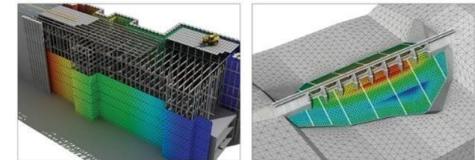# Seismic Time History Analysis for 2D Abutment

GTS NX

Geo-Technical analysis System New eXperience

Modified by Tadavarthi Sree Harsha

Integrated Solver Optimized for the next generation 64-bit platform
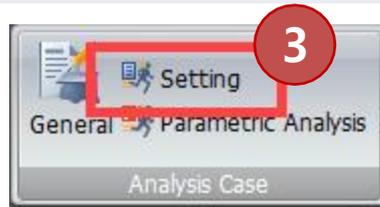**Finite Element Solutions for Geotechnical Engineering**

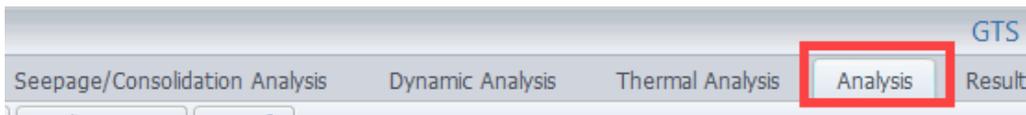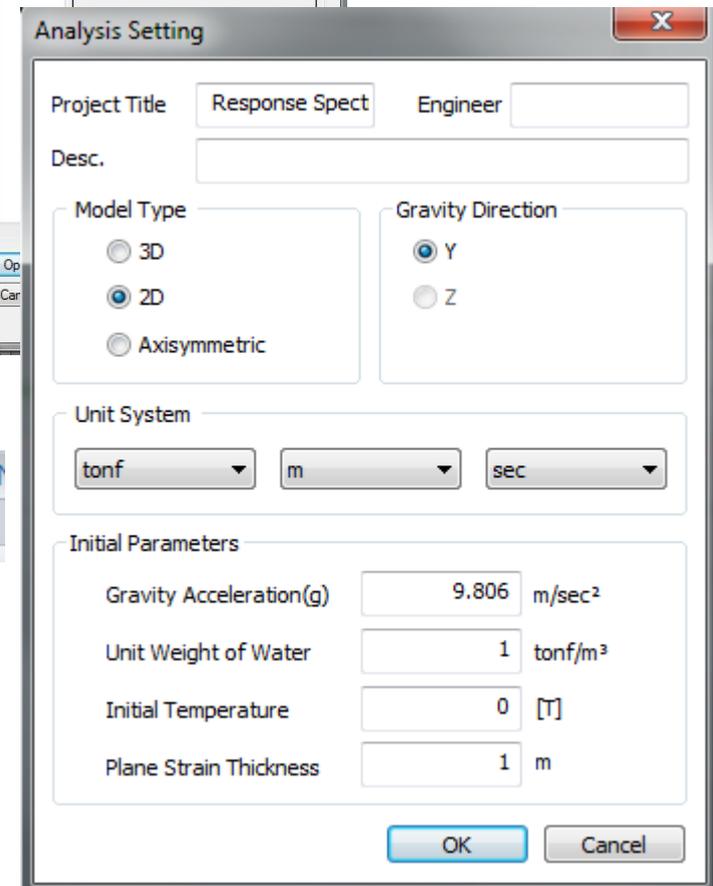**MIDAS**

Objectives

1. **Plane strain elements are used to model both the ground conditions and Abutment.**
2. **Pile elements are modeled as beam elements and embedded in Embankment, Clay and Soft Rock layers.**
3. **Model the load in surrounding ground generated by earthquake and evaluate dynamic behavior and vibration effect of ground and abutment.**
4. **Check the eigenvalue of ground through Eigenvalue analysis, Analyze ground dynamic behavior affected by earthquake.**
5. **Starting Files Required: GTS NX 2D Abutment Time History_start.gts**
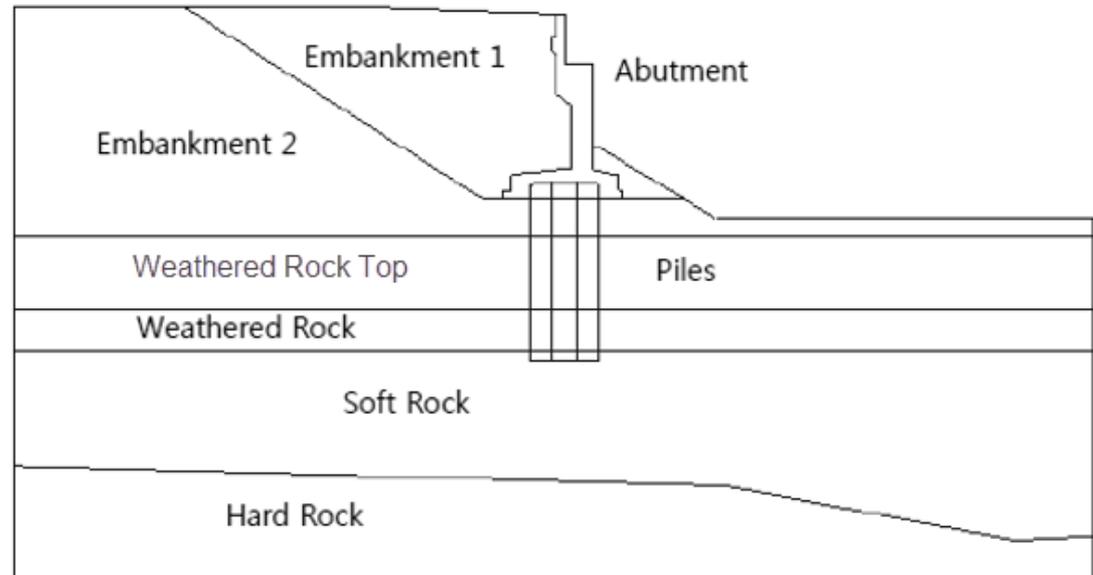
Procedure

1. **Main Menu > File>Open…**
2. **Double click 'GTS NX 2D Abutment Time History_start.gts'.**
3. **Main Menu > Analysis > Analysis Case > Setting > Unit System> tonf>m>s**
4. **Click [OK].**

Open dialog

Look in: 28_2D Seismic Response Analysis of Bridge A

Name	Date modified	Type
GTS NX 2D Tutorials 28_start	10/26/2014 4:26 PM	GTS NX D

☑ Read Result File

No Image

File name:

Files of type: GTS NX Files(*.gts)

Analysis Setting

Project Title: Response Spect   Engineer:

Desc.:

Model Type
- ○ 3D
- ● 2D
- ○ Axisymmetric

Gravity Direction
- ● Y
- ○ Z

Unit System
tonf   m   sec

Initial Parameters

Gravity Acceleration(g)	9.806	m/sec²
Unit Weight of Water	1	tonf/m³
Initial Temperature	0	[T]
Plane Strain Thickness	1	m

OK   Cancel

GTS N

Seepage/Consolidation Analysis   Dynamic Analysis   Thermal Analysis   Analysis   Result

3

Setting
General   Parametric Analysis
Analysis Case

Material Table

ID	Name	Type	Modulus of Elasticity(E) [tonf/m2]	Poisson's Ratio(v)	Unit Weight(Y) [tonf/m3]	Unit Weight (Saturated) [tonf/m3]	Cohesion (c) [tonf/m2]	Friction Angle [Φ]	K0
1	Embankment	Mohr Coulomb	4,000	0.35	1.8	1.85	1.5	25	1
2	Weathered Rock Top	Mohr Coulomb	10920	0.3	2	2	15.92	30	1
3	Weathered Rock	Mohr Coulomb	15000	0.35	2	2	20	32.5	1
4	Soft Rock	Mohr Coulomb	30,000	0.27	2.4	2.4	45	35	1
5	Hard Rock	Mohr Coulomb	300,000	0.2	2.5	2.5	170	38	1
6	Abutment	Mohr Coulomb	232,000	0.19	2.5	2.5	300	36	1
7	Steel	Elastic	21,000,000	0.3	7.8	-	-	-	-

Properties

ID	Name	Type	Subtype
1	Embankment	2D	Plane Strain
2	Weathered Rock Top	2D	Plane Strain
3	Weathered Rock	2D	Plane Strain
4	Soft Rock	2D	Plane Strain
5	Hard Rock	2D	Plane Strain
6	Abutment	2D	Plane Strain
7	Pile	1D	Beam

Procedure

1. **All the Materials and Properties are pre-defined in the 'GTS NX 2D Abutment Time History_start.gts' file.**
2. **Kindly go through the input values to have a better understanding of the material.**

Comp. Prop.
CSys
Function
Hinge
Material Property
Prop./CSys./Func.
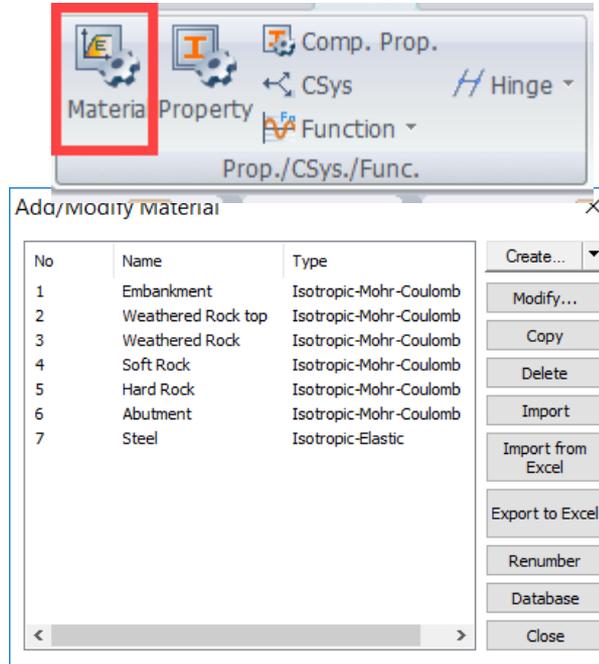
Add/Modify Material

No	Name	Type
1	Embankment	Isotropic-Mohr-Coulomb
2	Weathered Rock top	Isotropic-Mohr-Coulomb
3	Weathered Rock	Isotropic-Mohr-Coulomb
4	Soft Rock	Isotropic-Mohr-Coulomb
5	Hard Rock	Isotropic-Mohr-Coulomb
6	Abutment	Isotropic-Mohr-Coulomb
7	Steel	Isotropic-Elastic

Create...
Modify...
Copy
Delete
Import
Import from Excel
Export to Excel
Renumber
Database
Close

Comp. Prop.
CSys
Function
Hinge
Material Property
Prop./CSys./Func.
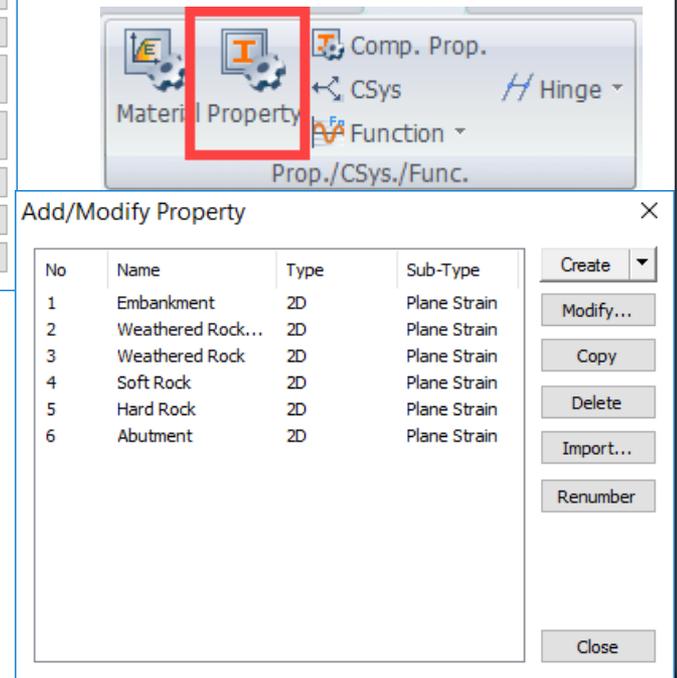
Add/Modify Property

No	Name	Type	Sub-Type
1	Embankment	2D	Plane Strain
2	Weathered Rock...	2D	Plane Strain
3	Weathered Rock	2D	Plane Strain
4	Soft Rock	2D	Plane Strain
5	Hard Rock	2D	Plane Strain
6	Abutment	2D	Plane Strain

Create
Modify...
Copy
Delete
Import...
Renumber
Close

Procedure

1. **Main Menu Model > Property**
2. **Click on [Add] > Select [1D].**
3. **ID '7', Name 'Pile'.**
4. **Element Type > [Beam].**
5. **Select 'Steel' Material**
6. **Click-on 'Sectional Library'**
7. **Select [Pipe], D > Enter '0.508'm, tw > enter '0.012'm.**
8. **Click [OK], Click [OK].**

Add/Modify Property

No	Name	Type	Sub-Type
1	Embankment	2D	Plane Strain
2	Clay	2D	Plane Strain
3	Weathered Rock	2D	Plane Strain
4	Soft Rock	2D	Plane Strain
5	Hard Rock	2D	Plane Strain
6	Abutment	2D	Plane Strain

Create ▾

2

1D...
2D...
3D...
Other...

Import...
Renumber

Close

Create/Modify 1D Property

Pile	Geogrid(1D)	Only(1D)
Truss	Embedded Truss	Beam	edded Beam

4

ID 7   Name Pile   Color ▾

Material   **5**   7: Steel ▾

☐ Taper

	Section-i	Section-j	
Cross Sectional Area(A)	0.0186987595	0.0186987595	m²
Torsional Constant(Ix)	0.00115072166	0.00115072166	m^4
Torsional Stress Coeff.	0.254	0.254	m
Area Moment of Inertia(Iy)	0.0005753608	0.0005753608	m^4
Area Moment of Inertia(Iz)	0.0005753608	0.0005753608	m^4
Effective Shear Area(Ay)	0.0093493797	0.0093493797	m²
Effective Shear Area(Az)	0.0093493797	0.0093493797	m²
Shear Stress Coefficient(Gy)	0.0373829343	0.0373829343	1/m
Shear Stress Coefficient(Gz)	0.0373829343	0.0373829343	1/m

Stress...   Stress...

y Axis Variable   Constant ▾
z Axis Variable   **6**   onstant ▾

☑ Spacing   1.5   m
☑ Section...   Pipe

Section Template

Pipe ▾

D   0.508   m
tw   .012   m

7

Procedure

1. **Main Menu > Mesh > Generate> 2D> Auto Area**
2. **Select Object Edges > Select the edges as shown.**
3. **Mesh Size > Element Size '1',**
4. **Property > Select 'Embankment' Name > 'Embankment'.**
5. **Click [Advanced Option].**
6. **Type [Triangle], Check-off [Register Each Area Independently].Click [OK].**
7. **Click [Apply].**
8. **Similarly create the mesh sets for Weathered Rock Top, Weathered Rock, Soft Rock, Hard Rock & Abutment.**
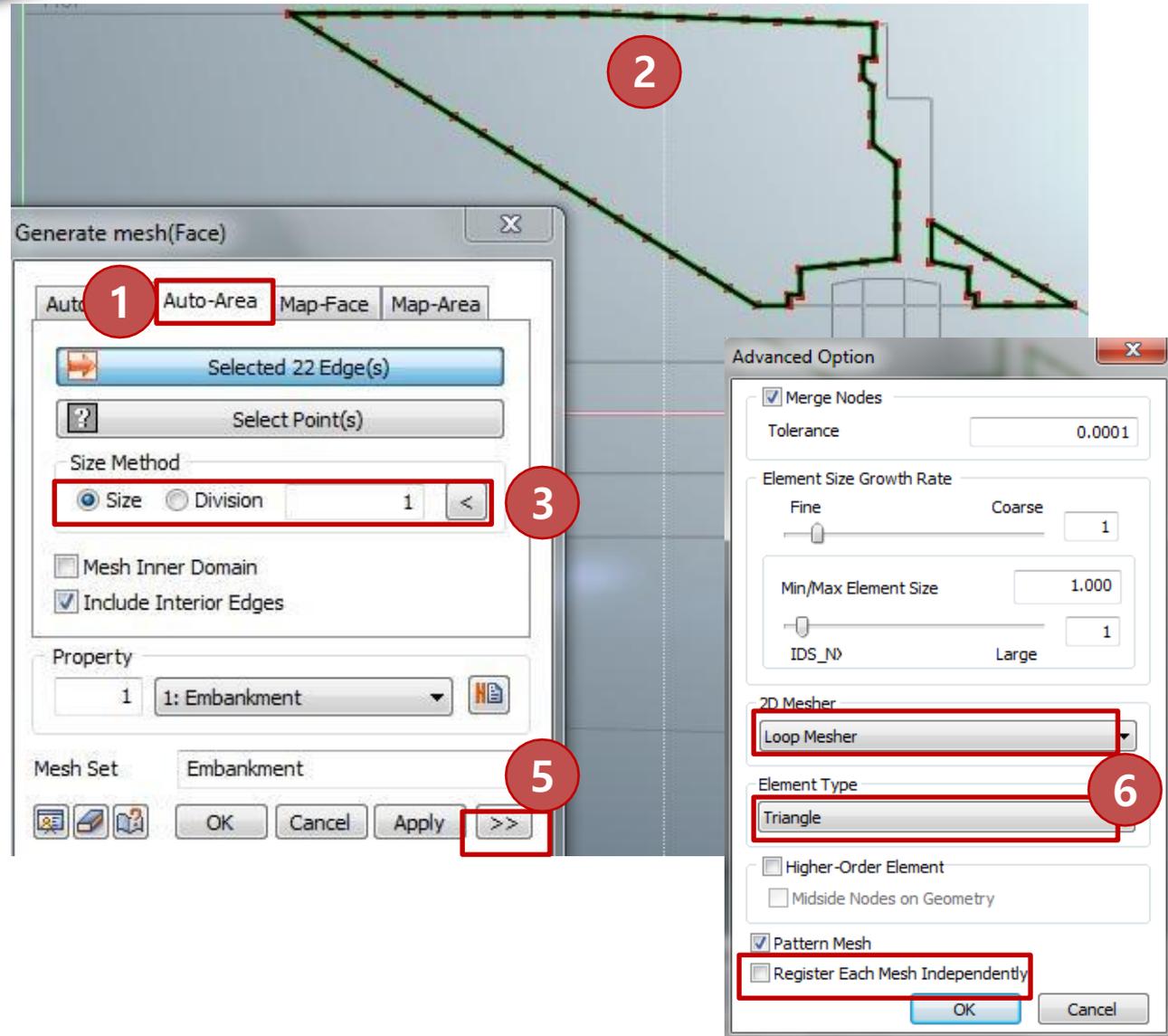
Generate mesh(Face)

Auto | Auto-Area | Map-Face | Map-Area

Selected 22 Edge(s)

Select Point(s)

Size Method
- Size ○ Division 1 <

☐ Mesh Inner Domain
☑ Include Interior Edges

Property
1 | 1: Embankment

Mesh Set Embankment

OK | Cancel | Apply | >>

Advanced Option

☑ Merge Nodes
Tolerance 0.0001

Element Size Growth Rate
Fine Coarse 1

Min/Max Element Size 1.000
IDS_N) Large 1

2D Mesher
Loop Mesher

Element Type
Triangle

☐ Higher-Order Element
☐ Midside Nodes on Geometry

☑ Pattern Mesh
☐ Register Each Mesh Independently

OK | Cancel
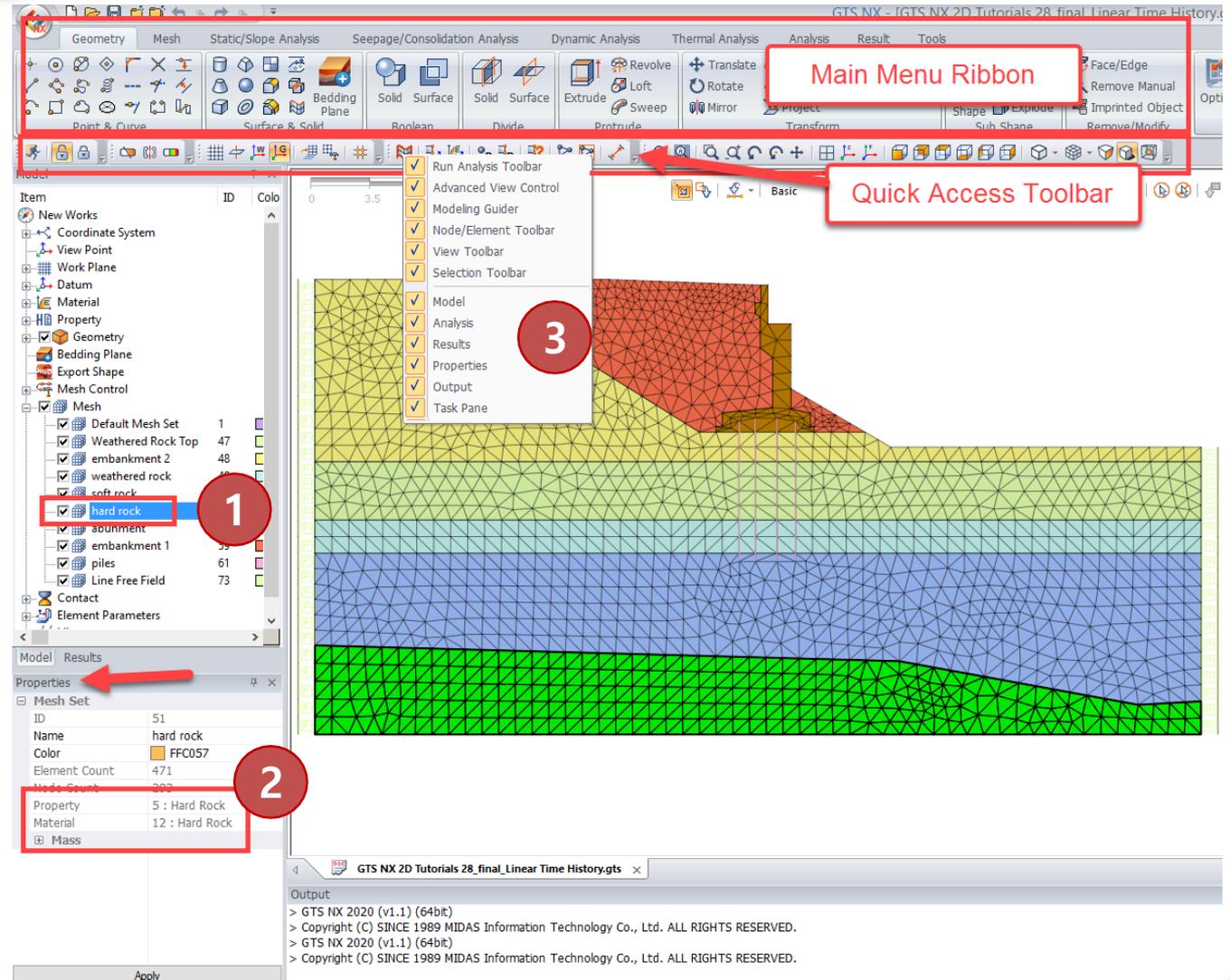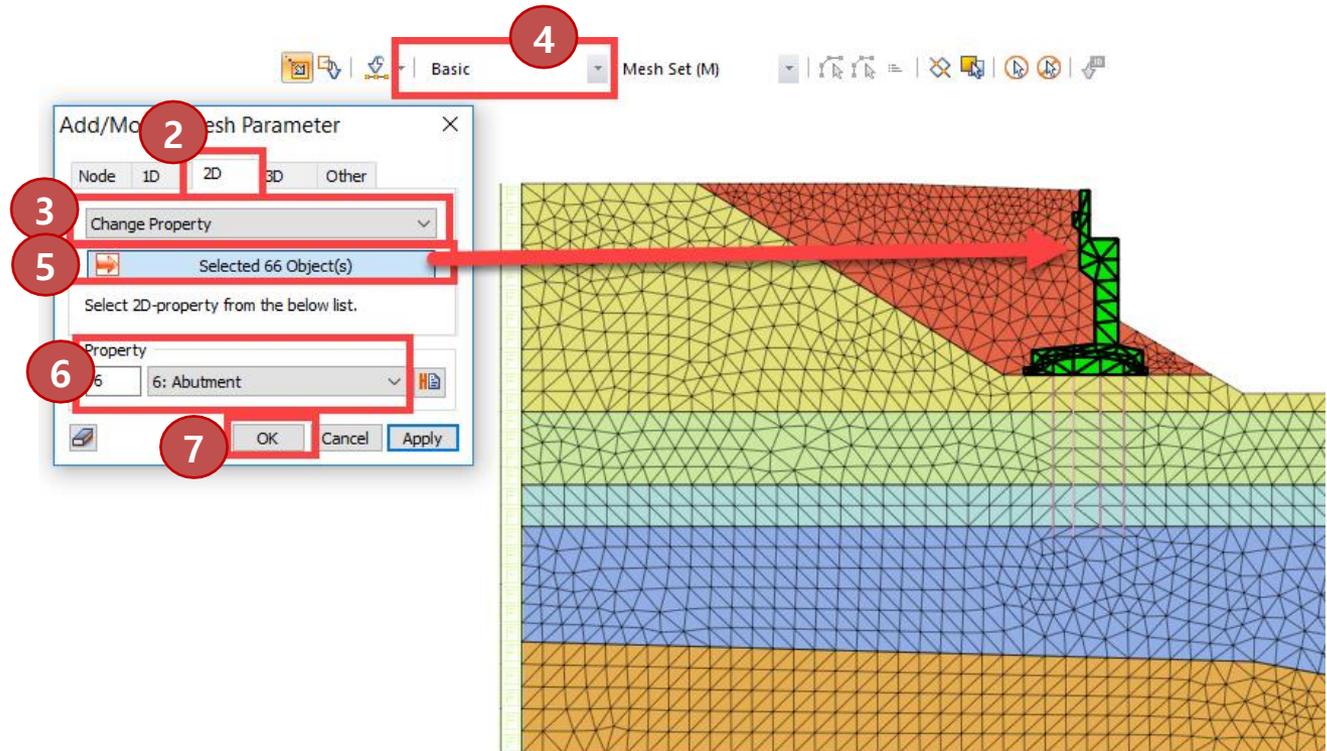
Procedure

The purpose of the Additional information slide is to provide the user a better understanding of the Graphic User Interface.

Let's see the trick to mesh the piles without actually 'meshing', but by 'Extracting'.

1. The idea is to extract the piles mesh from the existing 2D elements and nodes.
2. Hence while meshing the soil layers, we need select the piles geometry (lines) along with the soil layer boundaries.
3. By doing this, we can find nodes connecting piles geometry (lines).
4. This would help us in extracting the piles mesh. The procedure for 'Extraction' will be dealt in next page.
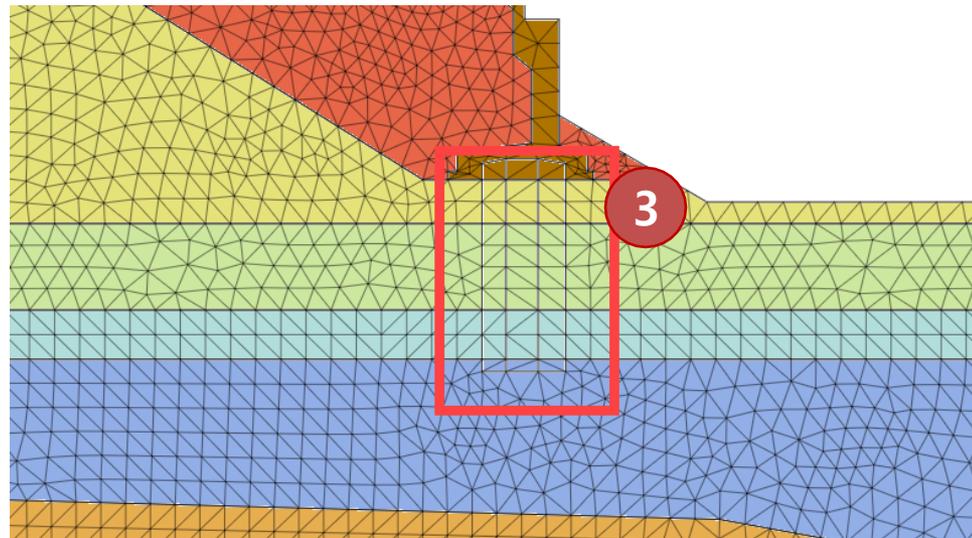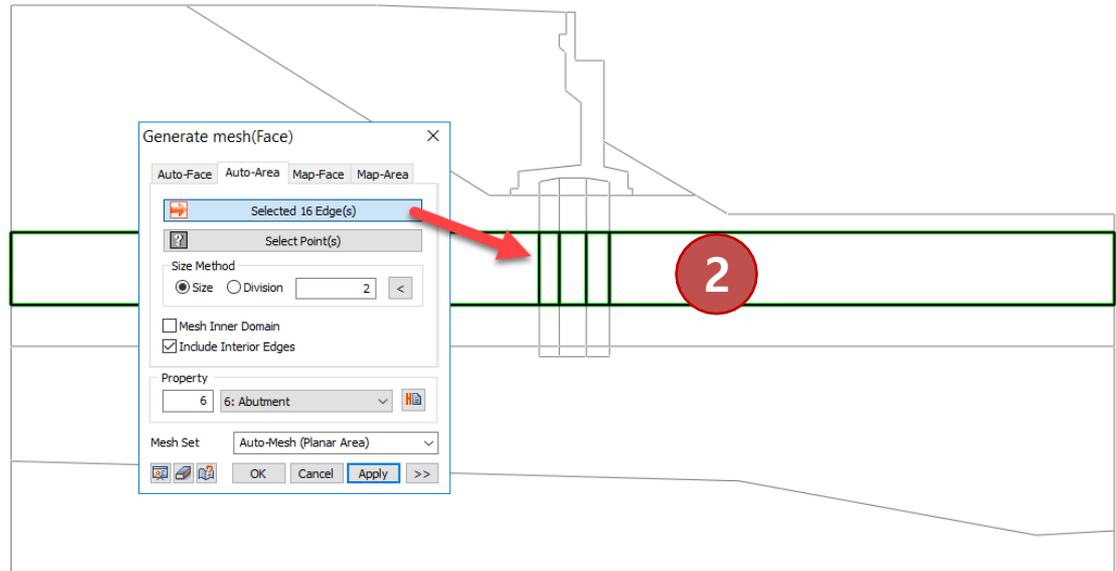
GTS NX

Procedure

1. **Main Menu >
   Mesh > Element
   >Extract Element**
2. **From Geometry >
   Select [Edge].**
3. **Select the 20
   edges of the Pile
   as shown.**
4. **Property ID > ' 7 :
   Pile '.**
5. **Mesh Set > Enter
   name 'Piles'.**
6. **Click [OK]**

Style Back

Create Modify Topo. **Extract**
Delete Para Divide In
Modify Connection Measure Pi

Element

Extract
Extract Element

Extract Element

Geometry | Mesh

ype Edge

Selected 20 Object(s)

Skip Duplicated Faces

Orientation (Element Z-Axis)

Beta Angle: 90 [deg]

Property

7 7: Pile

Mesh Set

☐ Register Based-on Object Shape

☐ Register Based-on Owner Shape

☐ Register Based-on Owner Mesh Set

Pile

OK Cancel Apply

Additional Information

Procedure

The purpose of the Additional information slide is to provide the user a better understanding of the Graphic User Interface.

Let's see how to check the material and property that was assigned to the Mesh sets.

1. Select any mesh set.
2. In the Works tree > Properties tab, we can find the assigned material and the property to the mesh set.
3. If the 'Properties Works tree' is not opened on your PC, then right click on the 'Quick Access Toolbar' to enable it.

Additional Information

Procedure

The purpose of the Additional information slide is to provide the user a better understanding of the Graphic User Interface.

Let's see how to 'change' the property when a incorrect one was assigned to the Mesh set.

1. Go to Mesh > Element > Parameters
2. Go to 2D.
3. In the dropdown menu, select 'Change Property'
4. Change the filter to 'Basic'
5. Select the mesh set to which property has to be changed.
6. Select the correct Property.
7. Click 'OK'

Procedure

1. **Works Tree > Mesh > Mesh set > Pile. Right Click to invoke Context Menu > Display > Element Csys.**
2. **As can be seen the Element Csys are not aligned in the same direction. This will result in wrong display of results.**
3. **Main Menu > Mesh > Element > Change Element Parameter.**
4. **In Selection filter select [1D]>> Change Cys.**
5. **"Base Element" select any element whose Z axis is parallel to Global X-axis**
6. **Select the "Pile" mesh set from the works tree.**
7. **Click [OK].**
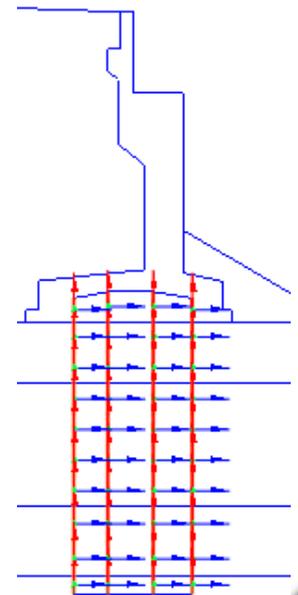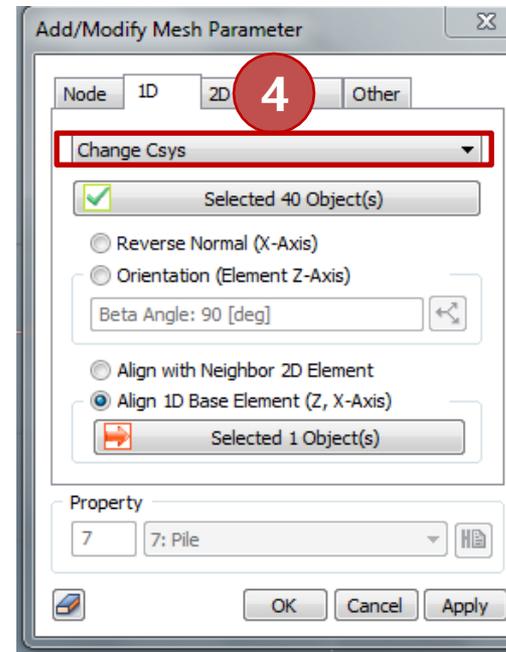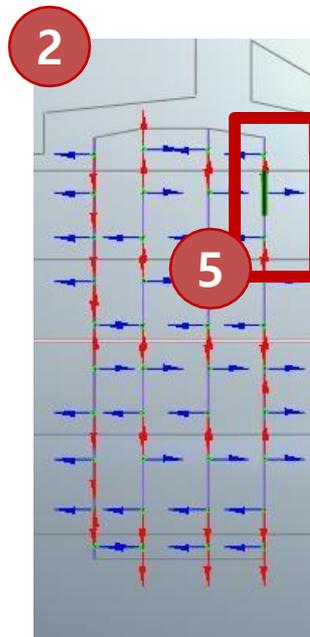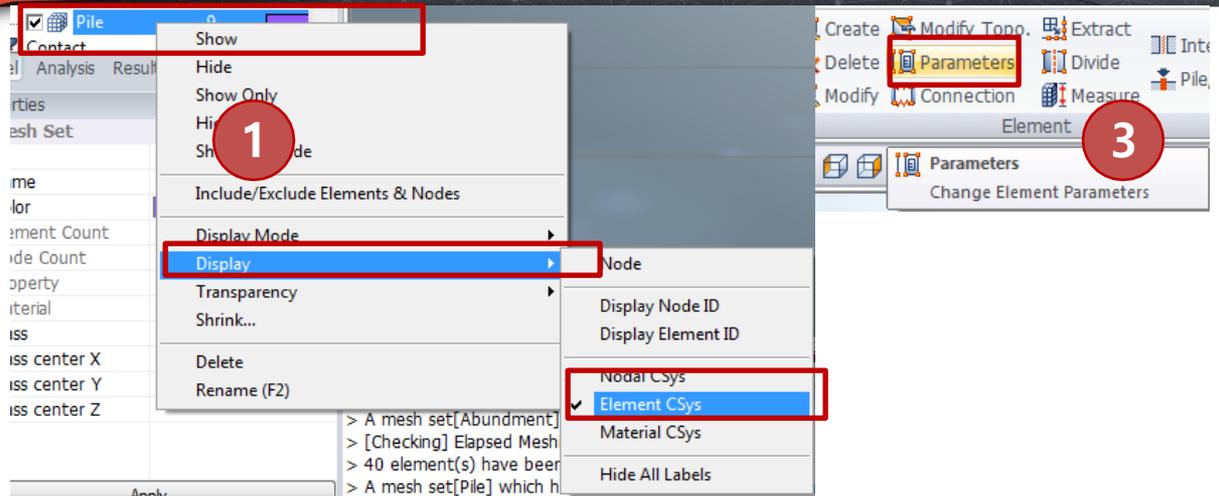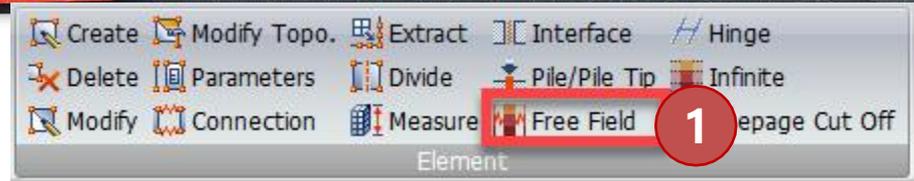
13
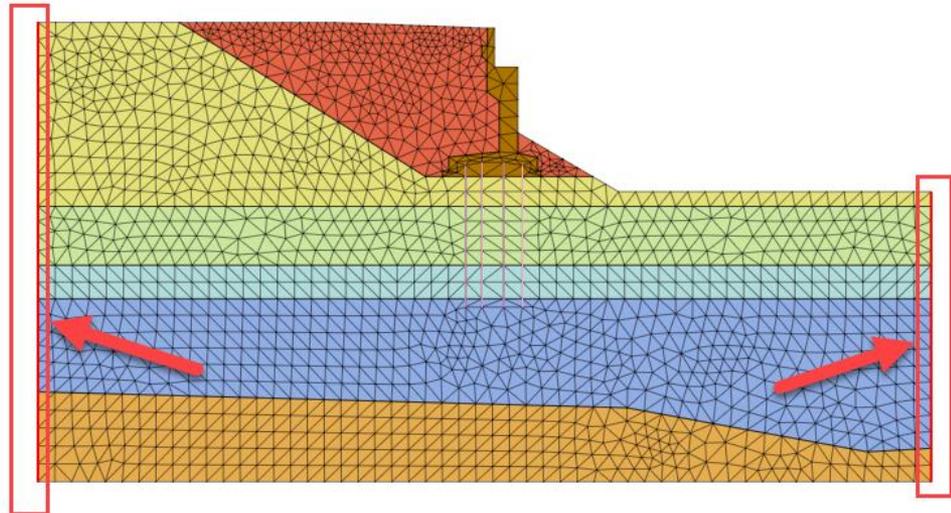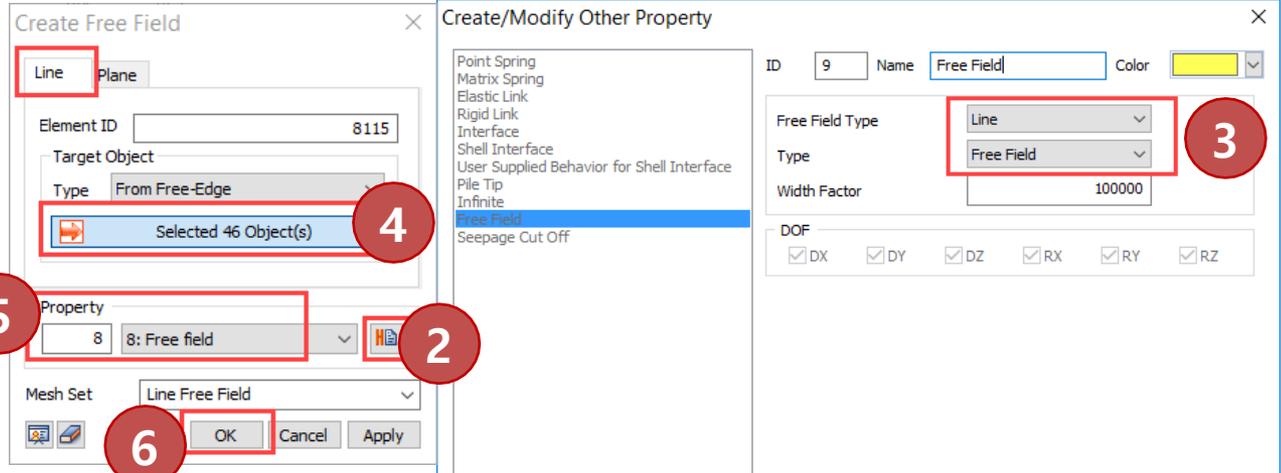
Procedure

1. **Main menu > Static/Slope Analysis > Boundary > Constraint**
2. **In the Constraint tab, go to 'Advanced' and select all the bottom nodes of the model including the bottom nodes of the free field elements as shown in the figure.**
3. **Enable Tx and Ty in the DOF column.**
4. **Enter the boundary set name as 'Fixed Bottom' and Click 'OK'**
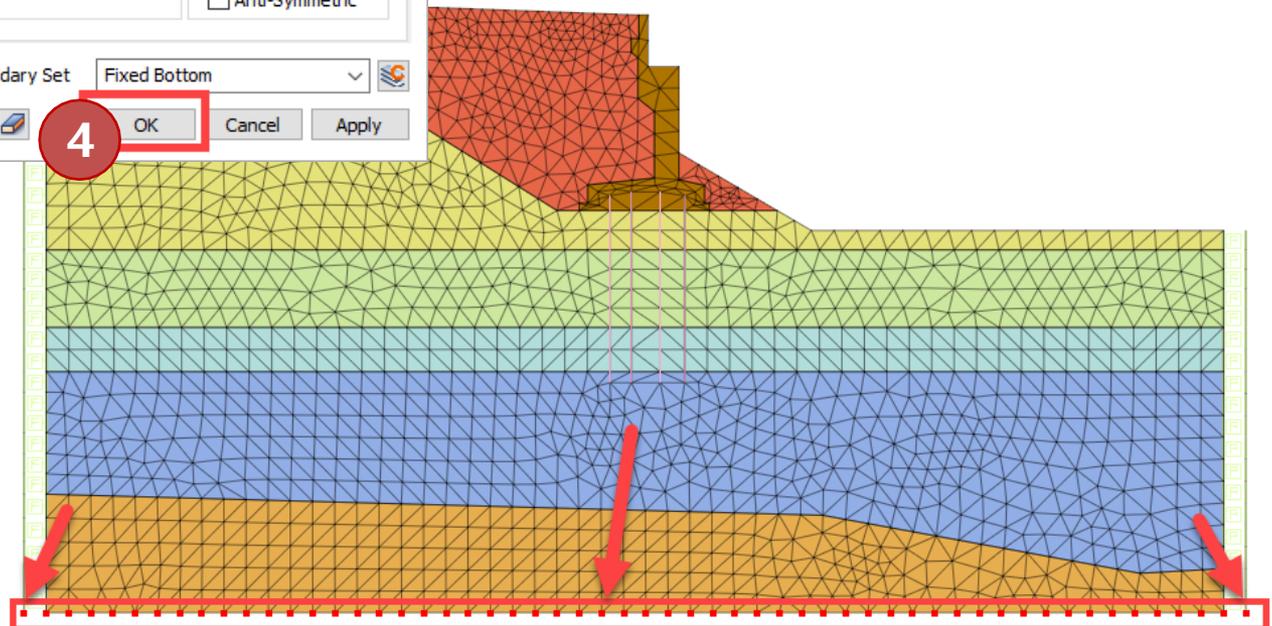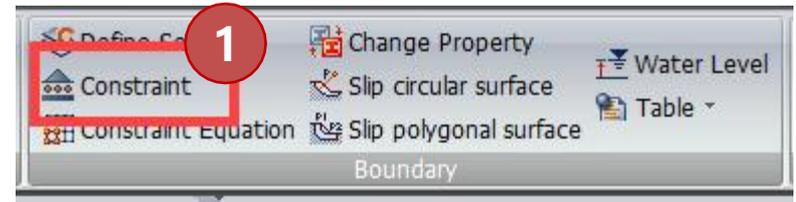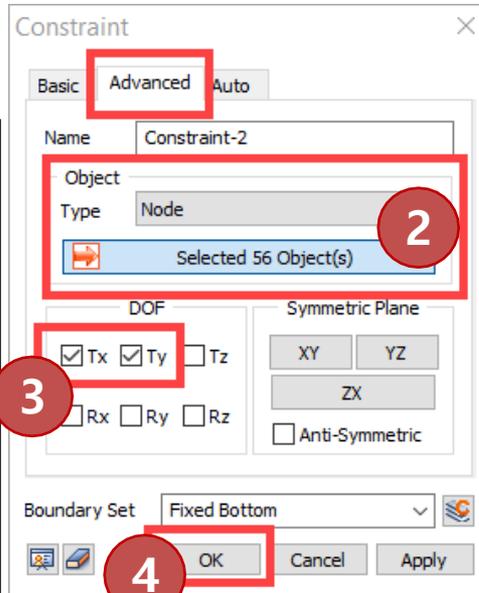
Procedure

1. **Main Menu > Analysis > General**
2. **Name 'Eigen Value', Analysis Type > Select [Eigenvalue].**
3. **Click >> to activate all the sets.**
4. **Click-on Analysis Control [ ].**
5. **Number of Frequencies '30'.**
6. **Activate 'Strum Sequence Check'.**
7. **Click 'OK'**
8. **Click 'OK'.**

Procedure

1. **Main Menu > Analysis > Perform...**
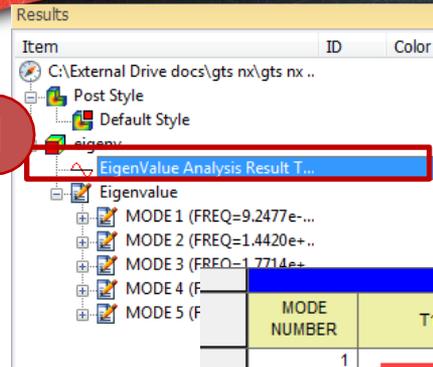2. **Check –on 'EigenValue'.**
3. **Click [OK].**

All the messages during the analysis will be shown in the Output Window. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The model is automatically saved before the analysis. The detail analysis information is also saved in a text file(*.OUT).
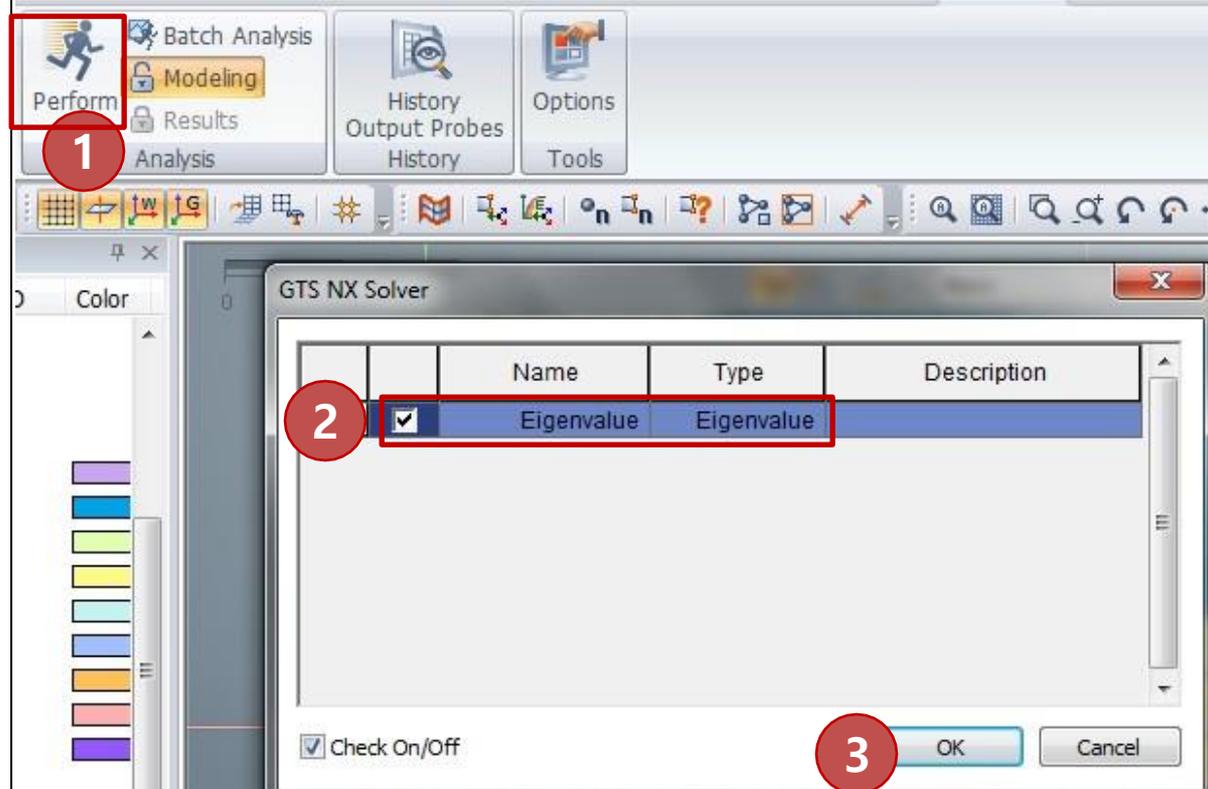
Procedure

1. Works tree> Result > Double Click on eigenvalue results table
2. We can find three tables. Go to 'Percentage Modal Effective Mass table' and find the first two mode numbers with more modal mass participation. In the current tutorial, Mode 2 and 6 have highest mass participation.
3. Then go to 'Real Eigenvalue Table' and check the frequencies of Mode 2 and Mode 6. Keep the record of frequencies of 1.628769 sec and 3.540236 sec.

1

Results

Item	ID	Color
C:\External Drive docs\gts nx\gts nx ..		
Post Style		
Default Style		
eigenv		
EigenValue Analysis Result T...		
Eigenvalue		
MODE 1 (FREQ=9.2477e-...		
MODE 2 (FREQ=1.4420e+..		
MODE 3 (FREQ=1.7714e+		
MODE 4 (F		
MODE 5 (F		

Based on the number of the degrees of freedom in the model, we can find 'n' number of natural frequencies. But all the frequencies are not important. We need to consider the frequencies of the modes with high effective mass participation.

2

PERCENTAGE MODAL EFFECTIVE MASS

MODE NUMBER	T1	T2	T3	R1	R2	R3
1	0.01%	0.00%	0.00%	0.00%	0.00%	0.00%
2	26.09%	0.00%	0.00%	0.00%	0.00%	3.57%
3	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%
4	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%
5	0.00%	27.80%	0.00%	0.00%	0.00%	17.58%
6	26.68%	0.00%	0.00%	0.00%	0.00%	0.03%
7	0.11%	0.00%	0.00%	0.00%	0.00%	0.00%
8	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%
9	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%
10	10.10%	0.00%	0.00%	0.00%	0.00%	0.02%
11	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%
12	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%

3

REAL EIGENVALUES

MODE NUMBER	EIGENVALUE	RADIANS	CYCLES	PERIOD	GENERALIZED MASS	GENERALIZED STIFFNESS	ORTHOGONALITY LOSS	ERROR MEASURE
1	1.020687e+002	1.010290e+001	1.607927e+000	6.219188e-001	1.000000e+000	1.020687e+002	0.000000e+000	4.283920e-012
2	1.047318e+002	1.023386e+001	1.628769e+000	6.139606e-001	1.000000e+000	1.047318e+002	0.000000e+000	0.000000e+000
3	3.220247e+002	1.794505e+001	2.856043e+000	3.501348e-001	1.000000e+000	3.220247e+002	0.000000e+000	1.591982e-011
4	4.197824e+002	2.048859e+001	3.260861e+000	3.066675e-001	1.000000e+000	4.197824e+002	0.000000e+000	4.400179e-012
5	4.307707e+002	2.075502e+001	3.303264e+000	3.027309e-001	1.000000e+000	4.307707e+002	0.000000e+000	0.000000e+000
6	4.947937e+002	2.224396e+001	3.540236e+000	2.824670e-001	1.000000e+000	4.947937e+002	0.000000e+000	0.000000e+000
7	4.972431e+002	2.229895e+001	3.548988e+000	2.817705e-001	1.000000e+000	4.972431e+002	0.000000e+000	0.000000e+000
8	5.792484e+002	2.406758e+001	3.830474e+000	2.610643e-001	1.000000e+000	5.792484e+002	0.000000e+000	5.366065e-012
9	6.868612e+002	2.620804e+001	4.171139e+000	2.397427e-001	1.000000e+000	6.868612e+002	0.000000e+000	2.199504e-012
10	7.357477e+002	2.712467e+001	4.317025e+000	2.316410e-001	1.000000e+000	7.357477e+002	0.000000e+000	0.000000e+000
11	8.266823e+002	2.875208e+001	4.576036e+000	2.185297e-001	1.000000e+000	8.266823e+002	0.000000e+000	9.198082e-012
12	1.111603e+003	3.334071e+001	5.306339e+000	1.884538e-001	1.000000e+000	1.111603e+003	0.000000e+000	4.488273e-012
13	1.253495e+003	3.540472e+001	5.634837e+000	1.774674e-001	1.000000e+000	1.253495e+003	0.000000e+000	1.499959e-011
14	1.443552e+003	3.799410e+001	6.046949e+000	1.653727e-001	1.000000e+000	1.443552e+003	0.000000e+000	1.110777e-011
15	1.584032e+003	3.979990e+001	6.334350e+000	1.578694e-001	1.000000e+000	1.584032e+003	0.000000e+000	5.088320e-012
16	1.662907e+003	4.077875e+001	6.490140e+000	1.540799e-001	1.000000e+000	1.662907e+003	0.000000e+000	0.000000e+000

GTS **NX**

Procedure

1. **Return to the Pre Mode**
2. **Main Menu > Dynamic Analysis> Load > Ground Acceleration**
3. **Enable 'X' direction**
4. **Click the '⊞' option next to function.**
5. **Click 'Add Time Function'**
6. **Click 'Earthquake'**
7. **In the GTS NX earthquake database, select 1966, Parkfield Cholame, Shandon, 40 deg.**
8. **Click 'OK'**
9. **Click 'OK'**
10. **Click 'Close'**
11. **Select the 'Park040' in the drop down menu.**
12. **Click 'OK'**

Procedure

1. **Main Menu > Analysis Case > General**
2. **Title 'Time History Analysis'.**
3. **Solution Type: Linear Time History (Direct)**
4. **Activate All Sets**
5. **Click on 'Time Step'**
6. **Click on 'Define Time Step'**
7. **Input the following, Name : Time Steps Time Duration: 15 sec (You can control the end time of the earthquake in this column) Time Increment: 0.02 sec (Time increments effects the analysis results.)**
8. **Click 'Add'**
9. **Click 'Close'**
10. **Click 'OK'**
11. **Click 'Analysis Control'**
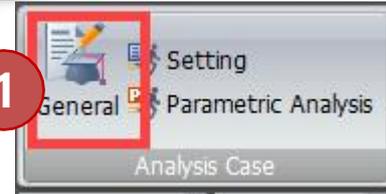
Add/Modify Analysis Case

Analysis Case Setting

Title	Time History
Description	
Solution Type	Linear Time History(Direct)
Construction Stage Set	

Time Step
Analysis Control
Output Control

Analysis Case Model

All Sets   <<   >>   Active Set

Mesh
　Default Mesh Set
　Line Free Field
　Weathered Rock Top
　abunment
　embankment 1
　embankment 2
　hard rock
　piles
　soft rock
　weathered rock
Boundary Condition
　Fixed Bottom
Dynamic Load
　Dynamic Load Set-1
Contact Pair

Mesh
　Default Mesh Set
　Line Free Field
　Weathered Rock Top
　abunment
　embankment 1
　embankment 2
　hard rock
　piles
　soft rock
　weathered rock
Boundary Condition
　Fixed Bottom
Dynamic Load
　Dynamic Load Set-1
Contact Pair

Solve Each Load Set Independently   Sorting   Name   OK   Cancel   Apply

Time Step

Time Step for Analysis and Output
Define Time Step

Summary
Total Time Duration   15
Total Time Steps   750

OK   Close

Define Time Step

Name	Time Steps		Add
Time Duration	15	sec	Insert
Time Increment	0.02	sec	Modify
Intermediate Output(Every N Time	1		Delete

No	Name	Time Duration	Time Increment	Inter. Output
1	Time Steps	15	0.02	1

Close

Procedure

1. **In General tab > Enable 'Include In-situ Analysis' and enable 'Clear Displacement/Strain'**
2. **Click on the Dynamic tab**
3. **Click on damping method [🖱] .**
4. **Select 'Calculate from Modal Damping' and Select frequency [sec]. Enter the frequencies for 2nd and 6th modes from previous eigenvalue analysis. Enter 0.05 in Damping Ratio for both Mode 1 and 2**
5. **Click 'OK'.**
6. **Click 'OK'**
7. **Click 'OK in the main analysis case.**
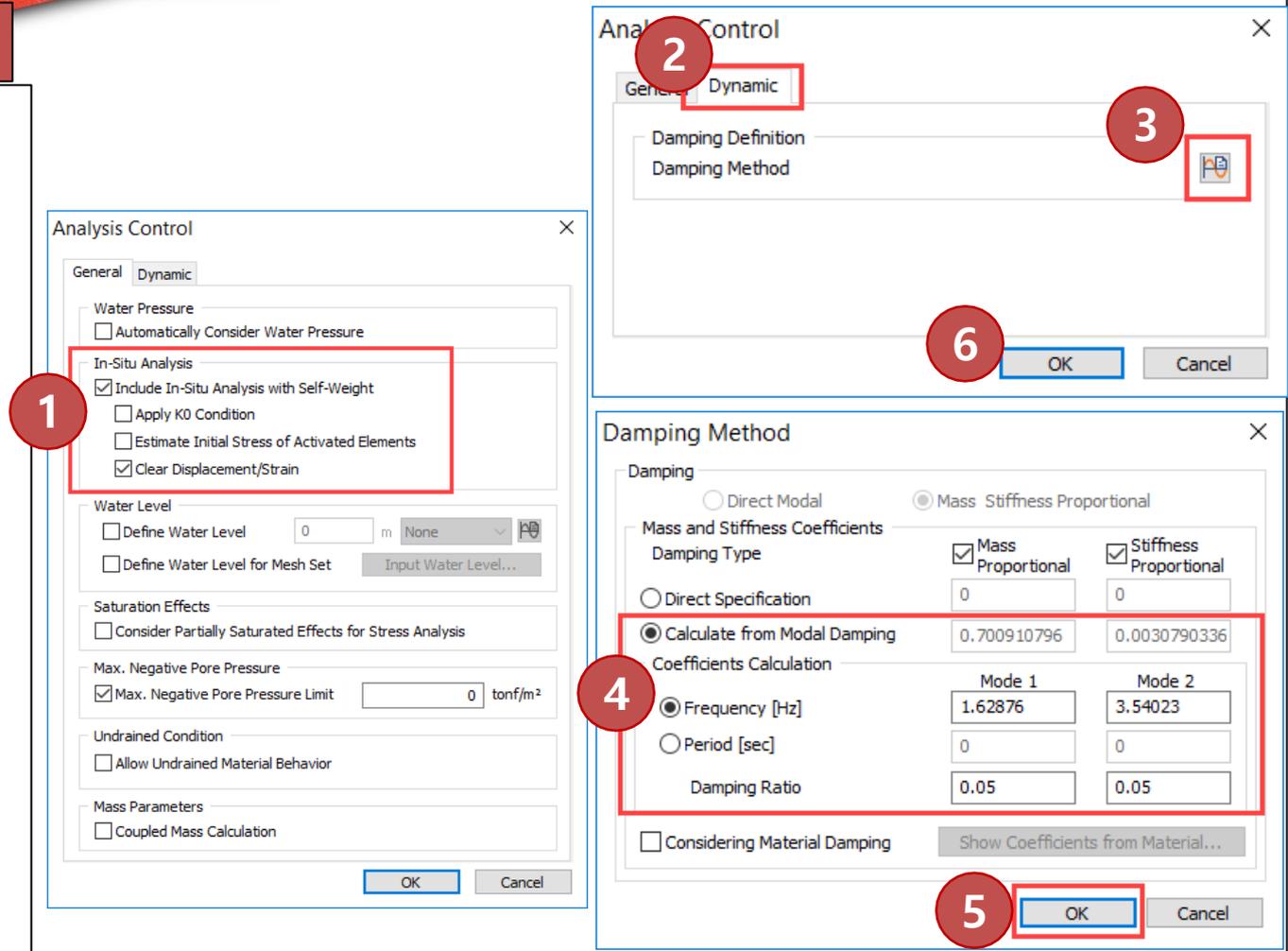
Analysis Control

General | Dynamic

Water Pressure
☐ Automatically Consider Water Pressure

In-Situ Analysis
☑ Include In-Situ Analysis with Self-Weight
☐ Apply K0 Condition
☐ Estimate Initial Stress of Activated Elements
☑ Clear Displacement/Strain

Water Level
☐ Define Water Level   0   m None
☐ Define Water Level for Mesh Set   Input Water Level...

Saturation Effects
☐ Consider Partially Saturated Effects for Stress Analysis

Max. Negative Pore Pressure
☑ Max. Negative Pore Pressure Limit   0   tonf/m²

Undrained Condition
☐ Allow Undrained Material Behavior

Mass Parameters
☐ Coupled Mass Calculation

OK   Cancel

Analysis Control

General | Dynamic

Damping Definition
Damping Method   🖱

OK   Cancel

Damping Method

Damping
○ Direct Modal   ○ Mass Stiffness Proportional

Mass and Stiffness Coefficients
Damping Type   ☑ Mass Proportional   ☑ Stiffness Proportional
○ Direct Specification   0   0
● Calculate from Modal Damping   0.700910796   0.0030790336

Coefficients Calculation
	Mode 1	Mode 2
● Frequency [Hz]	1.62876	3.54023
○ Period [sec]	0	0
Damping Ratio	0.05	0.05

☐ Considering Material Damping   Show Coefficients from Material...

OK   Cancel

Procedure

1. **Main Menu > Analysis > Perform…**
2. **Check –on 'Response Spectrum' only.**
3. **Click [OK].**

All the messages during the analysis will be shown in the Output Window. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The model is automatically saved before the analysis. The detail analysis information is also saved in a text file(*.OUT).

GTS NX Solver

	Name	Type	Description
☐	eigenv	Eigenvalue	
☑	Time History	Linear Time History(...	

☐ Check On/Off

[OK]   [Cancel]
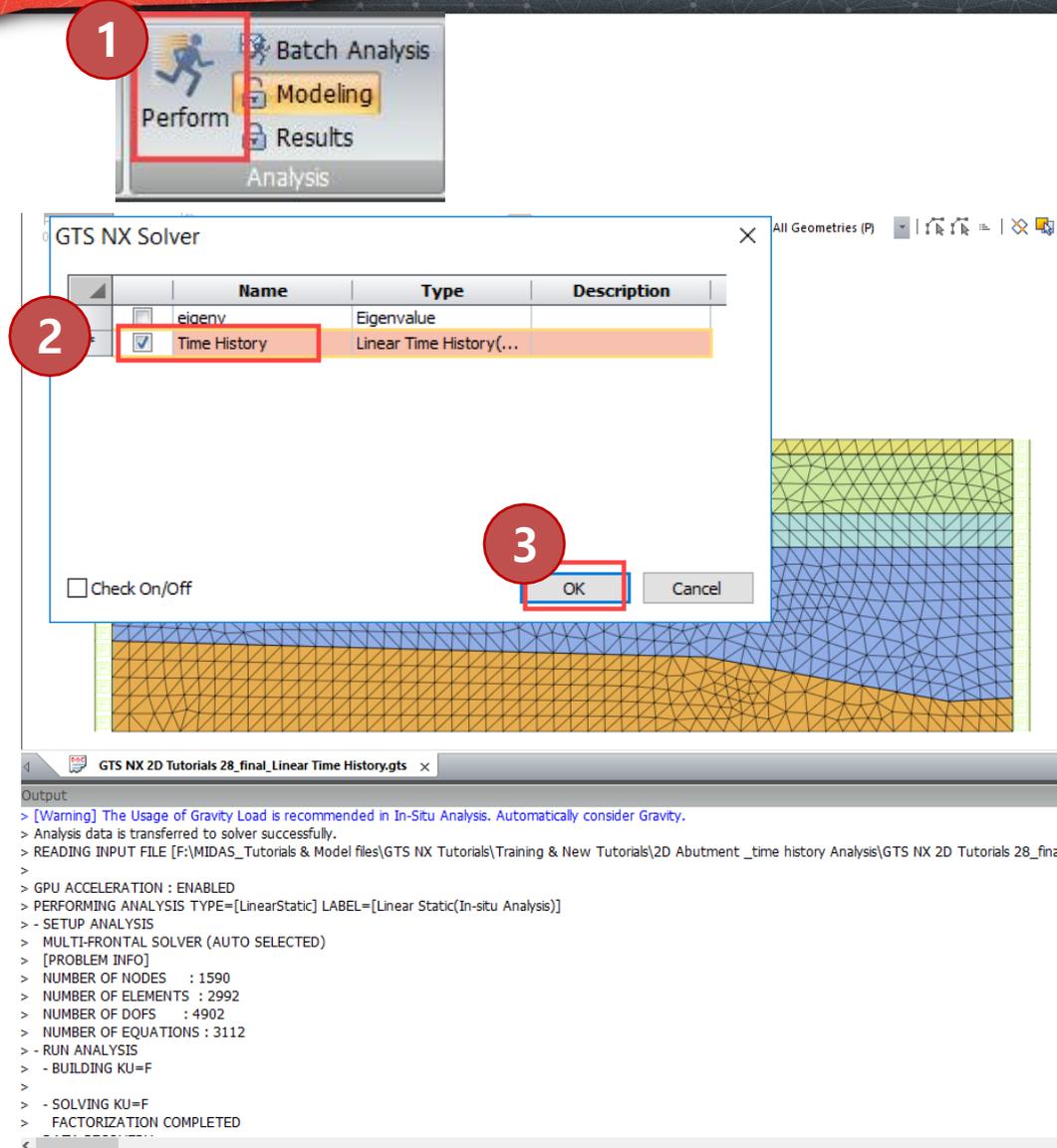
Output

> [Warning] The Usage of Gravity Load is recommended in In-Situ Analysis. Automatically consider Gravity.
> Analysis data is transferred to solver successfully.
> READING INPUT FILE [F:\MIDAS_Tutorials & Model files\GTS NX Tutorials\Training & New Tutorials\2D Abutment _time history Analysis\GTS NX 2D Tutorials 28_fina
>
> GPU ACCELERATION : ENABLED
> PERFORMING ANALYSIS TYPE=[LinearStatic] LABEL=[Linear Static(In-situ Analysis)]
> - SETUP ANALYSIS
>   MULTI-FRONTAL SOLVER (AUTO SELECTED)
>   [PROBLEM INFO]
>   NUMBER OF NODES   : 1590
>   NUMBER OF ELEMENTS : 2992
>   NUMBER OF DOFS   : 4902
>   NUMBER OF EQUATIONS : 3112
> - RUN ANALYSIS
>   - BUILDING KU=F
>
>   - SOLVING KU=F
>   FACTORIZATION COMPLETED

GTS NX 2D Tutorials 28_final_Linear Time History.gts

Procedure

1. **Post Works Tree > Time History > Absolute MAX > Relative Displacements > TX**
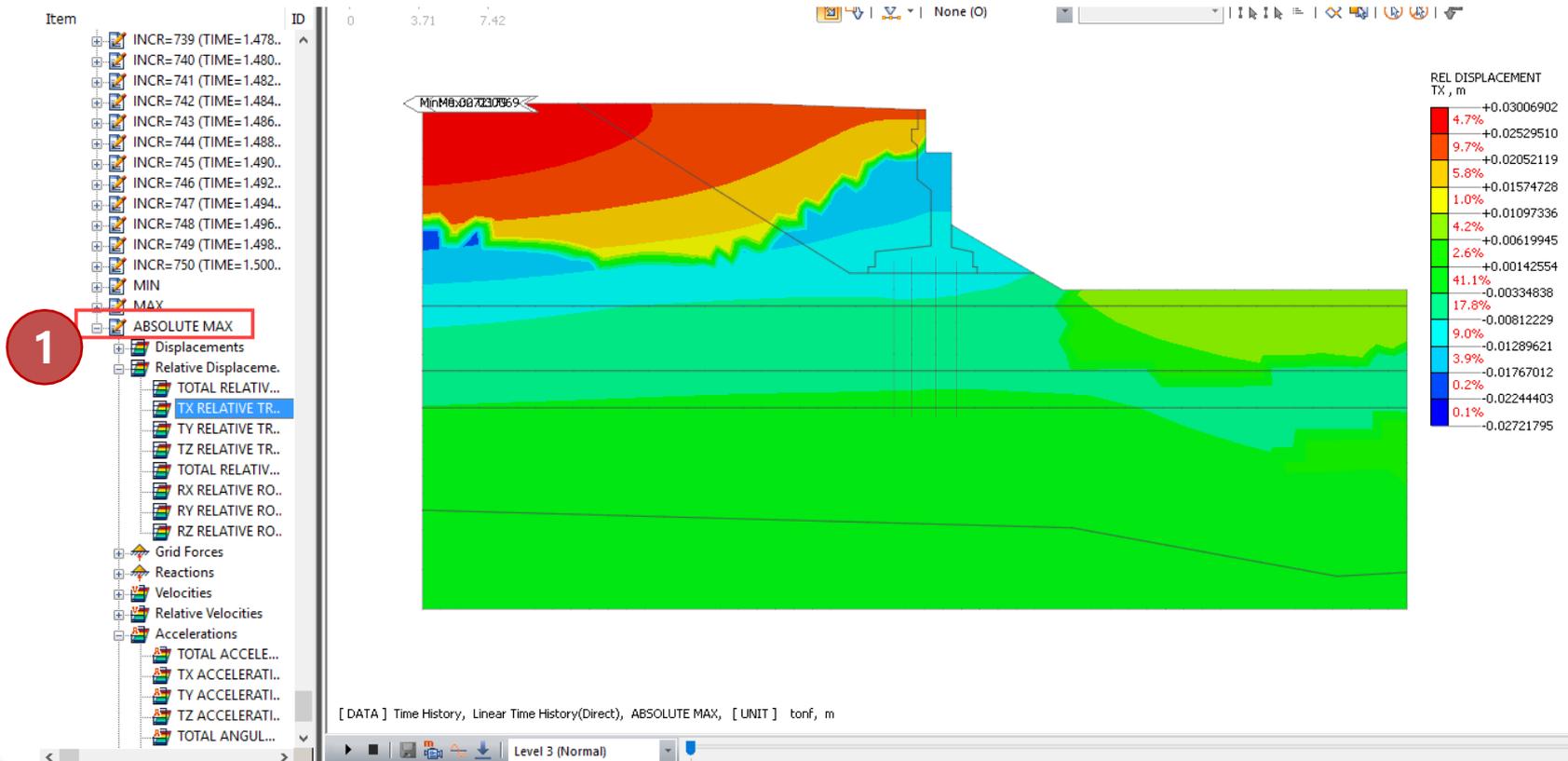
Procedure

1. **Post Works Tree > Time History > Absolute MAX > Acceleration> TX**
2. **Main menu > Result > Advanced > Extract**
3. **Select the following in the Extract Results Tab:**
   Analysis Set: Time History
   Result Type: Accelerations
   Results: TX Accelerations
4. **Select 'Step' in Order**
5. **Node Selection: Using the cursor select the top node of the abutment and any bottom node of the model as shown in the figure.**
6. **Click 'Table'**

Procedure

7. **Select three columns as shown in the picture. Right click and click on the 'Show Graph'**
8. **Input the Y label (value) as Accelerations and Graph Title as 'Acceleration vs Time.**
9. **Click 'OK'**
10. **In the graph as shown in the picture, red curve represents the input motion and green curve represents the 'amplified wave' recorded at the top node of the abutment.**
11. **Record the peak values in both the curves.**
Amplification = Abs(
$$\frac{\text{Output peak acceleration of } Amplified\ wave}{\text{Input Peak acceleration}})$$

Acceleration vs Time

Happy Modeling

GTS NX Certification task for participants involves submission of file (.docx or .pdf) with :

- Generating the model as shown in tutorial.

- Acceleration vs Time (Amplification) – Graph Picture

- Short summary of model creation, and results.

KINDLY SUBMIT YOUR FINAL RESULTS IN THE PROVIDED WORD FILE FORMAT.