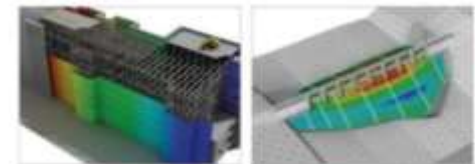




2D Liquefaction Analysis for Bridge Abutment

GTS NX
Geo-Technical analysis System New eXperience



Tutorial by Angel Francisco Martinez

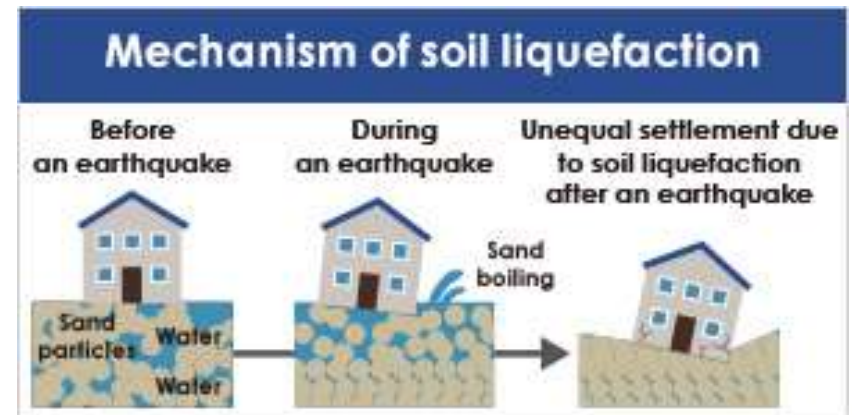
Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering

MIDAS

- Soil liquefaction is a phenomenon in which the strength and stiffness of a saturated soil/sand is reduced by earthquake shaking or other rapid loading.
- The pressures generated during large earthquake shaking can cause the liquefied sand and excess water to force its way to the ground surface.
- Soil particles can no longer support all the weight.
- Bridges and large buildings constructed on pile foundations may lose support from the adjacent soil and come to rest at a tilt after shaking.
- Sloping ground and ground next to rivers and lakes may slide on a liquefied soil layer ('lateral spreading').

2D Seismic Analysis Methods

- **Pseudostatic Analysis**
 - Limit Equilibrium Method
 - + MIDAS Soilworks
 - + Slide
- **Numerically Based Analysis**
 - Finite Element Method
 - + MIDAS GTS NX
 - + Quake/W
 - + Plaxis
 - Finite Difference Method
 - + FLAC



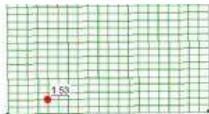
<http://www.cti.co.jp/en/solution/cae/cae2/>

Pseudostatic Analysis

GTS NX

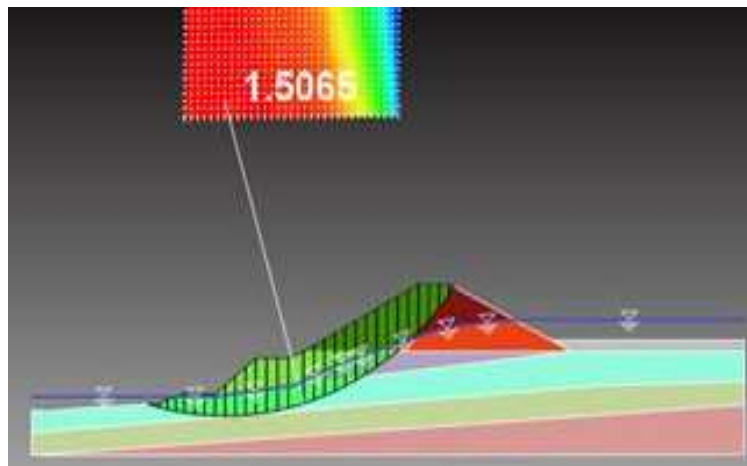
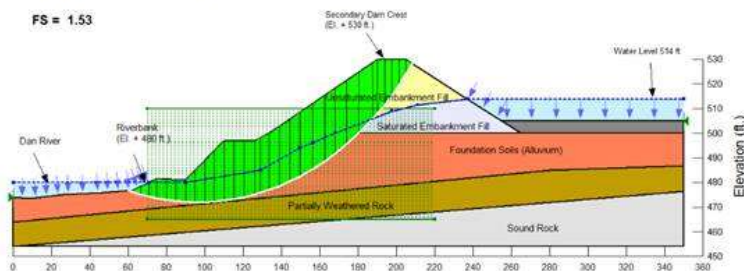


- Representation of the complex, transient, dynamics of earthquake shaking by a single, constant, unidirectional load is crude.
- Method has been shown to be unreliable for soils with significant pore pressure buildup during cycling (i.e., not valid for liquefaction).
- Cannot predict deformation.
- Is only a relative index of slope stability
- Outdated and should only be used for **screening purposes**.
- More elaborate techniques are generally warranted and are rather easy to do with modern computing software.



Load Condition: Static, Steady-State Seepage
Search Method: Grid & Radius
Location: Secondary Dam, Section C-C, Downstream

FS = 1.53



Midas SoilWorks

- 

FEM

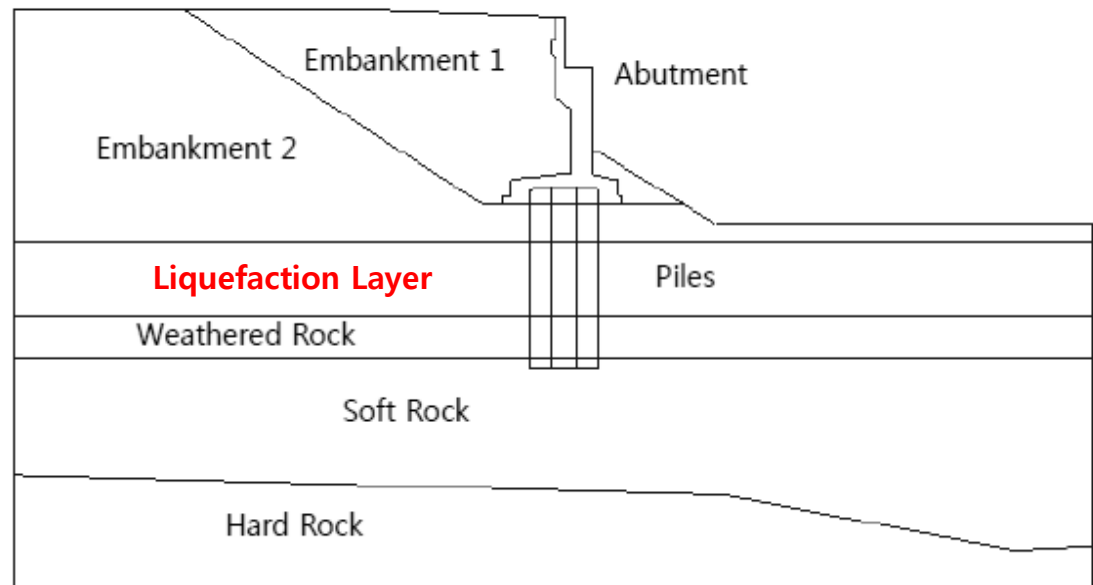
2D Liquefaction Analysis of Bridge Abutment

GTS NX



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, UBC Sand, 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 liquefaction tutorial_start.gtb
6. Check Liquefied Layers



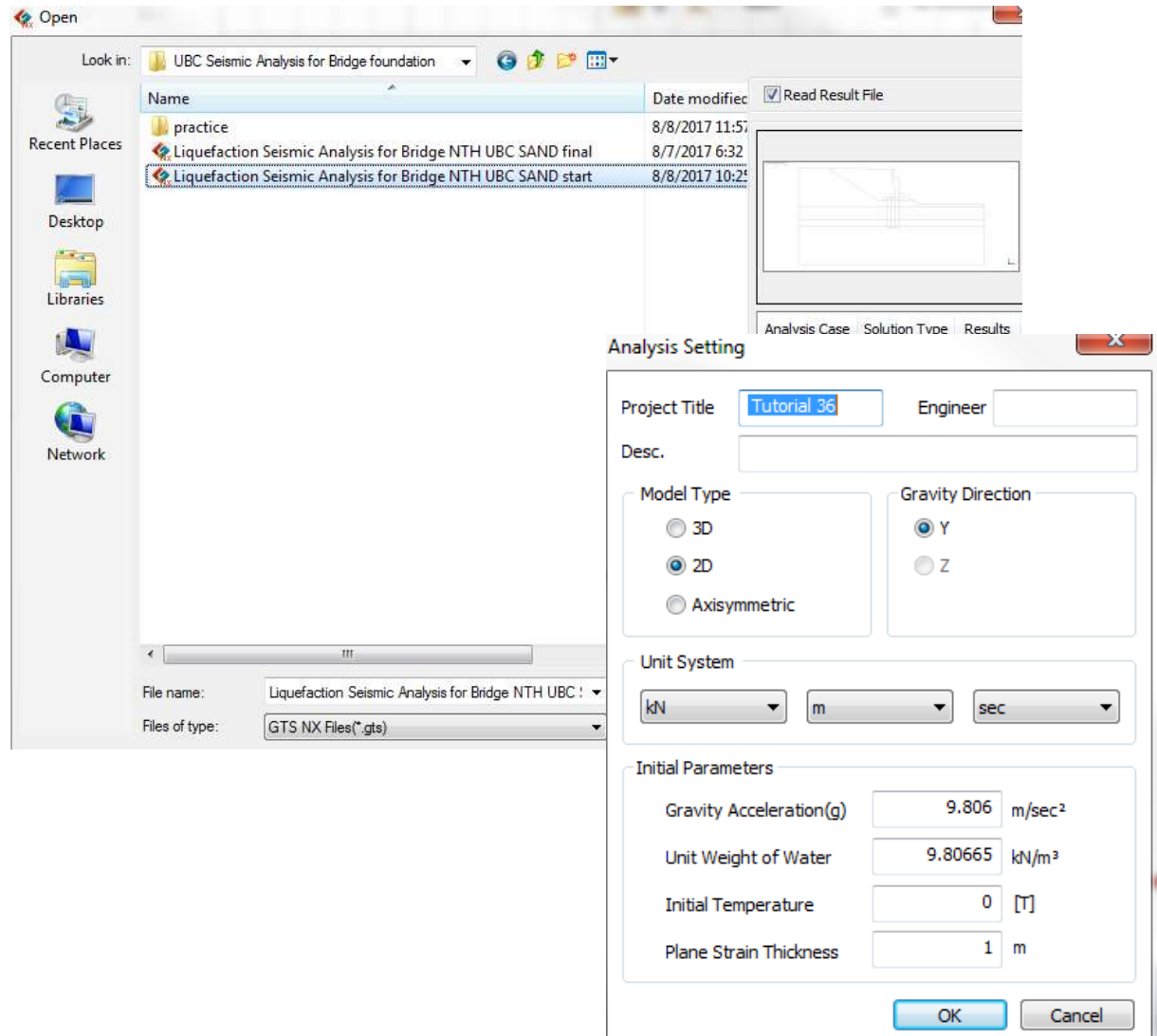
Step 01 File > Open

GTS NX



Procedure

1. Main Menu > File>Open...
2. Double click 'GTS NX 2D
liquefaction_start.gtb
3. Main Menu > Analysis > Setting > Unit System> kN>m>s
4. Click [OK].





Materials

ID	Name	Type	Modulus of Elasticity(E) [kN/m ²]	Poisson's Ratio(v)	Unit Weight(Y) [kN/m ³]	Unit Weight (Saturated) [kN/m ³]	Cohesion (c) [kN/m ²]	Friction Angle [Φ]	K0
1	Embankment	Mohr Coulomb	40,000	0.35	18	19	15	25	1
2	Liquefaction Layer	UBC SAND	60,000	0.02	20	21	-	-	-
3	Weathered Rock	Mohr Coulomb	100,000	0.3	20	21	35	33	1
4	Soft Rock	Mohr Coulomb	900,000	0.25	24	25	150	37	1
5	Abutment	Mohr Coulomb	21,000,000	0.18	25	-	-	-	-
6	Pile	Elastic	210,000,000	0.3	78	-	-	-	-

Step 02: Mesh > Material > UBC

GTS NX



The Modified UBCSAND model is developed to simulate liquefaction phenomenon using **plastic theory based on explicit method for 2D effective stress state**. It is extended to enable **implicit nonlinear analysis for 3D stress state** based on the constitutive model^{9, 10}

In elastic region, nonlinear elastic behavior can be simulated, elastic modulus changes according to the effective pressure applied. In **plastic region, the behavior is defined by three types of yield functions** : shear (shear hardening), compression (cap hardening), and pressure cut-off. In case of shear hardening, soil densification effect can be taken into account by cyclic loading.

* Note - **Implicit Method** : Explicit methods calculate the state of a system at a later time from the state of the system at the current time, while implicit methods find a solution by solving an equation involving both the current state of the system and the later one.

9 Beaty, M. and Byrne, PM., "An effective stress model for predicting liquefaction behaviour of sand," In Geotechnical earthquake engineering and soil dynamics III, Americal Society of Civil Engineers, Geotechnical Special Publication 75(1), 1998, pp. 766-777.

10 Puebla, H., Byrne, PM., and Phillips, R., "Analysis of CANLEX liquefaction embankments: prototype and centrifuge models," Canadian Geotechnical Journal, 34, 1997, pp 641-657.



Drainage parameters

Model Type **Modified UBCSAND** ☐ Structure

General Porous Non-Linear

Unit Weight(Saturated) kN/m³

Initial Void Ratio(eo)

☐ Unsaturated Property ...

Drainage Parameters

Undrained(Effective Stiffness/Effective Strength)

☒ Undrained Poisson's Ratio

☐ Skempton's B Coefficient

Seepage & Consolidation Parameters

Permeability Coefficients

kx	ky	kz
<input type="text" value="1e-005"/>	<input type="text" value="1e-005"/>	<input type="text" value="1e-005"/>

m/sec

☐ Void Ratio Dependency of Permeability(ck)

Specific Storativity(Ss) 1/m

The pore water pressure in stress analysis can be divided into normal state pore water pressure and abnormal state pore water pressure - **the excess pore water pressure generated between soil particles due to external loading under undrained conditions**. An excess pore water pressure of nearly 0 is called the drainage condition.

Undrained Material Type: Effective stiffness/effective strength

This is the most general case where the input stiffness parameters and strength parameters are the parameters of the ground skeleton. Like drained analysis, **GTS NX uses the input stiffness/strength parameters for undrained analysis**. The disadvantage is that the effective strength parameters in the undrained state are hard to obtain through experimentation.

Undrained Poisson's ratio and Skempton (B) coefficient are parameters used to calculate the bulk modulus of elasticity for water. **The undrained Poisson's ratio has a standard value of 0.495** with a compressibility of nearly '0 (zero)' and the Skempton coefficient expresses the saturation, with 1 meaning full saturation.

Step 02: Mesh > Material > UBC

GTS NX



Material

ID: Name: Color:

Model Type: ☐ Struct.

General Porous Non-Linear

Reference Pressure: kN/m²

Elastic

☐ Linear Elastic ☒ Power Law

Elastic Shear Modulus Number($K_{G,e}$):

Elastic Shear Modulus Exponent(n_e):

Plastic/Shear

Peak Friction Angle(ϕ_{Op}): [deg]

Constant Volume Friction Angle(ϕ_{Ocv}): [deg]

Cohesion(c): kN/m²

Plastic Shear Modulus Number($K_{G,p}$):

Plastic Shear Modulus Exponent(n_p):

Failure Ratio(R_f):

Post Liquefaction Calibration:

☒ Cyclic Behavior

Soil Densification Calibration Factor:

☐ Plastic/Pressure Cutoff

Tensile Strength: kN/m²

☐ Plastic/Cap

Cap Bulk Modulus Number:

Plastic Cap Modulus Exponent:

Over-Consolidation Ratio(OCR):

Parameter	Description	Reference
P_{ref}	Reference Pressure	In-situ horizontal stress at mid-level of soil layer
Elastic (Power Law)		
K_G^e	Elastic shear modulus number	Dimensionless
n_e	Elastic shear modulus exponent	Dimensionless
Plastic / Shear		
ϕ_p	Peak Friction Angle	Failure parameter as in MC model
ϕ_{cv}	Constant Volume Friction Angle	-
c	Cohesion	Failure parameter as in MC model
K_G^p	Plastic shear modulus number	Dimensionless
n_p	Plastic shear modulus exponent	Dimensionless
R_f	Failure ratio (q_f / q_a)	0.7~0.98 (< 1), decreases with increasing relative density
F_{post}	Post Liquefaction Calibration Factor	Residual shear modulus
F_{dens}	Soil Densification Calibration Factor	Cyclic Behavior
Advanced parameters		
P_{cut}	Plastic/Pressure Cutoff (Tensile Strength)	-
K_b^p	Cap Bulk Modulus Number	-
m_p	Plastic Cap Modulus Exponent	-
OCR	Over Consolidation Ratio	Normal stress / Pre-overburden pressure

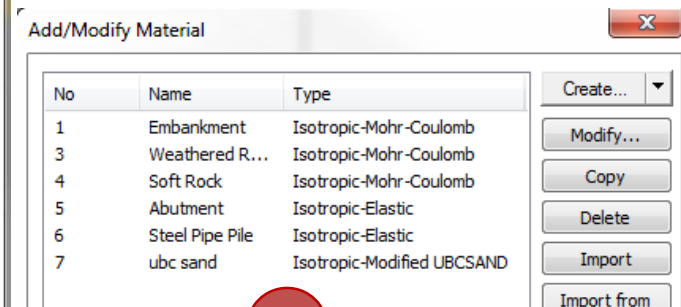
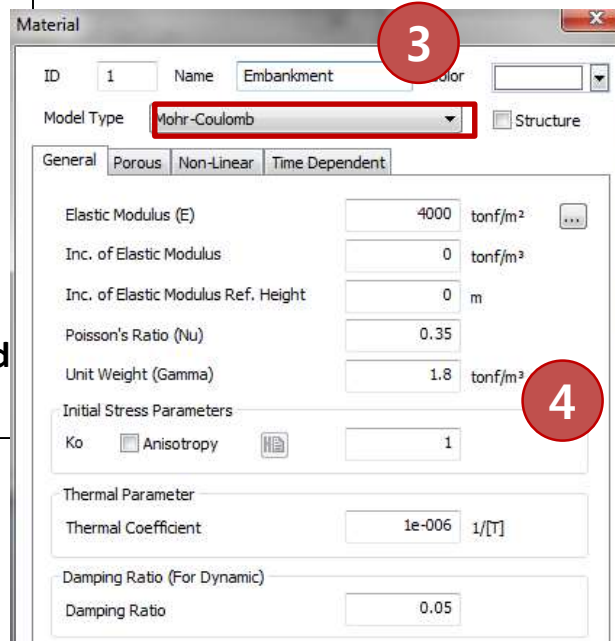
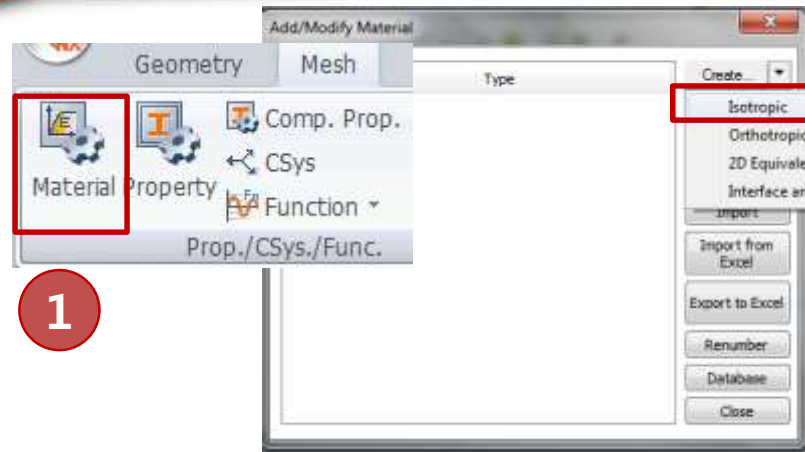
Step 02: Mesh > Materials

GTS NX



Procedure

1. Main Menu Model > materials
2. [Create]>Select Isotropic.
3. ID '1', Name 'Embankment', Model Type [Mohr Coulomb].
4. Enter the material properties as shown.
5. Click [OK].
6. Click [Apply].
7. Similarly create the materials for all the soil layers – Weathered Rock, Liquefaction, Soft Rock, and Abutment.





Properties

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

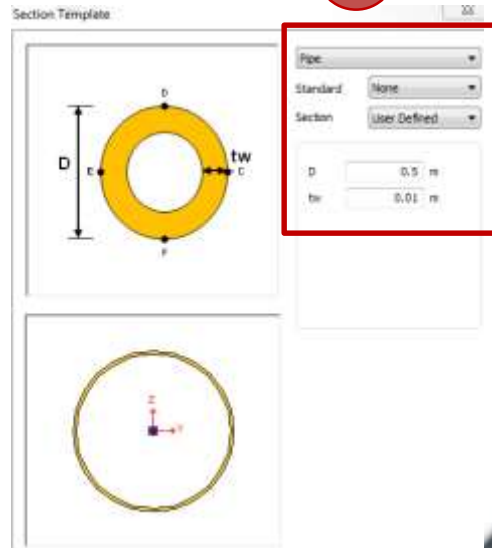
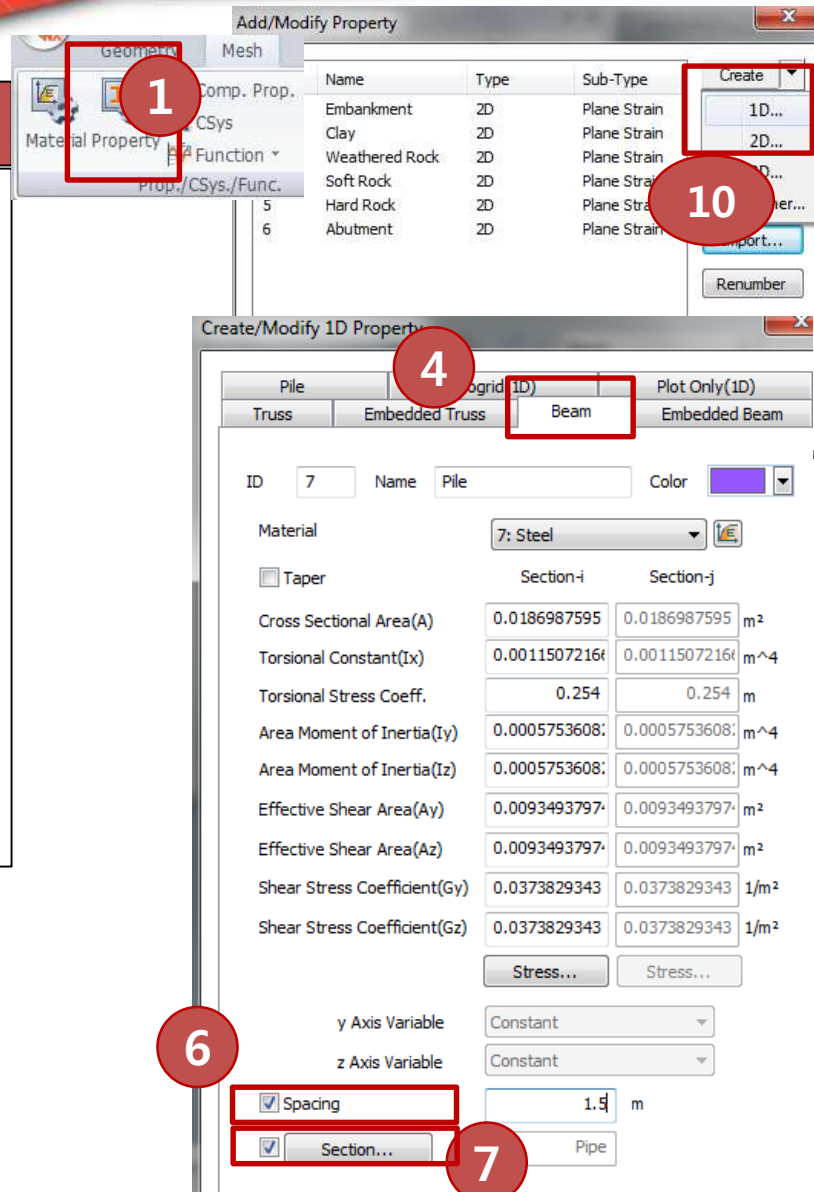
Step 03: Mesh > Property

GTS NX



Procedure

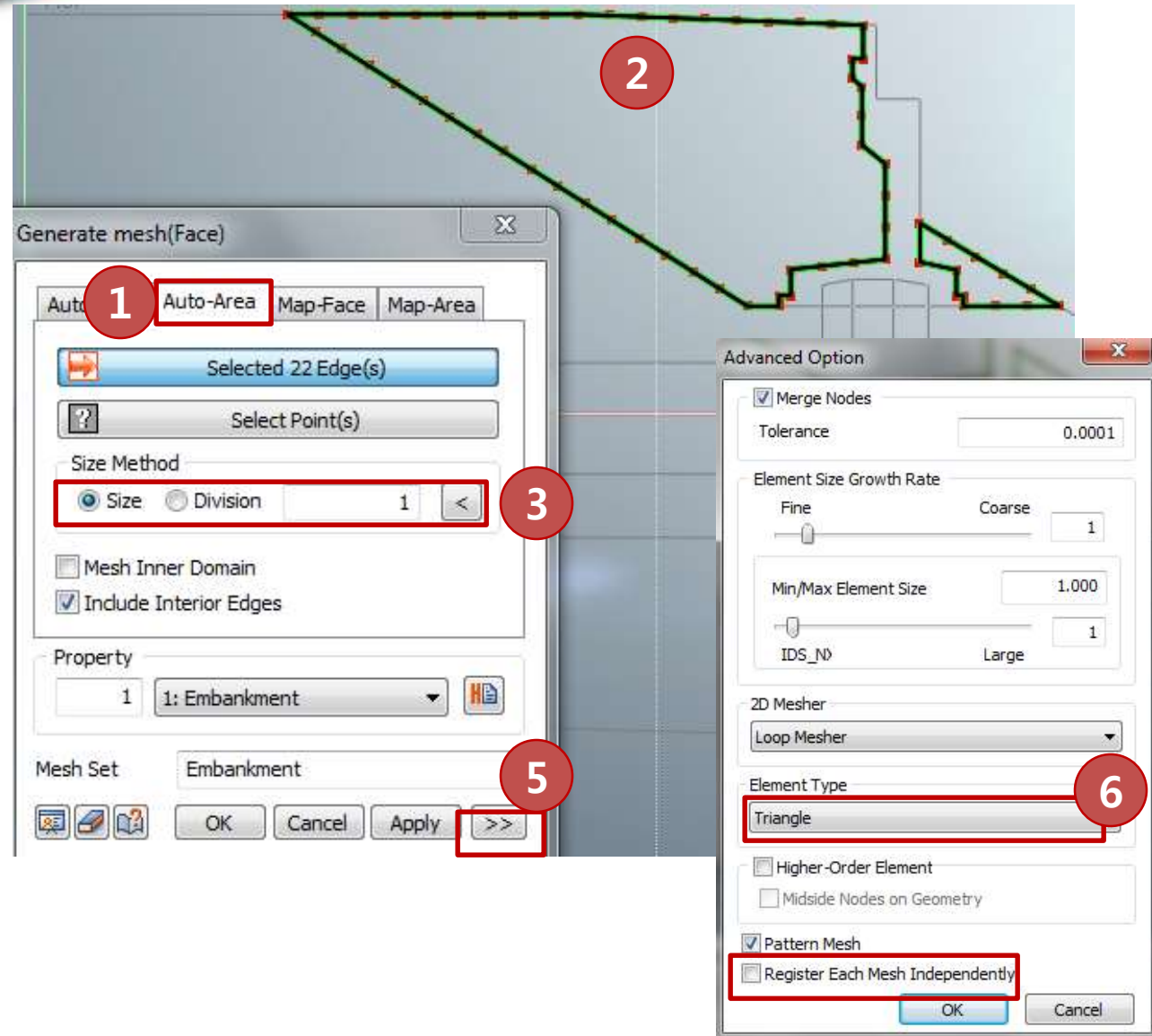
1. Main Menu Model > Property
2. Click on [Add] > Select [1D].
3. ID '7', Name 'Pile'.
4. Element Type > [Beam].
5. Material > 'Steel'
6. Spacing 1.5m
7. Click-on 'Sectional Library'
8. Select [Pipe], D > Enter '0.5'm, tw > enter '0.01'm.
9. Click [OK], Click [OK].
10. Repeat for 2D Plane Strain soil layers



Step 04 Mesh > 2D Mesh

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 Liquefaction, Weathered Rock, Soft Rock, & Abutment.



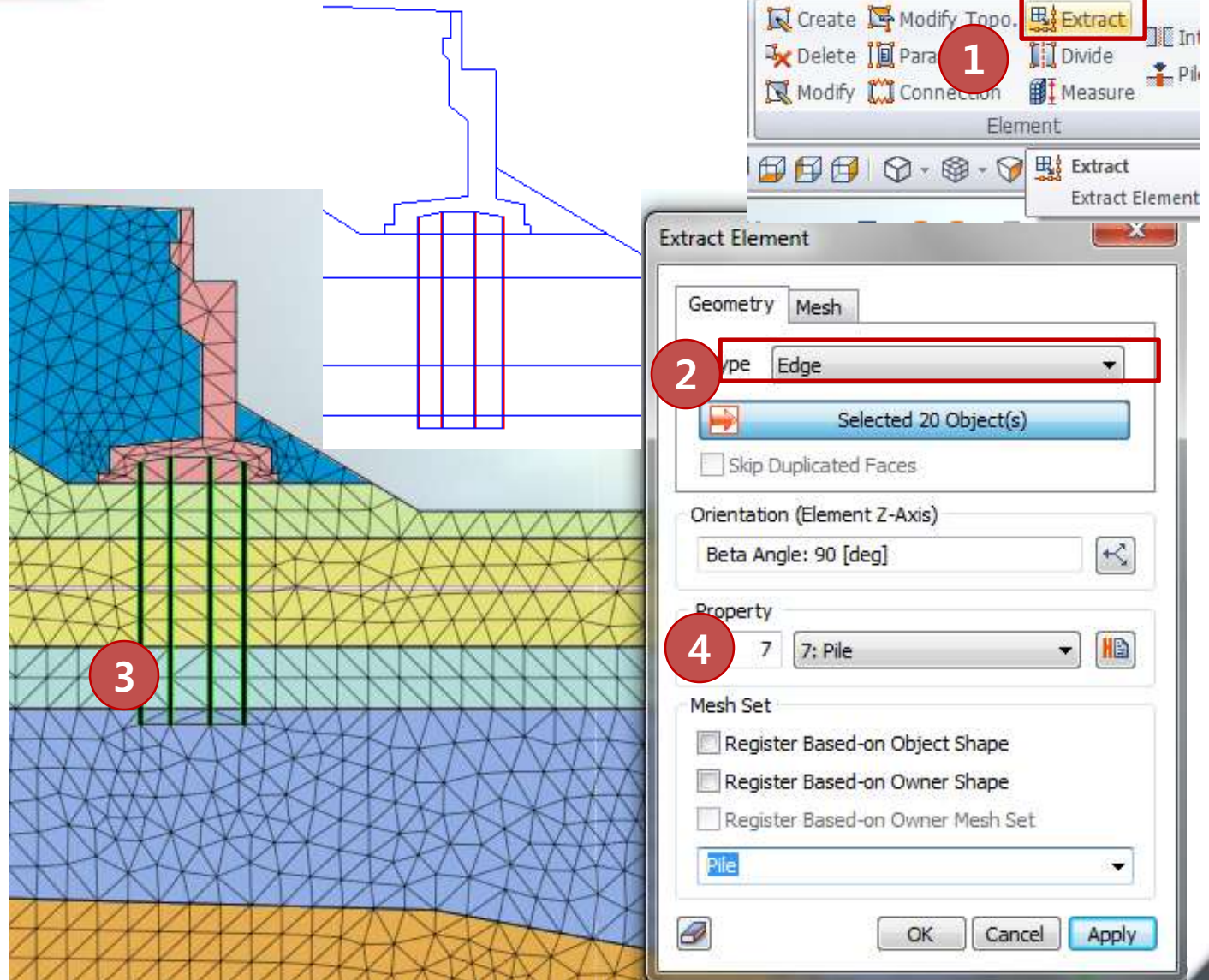
Step 05 Mesh > Element>Extract

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]



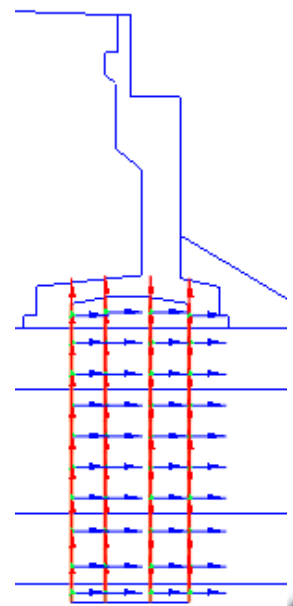
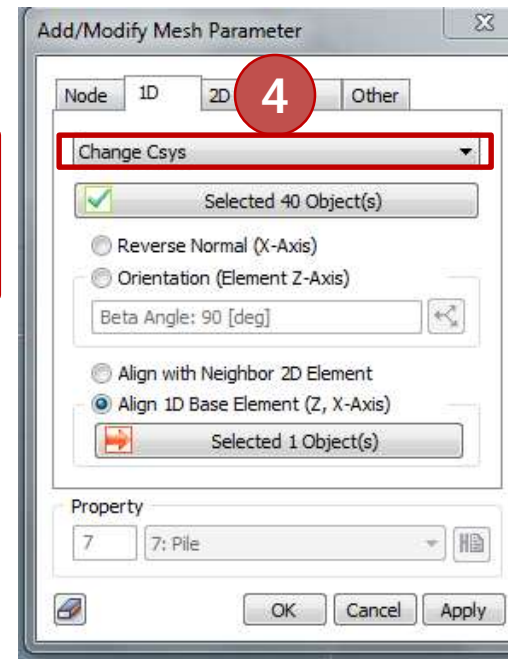
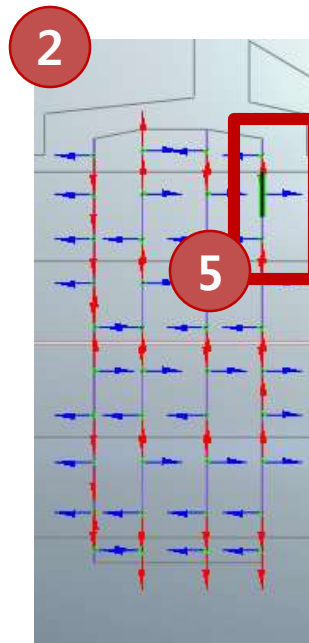
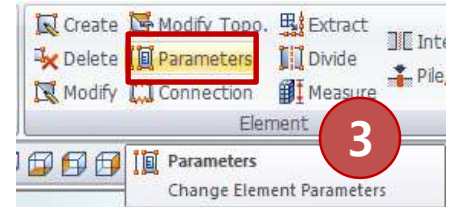
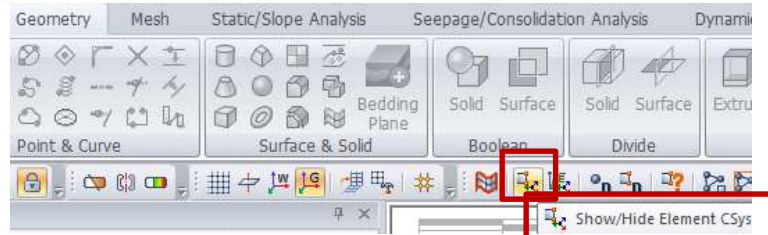
Step 06 Mesh > Element > Parameter

GTS NX



Procedure

1. Main 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].



Step 07 Mesh > Element > Create Ground Surface Spring

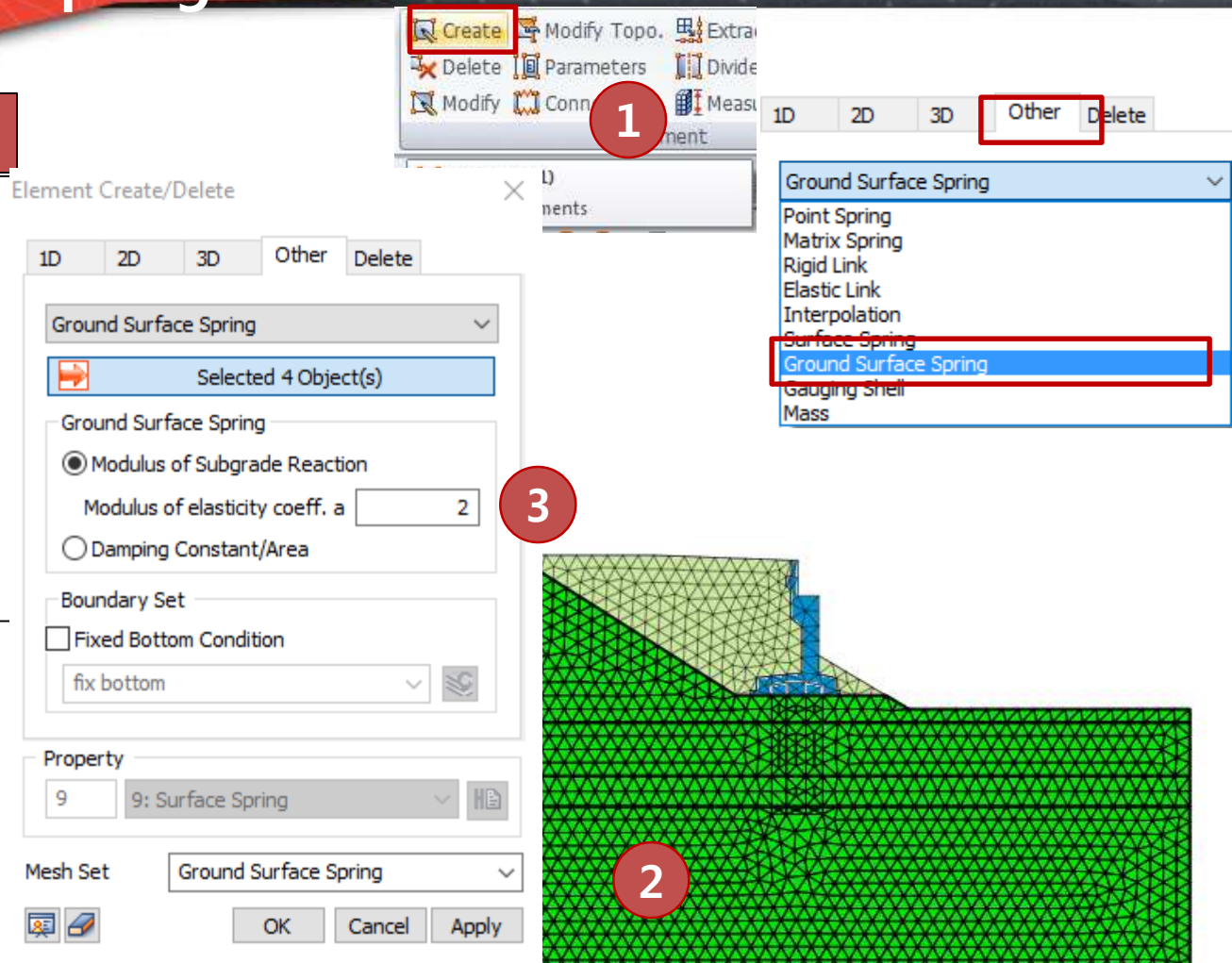
GTS NX



Procedure

1. Main Menu > Mesh > Element > Other > Ground Surface Spring
2. Select the 4 bottom ground layers as shown
3. Type the value of "2" for modulus of elasticity coefficient
4. Click [Apply].

<input checked="" type="checkbox"/>	Mesh	
<input checked="" type="checkbox"/>	Default Mesh Set	
<input checked="" type="checkbox"/>	Abutment	
<input checked="" type="checkbox"/>	Embankment 1	
<input checked="" type="checkbox"/>	Embankment 2	
<input checked="" type="checkbox"/>	Liquefaction Layer	
<input checked="" type="checkbox"/>	Weathered Rock	
<input checked="" type="checkbox"/>	Soft Rock	208
<input checked="" type="checkbox"/>	Pile	209
<input checked="" type="checkbox"/>	Ground Surface Spring	230



Modulus of Subgrade Reaction

GTS NX



The 'Create Ground Surface Spring' feature can generate automatically for dynamic analysis.

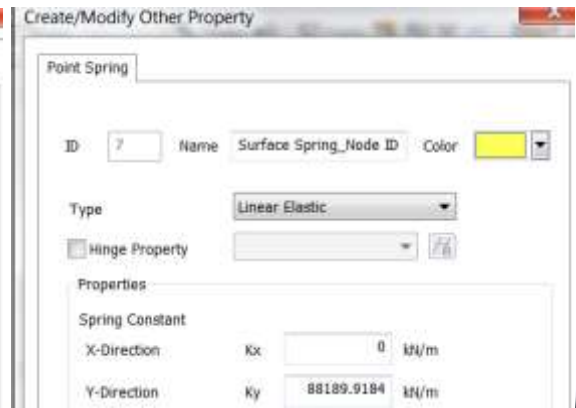
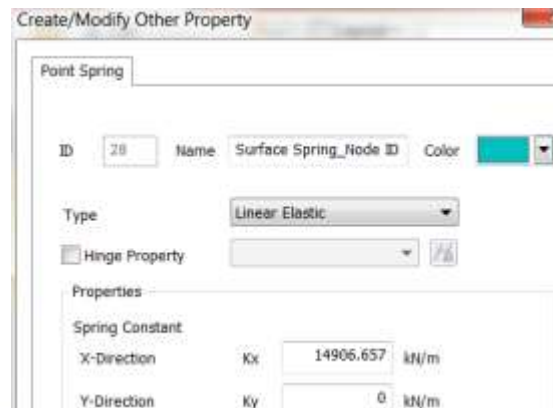
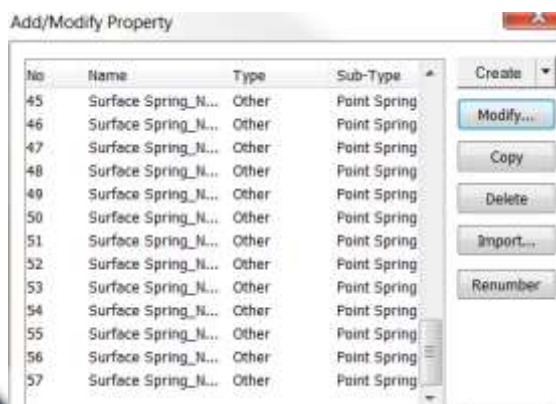
The modulus of subgrade reaction is calculated by equations in the 'Create Ground Surface Spring' feature.

Modulus of Vertical Subgrade Reaction : $k_v = k_{v0} \cdot \left(\frac{B_v}{30}\right)^{-3/4} \quad (\text{kgf/cm}^3)$

Modulus of Horizontal Subgrade Reaction : $k_h = k_{h0} \cdot \left(\frac{B_h}{30}\right)^{-3/4} \quad (\text{kgf/cm}^3)$

Where, $k_{v0} = \frac{1}{30} \cdot \alpha \cdot E_0 = k_{h0}$, $B_v = \sqrt{A_v}$, $B_h = \sqrt{A_h}$

α : Estimated factor for Modulus of subgrade reaction (use between 2.0~4.0)



Step 08 Analysis > General

Procedure

1. Main Menu > Analysis > General
2. Name 'Eigen Value', Analysis Type > Select [Eigenvalue].
3. Click-on Analysis Control [].
4. Water Level 2m Number of Frequencies '30'.
5. Check ON: Sturm Sequence Check, Max Negative Pore Pressure (0), and Allow Undrained Behavior

Click [OK], Click [OK].

Add/Modify Analysis Case

Analysis Case Setting

Title: Eigenvalue 3

Description:

Solution Type: Eigenvalue 2

Construction Stage Set: Construction Stage Set-1

Analysis Case Model

All Sets

Mesh

- Abutment
- Default Mesh Set
- Embankment 1
- Embankment 2
- Ground Surface Spring
- Liquefaction Layer
- Pile
- Soft Rock
- Weathered Rock
- Boundary Condition
- bottom fix
- Contact Pair

Mesh

- Abutment
- Default Mesh Set
- Embankment 1
- Embankment 2
- Ground Surface Spring
- Liquefaction Layer
- Pile
- Soft Rock
- Weathered Rock
- Boundary Condition
- bottom fix
- Contact Pair

Analysis Control

General

Initial Temperature

☐ Initial Temperature By Value: 0 [T]

Water Level

☒ Define Water Level: 2 m 4

☐ Define Water Level for Mesh Set: Inpt...

Eigenvectors

☒ Number of Modes: 30

Frequency Range of Interest

☐ Lowest: 0 ☐ Highest: 1000 Unit: [Cycle]/ sec

☒ Sturm Sequence Check

Saturation Effects

☐ Consider Partially Saturated Effects for Stress Analysis

Max. Negative Pore Pressure

☒ Max. Negative Pore Pressure Limit: 0 kN/m² 5

Undrained Condition

☒ Allow Undrained Material Behavior

Mass Parameters

☐ Coupled Mass Calculation

OK Cancel

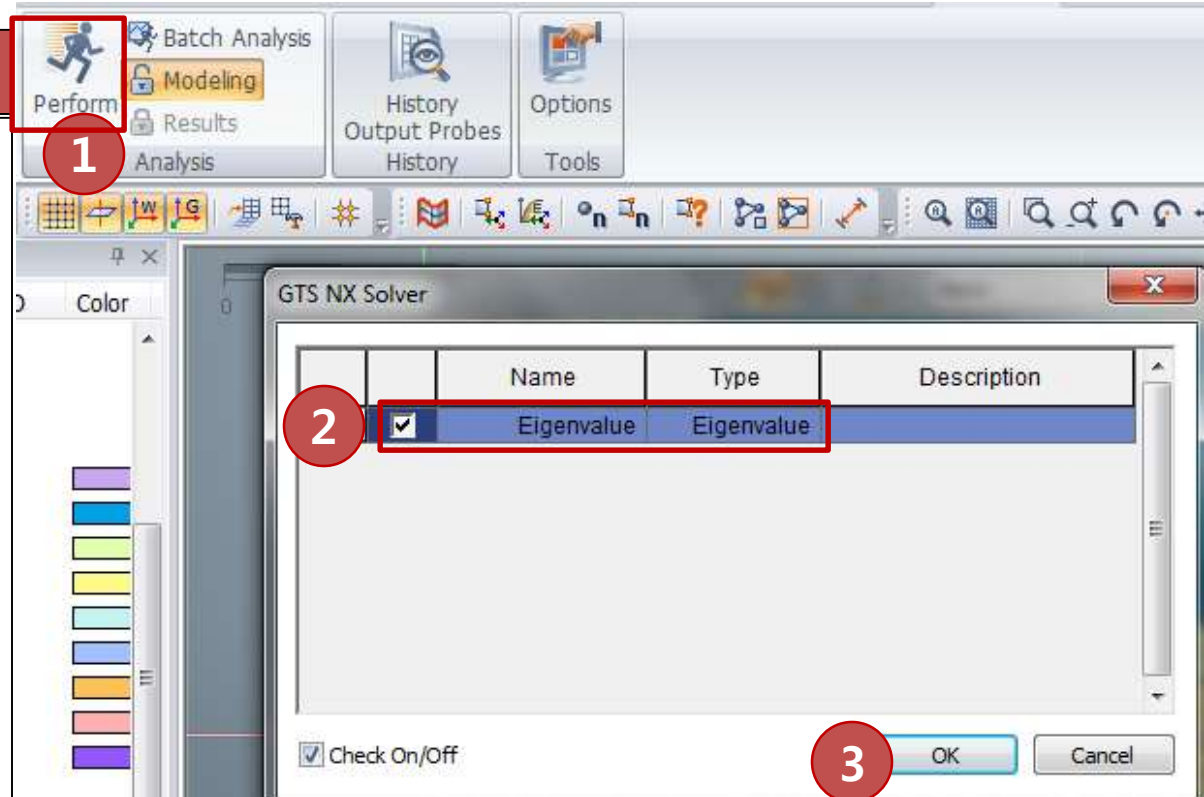
Step 9 Analysis > Perform

GTS NX



Procedure

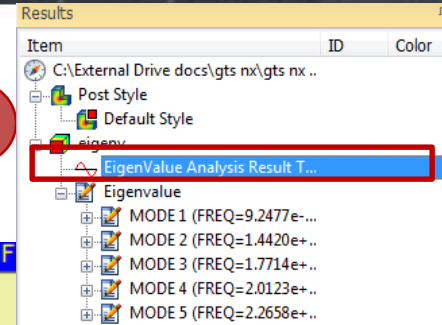
1. Main Menu > Analysis > Perform...
2. Check –on 'EigenValue'.
3. Click [OK].
4. All the messages during the analysis will be shown in the Output Window. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The model is automatically saved before the analysis. The result is saved as binary file(*.TA*) in the same folder as the model. The detail analysis information is also saved in a text file(*.OUT).



> [SYSTEM INFO]
> NUMBER OF THREADS : 4
> MAXIMUM MEMORY USAGE : 206 MB
> AVAILABLE MEMORY : 6262 MB
> TOTAL CPU TIME : 2.683 sec
> WALL CLOCK TIME : 1.498 sec
> TOTAL WARNINGS : 0

Step 10 Results > Vibration Mode

GTS NX



Procedure

1. Main Menu > Result > Vibration Mode Shape...
2. Check the periods of 1st and 6th modes where mass participation is the largest.
3. Keep the record of periods of 0.407 sec and 0.188 sec.

2		PERCENTAGE MODAL EFF						
MODE NUMBER	T1	T2	T3	R1				
1	27.89%	0.44%	0.00%	0.00%	0.00%	0.40%		
2	0.07%	6.68%	0.00%	0.00%	0.00%	16.61%		
3	0.01%	12.27%	0.00%	0.00%	0.00%	4.55%		
4	1.14%	0.69%	0.00%	0.00%	0.00%	0.02%		
5	0.82%	3.77%	0.00%	0.00%	0.00%	0.23%		
6	14.68%	0.87%	0.00%	0.00%	0.00%	1.01%		
7	0.01%	0.98%	0.00%	0.00%	0.00%	0.00%		

MODE 2 (FREQ=1.4420e+..)

MODE 3 (FREQ=1.7714e+..)

MODE 4 (FREQ=2.0123e+..)

MODE 5 (FREQ=2.2658e+..)

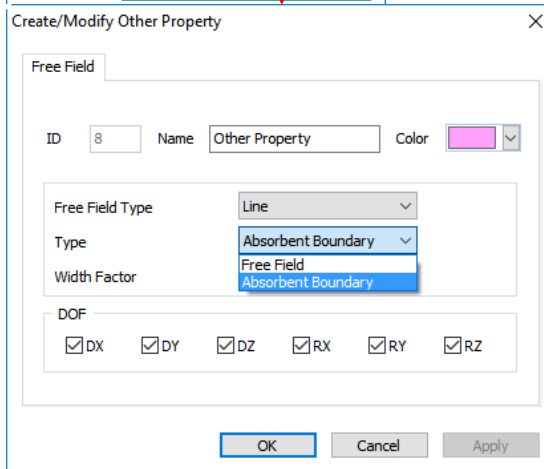
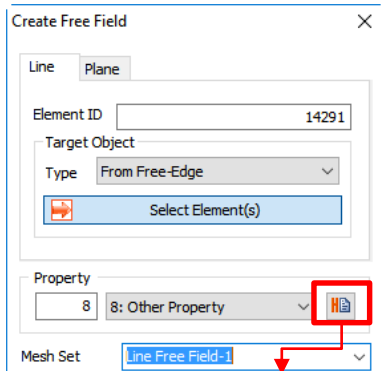
MODE NUMBER	REAL EIGENV			
	EIGENVALUE	RADIANS	CYCLES	PERIOD
1	2.376695e+002	1.541653e+001	2.453618e+000	4.075614e-001
2	4.462398e+002	2.112439e+001	3.862051e+000	2.974375e-001
3	4.780094e+002	2.186343e+001	4.73e+000	2.873833e-001
4	7.722640e+002	2.778964e+001	5.2858e+000	2.260981e-001
5	8.290507e+002	2.879324e+001	4.582587e+000	2.182174e-001
6	1.111551e+003	3.333993e+001	5.306214e+000	1.884583e-001

Step 11 Mesh > Element > Free Field

GTS NX



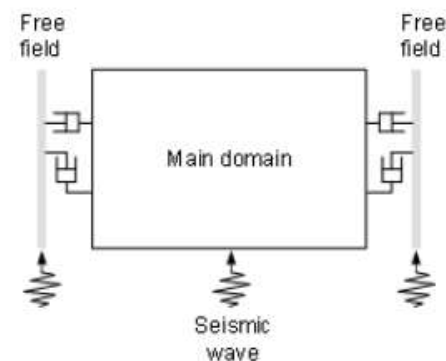
Free Field (Infinite boundary, Absorbent boundary)



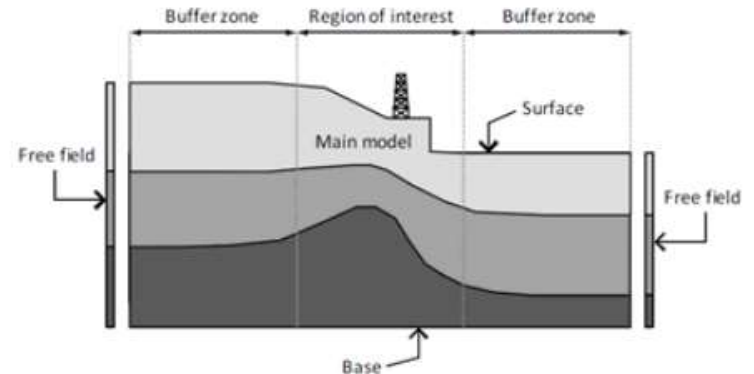
For the seismic analysis, users need to model infinite ground to eliminate the boundary effect caused by reflection wave. Since it is not possible to model infinite ground, users can apply Free Field Element at the boundary.

Absorbent Boundary : Enable to eliminate reflection wave at the ground boundary

Width Factor (Penalty Parameter) : In order to minimize the size effect of the model, users have to input more than 10^4 , This value will be multiplied by model width (In case of 2D, this is plain strain thickness (unit width)).



[Schematic Overview of Free Field Element]



[Free field effect (O), Absorb reflection (O)]

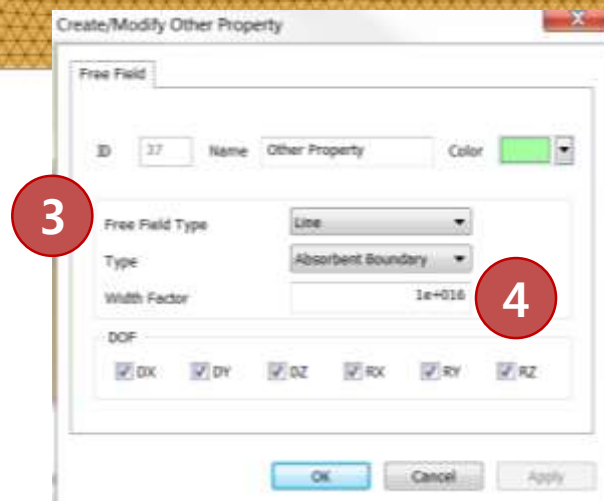
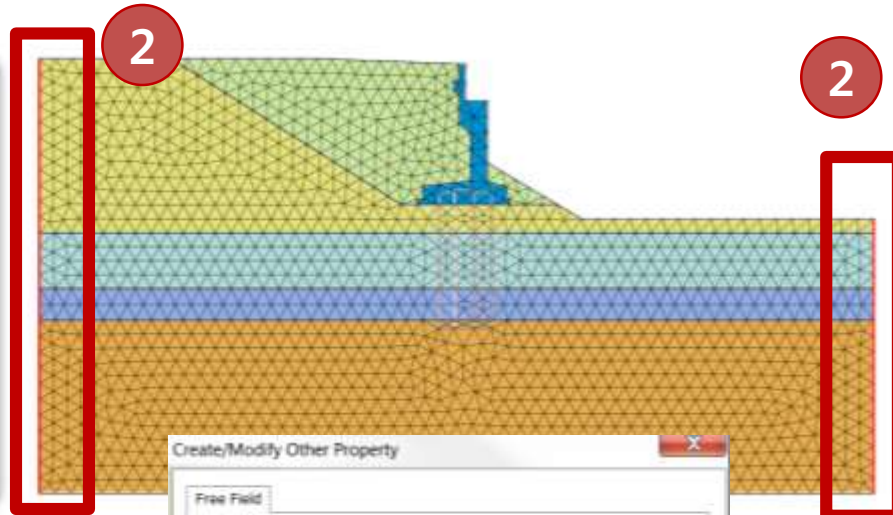
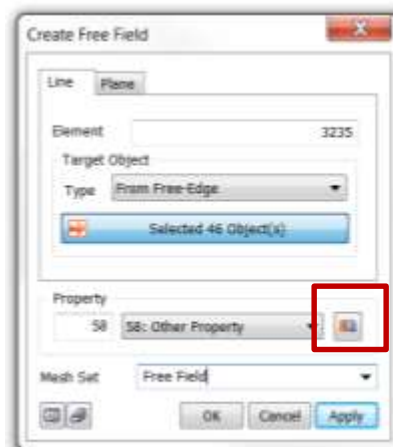
Step 11 Mesh > Element > Free Field

GTS NX



Procedure

1. Main Menu > Mesh > Element > Free Field >
2. Select the 2 side boundary nodes
3. Select Property as Absorbent
4. Width Factor 1×10^{16}
5. Click [Apply].



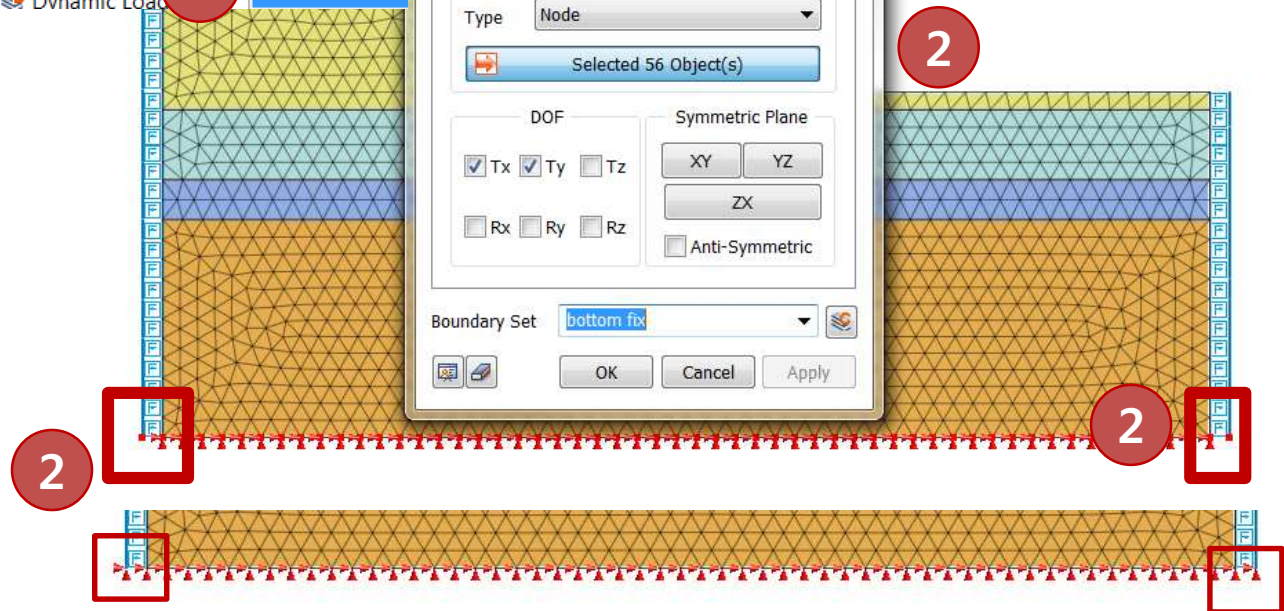
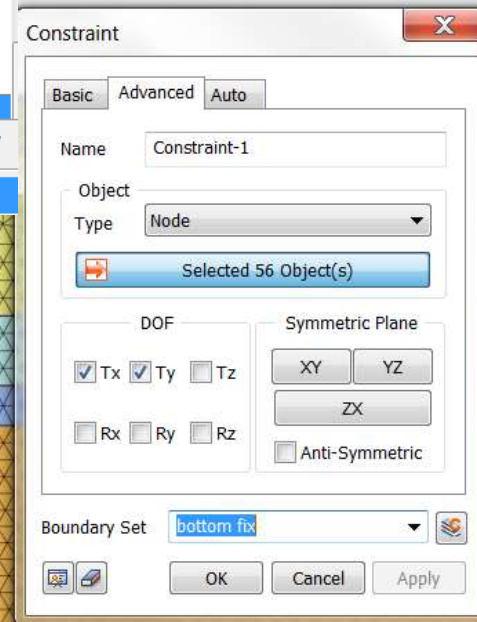
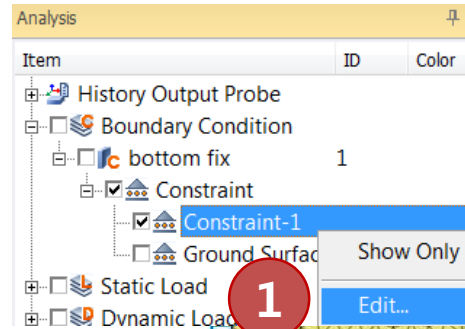
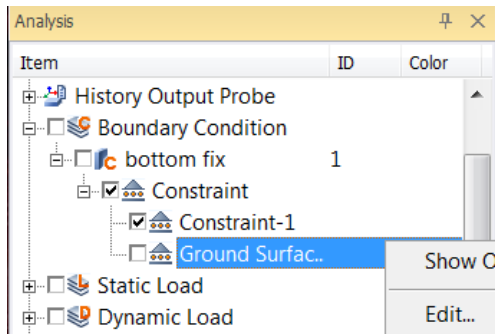
Step 11 Boundary for Free Field

GTS NX



Procedure

1. Tree Menu > Analysis > Boundary Conditions > Bottom Fix > Constraint > Edit
2. Select the 2 Free Field Elements
3. Delete the Ground Surface Spring



Step 12 Dynamic Analysis > Ground Acceleration

GTS NX



Procedure

1. Return to the Pre Mode
2. Main Menu > Dynamic Analysis> Load > Ground Acceleration
3. Activate X direction and click icon
4. Add Time Functions
Copy Paste function from Excel
5. Type Hachinohe in Name
6. Click OK OK.

The screenshot illustrates the steps for setting up Ground Acceleration in GTS NX. The main menu shows 'Dynamic Analysis' > 'Load' > 'Ground Acceleration'. The 'Ground Acceleration' dialog box shows 'X Direction' selected. The 'Time Function' dialog box shows 'Add Time Function' button. The 'Time History Load Function' dialog box shows 'Hachinohe' as the name and 'Normalized Acceleration' as the data type. The 'Import' tab shows a table of time and value data. The 'Graph Option' section shows 'X-axis Log Scale' and 'Y-axis Log Scale' options. The 'Seismic Function' section shows 'Seismic Function' selected. The graph shows the resulting acceleration data over time.

Time (sec)	Value (g)
1	0.0000
2	0.0620
3	0.1240
4	0.1860
5	0.2480
6	0.3100
7	0.3720
8	0.4340
9	0.4960
10	0.5580
11	0.6200
12	0.6820
13	0.7440

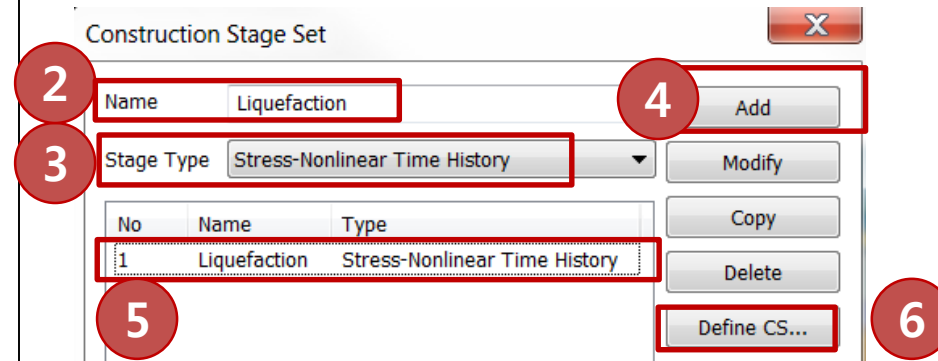
Step 13 Construction Stage

GTS NX



Procedure

1. Main Menu > Static / Slope Analysis > Construction Stage > Stage Set
2. Name: Liquefaction
3. Stage Type > Stress – Nonlinear Time History
4. Click > Add
5. Select "Liquefaction Set" from Table
6. Click > Define CS



Step 13 Construction Stage

GTS NX



Procedure

1. Stage name: Initial
2. Stage Type: Stress
3. Drag and Drop all data sets except Ground Surface Springs from Set Data to Activated Data
4. Set Water Level to 2m
5. Clear Displacement
6. Save > New

Define Construction Stage

Construction Stage Set Name: Liquefaction

Stage ID: 1: Initial

Stage Name: Initial

Stage Type: Stress

Set Data:

- Mesh
- Abutment
- Default Mesh Set
- Embankment 1
- Embankment 2
- Free Field
- Ground Surface Springs
- Liquefaction Layer
- Pile
- Soft Rock
- Weathered Rock
- Boundary Condition
- bottom fix
- Static Load
- Load Set-1
- Contact

Activated Data:

- Mesh
- Abutment
- Default Mesh Set
- Embankment 1
- Embankment 2
- Free Field
- Liquefaction Layer
- Pile
- Soft Rock
- Weathered Rock
- Boundary Condition
- bottom fix
- Contact
- Static Load
- Load Set-1

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Initial Condition:

☒ Define Water Level For Global

2 m None

☐ Define Water Level For Mesh Set

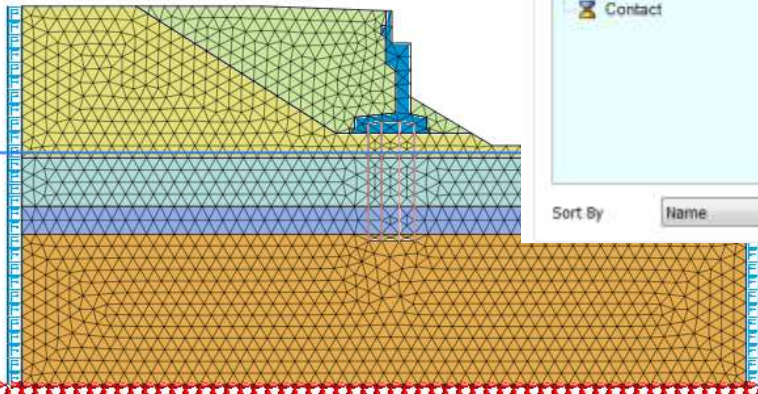
Input Water Level...

☒ Clear Displacement

Sort By: Name

Show Data Activate

Save Close



Step 13 Construction Stage

GTS NX



Procedure

1. Stage name: Time History
2. Stage Type: Time History
3. Drag and Drop Dynamic Load "Artificial" to Activated Data
4. Time Step > Define Time Step > Time Duration: 15s > Time Increment > 0.062 > Add
5. Analysis Control > General > Activate Undrained Behavior
6. Analysis Control > General > Dynamic > Damping Method
7. Select Calculate from Modal Damping Damping, and Select Period [sec]

Enter the periods for 0.407s and 0.188s from previous eigenvalue analysis.

Enter 0.05 in Damping Ratio for both Mode 1 and 2

Click [OK] > [Save] > [OK]

The screenshot shows the 'Define Construction Stage' dialog box in GTS NX. The 'Stage Set Name' is 'Liquefaction'. The 'Stage ID' is '2: seismic'. The 'Stage Name' is 'seismic'. The 'Type' is 'Nonlinear Time History'. The 'Time Step...' button is visible. The 'Activated Data' list includes 'Mesh', 'Boundary Condition', 'Contact', 'Dynamic Load', and 'Seismic Function'. The 'Deactivated Data' list includes 'Mesh', 'Boundary Condition', 'Static Load', and 'Contact'. The 'Initial Condition' section has options for 'Define Water Level For Global' and 'Define Water Level For Mesh Set'. The 'Define Time Step' dialog box is open, showing 'Name: time', 'Time Duration: 15 sec', 'Time Increment: 0.062 sec', and 'Intermediate Output (Every N Time): 1'. The 'Analysis Control' dialog box is open, showing the 'General' tab with 'Undrained Condition' checked. The 'Damping Method' dialog box is open, showing 'Damping' set to 'Mass Stiffness Proportional', 'Mass and Stiffness Coefficients' set to 'Calculate from Modal Damping', and 'Coefficients Calculation' set to 'Period [sec]'. The 'Damping Ratio' is set to 0.05 for both Mode 1 and Mode 2.

Red callouts 1 through 7 highlight the following steps:

1. Stage Set Name: Liquefaction
2. Stage ID: 2: seismic
3. Stage Name: seismic
4. Type: Nonlinear Time History
5. Time Step...
6. Define Time Step dialog box
7. Analysis Control dialog box

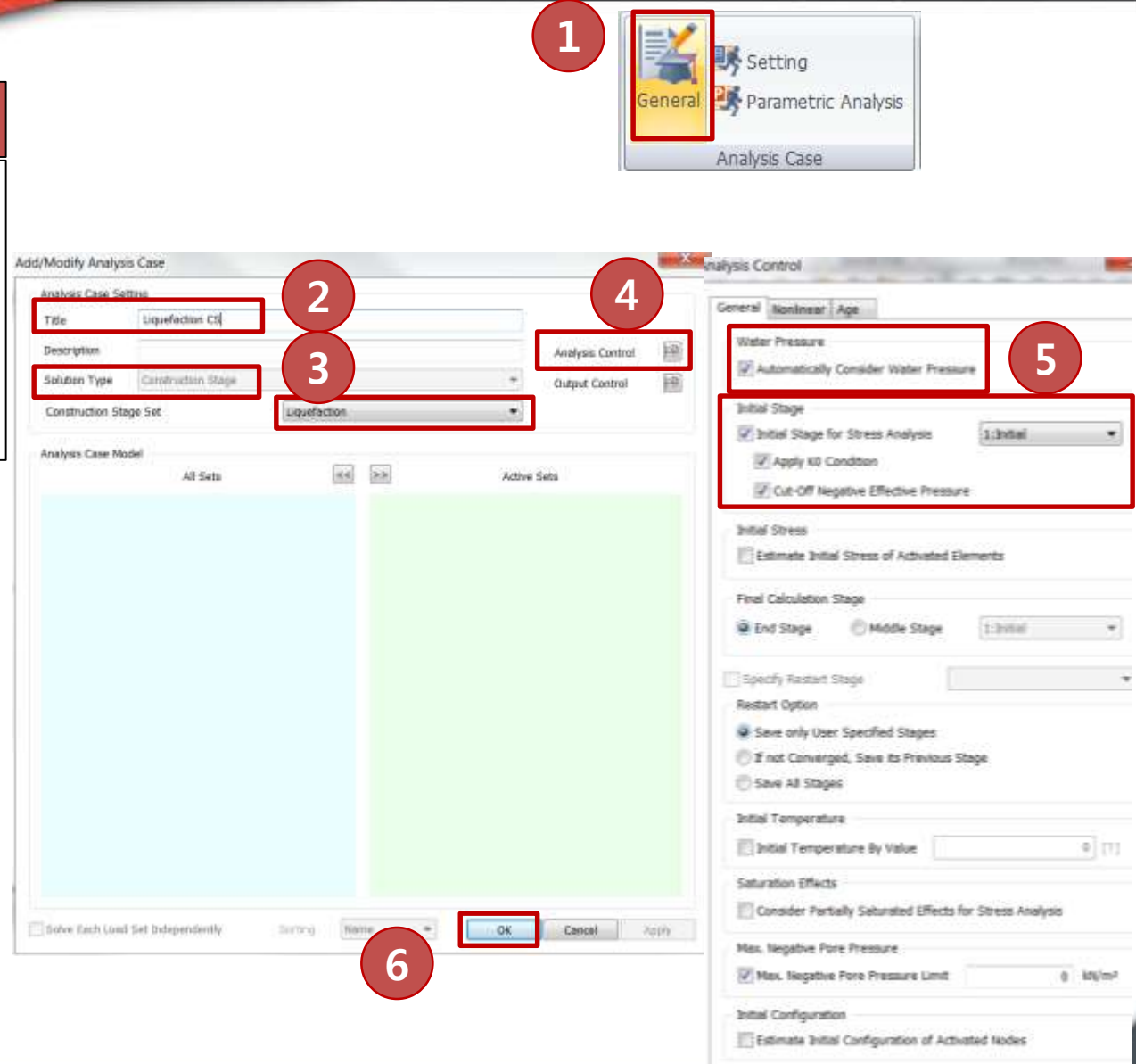
Step 14 Analysis Stage

GTS NX



Procedure

1. Analysis > Case > General
2. Title: Liquefaction CS
3. Solution Type: Construction Stage > Liquefaction
4. Analysis Control >
5. General > Set as shown
6. Ok > Ok



Step 15 Analysis > Perform

GTS NX

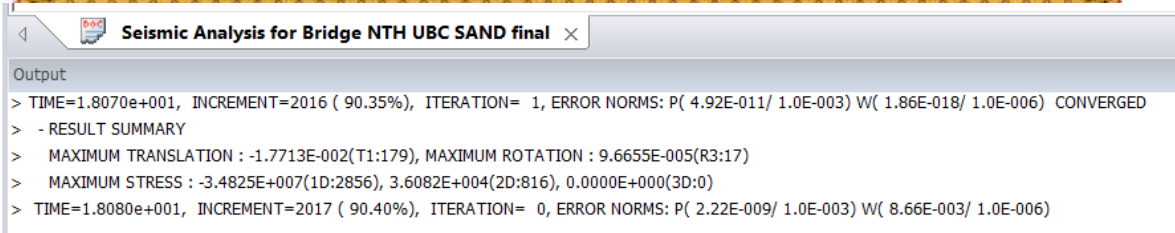
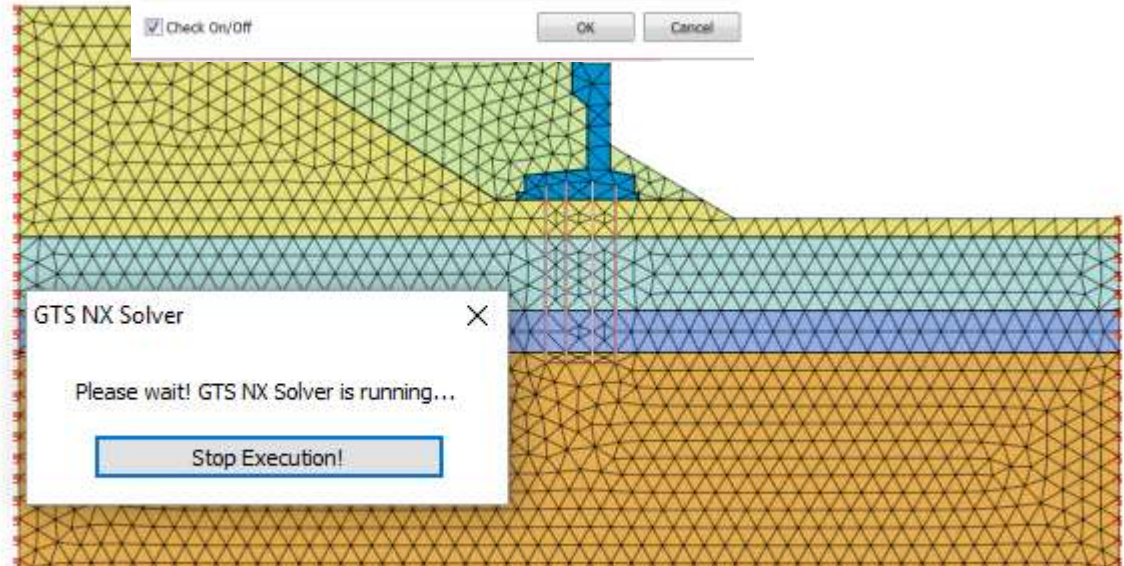
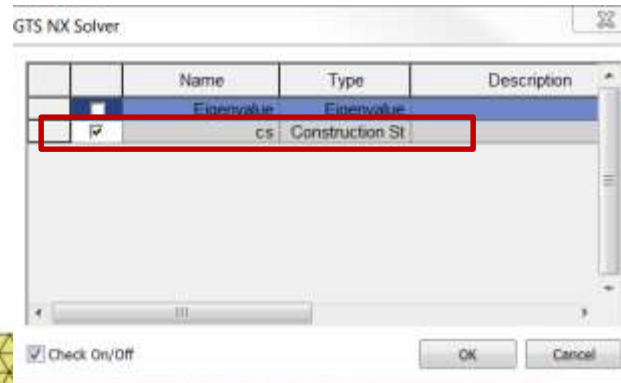


Procedure

1. Main Menu > Analysis > Perform...
2. Check –on 'CS' 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 result is saved as binary file(*.TA*) in the same folder as the model. The detail analysis information is also saved in a text file(*.OUT).

2



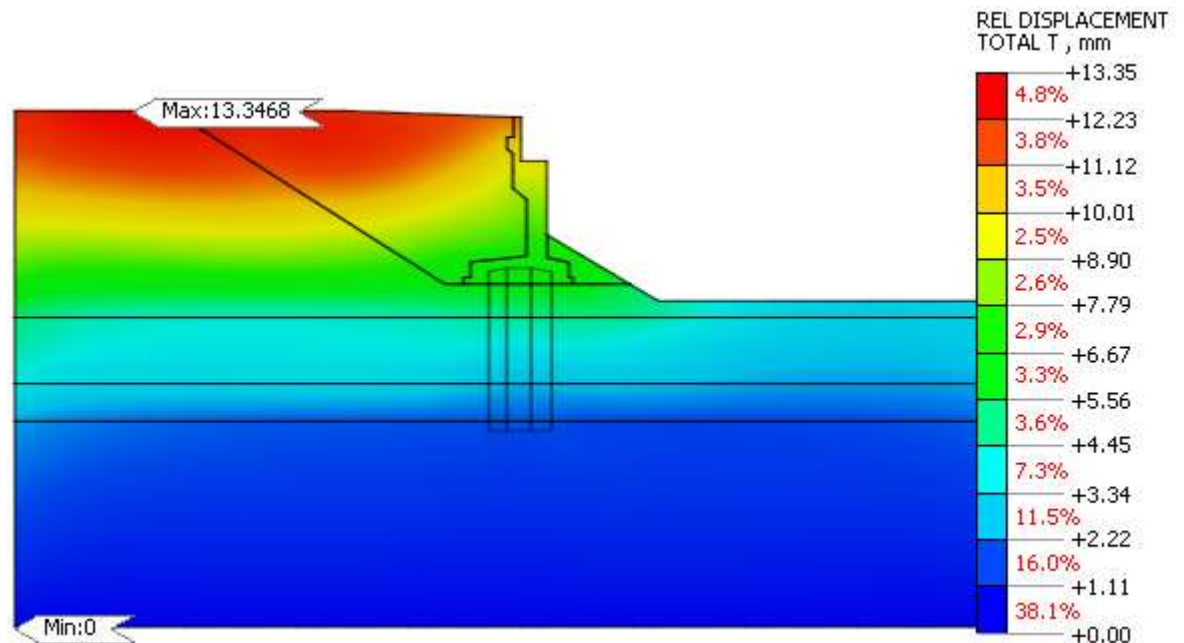
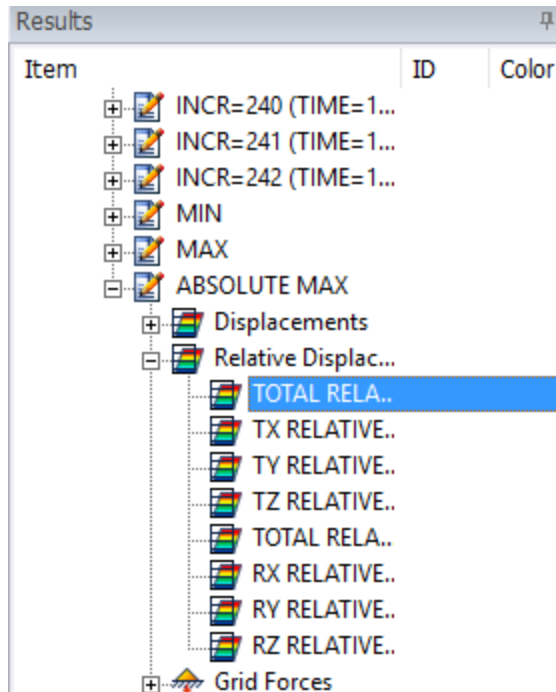
Step 16 Results > Displacements

GTS NX



Procedure

1. Post Works Tree > Time History> Absolute Max > Relative Displacement > Total



Step 16 Results > Beam Element Forces

GTS NX

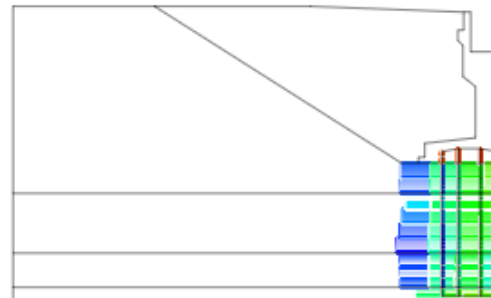


Procedure

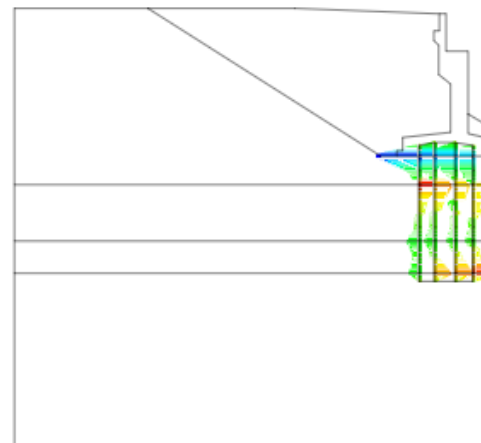
1. Post Works Tree > Time History > Absolute Max > Beam Force Element > Axial Force

Results

Item	ID	C
[-] ABSOLUTE MAX		
[+] Displacements		
[+] Relative Displac...		
[+] Grid Forces		
[+] Reactions		
[+] Velocities		
[+] Relative Velociti...		
[+] Accelerations		
[+] Relative Acceler..		
[+] Spring Element ..		
[+] Beam Element F..		
[+] AXIAL FORC..		
[+] SHEAR FOR...		
[+] SHEAR FOR...		
[+] TORQUE		
[+] BENDING M..		
[+] BENDING M..		



BEAM FORCE AXIAL FORCE , kN	
4.3%	+33.14
4.3%	-54.88
2.1%	-142.90
2.1%	-230.92
5.3%	-318.95
16.0%	-406.97
11.7%	-494.99
19.1%	-583.01
10.6%	-671.03
7.4%	-759.06
9.6%	-847.08
7.4%	-935.10
7.4%	-1023.12



BEAM FORCE BENDING MMNT Y , kN*m	
3.6%	+71.82
7.7%	+54.94
13.3%	+38.06
16.4%	+21.17
17.9%	+4.29
14.9%	-12.59
7.7%	-29.47
4.1%	-46.36
4.1%	-63.24
4.1%	-80.12
4.1%	-97.00
4.1%	-113.88
2.1%	-130.77

Step 16 Results > Accelerations

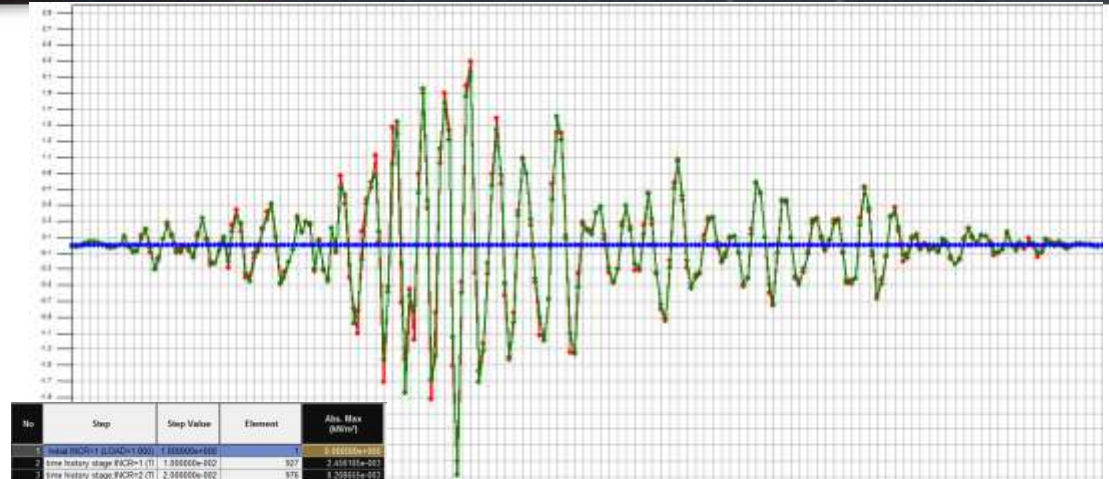
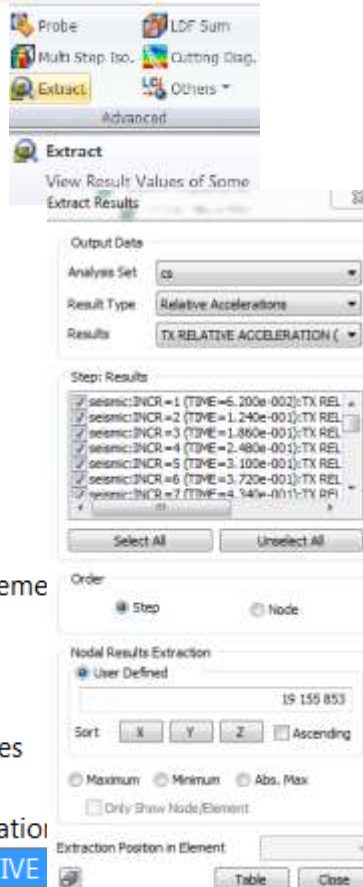
GTS NX



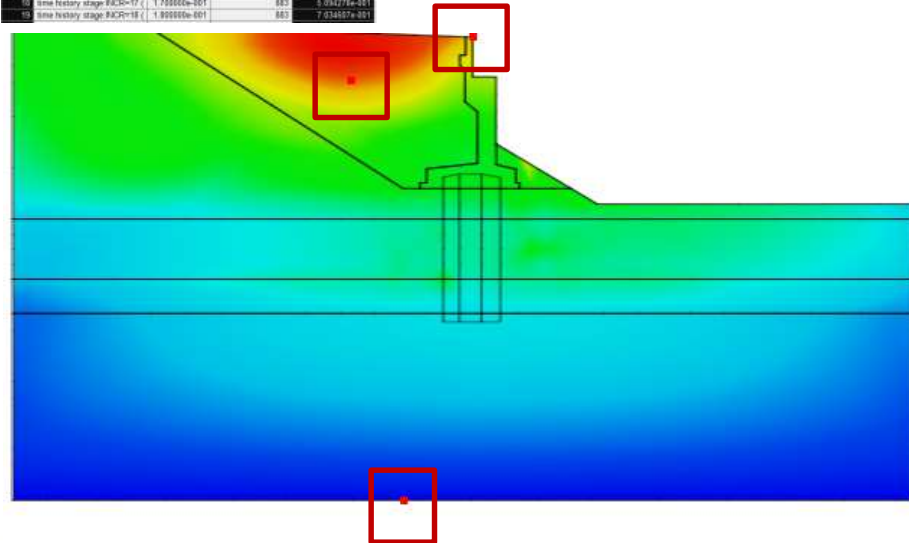
Procedure

Post Works Tree >
Time History >
Absolute Max >
Relative Accelerations
> Total
Results > Extract >
Relative Acc TX >
nodes 19, 129, 849 >
Table > Graph

- ABSOLUTE MAX
 - Displacements
 - Relative Displacements
 - Grid Forces
 - Reactions
 - Velocities
 - Relative Velocities
 - Accelerations
 - Relative Accelerations
 - TOTAL RELATIVE
 - TX RELATIVE ACCELERATION
 - TY RELATIVE ACCELERATION
 - TZ RELATIVE ACCELERATION
 - TOTAL RELATIVE ANGULAR
 - RX RELATIVE ACCELERATION
 - RY RELATIVE ACCELERATION
 - RZ RELATIVE ACCELERATION



No	Step	Step Value	Element	Abs. Max (MINI)
1	time history stage INCR=1 (1)	1.000000e-002	1	1.000000e-002
2	time history stage INCR=2 (1)	1.000000e-002	927	2.016115e-003
3	time history stage INCR=2 (1)	2.000000e-002	976	1.598615e-003
4	time history stage INCR=3 (1)	3.000000e-002	976	1.251817e-002
5	time history stage INCR=4 (1)	4.000000e-002	889	1.923384e-002
6	time history stage INCR=5 (1)	5.000000e-002	883	1.633726e-002
7	time history stage INCR=6 (1)	6.000000e-002		
8	time history stage INCR=7 (1)	7.000000e-002		
9	time history stage INCR=8 (1)	8.000000e-002		
10	time history stage INCR=9 (1)	9.000000e-002		
11	time history stage INCR=10 (1)	1.000000e-001		
12	time history stage INCR=11 (1)	1.100000e-001		
13	time history stage INCR=12 (1)	1.200000e-001	976	7.988617e-002
14	time history stage INCR=13 (1)	1.300000e-001	976	1.125126e-001
15	time history stage INCR=14 (1)	1.400000e-001	976	1.687851e-001
16	time history stage INCR=15 (1)	1.500000e-001	976	2.374762e-001
17	time history stage INCR=16 (1)	1.600000e-001	883	3.264815e-001
18	time history stage INCR=17 (1)	1.700000e-001	883	5.094277e-001
19	time history stage INCR=18 (1)	1.800000e-001	883	7.034867e-001



TOTAL T, m/sec^2
+3.28
1.3%
+3.00
1.3%
+2.73
1.7%
+2.46
1.6%
+2.18
2.8%
+1.91
3.0%
+1.64
10.6%
+1.37
10.2%
+1.09
13.1%
+0.82
22.6%
+0.55
19.8%
+0.27
12.2%
+0.00

Step 16 Results > Excessive Pore Stress

GTS NX



Procedure

Post Works Tree > Time History > Absolute Max > Plane Strain Stresses > Excessive Pore Stress Results > Extract > Excess Pore Stress > Abs Max > Table > Graph

- ☒ Plane Strain Stresses
 - ☒ S-XX TOTAL
 - ☒ S-YY TOTAL
 - ☒ S-ZZ TOTAL
 - ☒ S-XX
 - ☒ S-YY
 - ☒ S-ZZ
 - ☒ S-XY
 - ☒ S-MAJOR PRINCIPAL (V)
 - ☒ S-MINOR PRINCIPAL (V)
 - ☒ SAFETY FACTOR
 - ☒ S-MAX SHEAR
 - ☒ S-EQUIVALENT
 - ☒ S-VON MISES
 - ☒ PLASTIC STATUS
 - ☒ MEAN EFFECTIVE PRESSURE
 - ☒ MEAN TOTAL PRESSURE
 - ☒ PORE STRESS
 - ☒ EXCESSIVE PORE STRESS

Extract Results

Output Data

Analysis Set: CS

Result Type: Plane Strain Stresses

Results: EXCESSIVE PORE STRESS

Step: Results

☒ Initial: BOCR=1 (LOAD=1.00e+000): EXCESSIVE F

☒ seismic: BOCR=2 (TIME=6.200e-002): EXCE

☒ seismic: BOCR=3 (TIME=1.240e-001): EXCE

☒ seismic: BOCR=4 (TIME=2.480e-001): EXCE

☒ seismic: BOCR=5 (TIME=3.180e-001): EXCE

☒ seismic: BOCR=9 (TIME=3.720e-001): EXCE

Select All Unselect All

Order: ☒ Step ☐ Element

Element Result Extraction

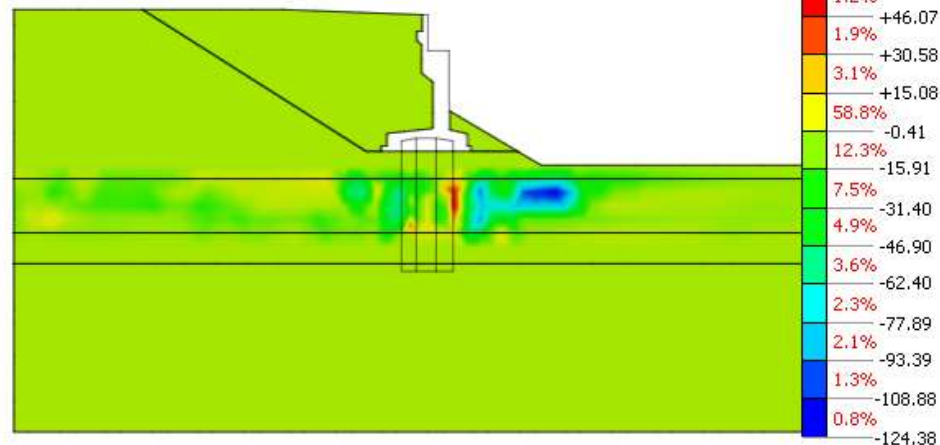
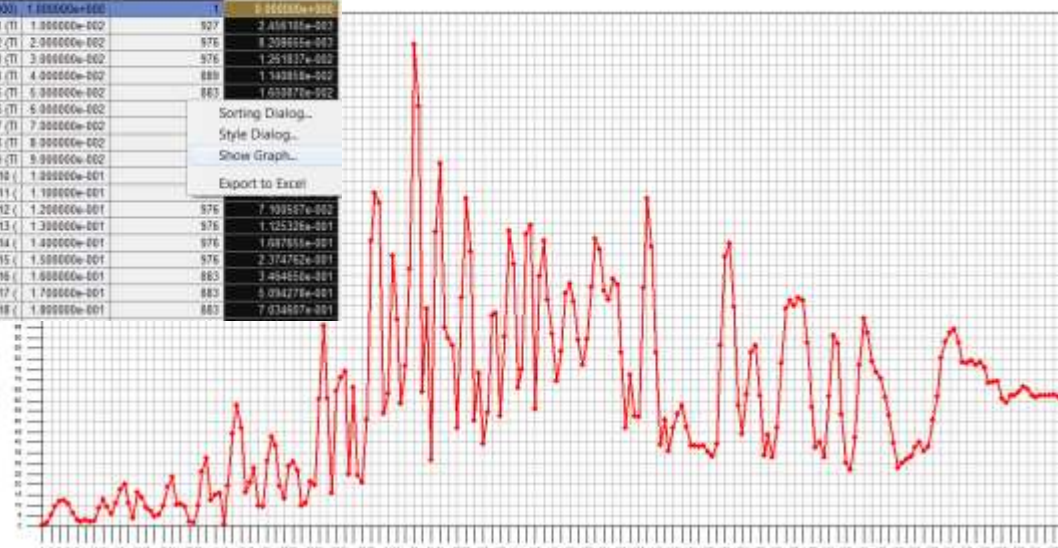
☒ User Defined

Sort: ☒ X ☐ Y ☐ Z ☐ Ascending

☐ Maximum ☐ Minimum ☒ Abs. Max

Step Value	Element	Abs. Max (kN/m ²)
BOCR=1 (0.00)	1.1111111e+000	1.1111111e+000
seismic: BOCR=1 (TIME=6.200e-002): EXCE	927	2.458184e-003
seismic: BOCR=2 (TIME=1.240e-001): EXCE	976	8.298666e-003
seismic: BOCR=3 (TIME=2.480e-001): EXCE	976	1.251817e-002
seismic: BOCR=4 (TIME=3.180e-001): EXCE	889	1.148816e-002
seismic: BOCR=5 (TIME=3.720e-001): EXCE	863	1.038770e-002
seismic: BOCR=6 (TIME=4.000e-001): EXCE		
seismic: BOCR=7 (TIME=4.000e-001): EXCE		
seismic: BOCR=8 (TIME=4.000e-001): EXCE		
seismic: BOCR=9 (TIME=4.000e-001): EXCE		
seismic: BOCR=10 (TIME=4.000e-001): EXCE		
seismic: BOCR=11 (TIME=4.000e-001): EXCE		
seismic: BOCR=12 (TIME=4.000e-001): EXCE	976	7.189567e-002
seismic: BOCR=13 (TIME=4.000e-001): EXCE	976	1.125330e-001
seismic: BOCR=14 (TIME=4.000e-001): EXCE	976	1.687891e-001
seismic: BOCR=15 (TIME=4.000e-001): EXCE	976	2.374762e-001
seismic: BOCR=16 (TIME=4.000e-001): EXCE	863	3.484650e-001
seismic: BOCR=17 (TIME=4.000e-001): EXCE	863	5.094271e-001
seismic: BOCR=18 (TIME=4.000e-001): EXCE	863	7.034897e-001

Sorting Dialog...
Style Dialog...
Show Graph...
Export to Excel



Step 16 Results > UBC Sand Results

GTS NX



■ Pore Pressure Ratio (PPR)

- The ratio of excessive pore pressure change and the initial effective pressure

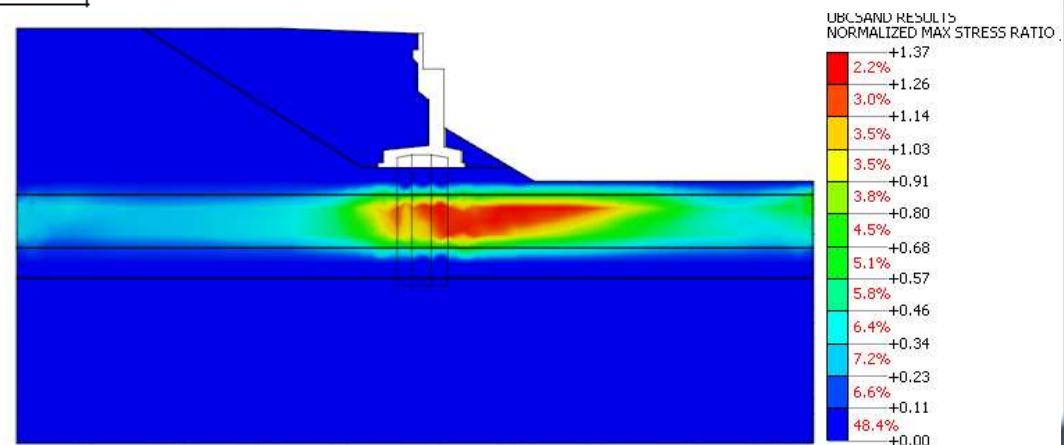
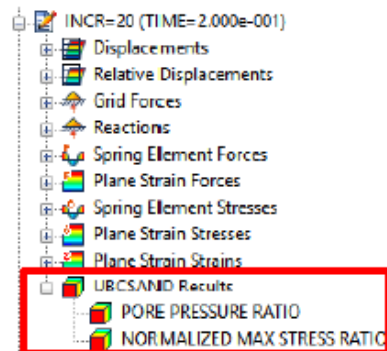
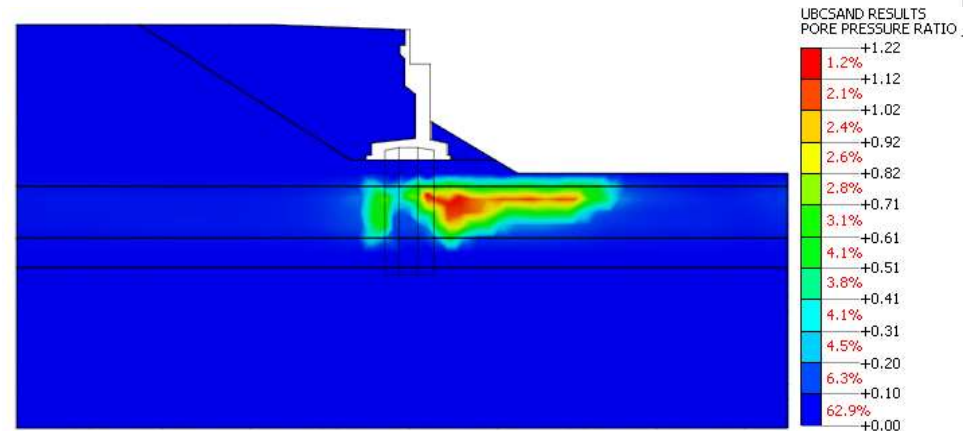
$$PPR = -\frac{\Delta p_w}{p_{init}} = \frac{p'_{init} - p'_{current}}{p_{init}}$$

Δp_w	Excessive Pore Pressure Change
p_{init}	Initial Effective Pressure
$p_{current}$	Current Effective Pressure

■ Normalized Max Stress Ratio

- The ratio of mobilized friction angle and the peak friction angle
- When the Max stress ratio is reached, the mobilized friction angle is close to the peak friction angle, liquefaction is triggered (1 = Liquefaction)

$\max \left(\frac{\sin \phi_m}{\sin \phi_p} \right)$	ϕ_m	Mobilized Friction Angle
	ϕ_p	Peak Friction Angle





End

Questions?

Global Help Desk
<http://globalsupport.midasuser.com/helpdesk/>