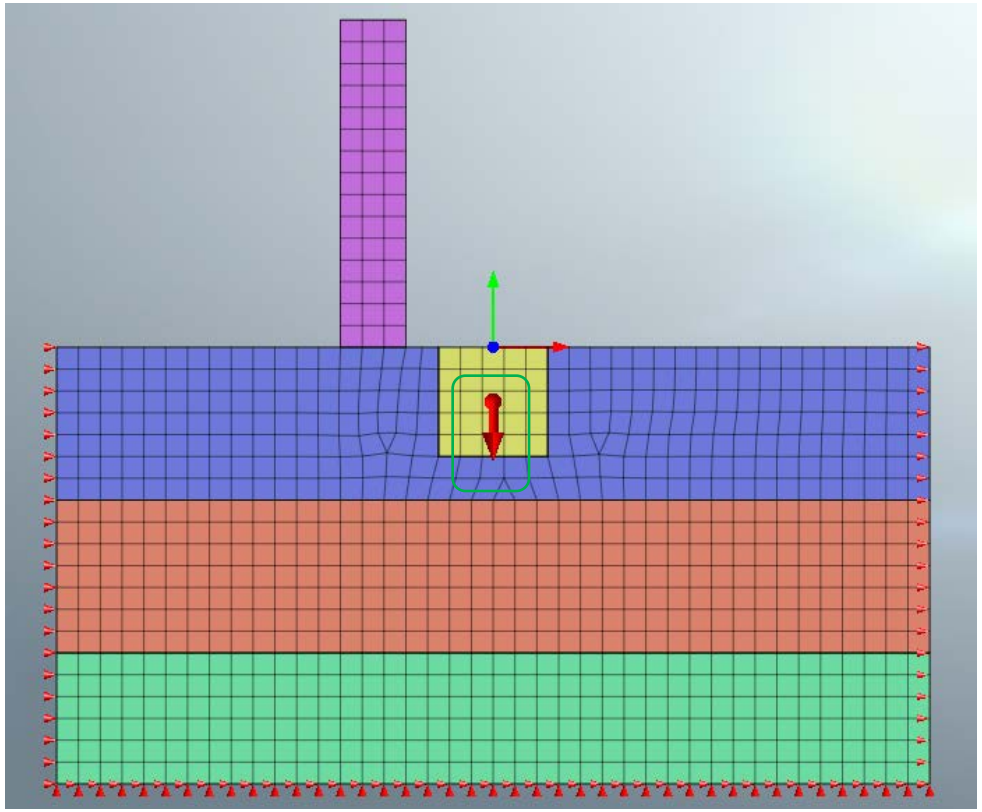
2. Set to auto

3. Rename

Note: The side of the wall will be set to a pin support while the bottom will be set to a fixed support

GRAVITY/SELF-WEIGHT





To define set the gravity/self-weight

1. Click Self-weight

2. Define the load to the axis of gravity

3. Rename

Note: Gravity/Self-weight is indicated in the diagram as the downward arrow

ANALYSIS CASE

To define the analysis case

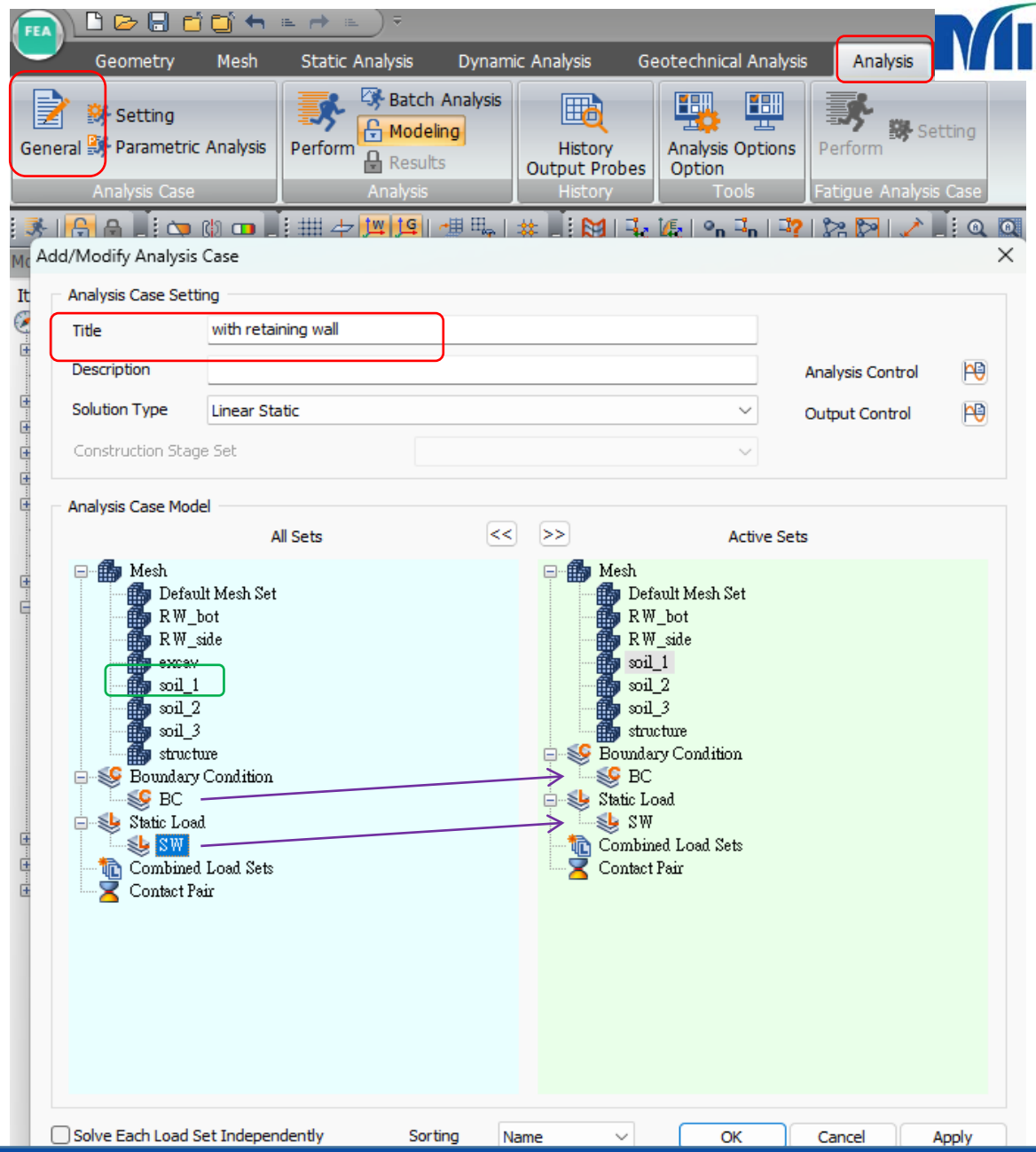
1. Click General in the analysis tab

2. Define a title depending on the project situation

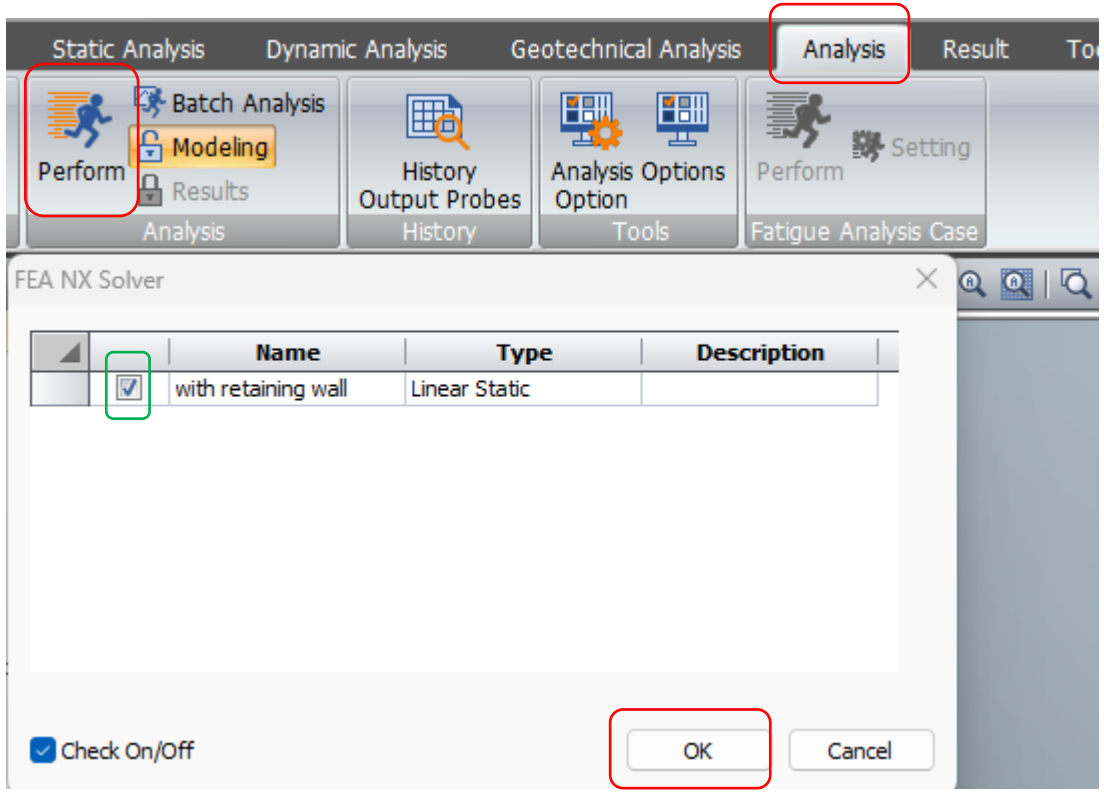
3. Remove excavation mesh from the active sets

4. Add the boundary condition

5. Add the self weight

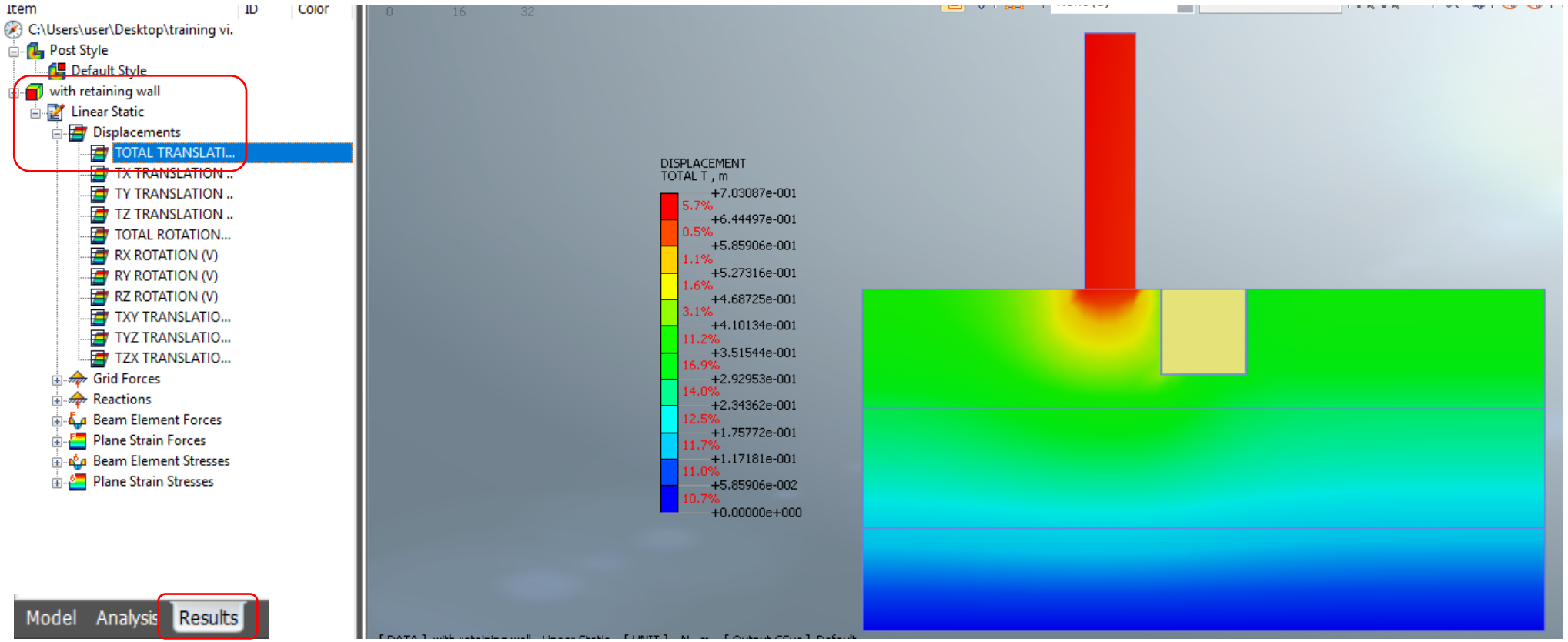


RUN THE CASE



To run the case
1. Click Perform in the analysis tab
2. Activate the analysis case that needed to be solved

RESULTS

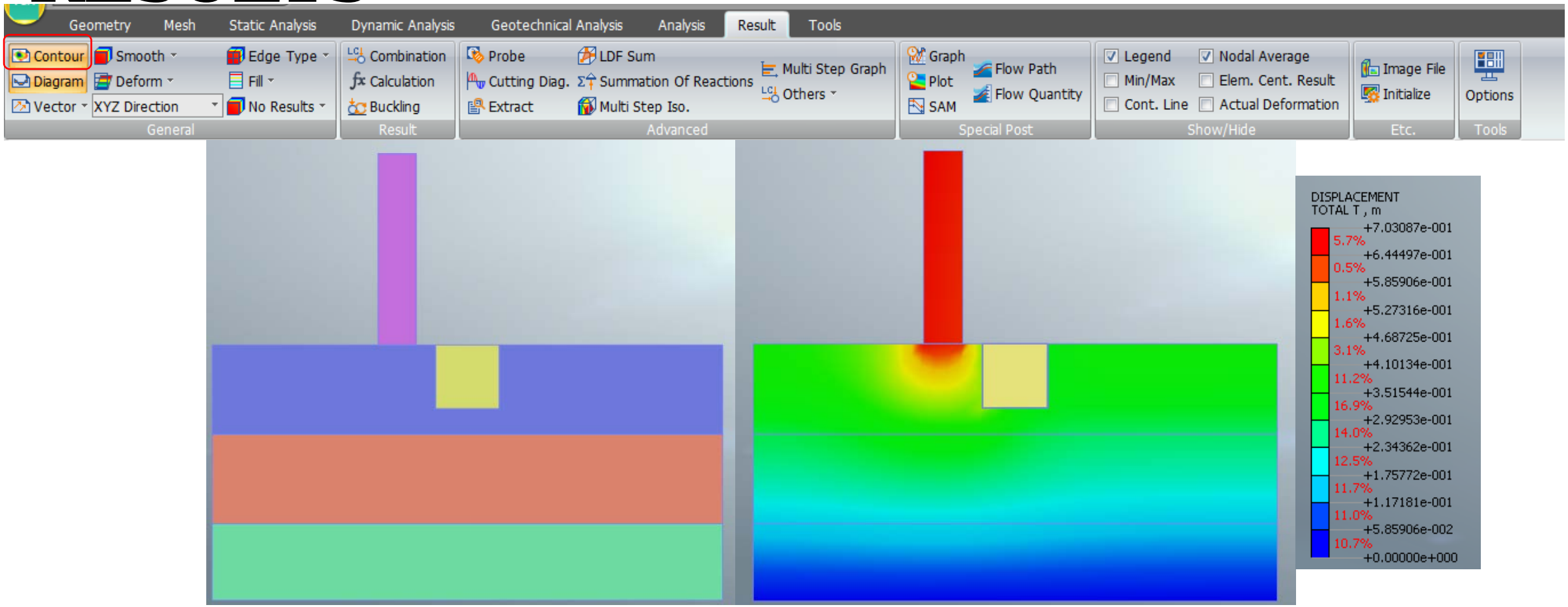


To view results

1. From the model tree, results tab, drop the analysis cases

2. Click the desired result

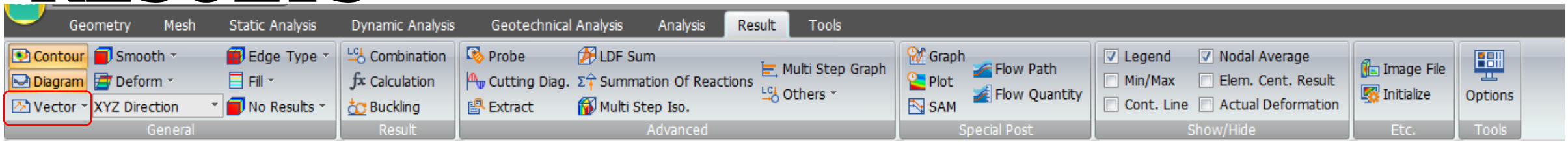
RESULTS

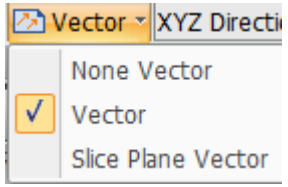


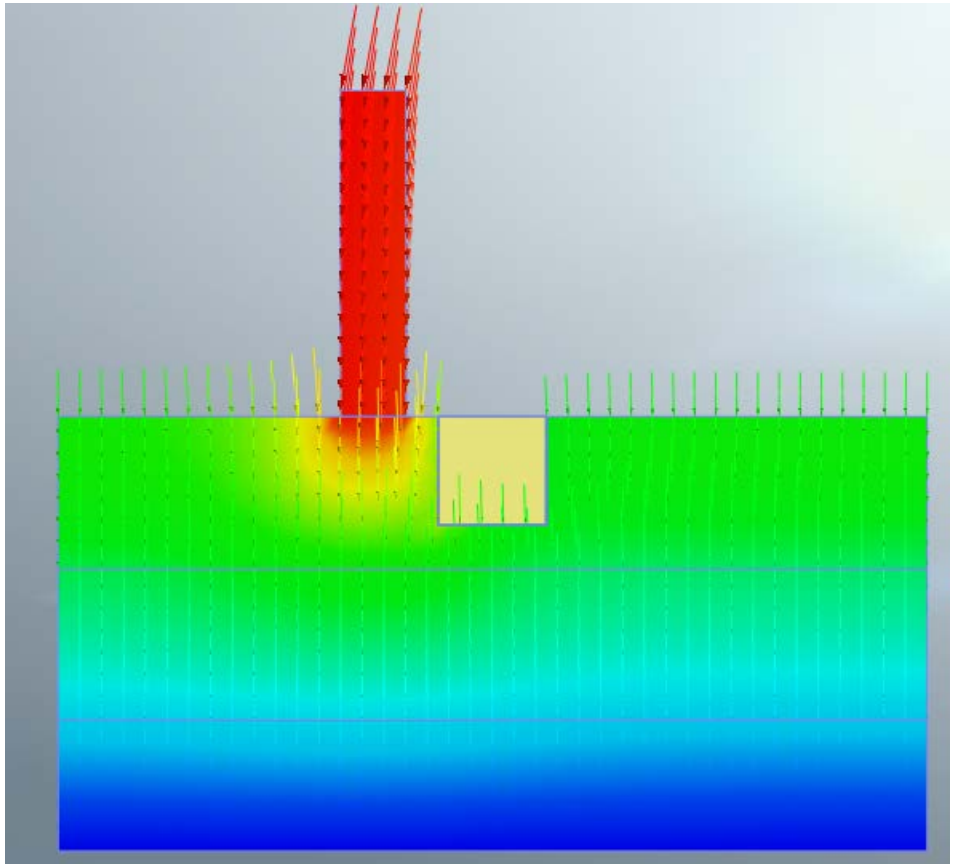
Contour

-Applies color mapping to the model based on the selected result type, such as displacement.

RESULTS



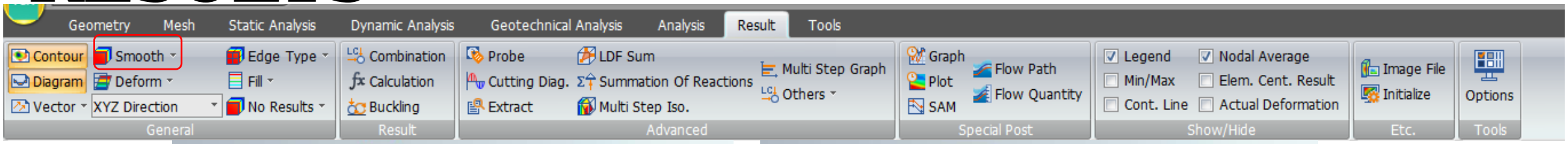


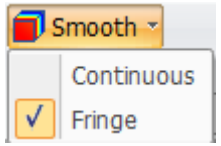


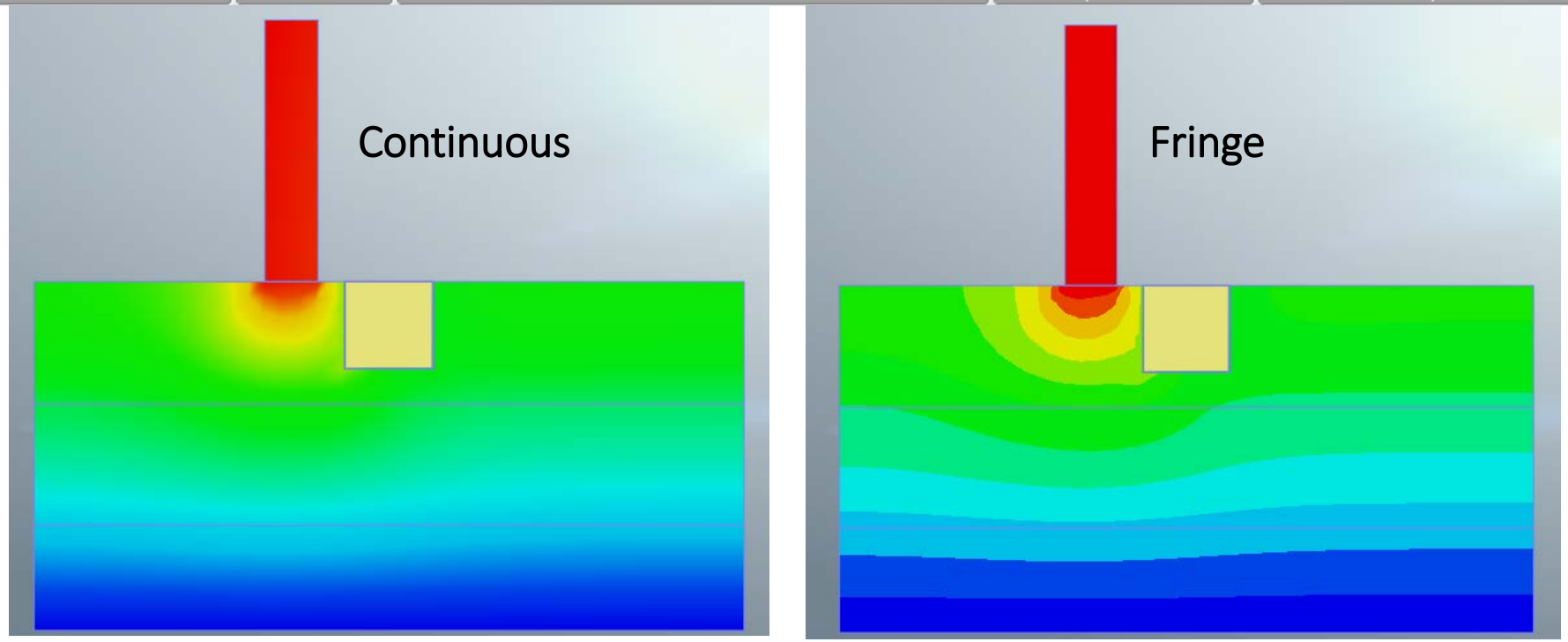
Vector

-Shows the force and direction affecting the diagram.

RESULTS



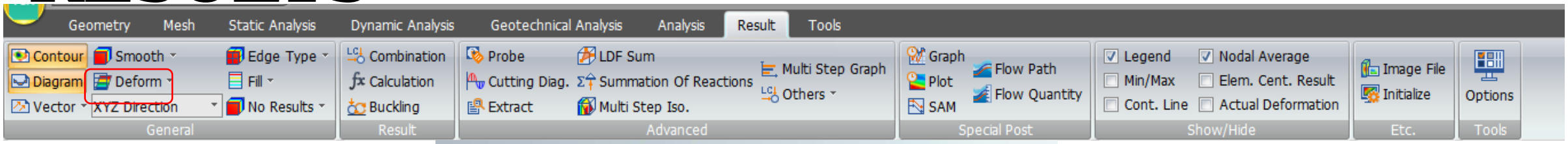




Smooth

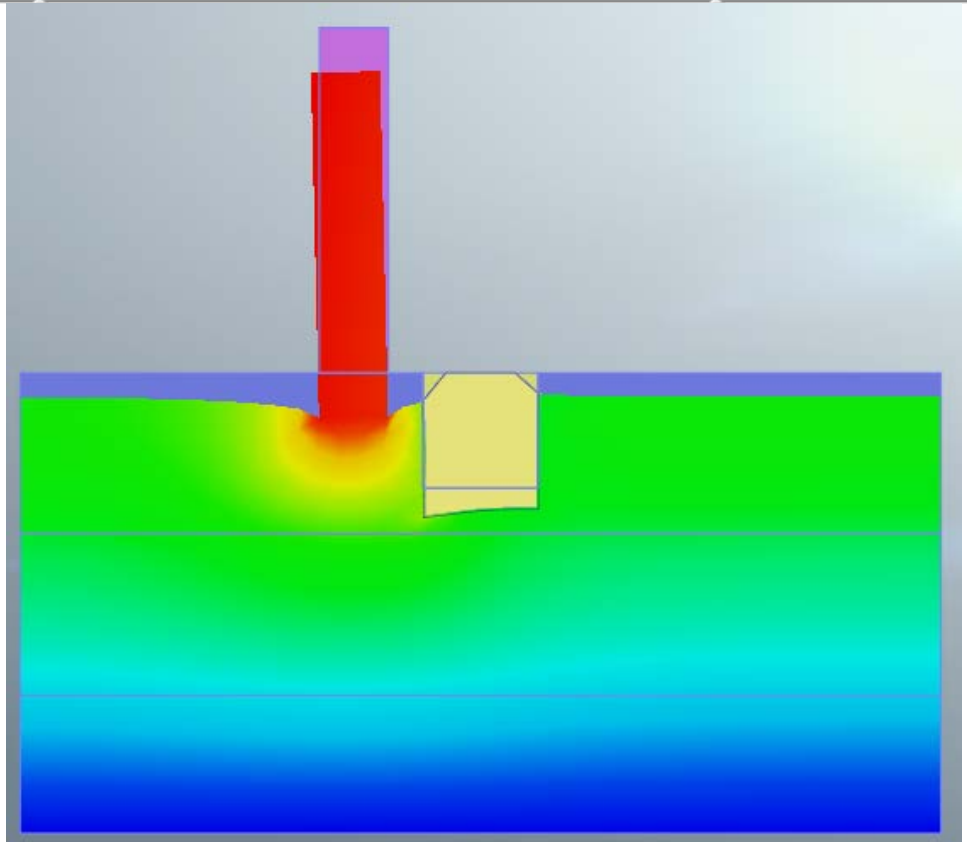
-Smooths the contour for a more refined appearance.

RESULTS

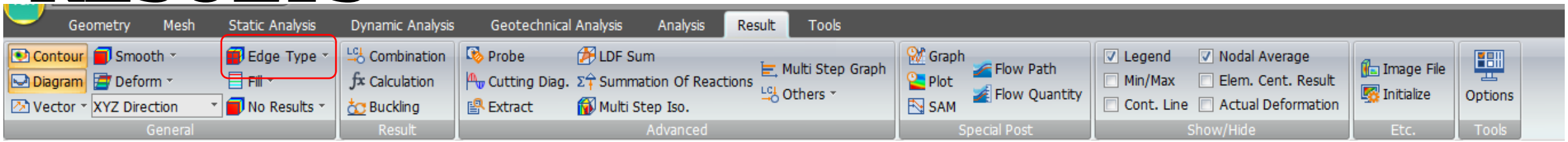


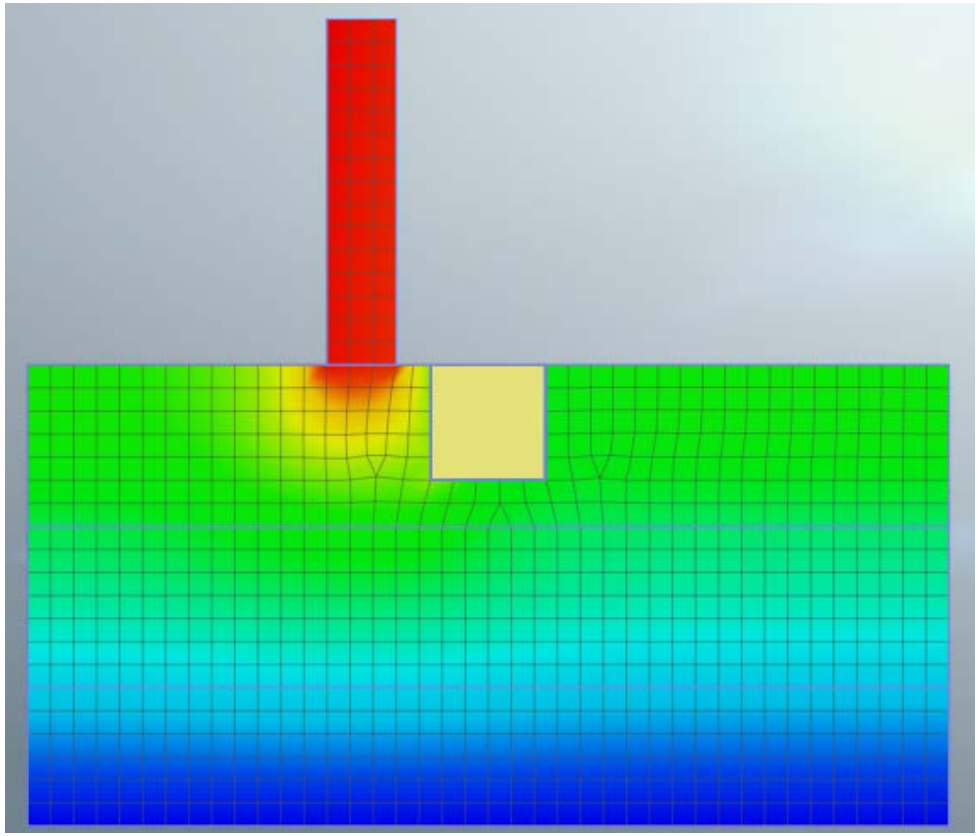
Deform

-Shows the deformed shape to compare easily from the original shape of the structure



RESULTS

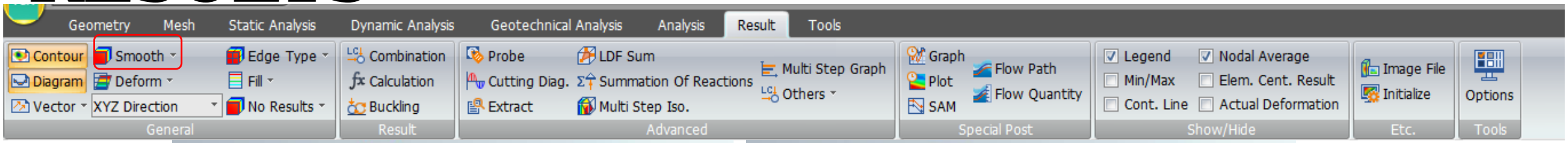


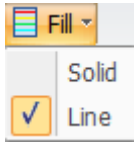


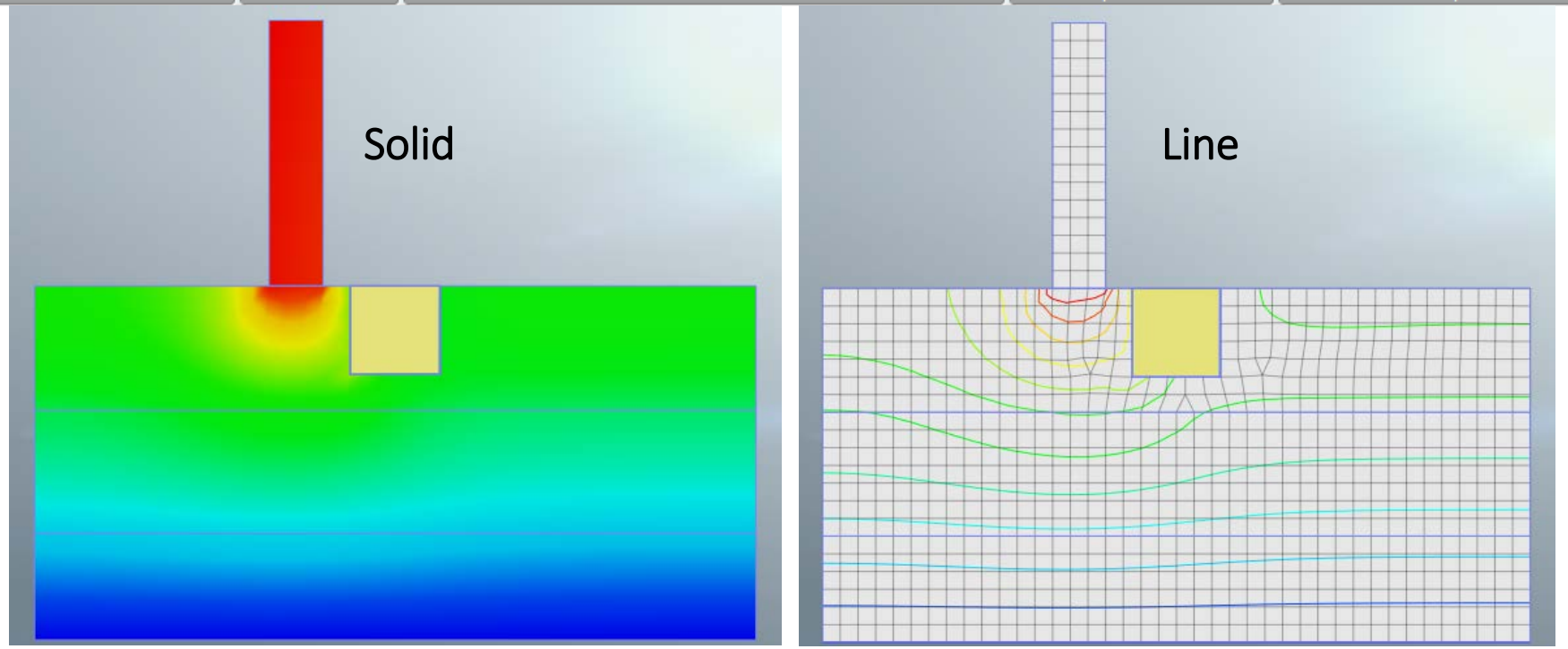
Edge Type

-Shows the mesh of the diagram

RESULTS



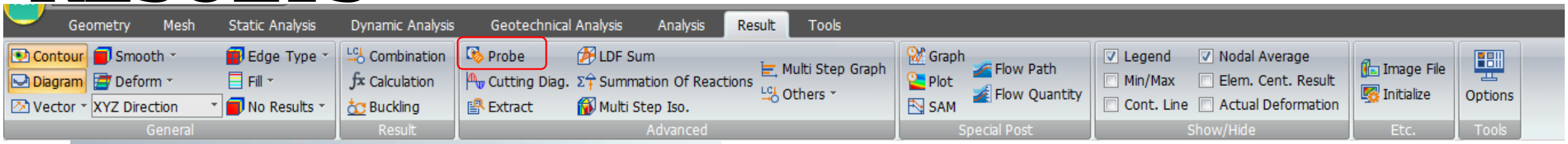


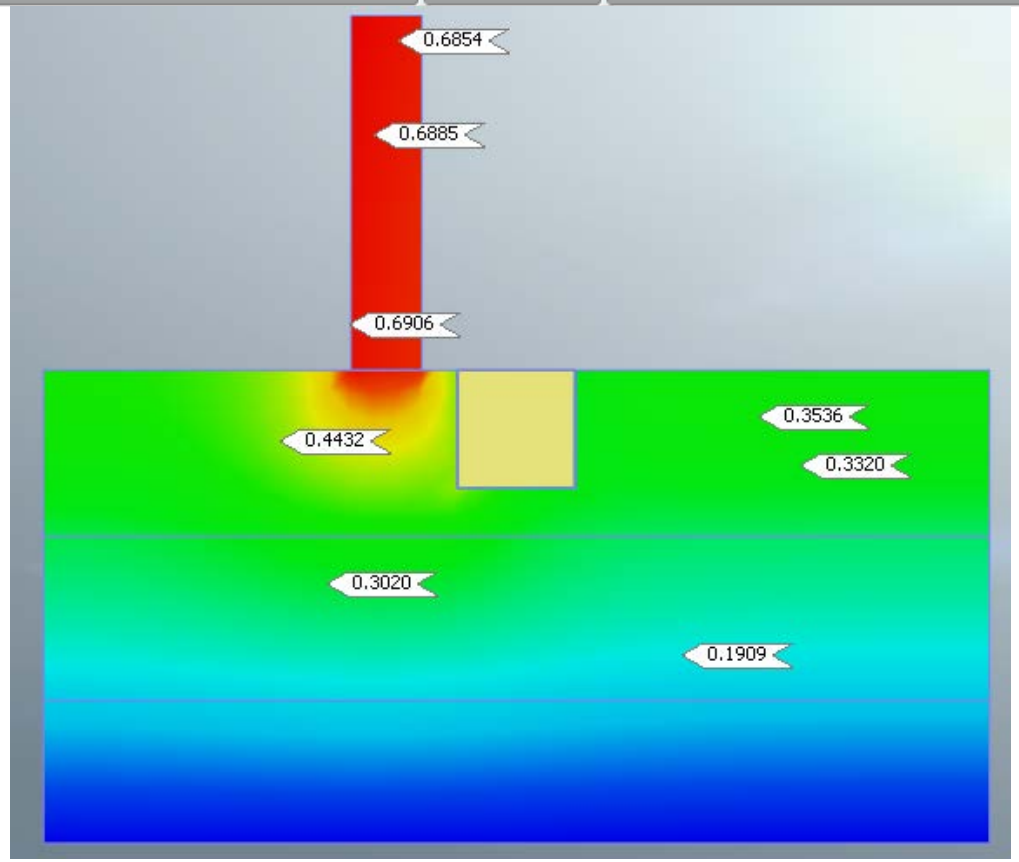


Fill

-Allows the option to fully color or just show the colored line of the diagram.

RESULTS

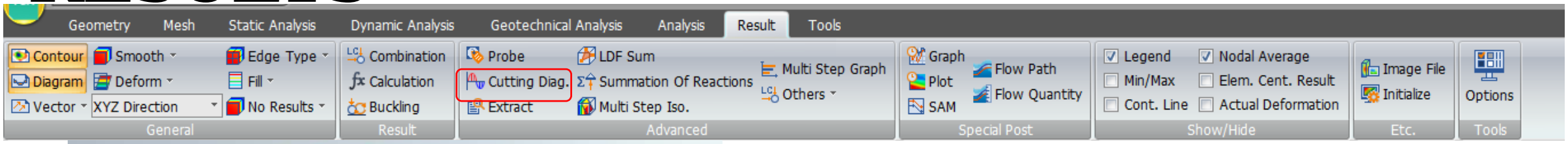


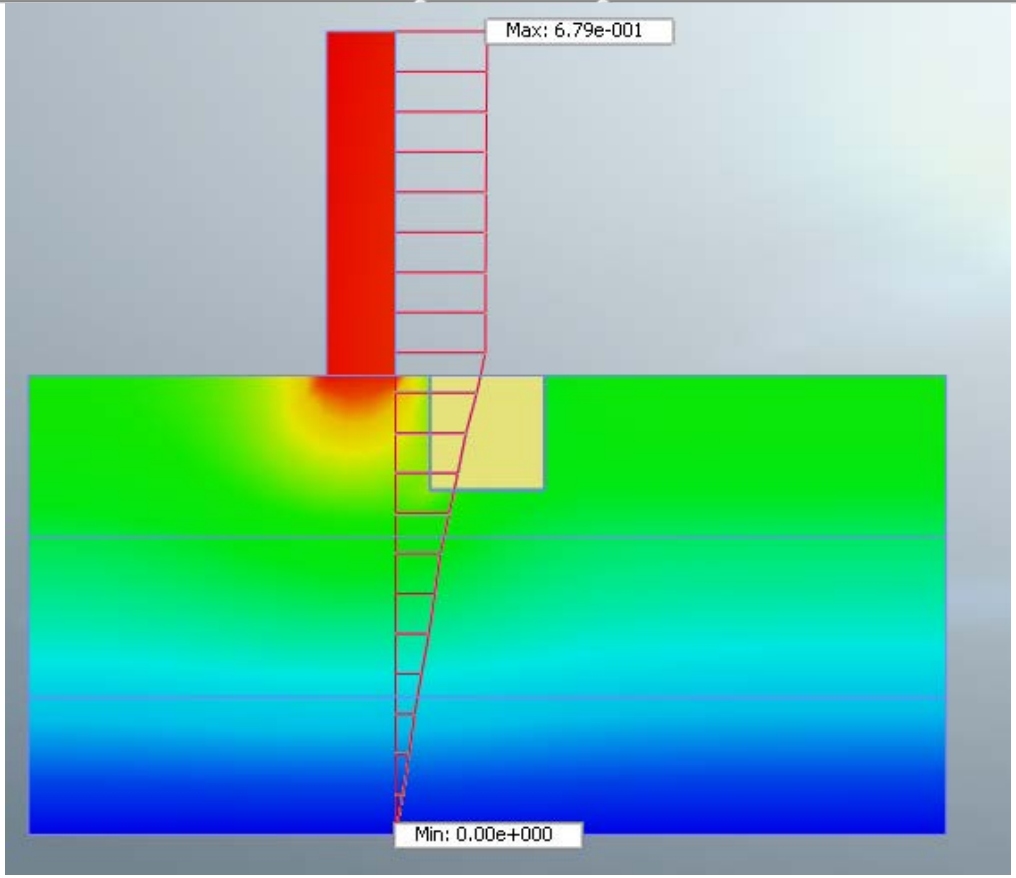


Probe

-Allows a point per point or element per element showing of the result.

RESULTS

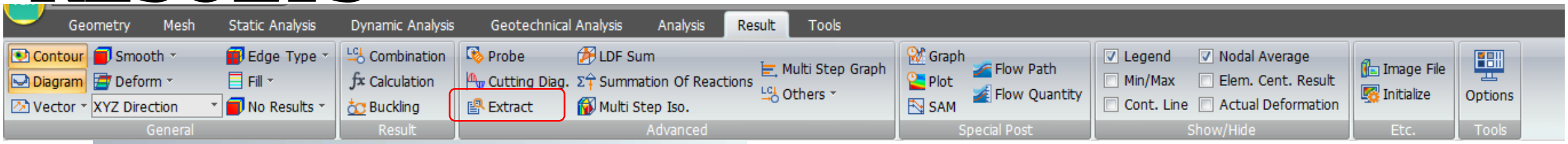


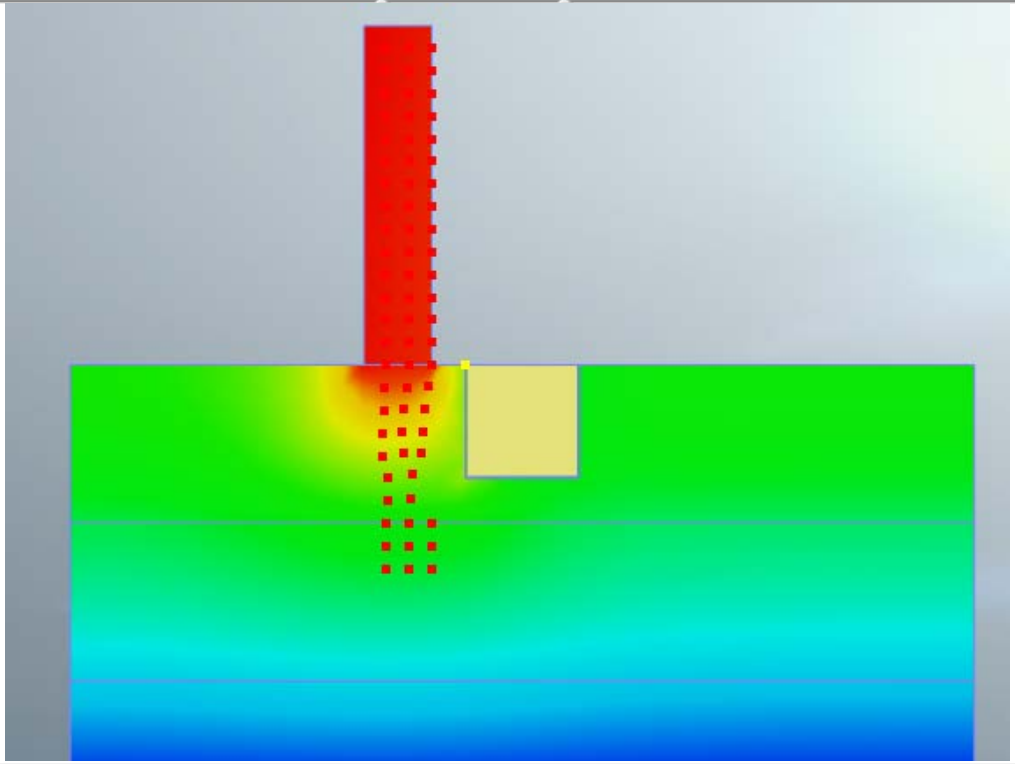


Cutting Diagram

-Shows a diagram between two points.

RESULTS



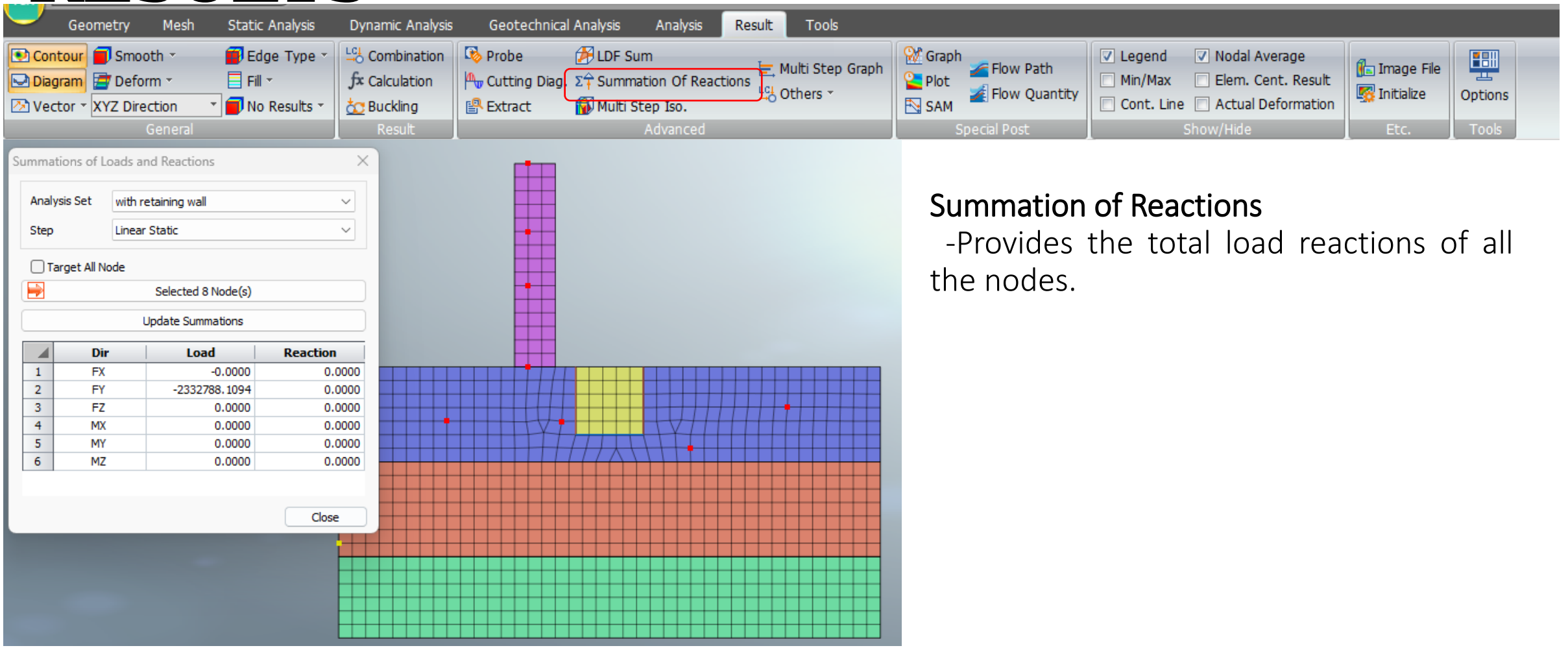


Extract

-Tabulates the displacements of all the node selected in the diagram.

le:83 SLATION (V) m)	Node:83 TZX TRANSLATION (V) (m)	Node:83 TOTAL ROTATION (V) ([rad])	Node:83 RX ROTATION (V) ([rad])	Node:83 RY ROTATION (V) ([rad])	Node:83 RZ ROTATION (V) ([rad])	Node:84 TOTAL TRANSLATION (V) (m)	Node:84 TX TRANSLATION (V) (m)	Node:84 TY TRANSLATION (V) (m)	Node:84 TZ TRANSLATION (V) (m)	Node:84 TXY TRANSLATION (V) (m)
6.651329e-001	4.300737e-003	0.000000e+000	0.000000e+000	0.000000e+000	0.000000e+000	6.736153e-001	-4.261994e-003	-6.736017e-001	0.000000e+000	6.736153e-001

RESULTS



Summation of Reactions

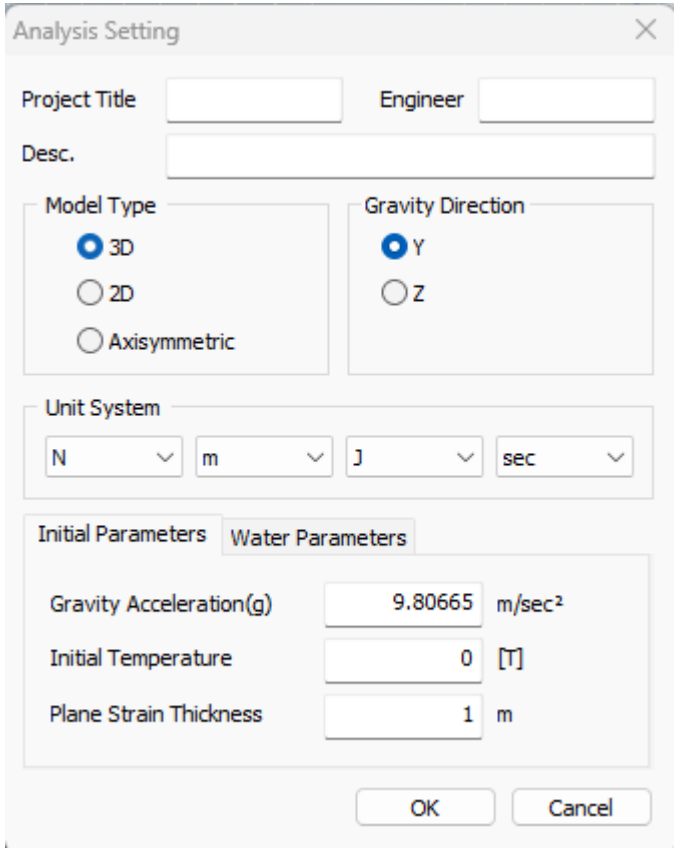
-Provides the total load reactions of all the nodes.

MIDAS

FEA NX

3D TUTORIAL

GEOMETRY SET-UP



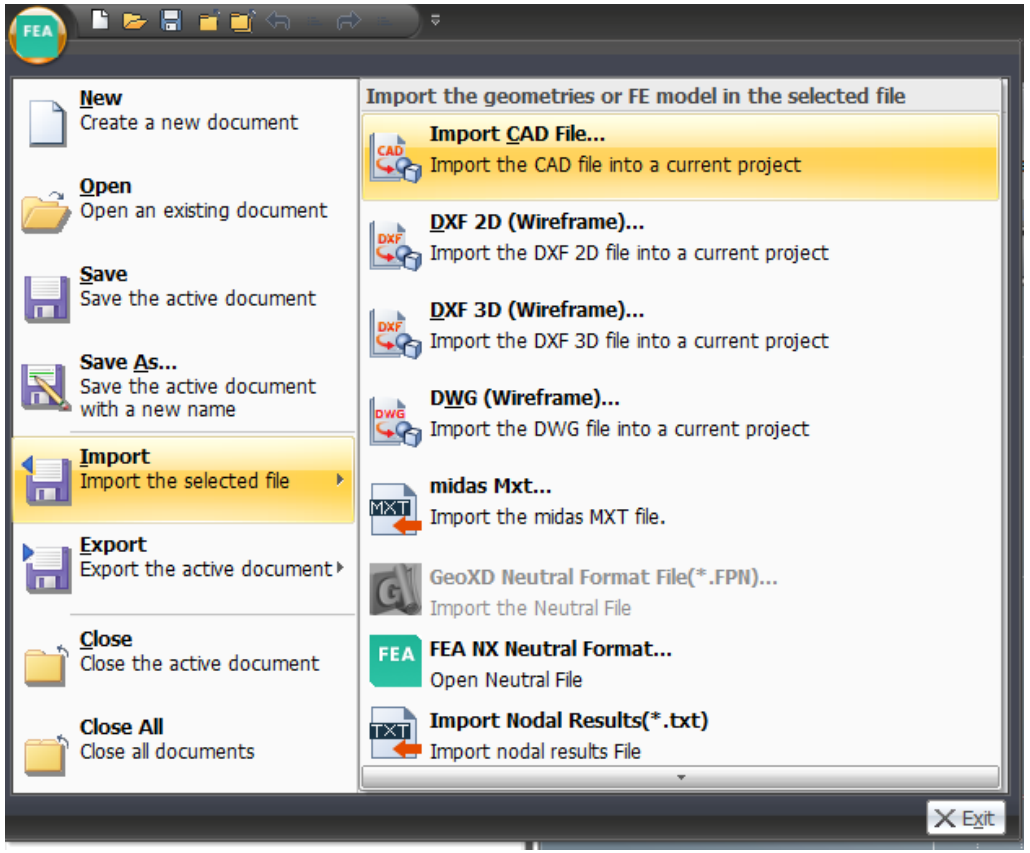
Analysis setting:

- Model Type : 3D
- Choose the preferred unit system

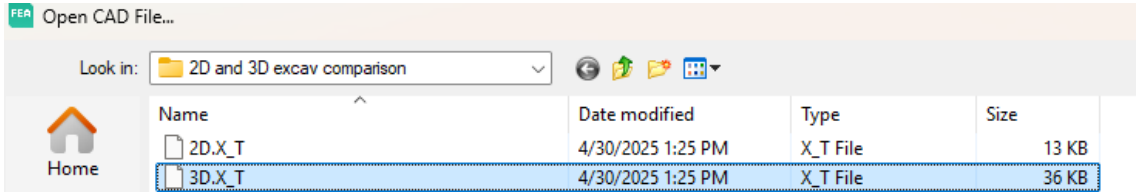
Note:

1. The axis of gravity can only be defined in this setting.
2. Unit system can be modified throughout the entire operation.

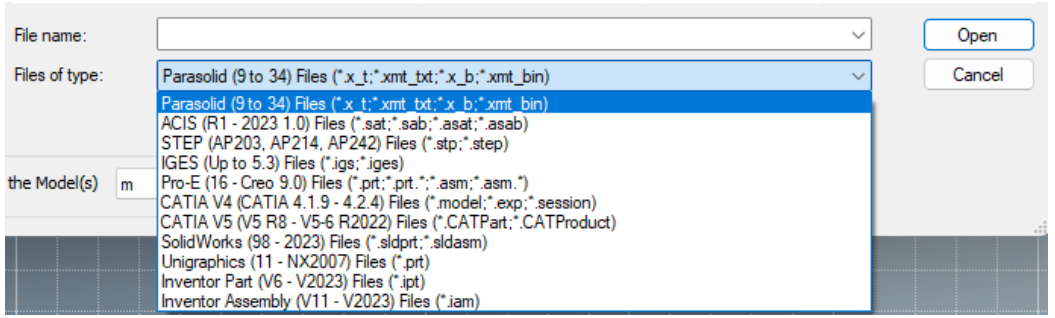
GEOMETRY SET-UP



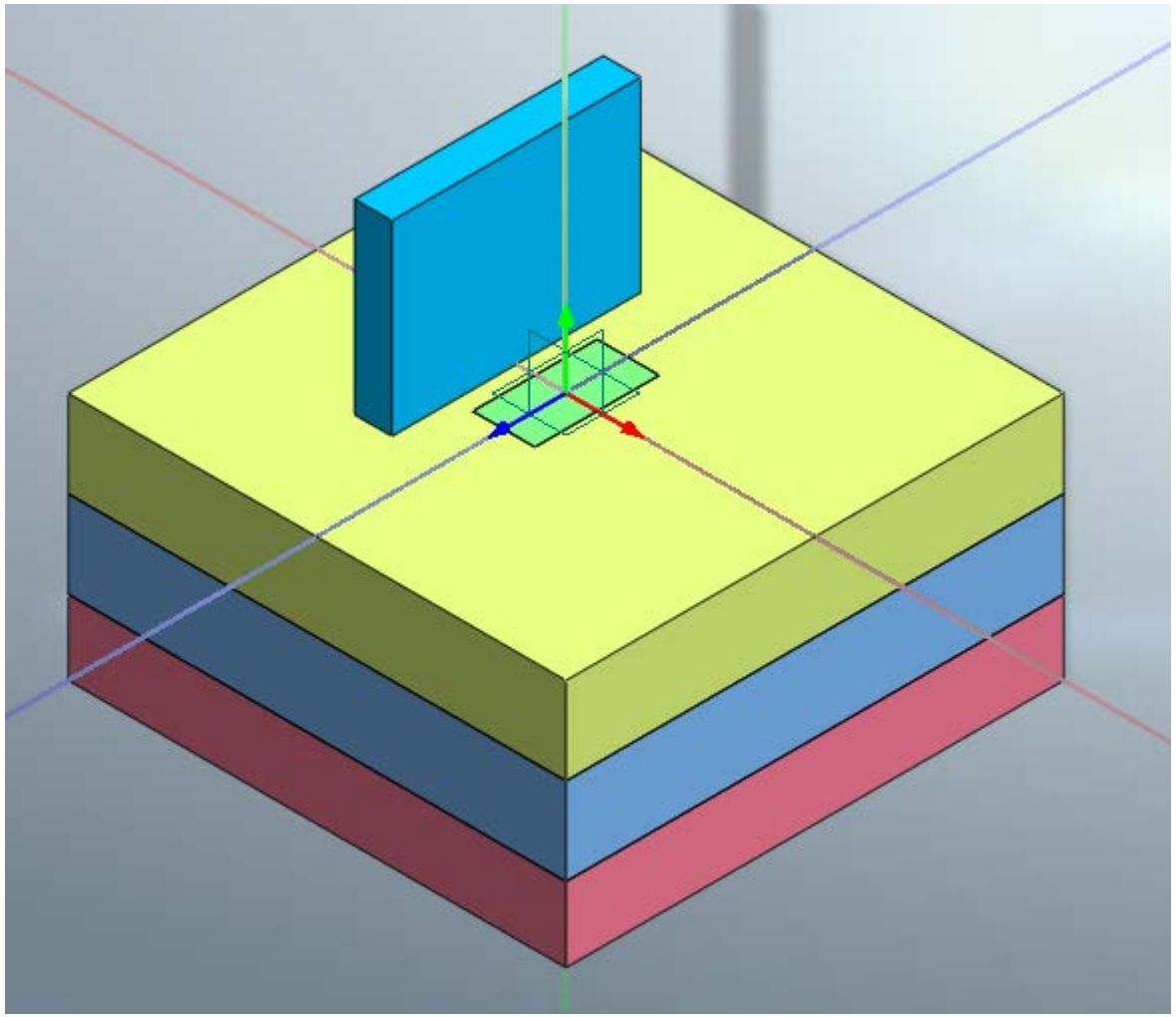
IMPORTING CAD FILE

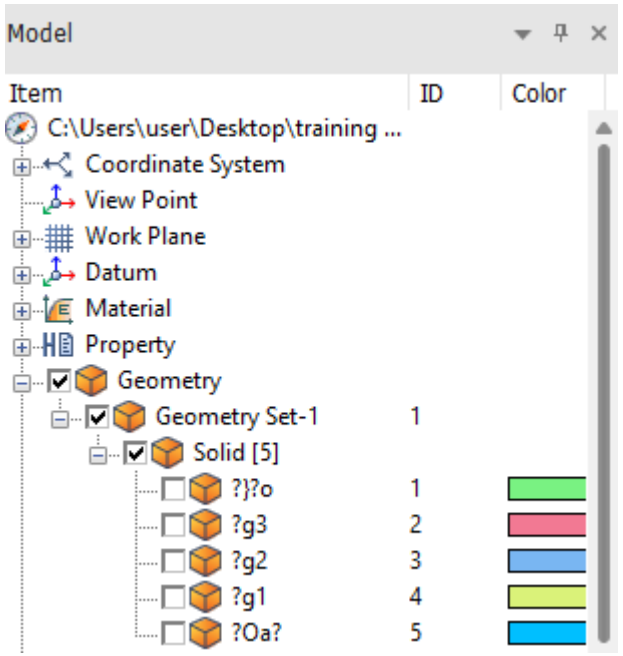


COMPATIBLE FILES



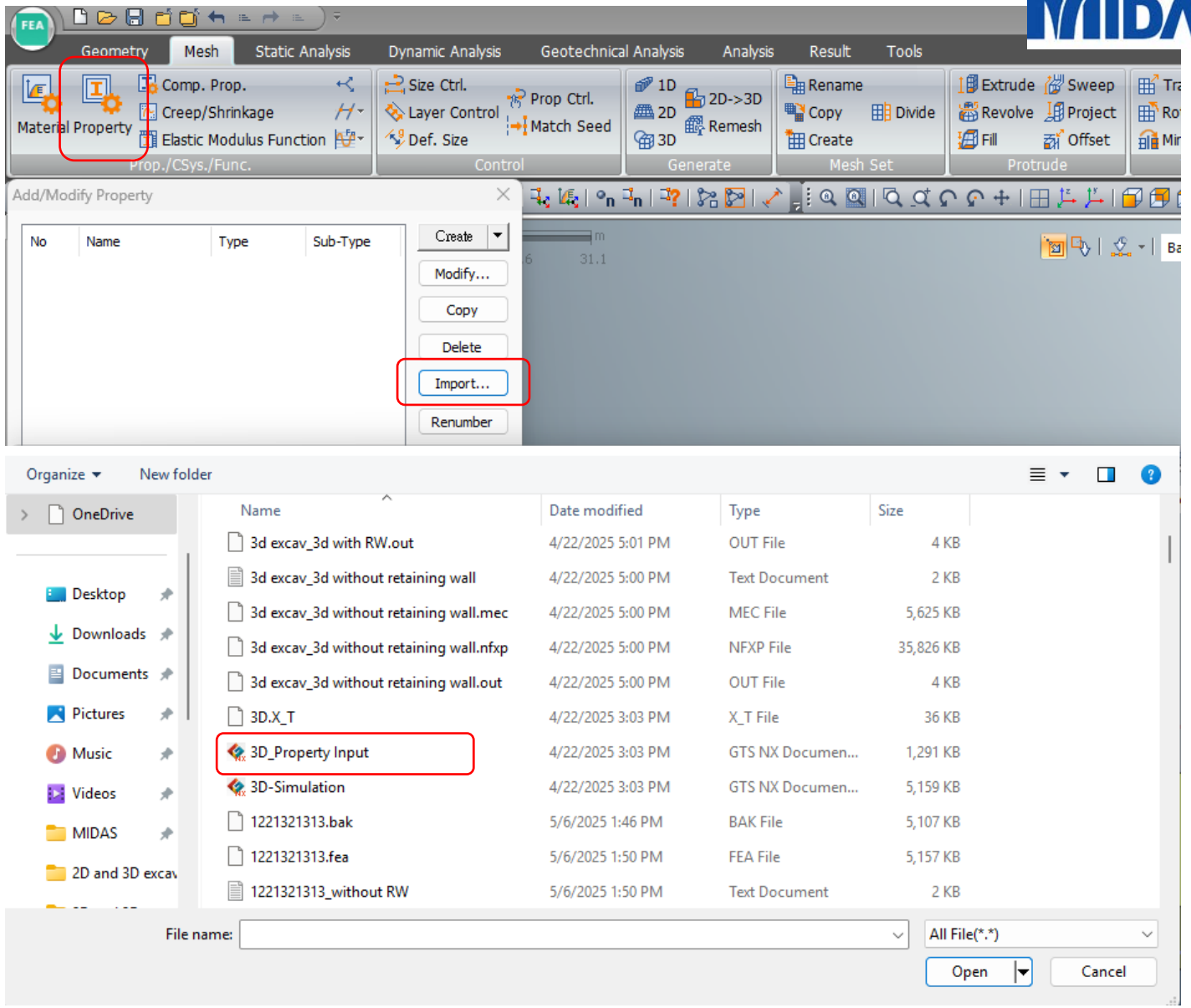
GEOMETRY SET-UP





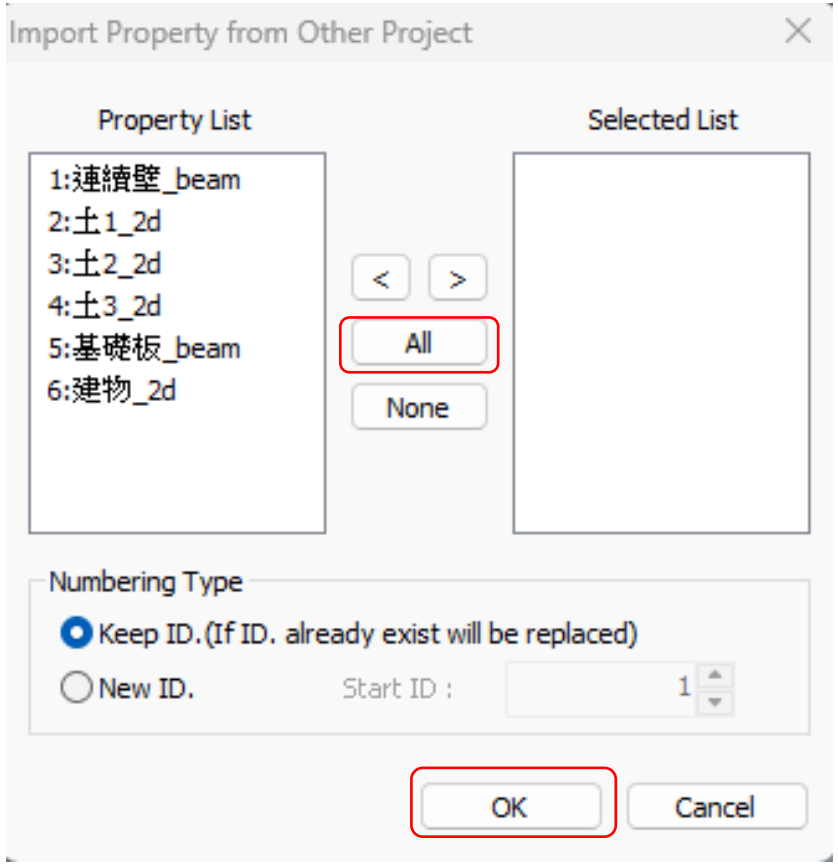
MESHING

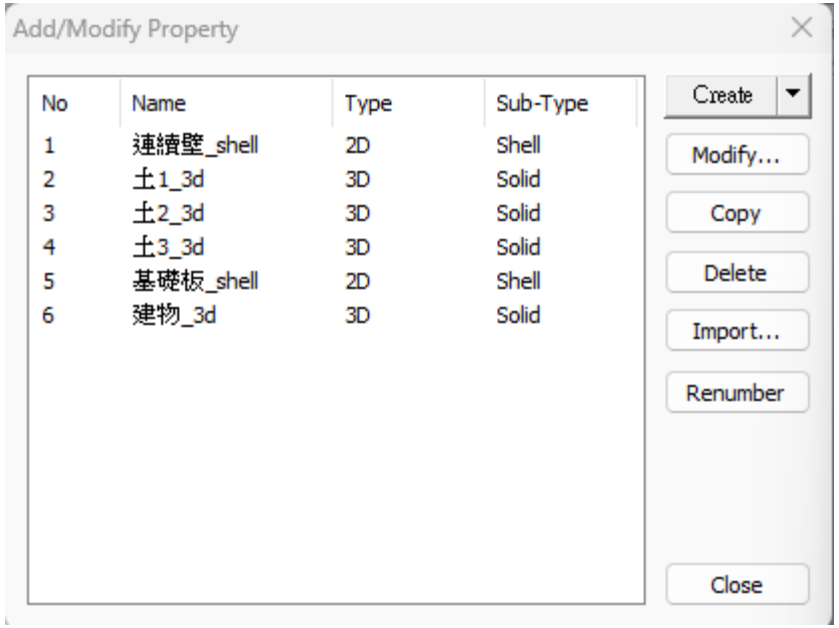
IMPORTING PROPERTY



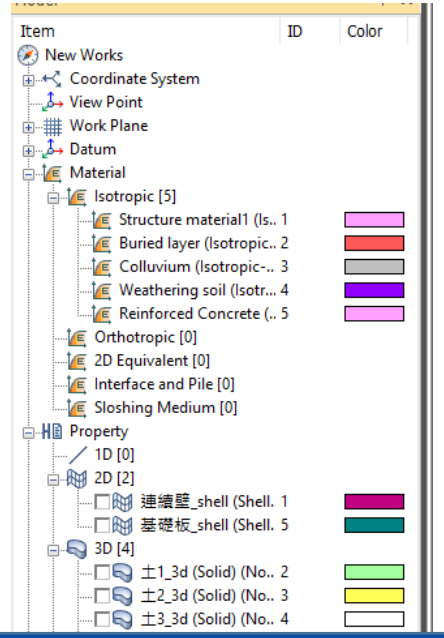
MESHING

IMPORTING PROPERTY





***Properties and materials can be manually added, imported or both.**

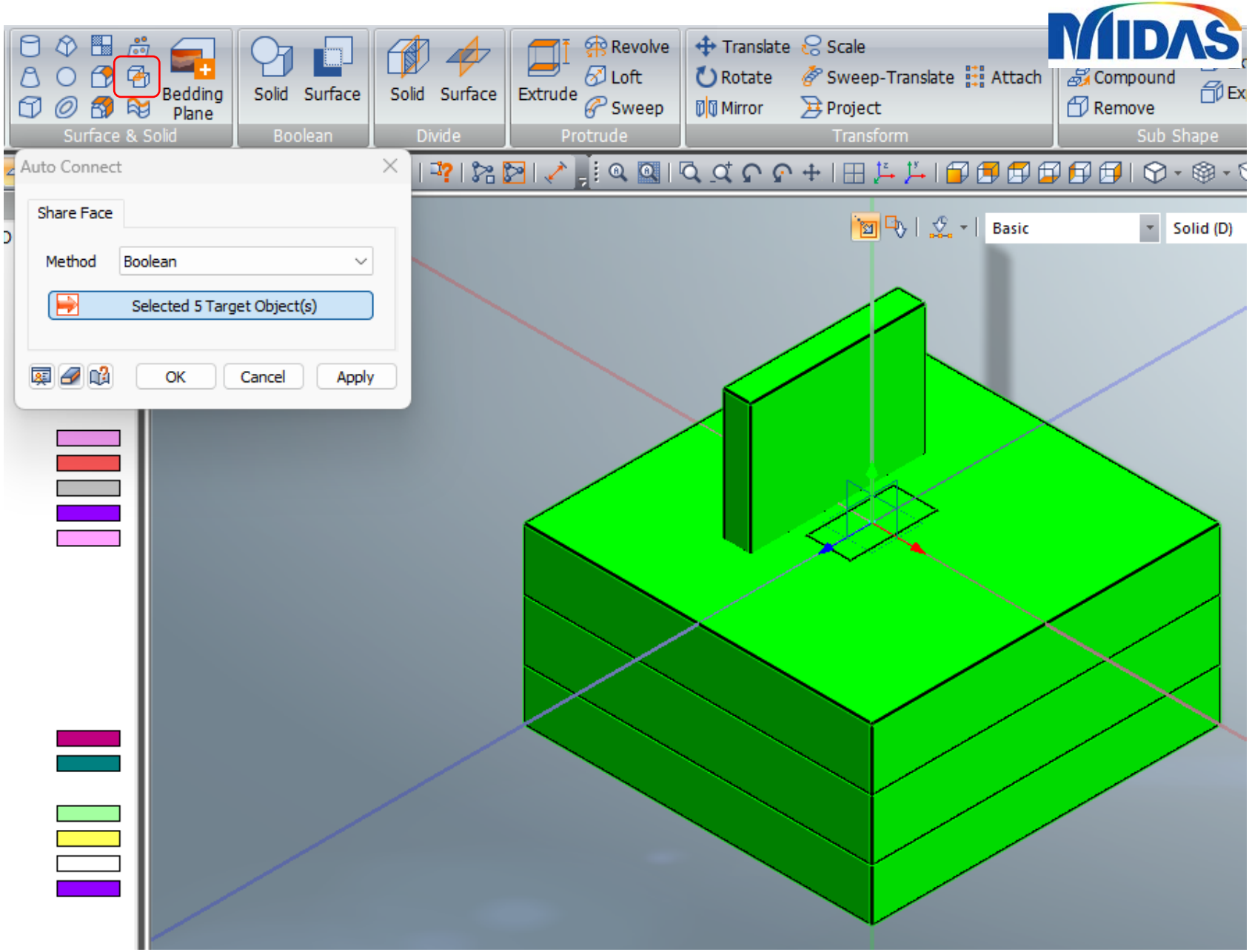


Materials and properties should be reflected in tree model

MESHING

AUTO CONNECT

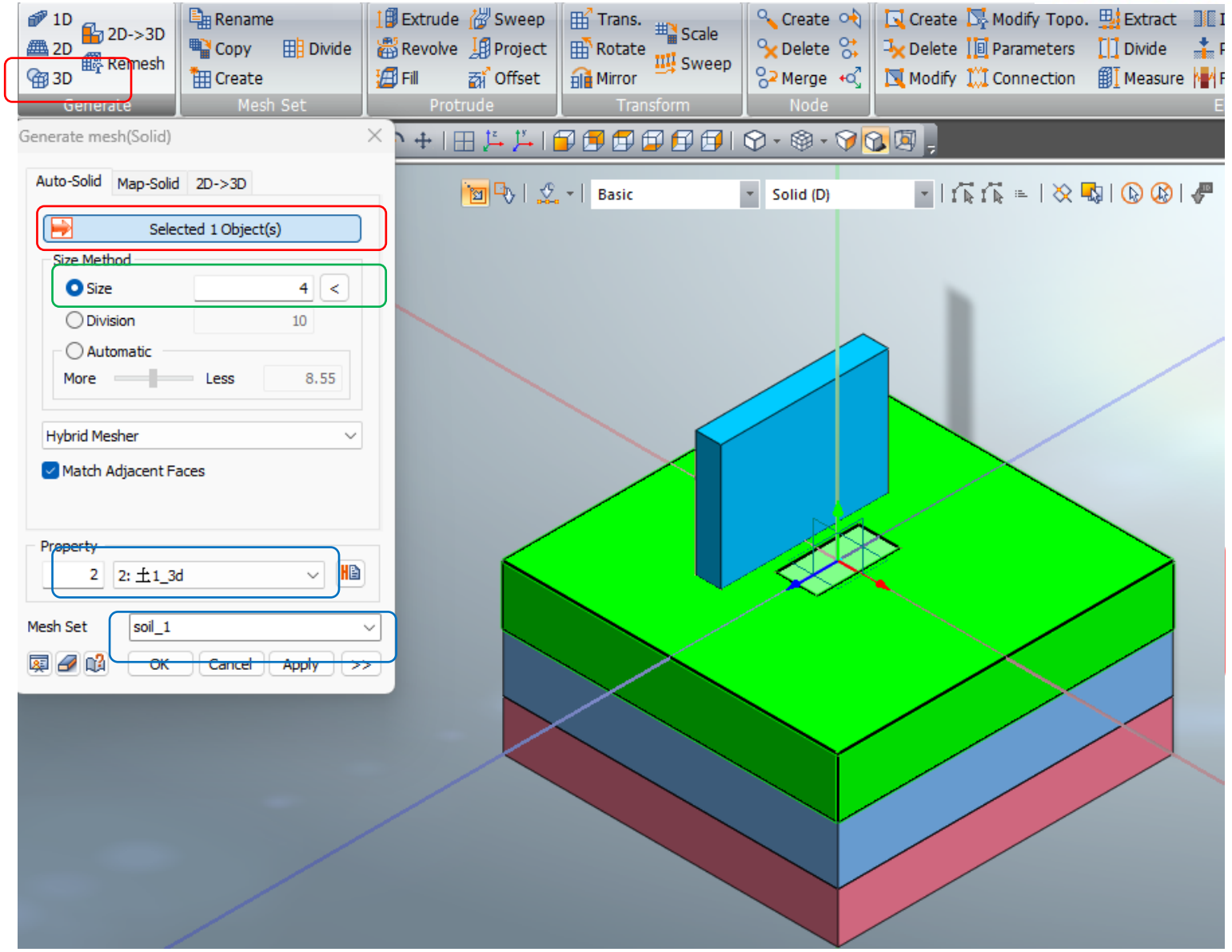
- Connects elements to make a cohesive diagram



MESHING

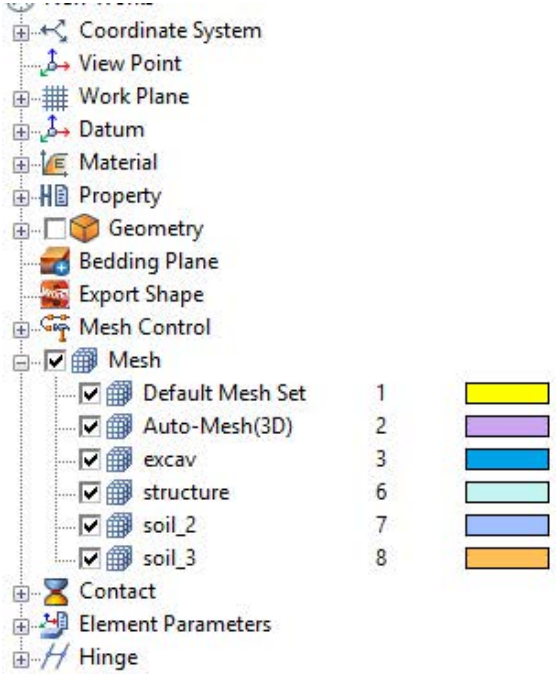
MESHING THE ELEMENTS

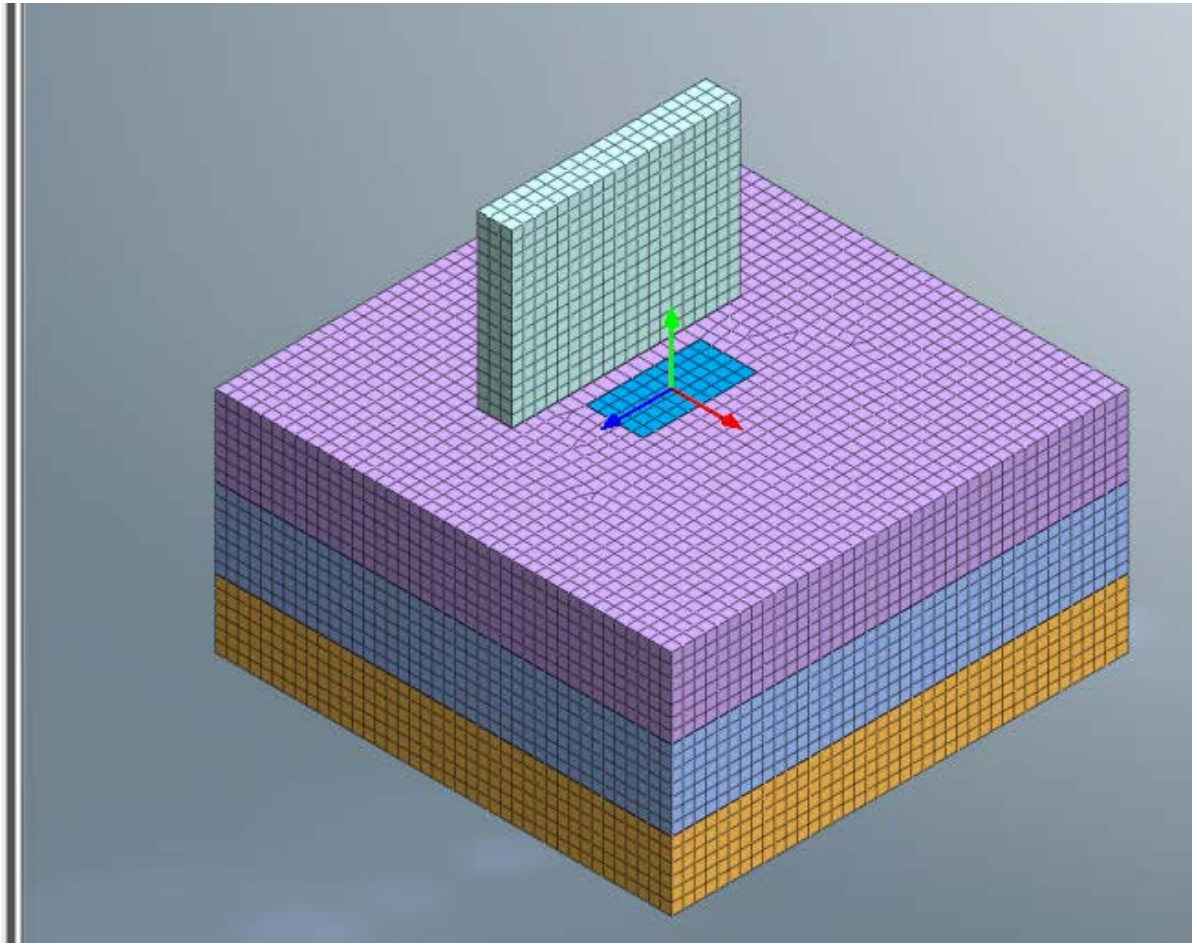
1. 3D > Select the object > highlight the element
2. Define the mesh size (The smaller the size, the more accurate the result but also the more difficult it is for the computer to process)
3. Select the appropriate property for the element
4. Rename the mesh
5. Repeat for all elements in the project



MESHING

MESHING THE ELEMENTS

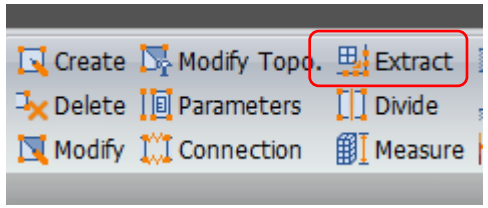


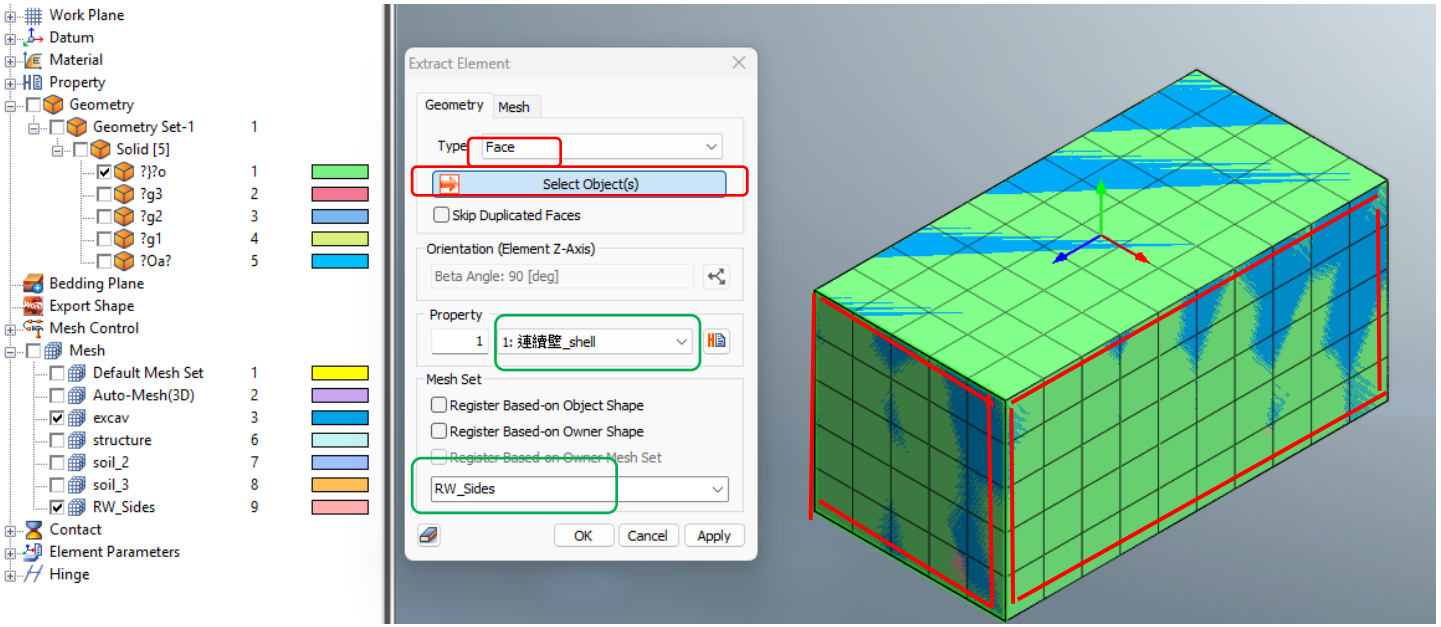


NOTE: After meshing all the elements, it should appear in the drawing, also in the model tree

MESHING

DEFINING THE RETAINING WALL



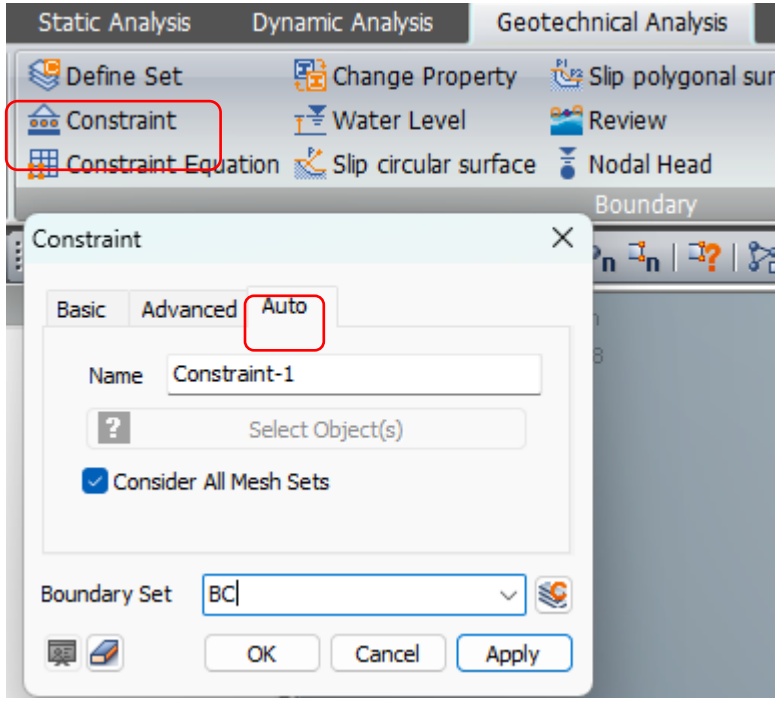


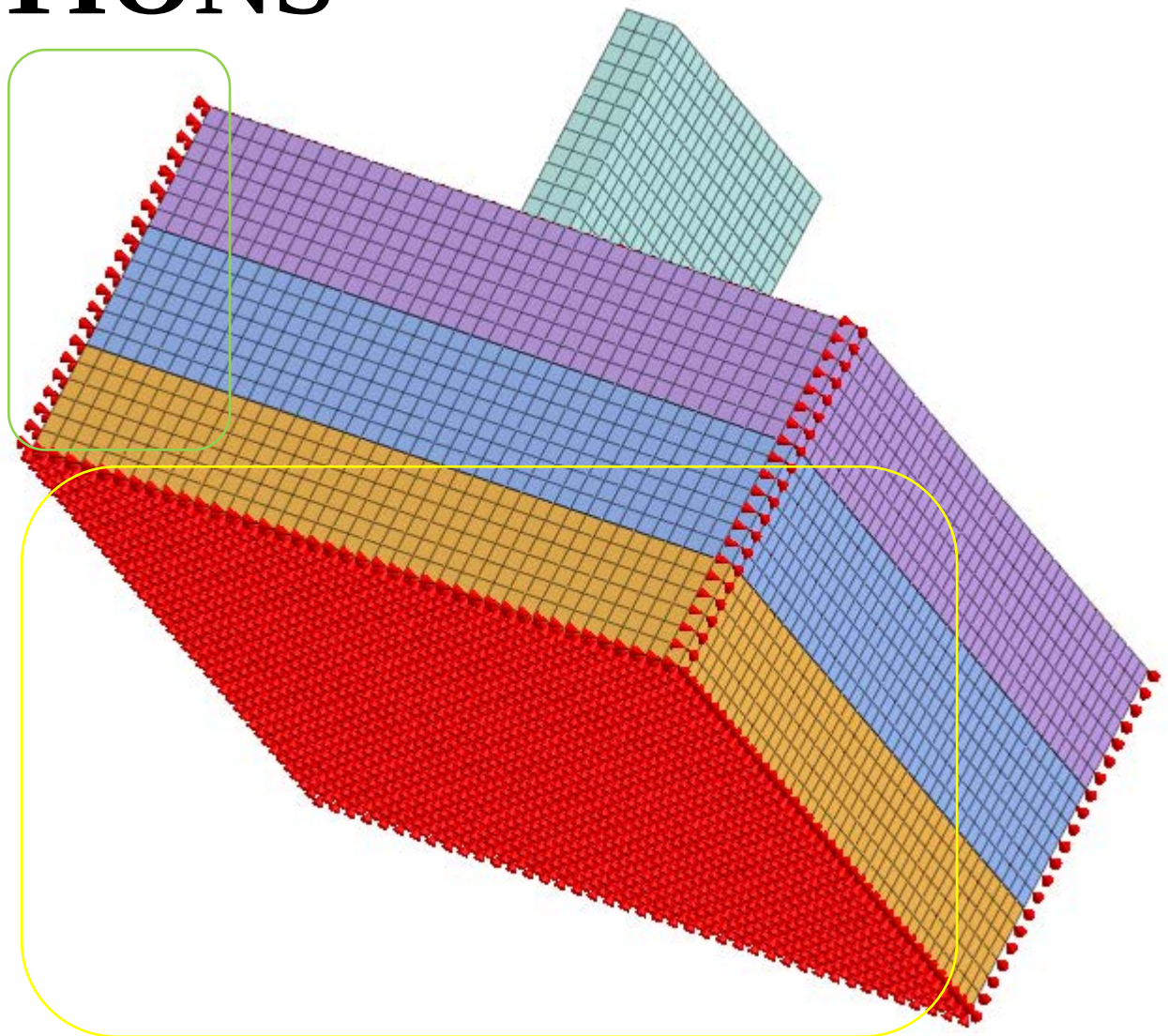
To define a retaining wall

1. Click Extract, use Face selection
2. Select objects/element to become a retaining wall
3. Define the property
4. Rename
5. Apply and repeat to all elements

***Turn off all geometry and mesh except for the excavation to extract the face easier**

BOUNDARY CONDITIONS





To set the boundary conditions

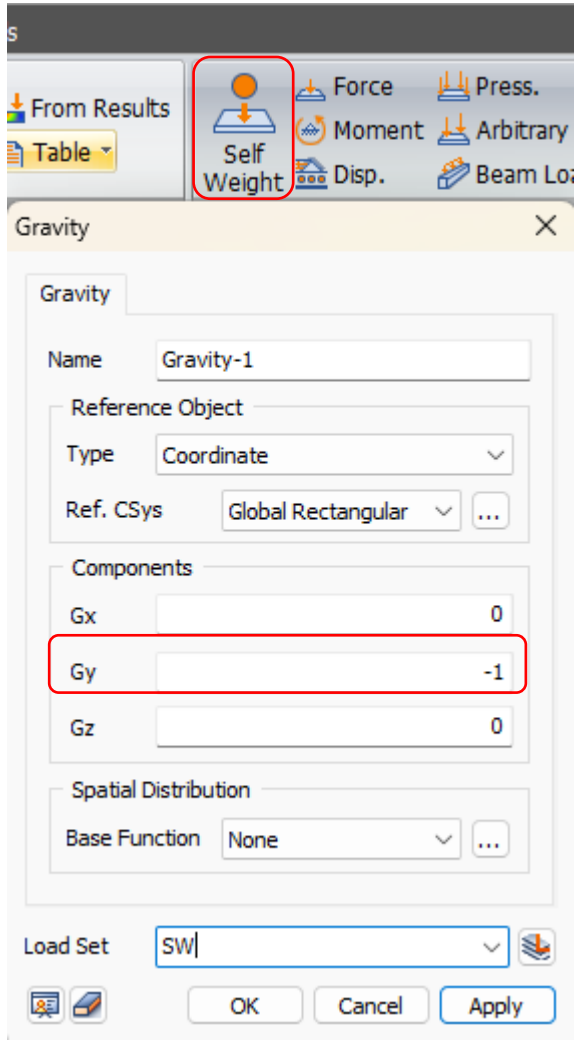
1. Click Constraint

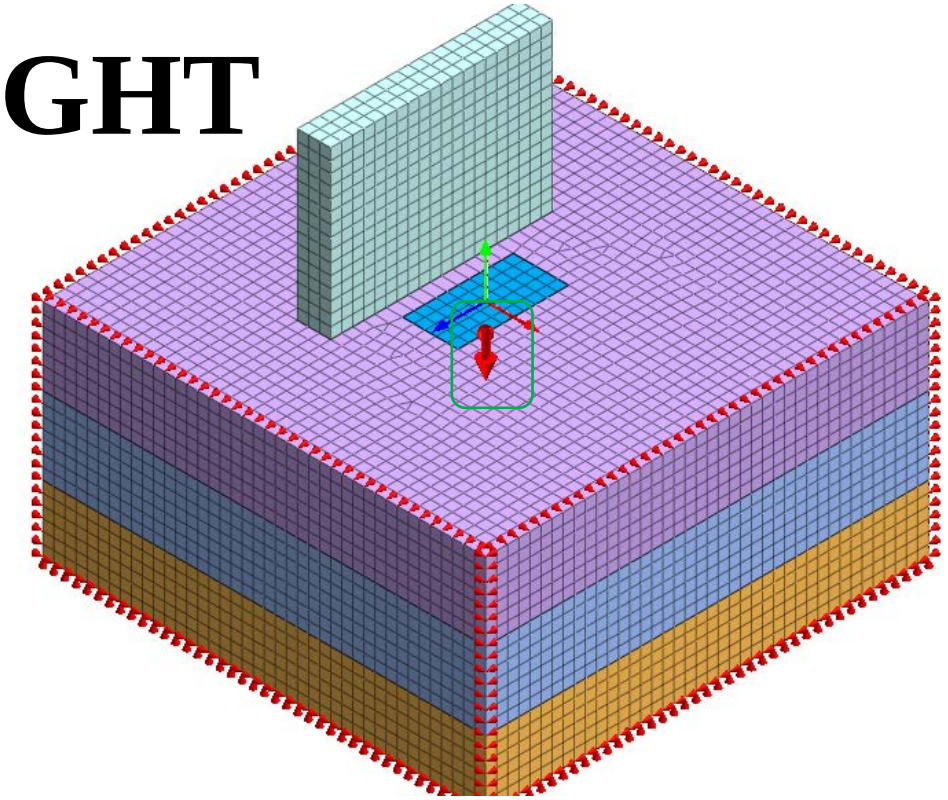
2. Set to auto

3. Rename

Note: The side of the wall will be set to a pin support while the bottom will be set to a fix support

GRAVITY/SELF-WEIGHT





To define set the gravity/self-weight
1. Click Self-weight
2. Define the load to the axis of gravity
3. Rename
Note: Gravity/Self-weight is indicated in the diagram as the downward arrow

ANALYSIS CASE

To define the analysis case

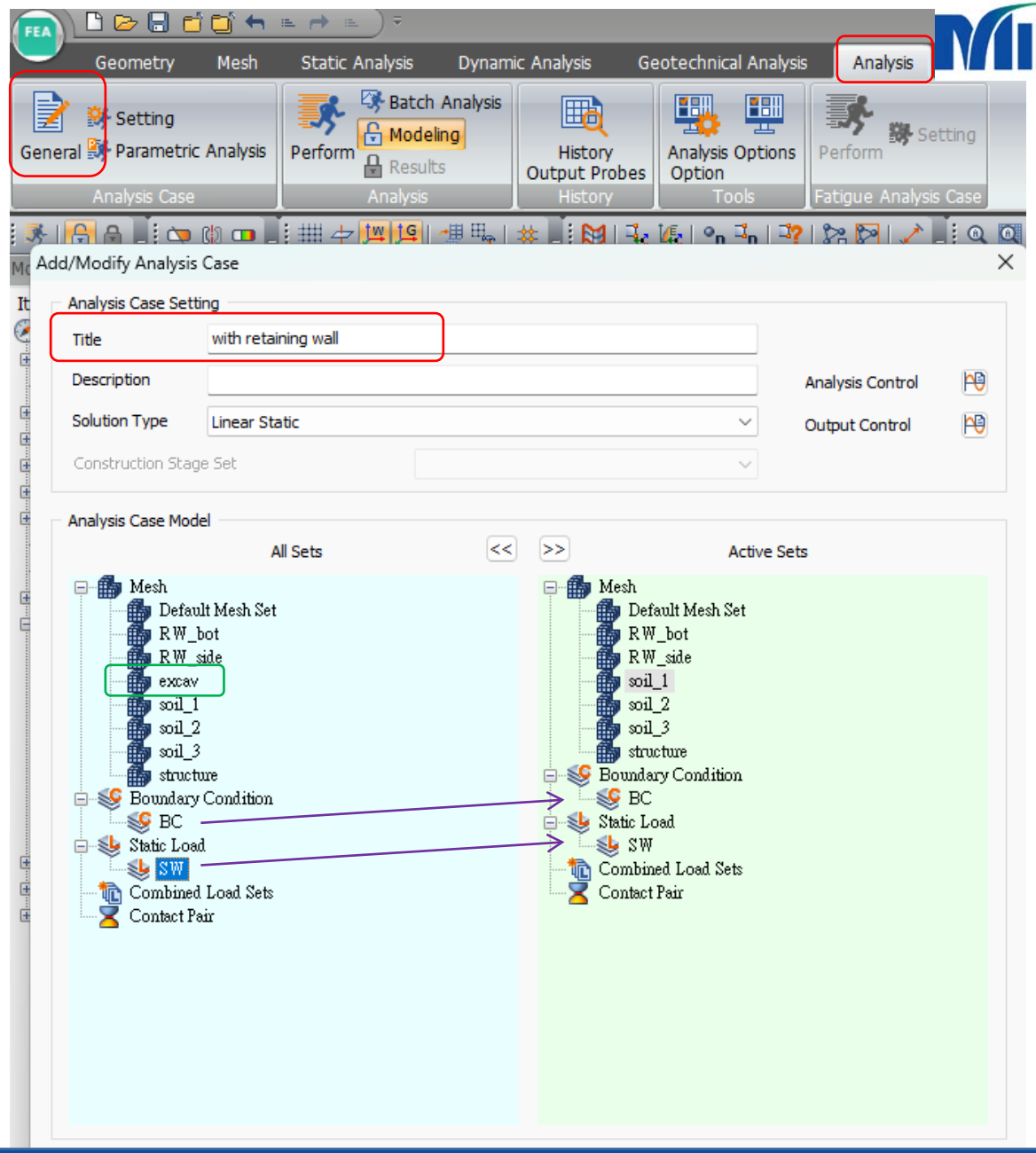
1. Click General in the analysis tab

2. Define a title depending on the project situation

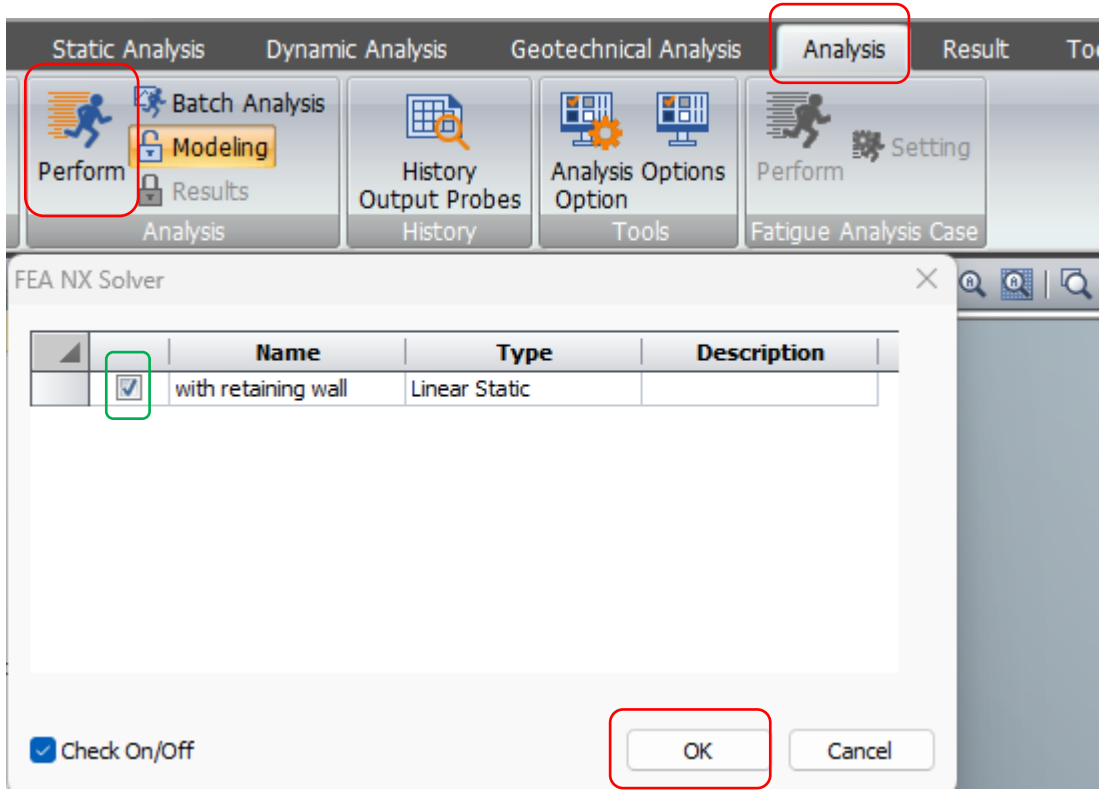
3. Remove excavation mesh from the active sets

4. Activate the boundary condition

5. Activate the self weight

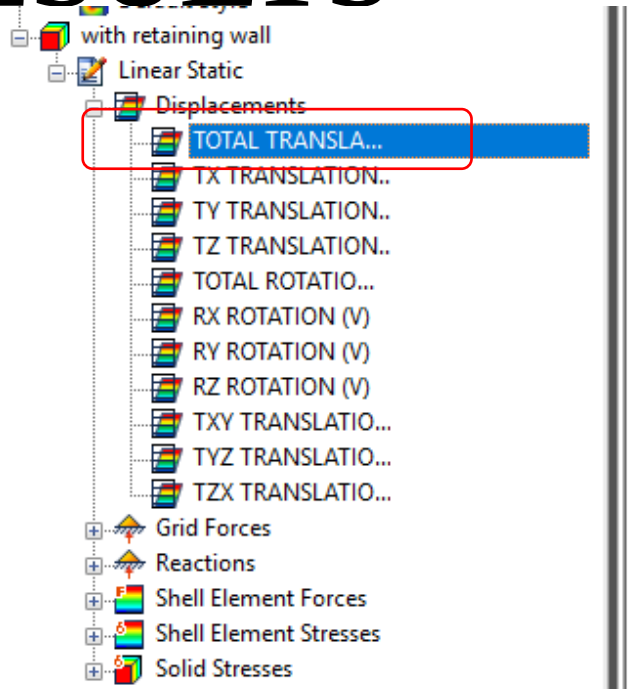


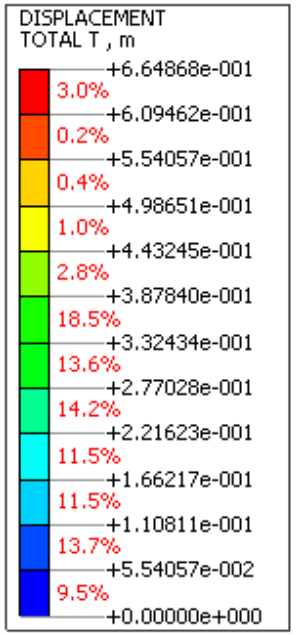
RUN THE CASE

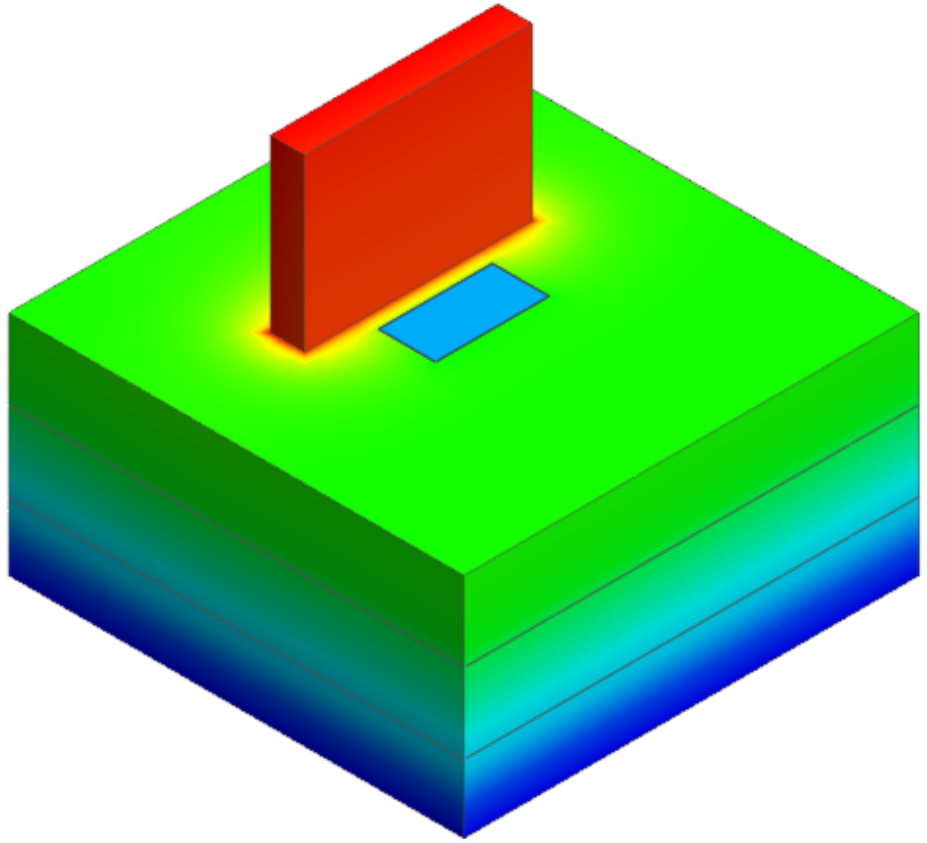


To run the case
1. Click Perform in the analysis tab
2. Activate the analysis case that needed to be solved

RESULTS





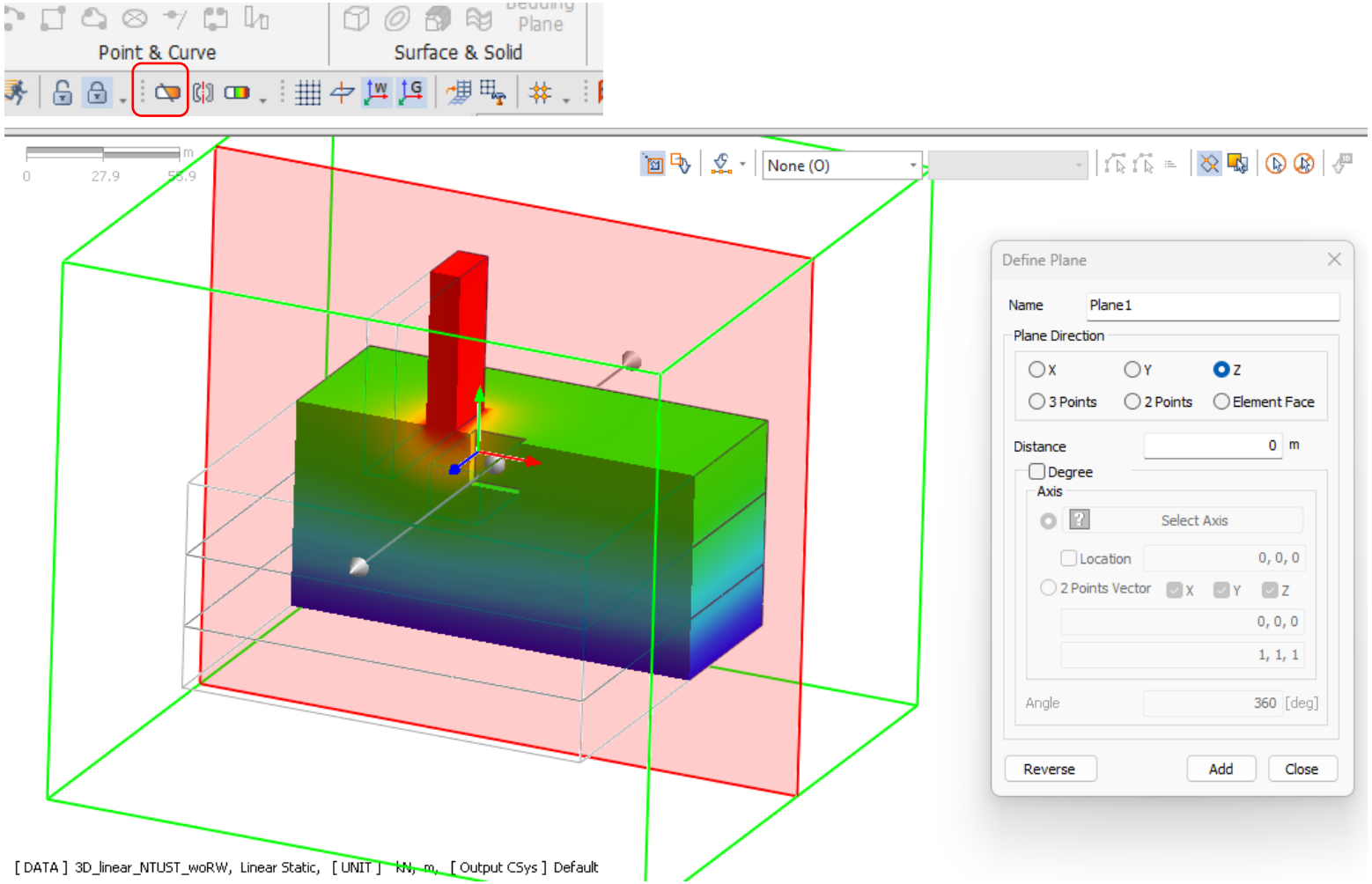


To view results

1. From the model tree, results tab, drop the analysis cases

2. Click the desired result

RESULTS - CLIPPING PLANE



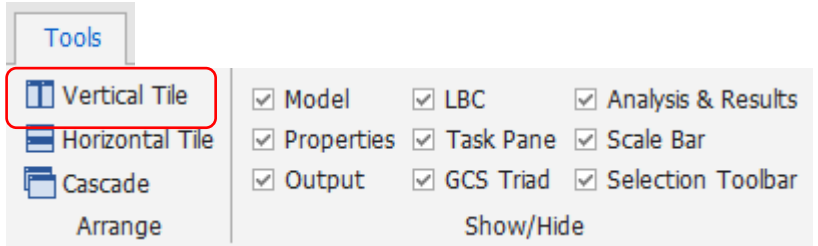
Clipping Plane

- to section along an axis

To use,

1. Select an axis on which the plane will run on to
2. Input a distance or drag the plane from the model.

RESULTS - VERTICAL TILE

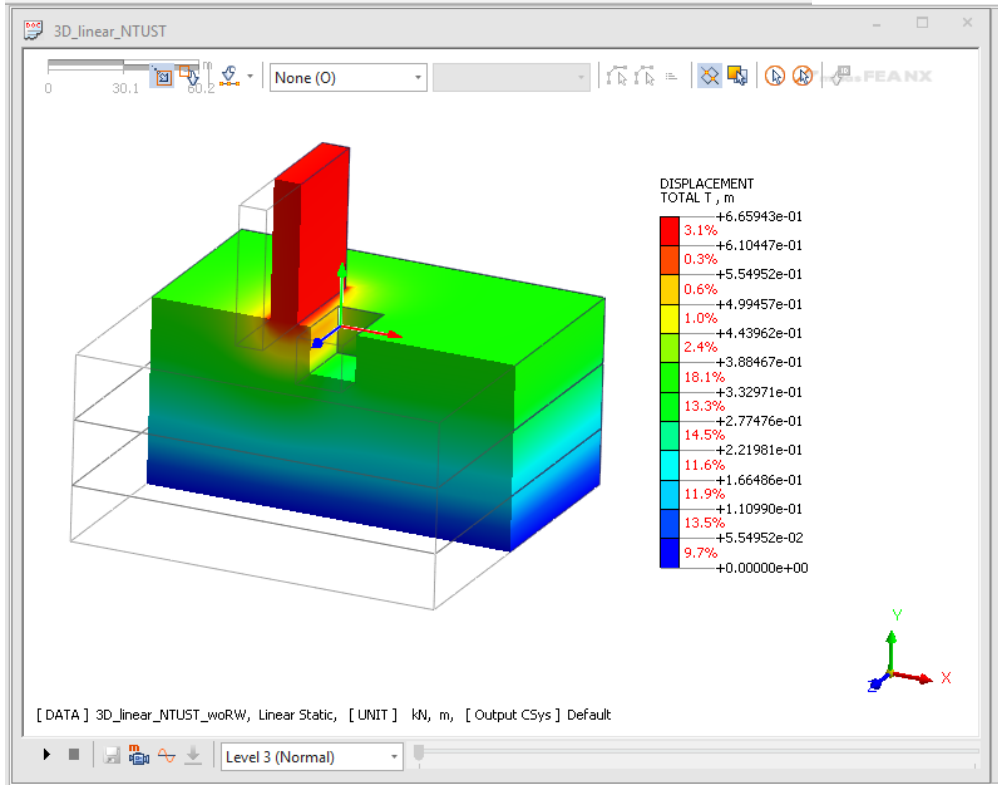


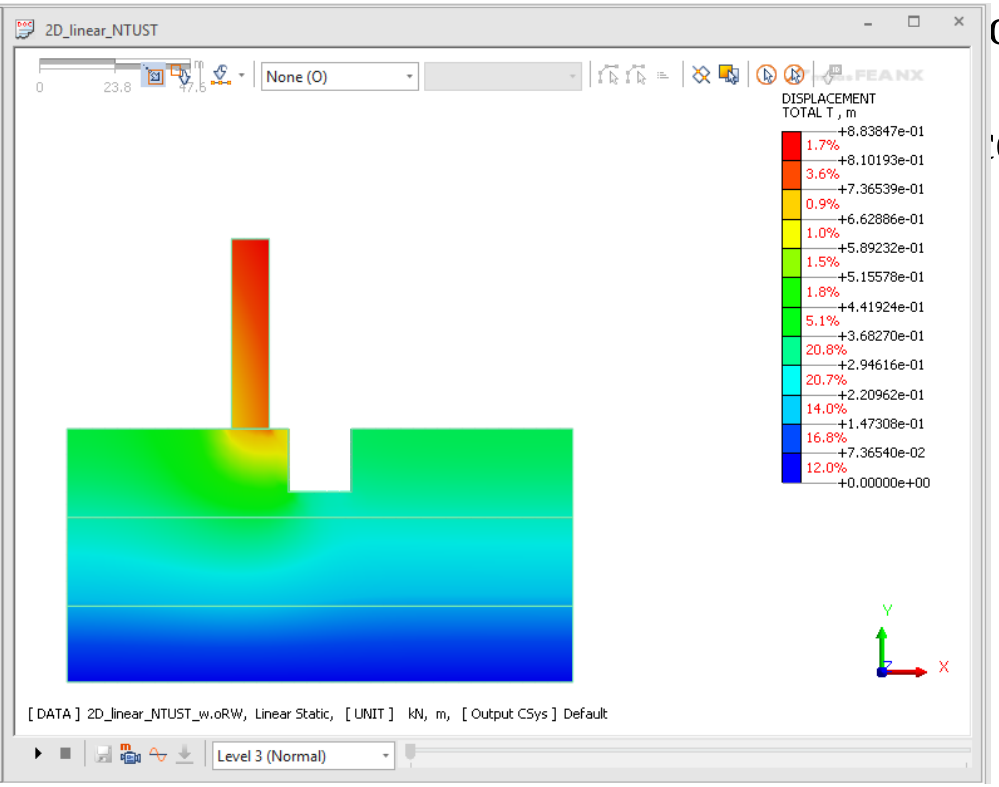
Clipping plane
- to section along an axis

To use,

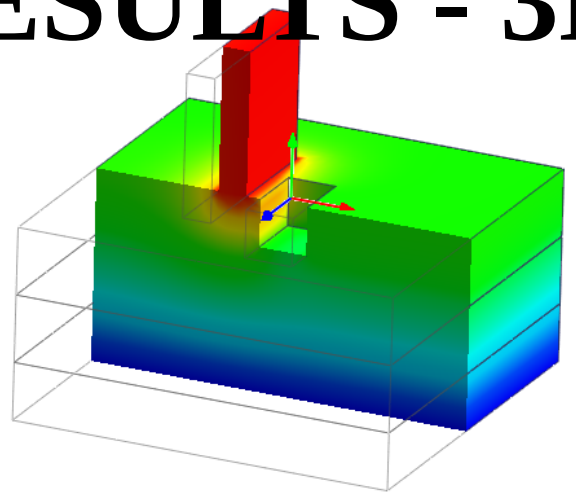
on which the plane

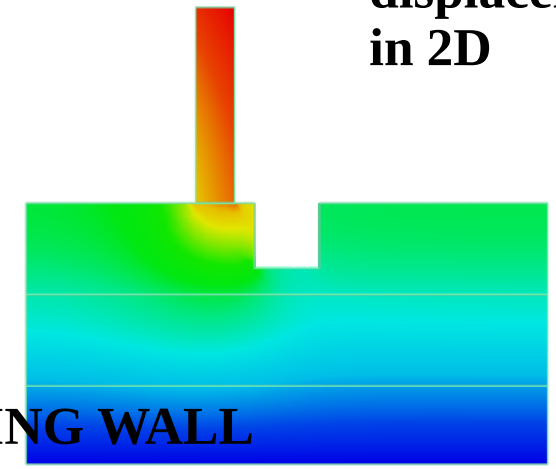
e or drag the plane

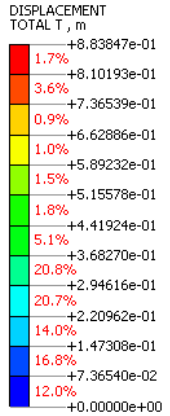




RESULTS - 3D vs 2D



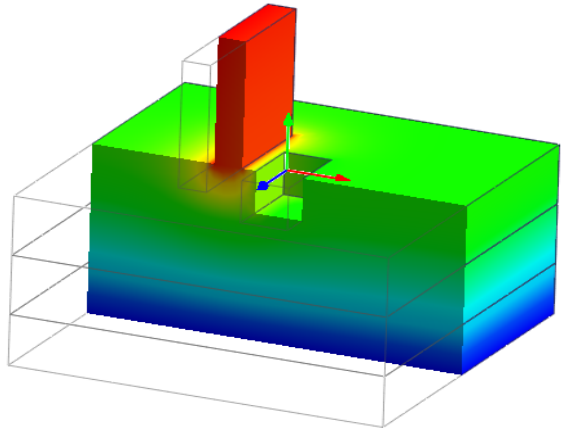


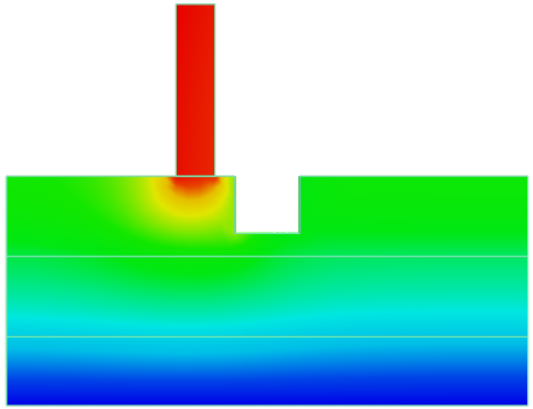


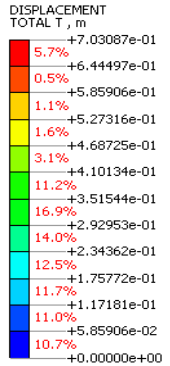
~25% more displacement in 2D

WITHOUT RETINING WALL

WITHOUT RETAINING WALL



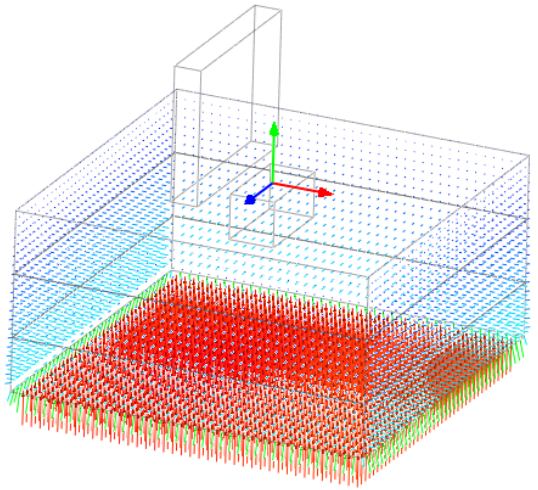


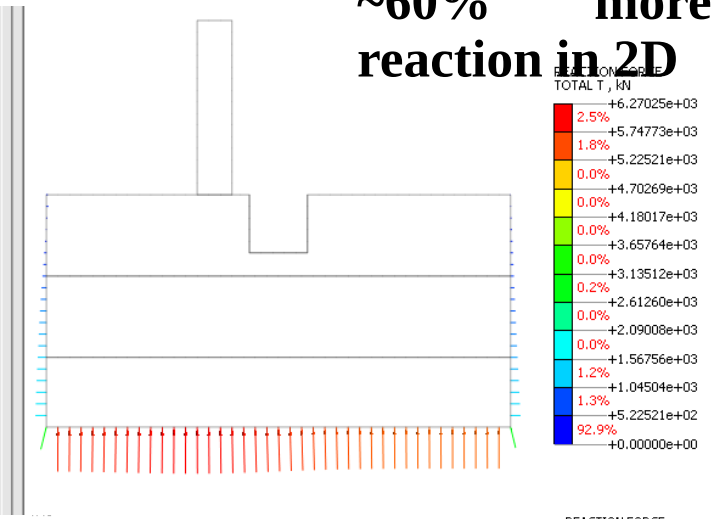


WITH RETAINING WALL

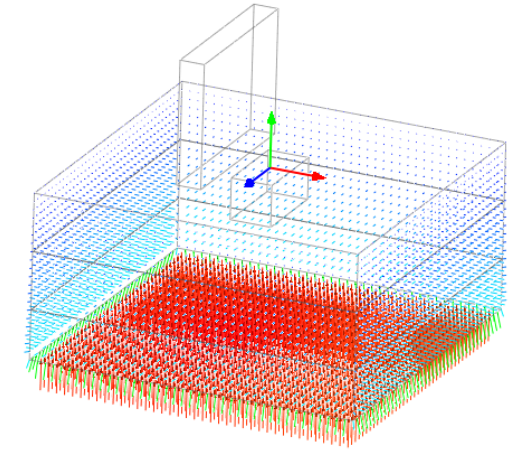
RESULTS - 3D vs 2D

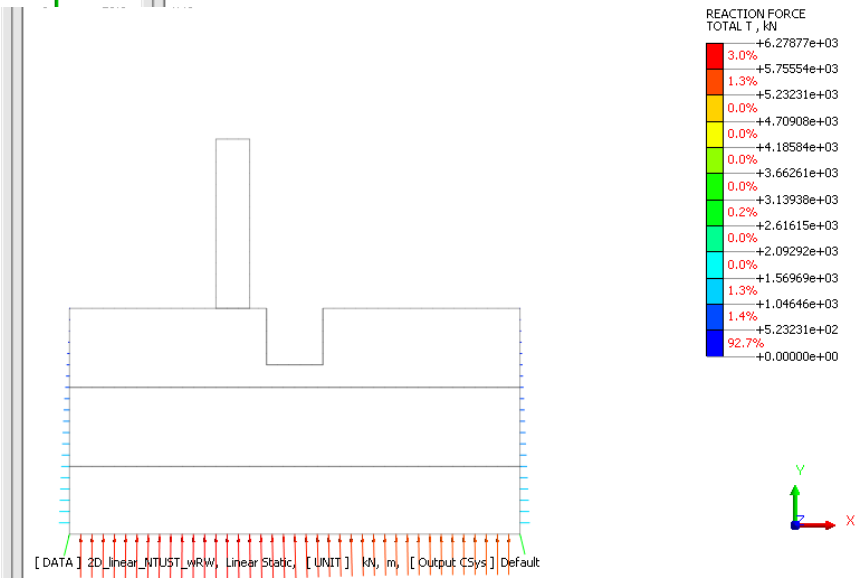
~60% more reaction in 2D





WITHOUT RETAINING WALL

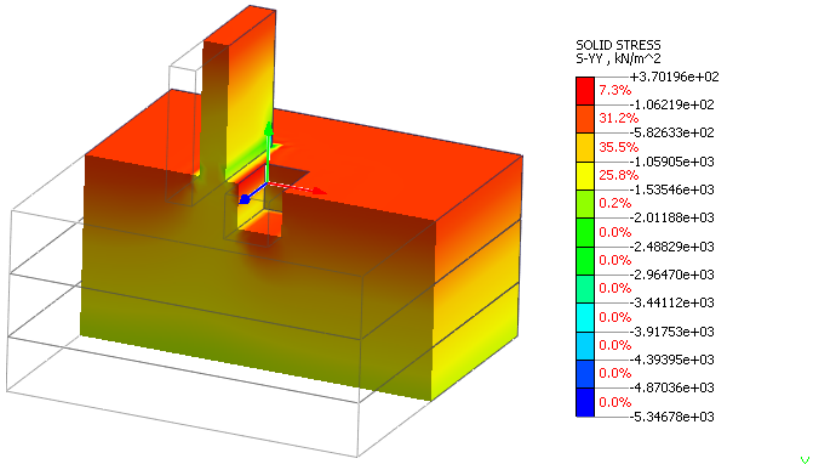




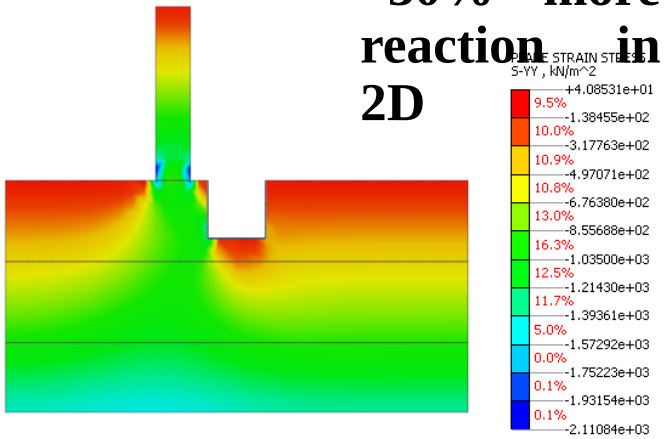
WITH RETAINING WALL

REACTION FORCE

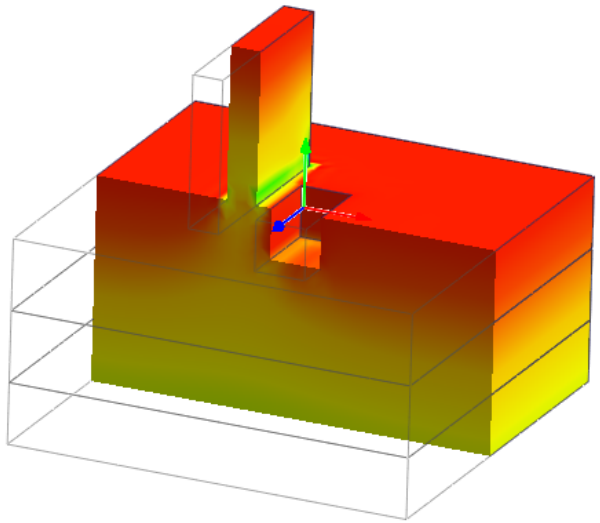
RESULTS - 3D vs 2D

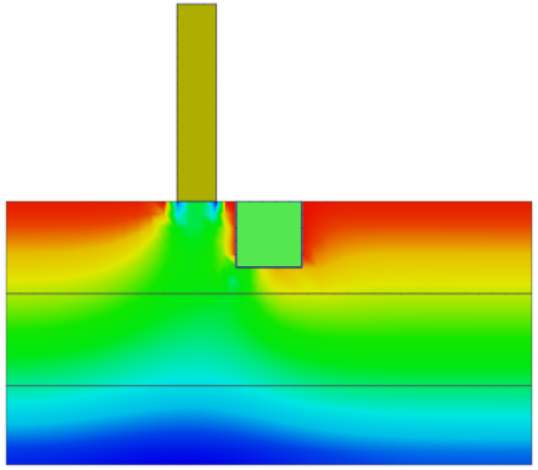


~30% more reaction in 2D



WITHOUT RETAINING WALL





WITH RETAINING WALL

STRESS YY