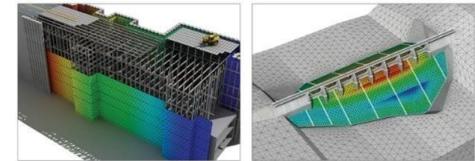


Consolidation Analysis using Vertical Drains 2D

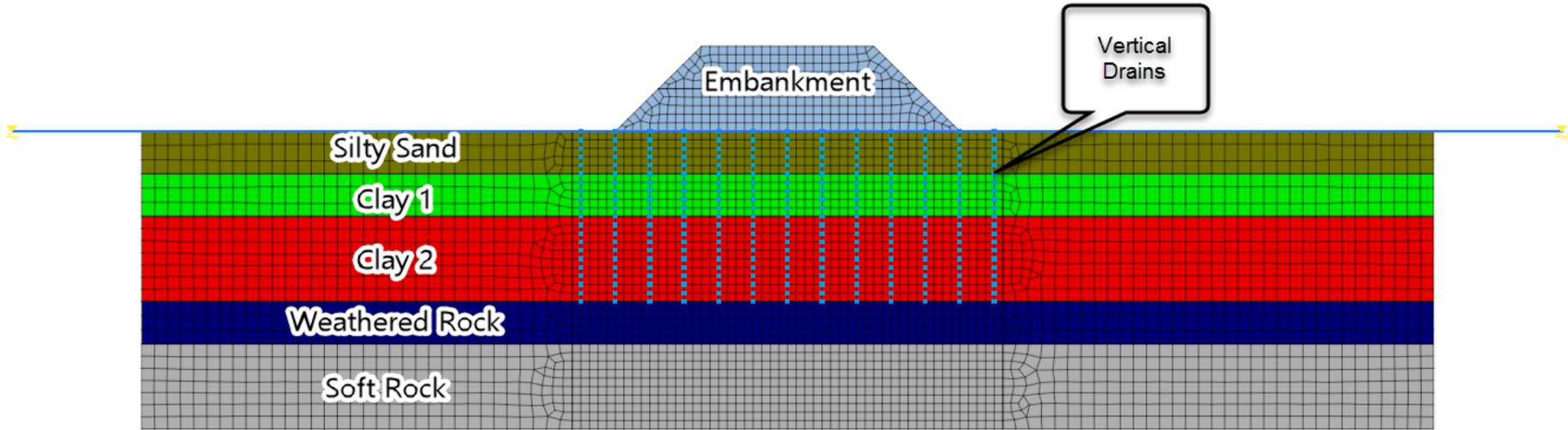
GTS NX
Geo-Technical analysis System New eXperience



by Tadavarthi Sree Harsha

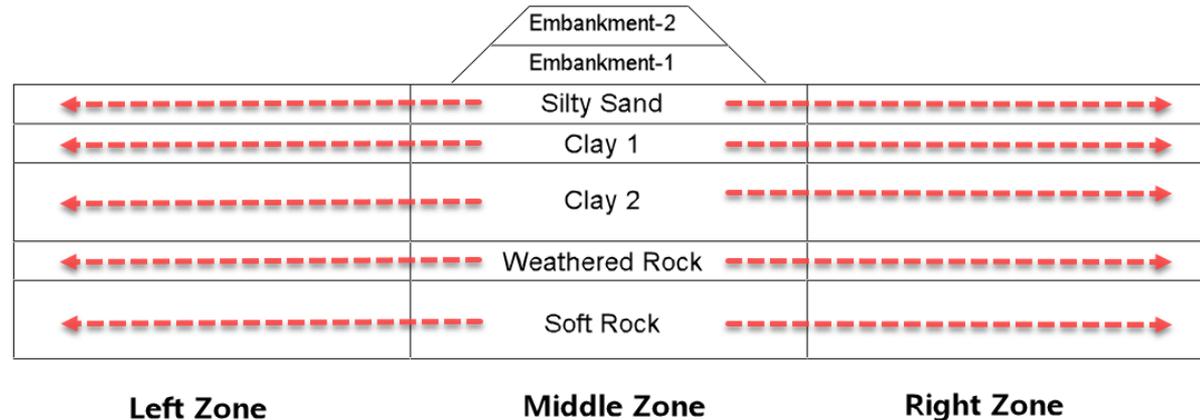
Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering





Objectives

1. Plane strain elements are used to model both the ground conditions and Embankment.
2. Vertical drains are not actually being modeled in this tutorial, but the effect of the vertical drains is modeled using 'Draining Function'.
3. An Embankment to be constructed in two stages to facilitate the excess pore pressure dissipation.
4. Modeling of vertical drains with 2m spacings will be done and results are analyzed. The middle zone shown in the figure is the ground improvement zone.
5. Starting Files Required:
Vertical Drains_2D_Start.gts



Material Table-1

	Embankment	Silty Sand	Weathered Rock	Soft Rock
Type	Elastic	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb
Elastic Modulus (kN/m ²)	10000	50000	150000	300000
Poisson's Ratio	0.33	0.3	0.2	0.15
Unit Weight(kN/m ³)	17	18	20	23
K0 (Manual)	0.5	0.6	1	1
Unit Weight Saturated (kN/m ³)	18	20	21	23
Initial Void Ratio	0.5	0.5	0.5	0.5
Drainage Parameters	Drained	Drained	Drained	Drained
Permeability Coefficients (Kx, Ky, & Kz) (m/day)	0.009	1.728	0.1728	0.01728
Void Ratio Dependency on Permeability (ck)	-	0.5	0.5	0.5
Cohesion (kN/m ²)	-	20	100	200
Friction Angle (degrees)	-	25	33	38

Material Table-2

	Clay 1	Clay 2
Type	Modified Cam Clay	Modified Cam Clay
Elastic Modulus (kN/m ²)	3450	7000
Poisson's Ratio	0.35	0.4
Unit Weight(kN/m ³)	16	17
K0 (Manual)	0.5	0.5
Unit Weight Saturated (kN/m ³)	16	17
Initial Void Ratio	1.76	1.5
Drainage Parameters	Undrained (Effective Stiffness and Effective Strength)	Undrained (Effective Stiffness and Effective Strength)
Permeability Coefficients (K _x , K _y , & K _z) (m/day)	0.001728	0.001728
Void Ratio Dependency on Permeability (ck)	0.5	0.5

	Clay 1	Clay 2
Type	Modified Cam Clay	Modified Cam Clay
OCR	2.05	1
Slope of Consol Line (λ)	0.313	0.32
Slope of Over Consol Line (k)	0.063	0.028
Slope of Critical State Line (M)	1.113	1
Allowable Tensile Stress (kN/m ²)	50	50

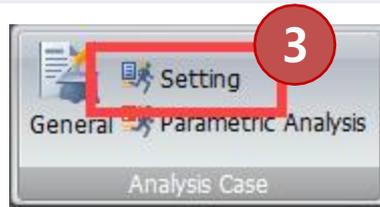
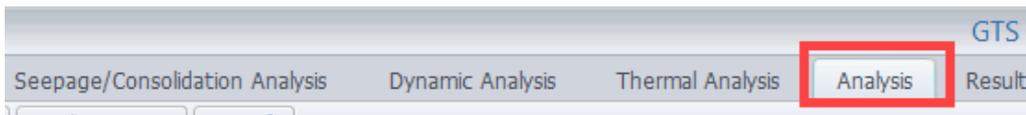
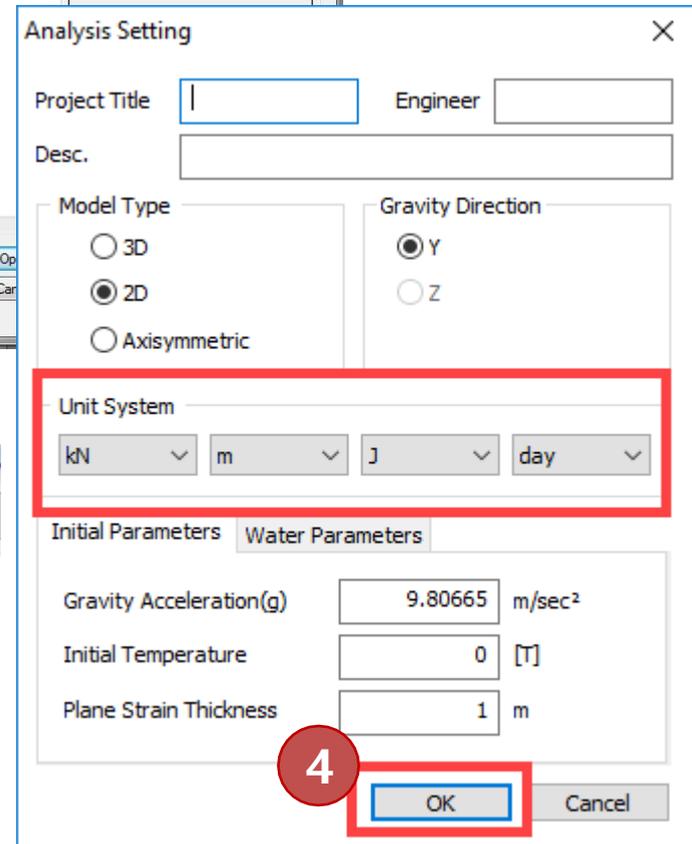
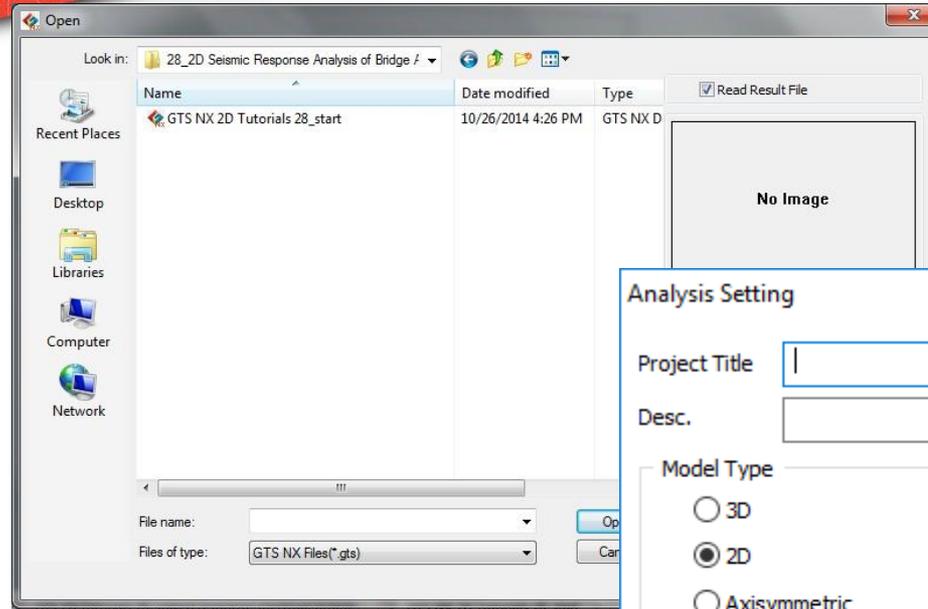
Properties

ID	Name	Type	Subtype
1	Embankment	2D	Plane Strain
2	Silty Sand	2D	Plane Strain
3	Clay 1	2D	Plane Strain
4	Clay 2	2D	Plane Strain
5	Weathered Rock	2D	Plane Strain
6	Soft Rock	2D	Plane Strain

Step 01-1 File > Open

Procedure

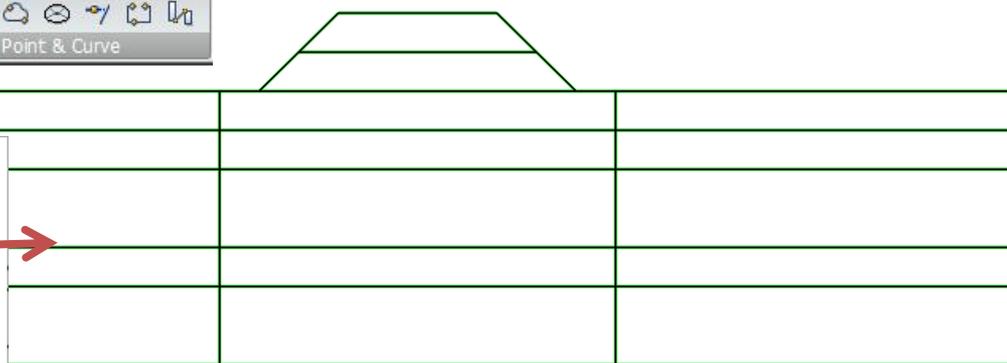
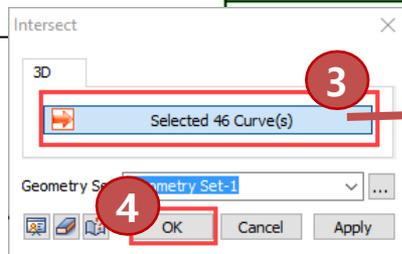
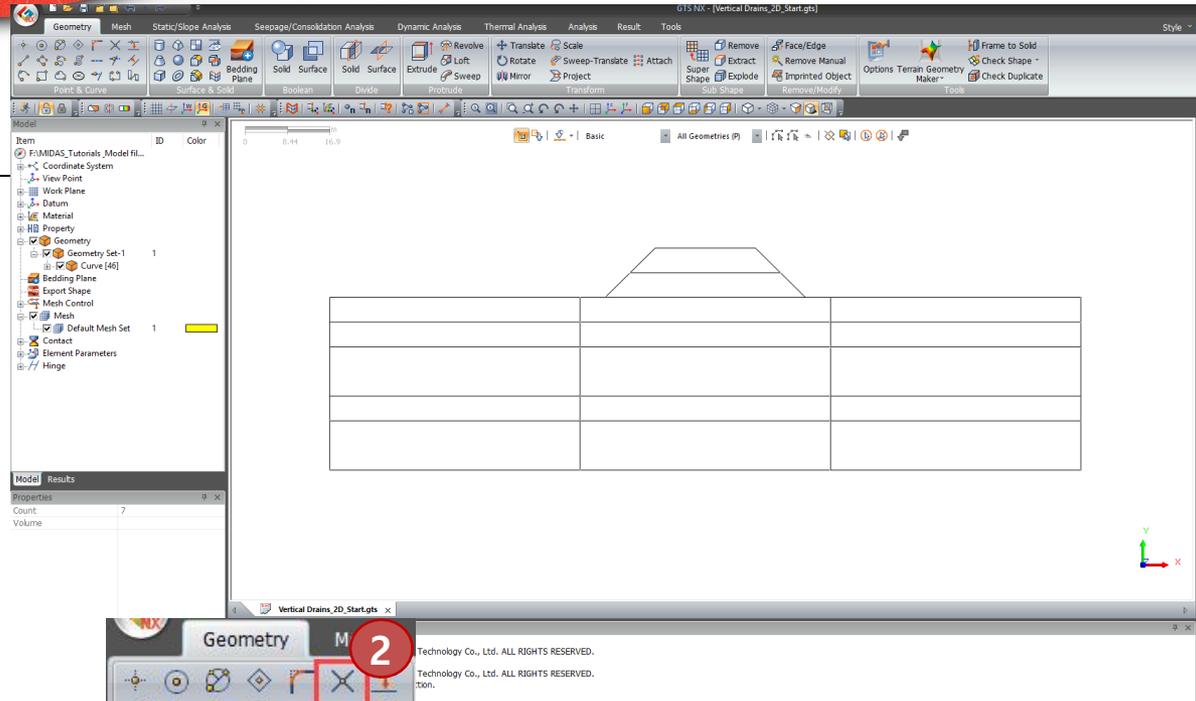
1. Main Menu > File>Open...
2. Double click 'Vertical Drains_2D_Start'.
3. Main Menu > Analysis > Analysis Case > Setting > Unit System > kN>m>Day
4. Click [OK].



Step 01-2 File > Open

Procedure

1. The Outline of the geometry is already modeled in the start file as shown in the picture.
2. Go to Geometry > Point & Curve > Intersect
3. Select all the curves
4. Click 'OK'
5. Please use this geometry to proceed for consolidation analysis.

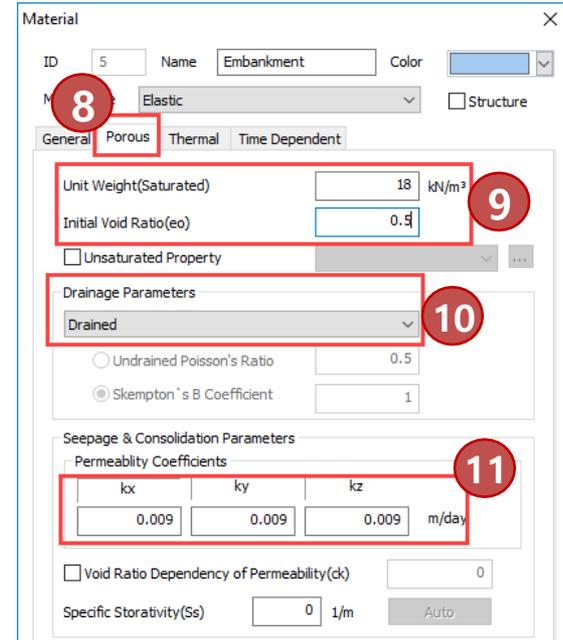
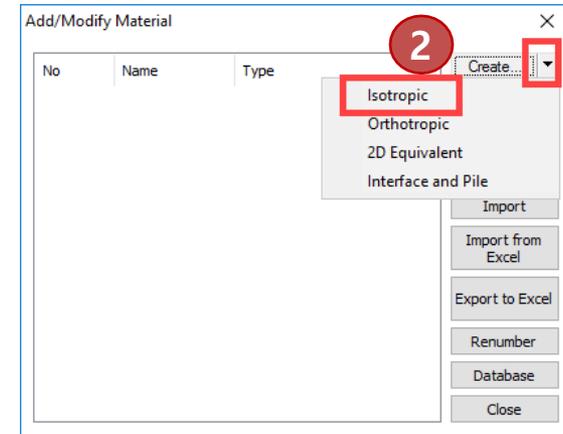
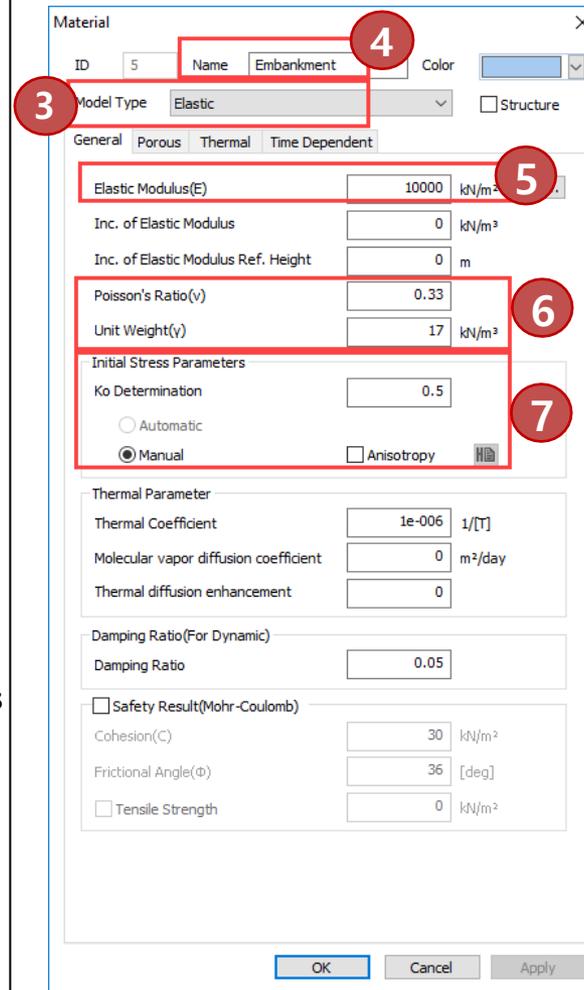
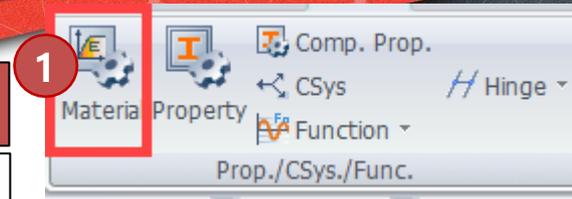


Step 02 Mesh > Material Definition

Procedure

1. Mesh > Prop./Csys./Func.> Material
2. Click on Create and select Isotropic
3. Choose the model type as Elastic
4. Give the name as 'Embankment'
5. Input the value of Elastic Modulus = 10000 kN/m²
6. Input Unit weight = 17 kN/m³
7. Input the poisson's ratio=0.33 and K0=0.5
8. Go to Porous tab
9. Input Saturated unit weight = 18 kN/m³ and Initial void ratio=0.5
10. Select Drained Conditions
11. Input permeability Coefficients = 0.009 m/day
12. Click 'OK'

Similarly define the inputs for Silty Sand, Weathered Rock, Soft Rock, Clay 1, and Clay 2 as per Material Table 1&2.



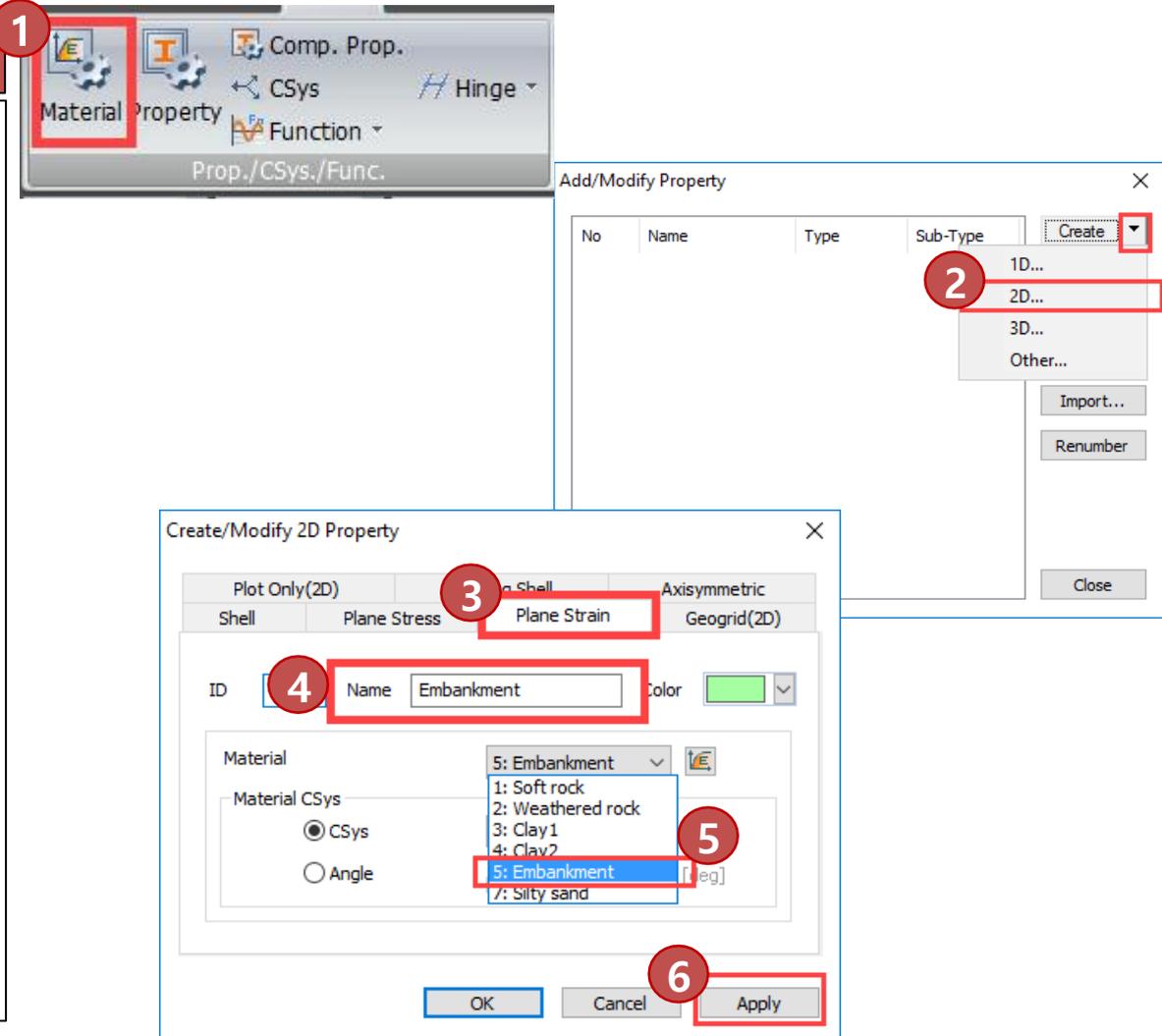
Step 03 Mesh > Property Definition

Procedure

1. Mesh > Prop./Csys./Func.> Property
2. Click on Create and select 2D.
3. Choose the property type as Plane Strain.
4. Give the name as 'Embankment'
5. Select Embankment material in the material dropdown menu.
6. Click 'Apply'.
7. Now that the Embankment Plane Strain property is defined.

Similarly define the property for Silty Sand, Weathered Rock, Soft Rock, Clay 1, and Clay 2 materials.

8. After defining all the properties, click 'Cancel'

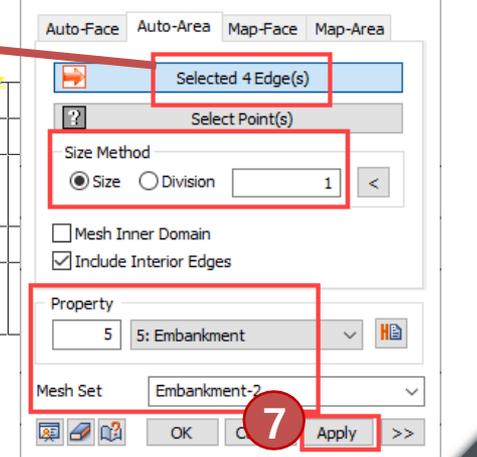
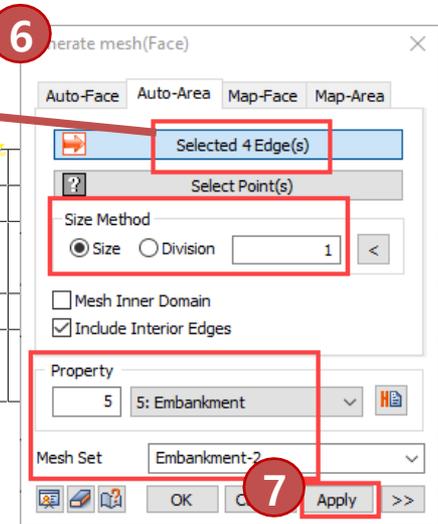
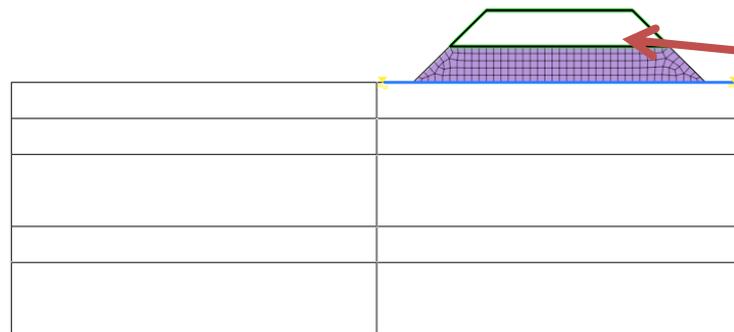
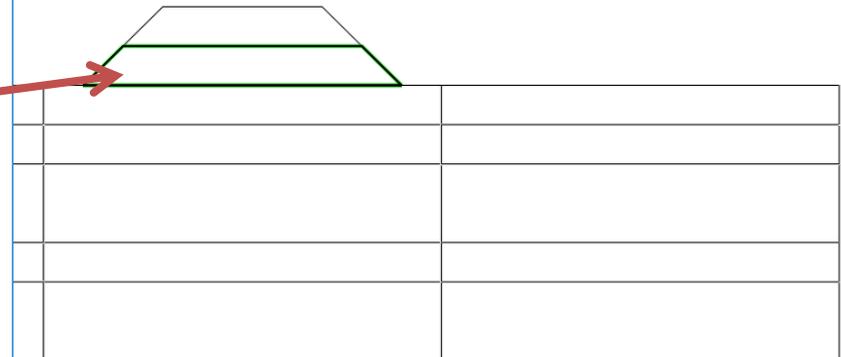
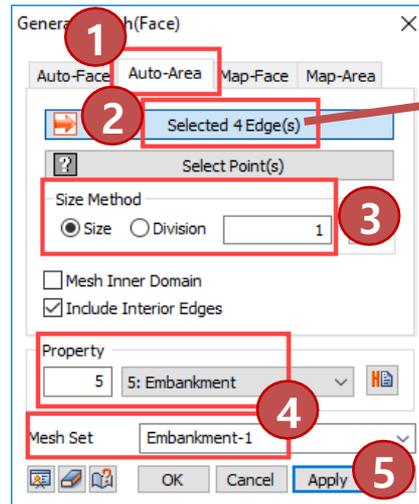
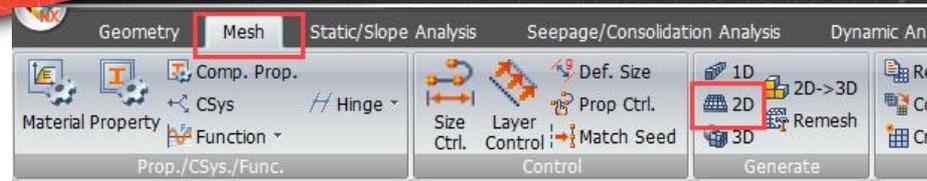


Step 04-1 Mesh > 2D Mesh

Procedure

The objective is to generate a Fine mesh for the middle zone and Moderate mesh for the left and right zones.

1. Main Menu > Mesh > Generate > 2D > Auto Area
2. Select Object Edges > Select the edges as shown.
3. Mesh Size > Element Size '1'm,
4. Property > Select 'Embankment' Name > 'Embankment-1'.
5. Click 'Apply'
6. Similarly select the top portion of embankment and generate a mesh with element size of 1m and name it as 'Embankment-2'
7. Click 'Apply'



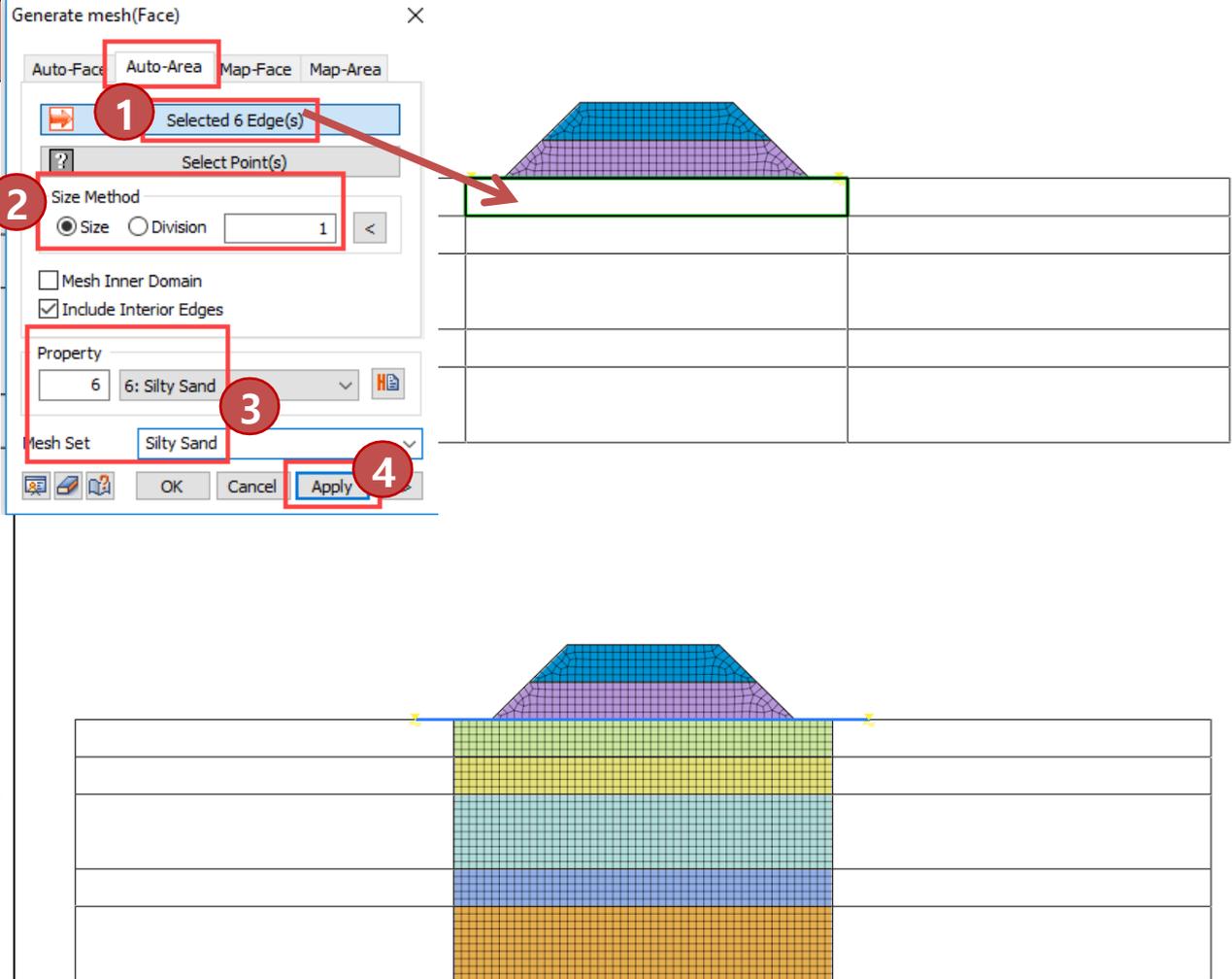
Step 04-2 Mesh > 2D Mesh

Procedure

The objective is to generate a Fine mesh for the middle zone and Moderate mesh for the left and right zones.

1. Select Object Edges > Select the edges as shown.
2. Mesh Size > Element Size '1'm,
3. Property > Select 'Silty Sand' Name > Silty Sand'.
4. Click 'Apply'

Similarly mesh the Clay-1, Clay-2, Weathered Rock and Soft Rock Middle zones using 1m mesh size and corresponding properties.



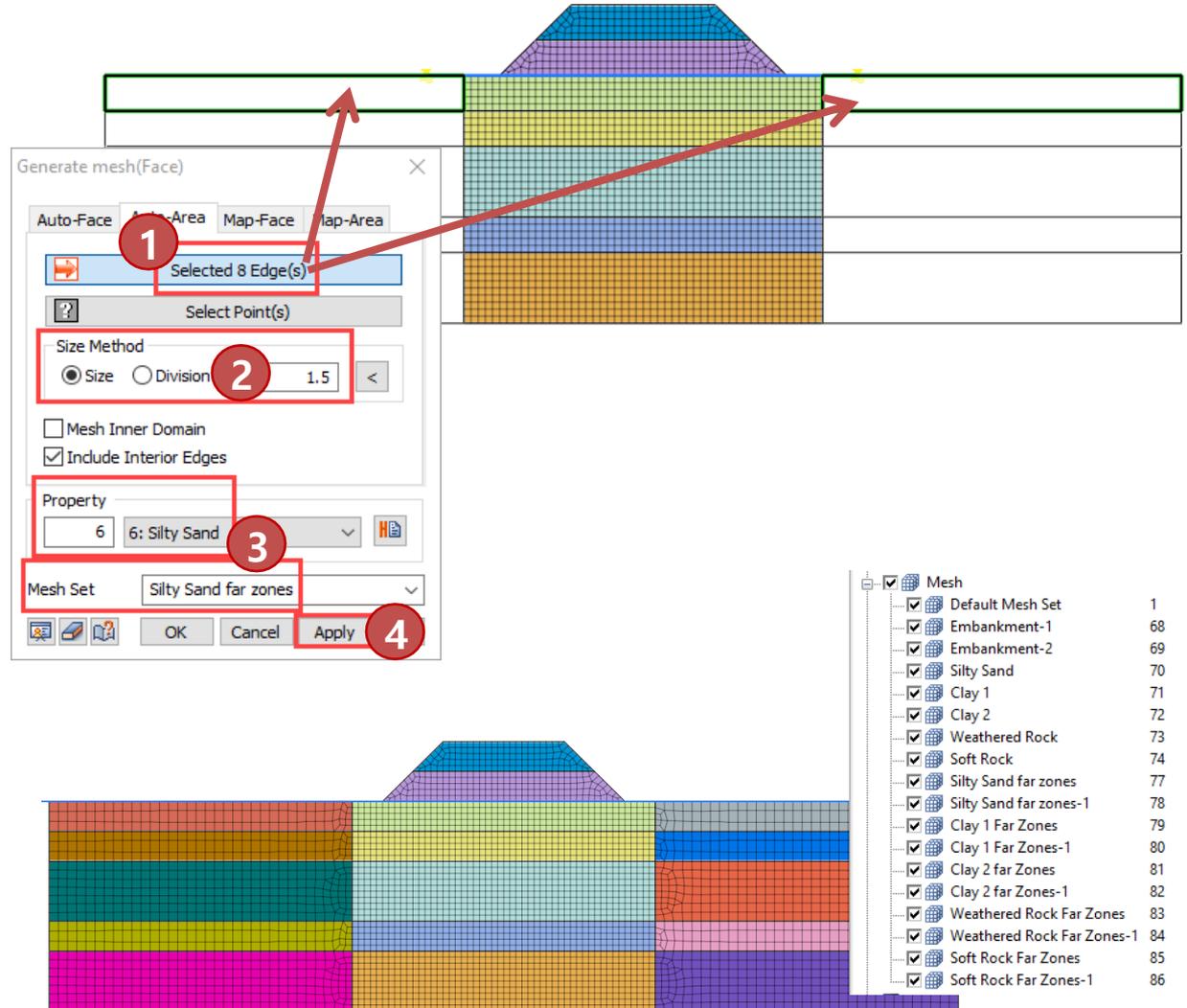
Step 04-3 Mesh > 2D Mesh

Procedure

The objective is to generate a Fine mesh for the middle zone and Moderate mesh for the left and right zones.

1. Select Object Edges > Select the edges as shown.
2. Mesh Size > Element Size '1.5'm,
3. Property > Select 'Silty Sand' Name > Silty Sand Far Zones'.
4. Click 'Apply'

Similarly mesh the Clay-1, Clay-2, Weathered Rock and Soft Rock Far zones using 1.5m mesh size and corresponding properties.



The image shows a 2D cross-section of a dam structure with a mesh applied to various zones. The mesh is color-coded by material type: blue for embankment, green for silty sand, yellow for clay, purple for weathered rock, and orange for soft rock. The mesh is finer in the middle zone and coarser in the left and right zones.

The 'Generate mesh(Face)' dialog box is shown with the following settings:

- Auto-Face: Area, Map-Face
- Selected 8 Edge(s)
- Select Point(s):
- Size Method: Size, Division, 1.5
- Mesh Inner Domain
- Include Interior Edges
- Property: 6, 6: Silty Sand
- Mesh Set: Silty Sand far zones
- Buttons: OK, Cancel, Apply

The mesh set list on the right side of the dialog box is as follows:

Mesh Set	Order
Mesh	1
Default Mesh Set	1
Embankment-1	68
Embankment-2	69
Silty Sand	70
Clay 1	71
Clay 2	72
Weathered Rock	73
Soft Rock	74
Silty Sand far zones	77
Silty Sand far zones-1	78
Clay 1 Far Zones	79
Clay 1 Far Zones-1	80
Clay 2 far Zones	81
Clay 2 far Zones-1	82
Weathered Rock Far Zones	83
Weathered Rock Far Zones-1	84
Soft Rock Far Zones	85
Soft Rock Far Zones-1	86

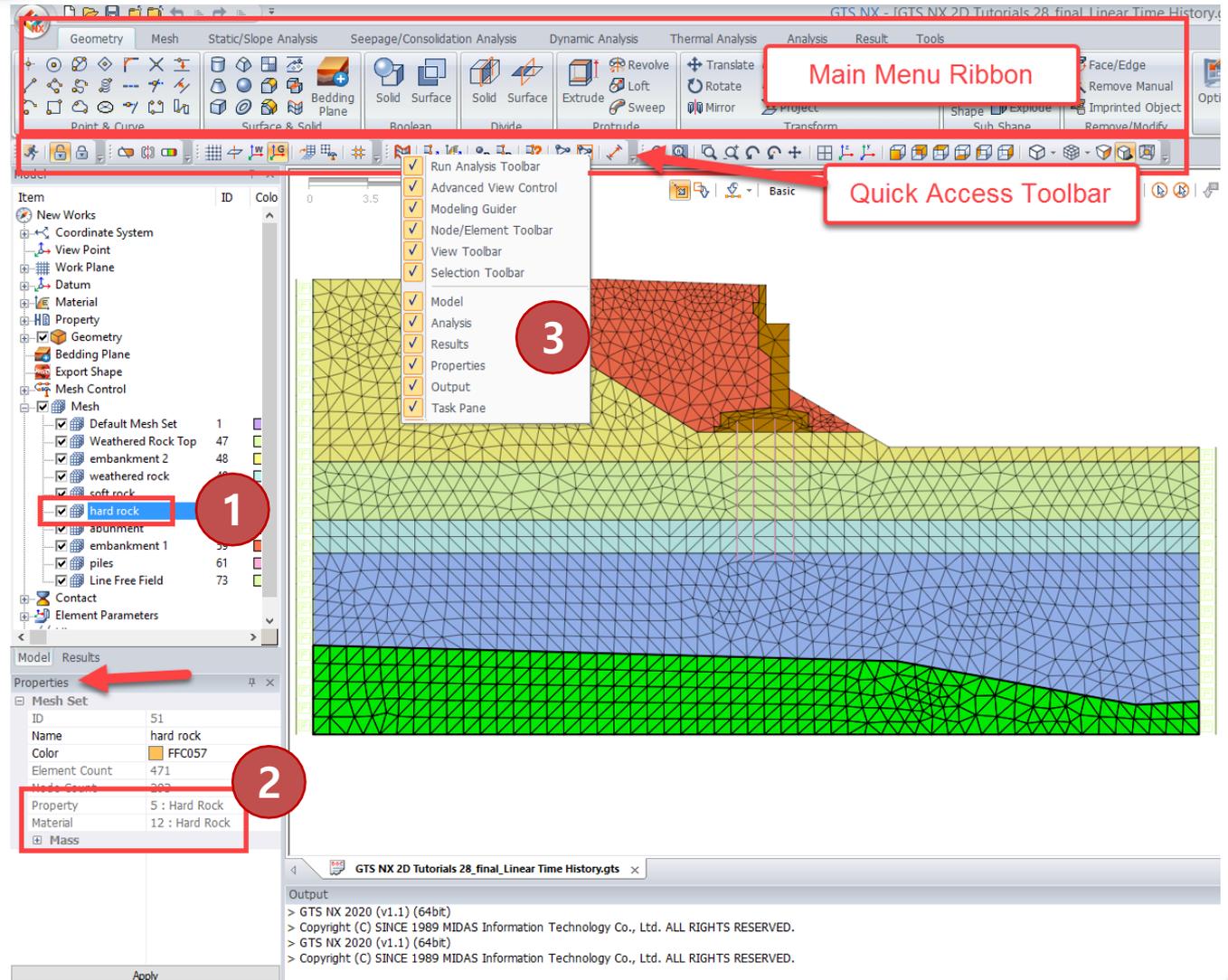
Additional Information

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.



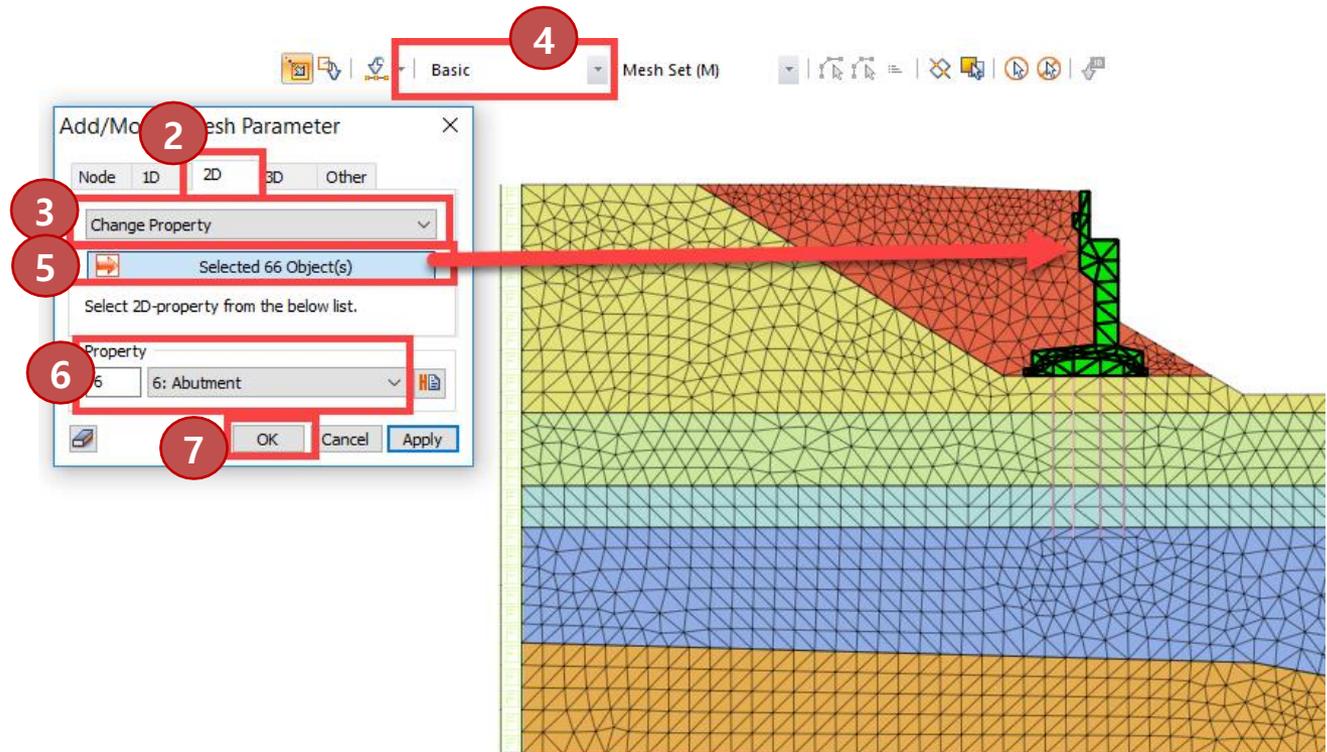
Additional Information

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'

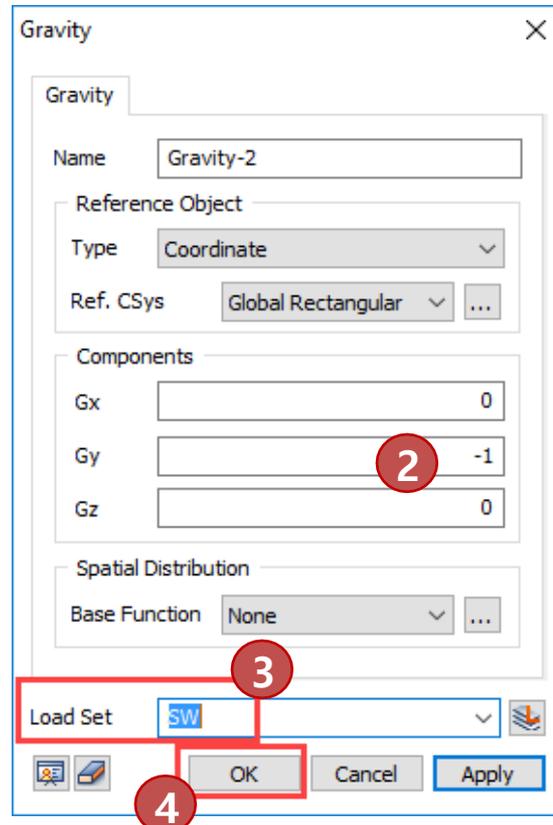
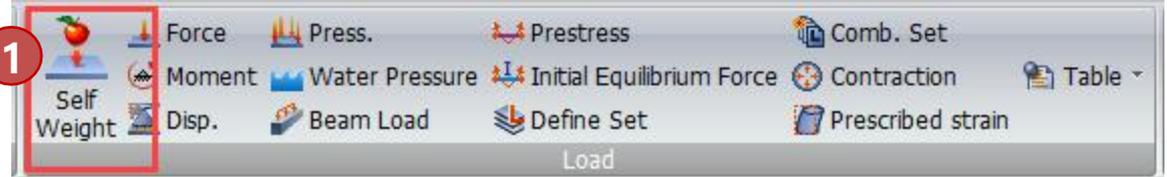


Step 05

Seepage/Consolidation Analysis > Load > Self Weight

Procedure

1. Main menu > Seepage/Consolidation Analysis > Load > Self Weight
2. Make sure that '-1' is inputted in the Gy Component.
3. Enter the Load set name as 'SW'.
4. Click 'OK'

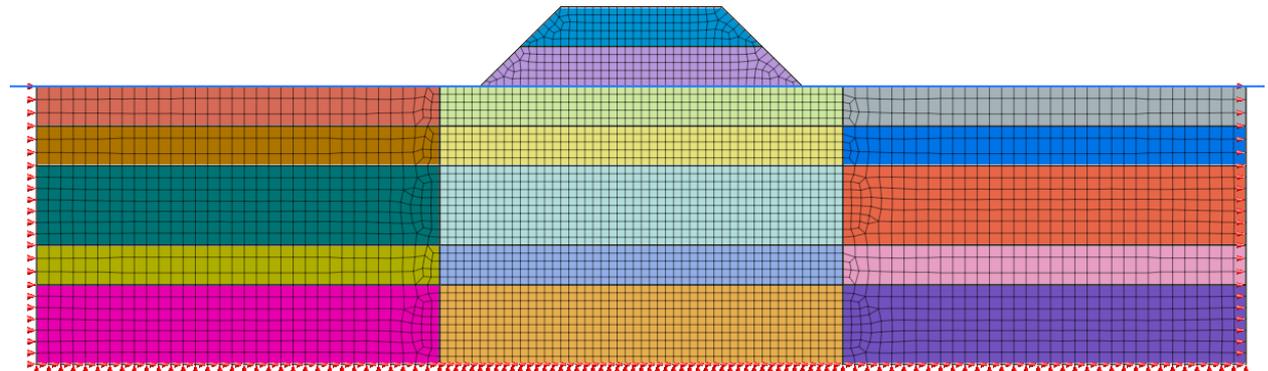
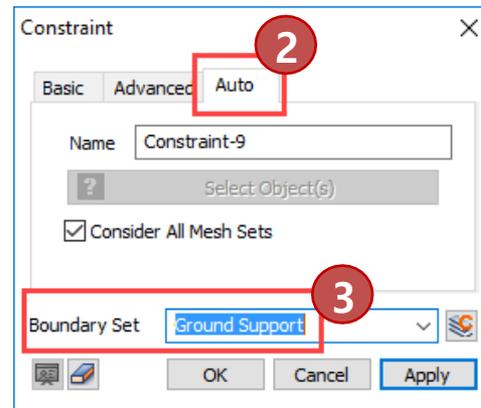


Step 06 Seepage/Consolidation Analysis > Boundary > Constraint

Procedure

1. Main menu > Seepage/Consolidation Analysis > Boundary > Constraint
2. In the Constraint tab, go to 'Auto'
3. Enter the boundary set name as 'Ground Support' and Click 'OK'

Boundary Supports will be automatically assigned to the model

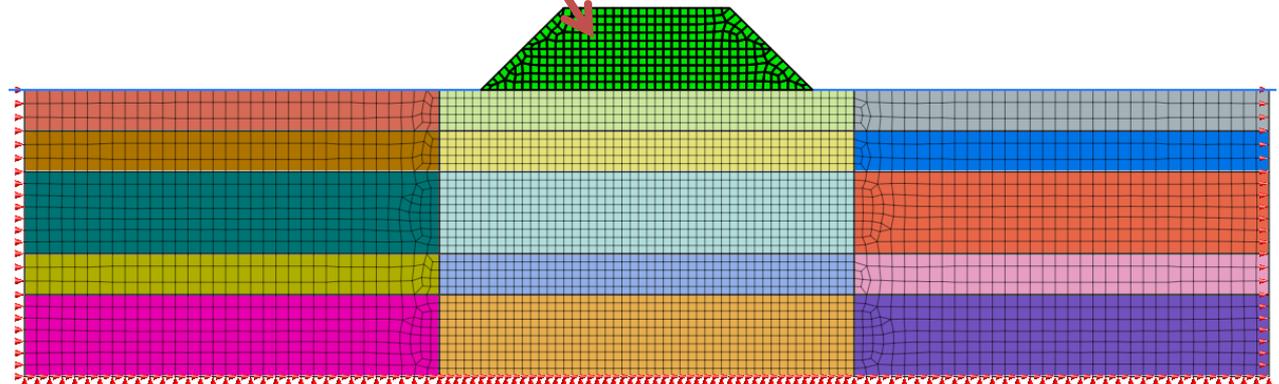
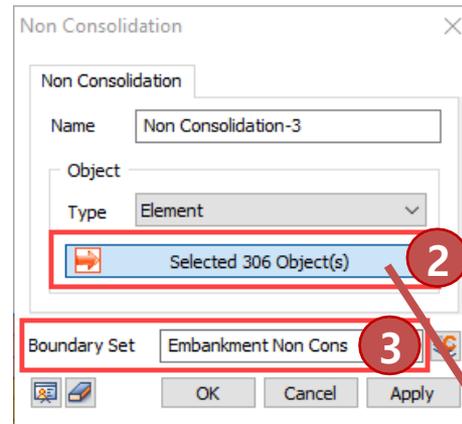


Step 07

Seepage/Consolidation Analysis > Boundary > Non-Consolidation

Procedure

1. Main menu > Seepage/Consolidation Analysis > Boundary > Non Consolidation
2. Select all the elements of the Embankment
3. Enter the boundary set name as 'Embankment Non Cons' and Click 'OK'

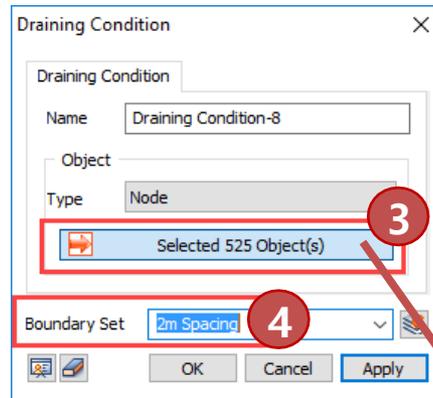
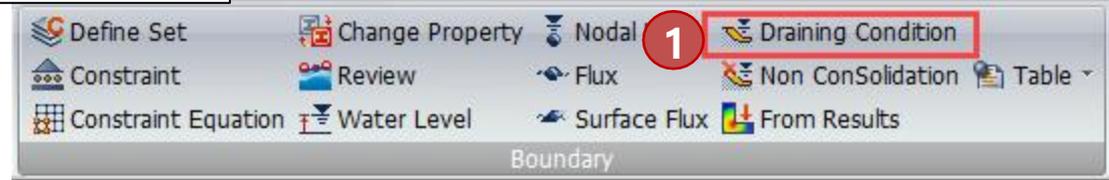


Step 08

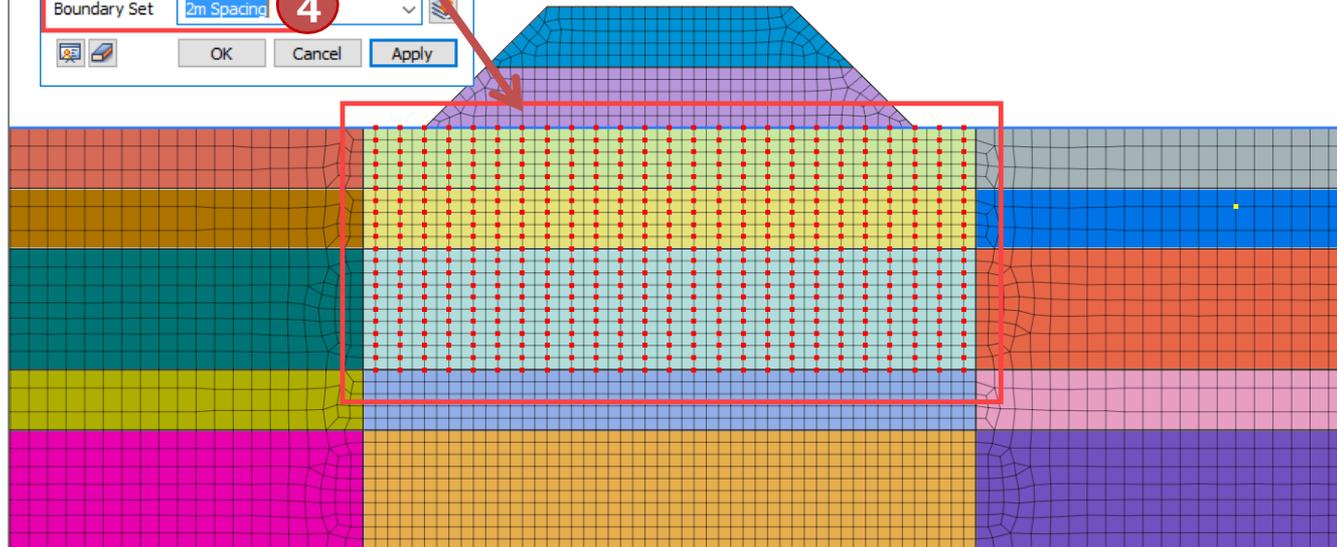
Seepage/Consolidation Analysis > Boundary > Draining Condition

Procedure

1. Main menu > Seepage/Consolidation Analysis > Boundary > Draining Condition
2. The drains should be applied in the middle zone and should reach a depth of up to Clay-2 Layer.
3. Hence select the nodes with a spacing of 2m horizontally as shown in the picture.
4. Name the boundary set as '2m Spacing'
5. Click 'OK'



Horizontal Spacing between the selected nodes is 2 m and the depth of node selection is up to Clay-2 layer.

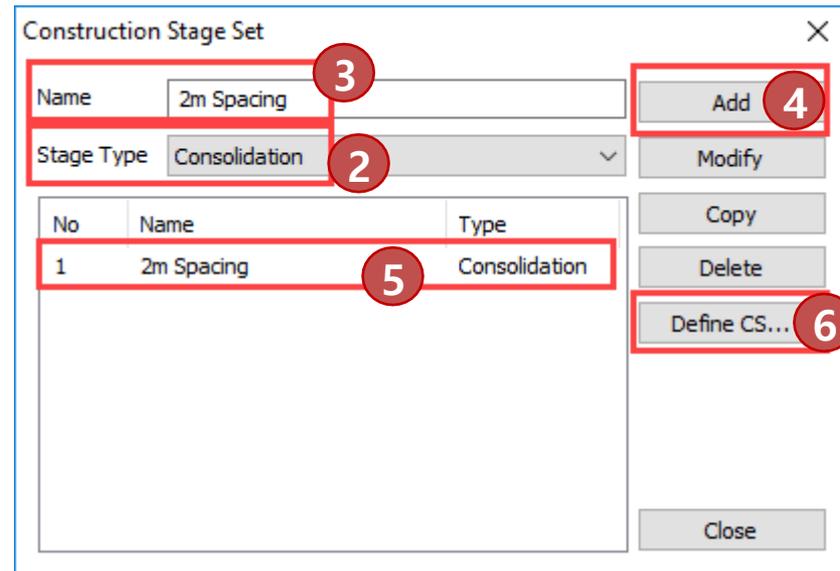


Step 09-1

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

1. Main menu > Seepage/Consolidation Analysis > Construction Stage > Stage Set
2. Select the Stage type 'Consolidation'
3. Enter the Name '2m Spacing'
4. Click 'Add'
5. Using the cursor click on the generated Construction Stage Set.
6. Click 'Define CS'



Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

1. Enter the Stage Name as 'Initial'
2. Activate all the mesh sets except 'Embankment 1&2'.
3. Activate 'Ground Support' and also activate load set 'SW'
4. We need to activate water level at origin level. Hence enter 0 m as the ground surface in current model is at Origin level.
5. Enable 'Clear Displacement'
6. Click on 'Time Step'
7. Enter duration of 1 day.
8. Enter Step number as 1.
9. Enable 'Save Result'
10. Click on 'Generate Step'
11. Click 'OK'
12. Click 'Save'
13. Click 'New'

Define Construction Stage

Construction Stage Set Name: 2m Spacing

Stage ID: 1: Initial

Stage Name: Initial

Stage Type: Consolidation

Buttons: Move, New, Insert, Delete, Move to Next, Time Step...

Set Data, Activated Data, Deactivated Data

Initial Condition: Define Water Level For Global (0 m), Clear Displacement

Time Step: Duration: 1 day, Step Number: 1, Save Result, Generate Step, OK

Bottom Buttons: Save, Close

Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

Steps Contd..

14. Enter the Stage Name as 'Vertical Drains'
15. Activate the '2m Spacing' boundary condition
16. Click 'Time Step'
17. Enter duration as 10 days (10 days of vertical Drain Construction)
18. Enter Step Number as 5
19. Enable 'Save result'
20. Click 'Generate Step'
21. Click 'OK'
22. Click 'Save'
23. Click 'New'

Define Construction Stage

Construction Stage Set Name: 2m Spacing

Stage ID: 14 Vertical Drains

Stage Name: Vertical Drains

Stage Type: Consolidation

Set Data: Mesh, Clay 1 Far Zones, Clay 1 Far Zones-1, Clay 2, Clay 2 far Zones, Clay 2 far Zones-1, Default Mesh Set, Embankment-1, Embankment-2, Silty Sand, Silty Sand far zones, Silty Sand far zones-1, Soft Rock, Soft Rock Far Zones, Soft Rock Far Zone-1, Weathered Rock, Weathered Rock-Far Zo, Weathered Rock Far Zo

Activated Data: Mesh, Boundary Condition, 2m Spacing, Static Load, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Contact

Time Step

Duration: 10 day

Step Number: 5

Save Result:

Generate Step

Time	Load Factor	Save Step
2.0000	0.2000	<input checked="" type="checkbox"/>
4.0000	0.2000	<input checked="" type="checkbox"/>
6.0000	0.2000	<input checked="" type="checkbox"/>
8.0000	0.2000	<input checked="" type="checkbox"/>
10.0000	0.2000	<input checked="" type="checkbox"/>
*		<input type="checkbox"/>

Auto Time Step

Initial Time Step: Auto 0 day

Max. Pore Pressure Changes per Step: 1 kN/m²

Ratio of Max Time Step to Initial: 5

Save Step: Last Increment

OK Close

Save Close

Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

Steps Contd..

24. Enter the Stage Name as 'Embankment-1'
25. Activate 'Embankment-1' mesh set and also activate 'Embankment Non Cons' boundary set.
26. Click 'Time Step'
27. Enter Duration as 5 days.
28. Enter Step Number as 1.
29. Enable 'Save Result'
30. Click 'Generate Step'
31. Click 'OK'
32. Click 'Save'
33. Click 'New'

Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

Steps Contd..

- 34. Enter the Stage Name as 'Leave Period-1'
- 35. Click 'Time Step'
- 36. Enter Duration as 30 days.
- 37. Enter Step Number as 10.
- 38. Enable 'Save Result'
- 39. Click 'Generate Step'
- 40. Click 'OK'
- 41. Click 'Save'

Define Construction Stage

Construction Stage Set Name: 2m Spacing

Stage ID: 4: Leave Period-1

Stage Name: Leave Period-1

Stage Type: Consolidation

Time Step...

Time Step

Duration: 30 day

Step Number: 10

Save Result

Generate Step

Time	Load Factor	Save Step
3.0000	0.1000	<input checked="" type="checkbox"/>
6.0000	0.1000	<input checked="" type="checkbox"/>
9.0000	0.1000	<input checked="" type="checkbox"/>
12.0000	0.1000	<input checked="" type="checkbox"/>
15.0000	0.1000	<input checked="" type="checkbox"/>
18.0000	0.1000	<input checked="" type="checkbox"/>
21.0000	0.1000	<input checked="" type="checkbox"/>
24.0000	0.1000	<input checked="" type="checkbox"/>

Save

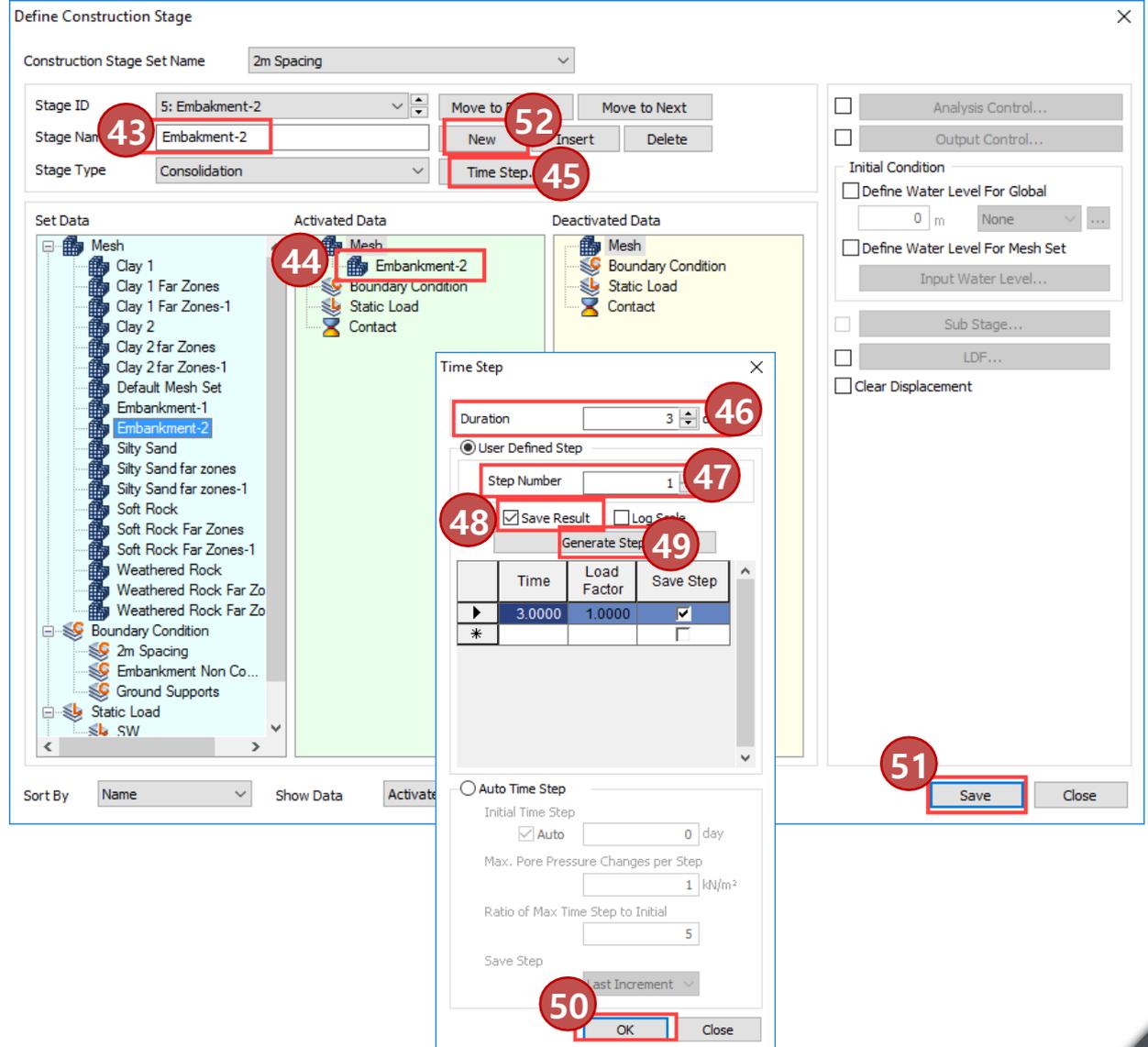
Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

Steps Contd..

43. Enter stage name as 'Embankment 2'
44. Activate 'Embankment 2' mesh set.
45. Click 'Time Step'
46. Enter Duration as 3 days.
47. Enter step number as 1.
48. Enable 'Save Result'
49. Click 'Generate Step'
50. Click 'OK'
51. Click 'Save'
52. Click 'New'



Define Construction Stage

Construction Stage Set Name: 2m Spacing

Stage ID: 5: Embankment-2

Stage Name: Embankment-2

Stage Type: Consolidation

Set Data:

- Mesh
 - Clay 1
 - Clay 1 Far Zones
 - Clay 1 Far Zones-1
 - Clay 2
 - Clay 2 far Zones
 - Clay 2 far Zones-1
 - Default Mesh Set
 - Embankment-1
 - Embankment-2
 - Silty Sand
 - Silty Sand far zones
 - Silty Sand far zones-1
 - Soft Rock
 - Soft Rock Far Zones
 - Soft Rock Far Zones-1
 - Weathered Rock
 - Weathered Rock Far Zo
 - Weathered Rock Far Zo
- Boundary Condition
 - 2m Spacing
 - Embankment Non Co...
 - Ground Supports
 - Static Load
 - SW

Activated Data:

- Mesh
 - Embankment-2
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Time Step

Duration: 3

User Defined Step

Step Number: 1

Save Result Log Scale

Generate Step

Time	Load Factor	Save Step
3.0000	1.0000	<input checked="" type="checkbox"/>
*		<input type="checkbox"/>

Auto Time Step

Initial Time Step: Auto 0 day

Max. Pore Pressure Changes per Step: 1 kN/m²

Ratio of Max Time Step to Initial: 5

Save Step: Last Increment

Save

Close

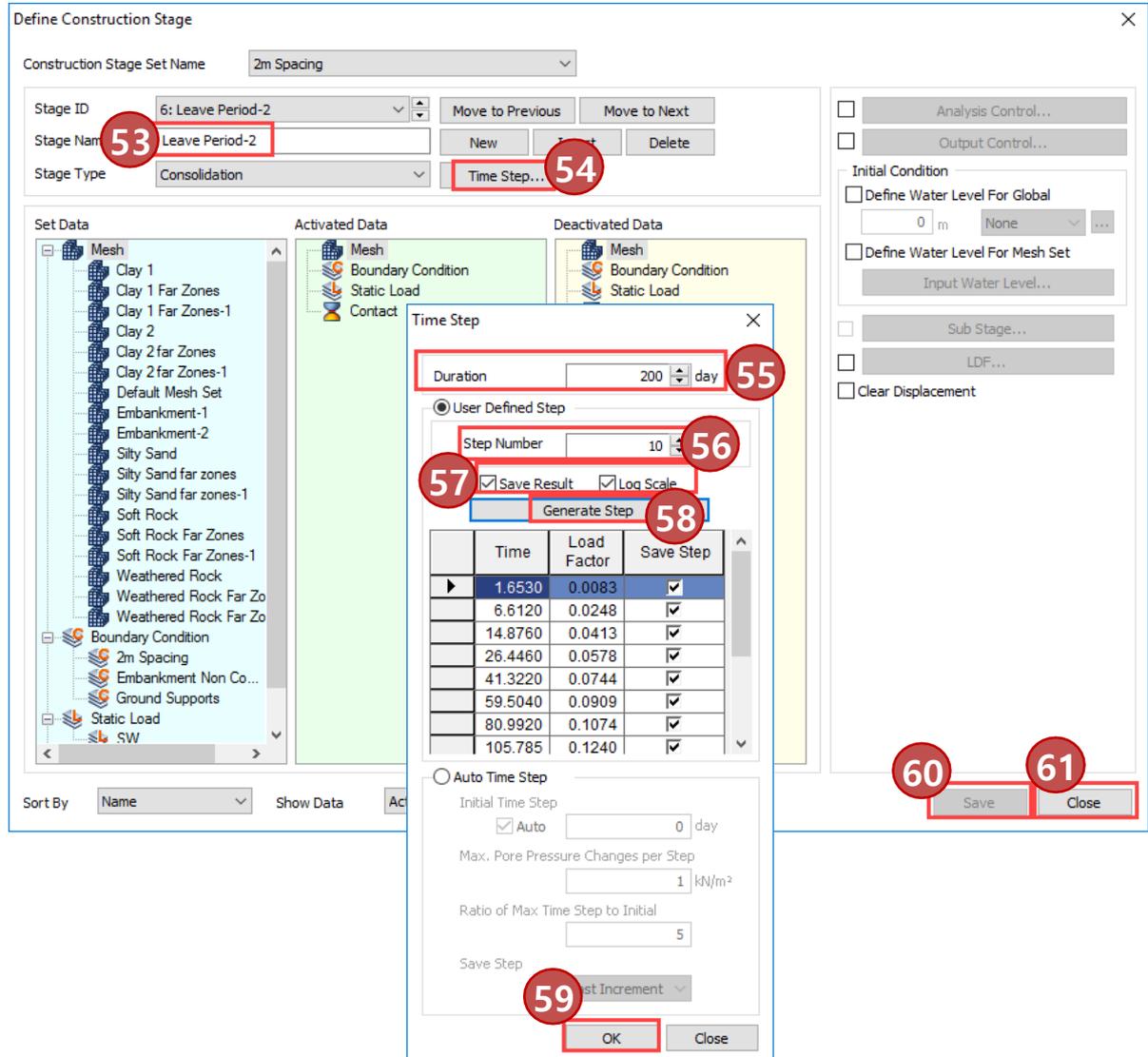
Step 09-2

Seepage/Consolidation Analysis > Construction Stage > Stage Set

Procedure

Steps Contd..

- 53. Enter stage name as 'Leave Period-2'
- 54. Click 'Time Step'
- 55. Enter Duration as 200 days.
- 56. Enter Step Number as 10
- 57. Enable 'Save Result' and 'Log Scale'
- 58. Click 'Generate Step'
- 59. Click 'OK'
- 60. Click 'Save'
- 61. Click 'Close'



Define Construction Stage

Construction Stage Set Name: 2m Spacing

Stage ID: 6: Leave Period-2

Stage Name: 53 Leave Period-2

Stage Type: Consolidation

Time Step... 54

Set Data: Mesh, Clay 1, Clay 1 Far Zones, Clay 1 Far Zones-1, Clay 2, Clay 2 far Zones, Clay 2 far Zones-1, Default Mesh Set, Embankment-1, Embankment-2, Silty Sand, Silty Sand far zones, Silty Sand far zones-1, Soft Rock, Soft Rock Far Zones, Soft Rock Far Zones-1, Weathered Rock, Weathered Rock Far Zo, Weathered Rock Far Zo

Activated Data: Mesh, Boundary Condition, Static Load, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load

Time Step

Duration: 200 day 55

Step Number: 10 56

Save Result 57 Log Scale 57

Generate Step 58

Time	Load Factor	Save Step
1.6530	0.0083	<input checked="" type="checkbox"/>
6.6120	0.0248	<input checked="" type="checkbox"/>
14.8760	0.0413	<input checked="" type="checkbox"/>
26.4460	0.0578	<input checked="" type="checkbox"/>
41.3220	0.0744	<input checked="" type="checkbox"/>
59.5040	0.0909	<input checked="" type="checkbox"/>
80.9920	0.1074	<input checked="" type="checkbox"/>
105.785	0.1240	<input checked="" type="checkbox"/>

Auto Time Step

Initial Time Step: Auto 0 day

Max. Pore Pressure Changes per Step: 1 kN/m²

Ratio of Max Time Step to Initial: 5

Save Step: First Increment

OK 59 Close

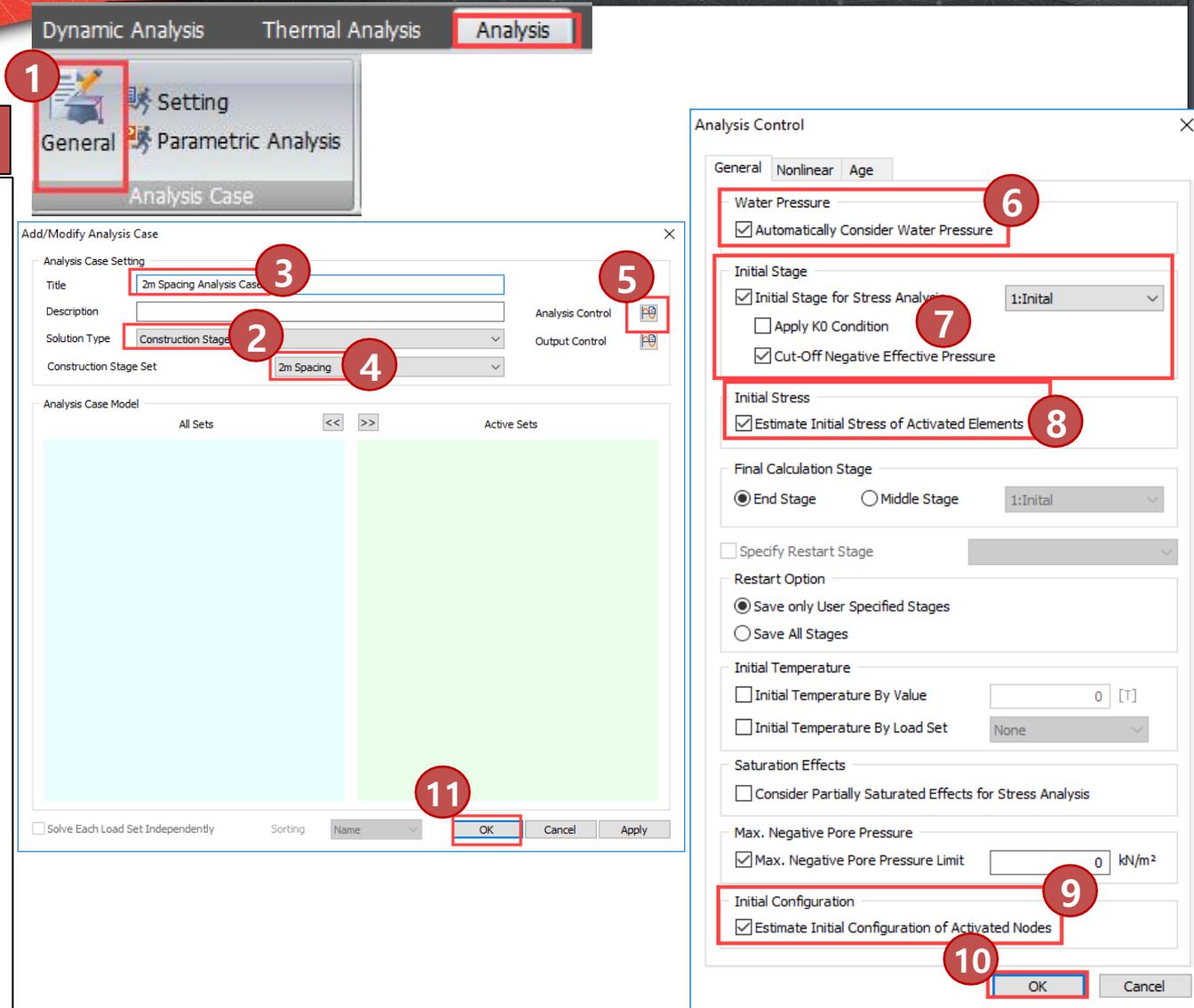
Analysis Control...
Output Control...
Initial Condition:
 Define Water Level For Global
 Define Water Level For Mesh Set
Input Water Level...
Sub Stage...
LDF...
Clear Displacement

60 Save 61 Close

Step 10 Analysis > General

Procedure

1. Main Menu > Analysis > General
2. Select solution type as 'Construction Stage'.
3. Enter the title as '2m Spacing Analysis Case'
4. Select '2m spacing' construction stage set.
5. Open Analysis Control.
6. Enable 'Automatic Consider water Pressure'
7. Enable 'Initial Stage' and 'Cut-Off Negative Effective Pressure'
8. Enable 'Estimate Initial Stress of Activated Elements'
9. Enable 'Estimate Initial Configuration of Activated Nodes'
10. Click 'OK'
11. Click 'OK'



The screenshot displays the GTS NX software interface. At the top, the 'Analysis' menu is highlighted. Below it, the 'Analysis Case' dialog box is open, showing the 'Add/Modify Analysis Case' window. The 'Title' field contains '2m Spacing Analysis Case', the 'Solution Type' is set to 'Construction Stage', and the 'Construction Stage Set' is '2m Spacing'. The 'Analysis Control' dialog box is also open, showing the 'General' tab. The 'Water Pressure' section has 'Automatically Consider Water Pressure' checked. The 'Initial Stage' section has 'Initial Stage for Stress Analysis' checked, 'Apply K0 Condition' unchecked, and 'Cut-Off Negative Effective Pressure' checked. The 'Initial Stress' section has 'Estimate Initial Stress of Activated Elements' checked. The 'Final Calculation Stage' is set to 'End Stage'. The 'Restart Option' is 'Save only User Specified Stages'. The 'Initial Temperature' section has 'Initial Temperature By Value' and 'Initial Temperature By Load Set' unchecked. The 'Saturation Effects' section has 'Consider Partially Saturated Effects for Stress Analysis' unchecked. The 'Max. Negative Pore Pressure' section has 'Max. Negative Pore Pressure Limit' checked and set to 0 kN/m². The 'Initial Configuration' section has 'Estimate Initial Configuration of Activated Nodes' checked. The 'OK' button is highlighted in both dialog boxes.

Dynamic Analysis Thermal Analysis **Analysis**

1 **Setting**
General Parametric Analysis
Analysis Case

Add/Modify Analysis Case

Analysis Case Setting

Title: 2m Spacing Analysis Case
Description:
Solution Type: Construction Stage
Construction Stage Set: 2m Spacing

Analysis Control
Output Control

Analysis Case Model

All Sets << >> Active Sets

Solve Each Load Set Independently Sorting: Name **OK** Cancel Apply

Analysis Control

General Nonlinear Age

Water Pressure
 Automatically Consider Water Pressure

Initial Stage
 Initial Stage for Stress Analysis
 Apply K0 Condition
 Cut-Off Negative Effective Pressure

Initial Stress
 Estimate Initial Stress of Activated Elements

Final Calculation Stage
 End Stage Middle Stage

Specify Restart Stage

Restart Option
 Save only User Specified Stages
 Save All Stages

Initial Temperature
 Initial Temperature By Value
 Initial Temperature By Load Set

Saturation Effects
 Consider Partially Saturated Effects for Stress Analysis

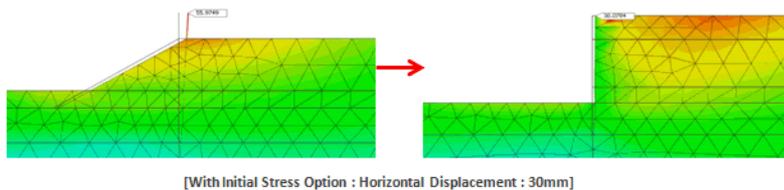
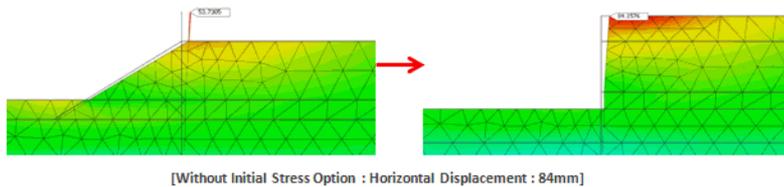
Max. Negative Pore Pressure
 Max. Negative Pore Pressure Limit

Initial Configuration
 Estimate Initial Configuration of Activated Nodes

OK Cancel

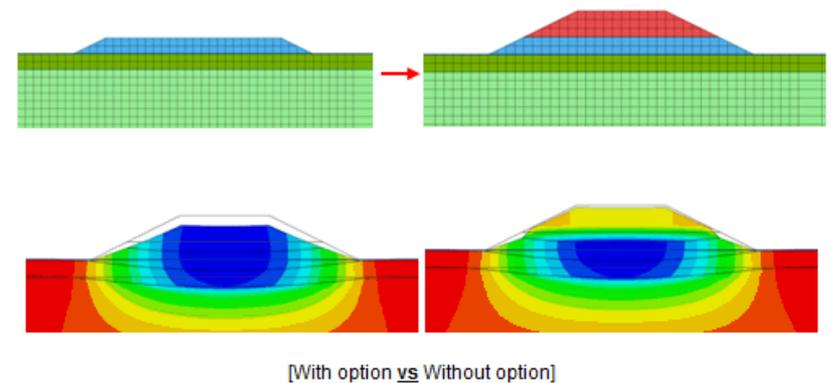
Estimate Initial Stress of Activated Elements

In order to calculate the initial stress of ground, GTS NX perform Linear Analysis even if nonlinear material is assigned to the elements. In this case, it can result in, sometimes, over-estimating the soil behavior (large displacement). Initial Stress Options can eliminate this problem especially for newly activated elements which are to simulate a fill-up ground such as backfill and embankment.



Estimate Initial Configuration of Activated Nodes

During construction, the newly activated nodes(elements) can be set to the position considering deformed shape in the previous stage. Following is the example of staged embankment to compare the settlement distribution between with and without applying the option.



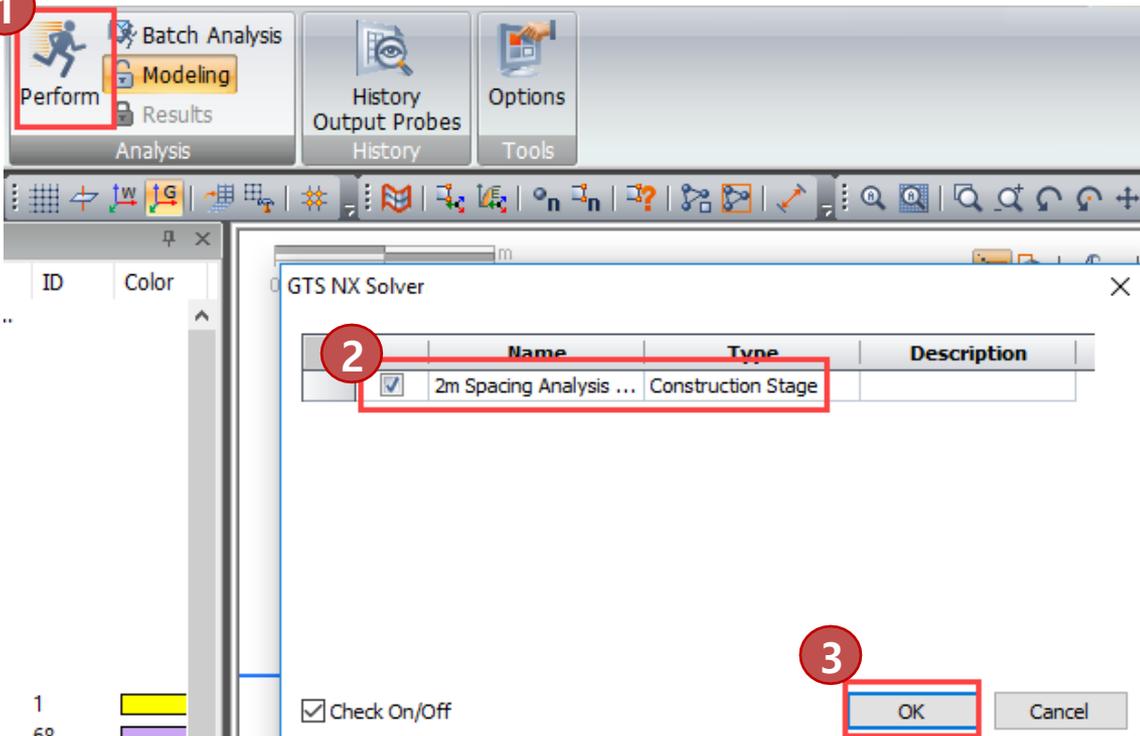
Step 11 Analysis > Perform

Procedure

1. Main Menu > Analysis > Perform...
2. Check –on '2m Spacing Analysis Case'.
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).



The screenshot shows the GTS NX software interface. The 'Perform' menu item is highlighted with a red box and a circled '1'. The 'GTS NX Solver' dialog box is open, showing a table with the following data:

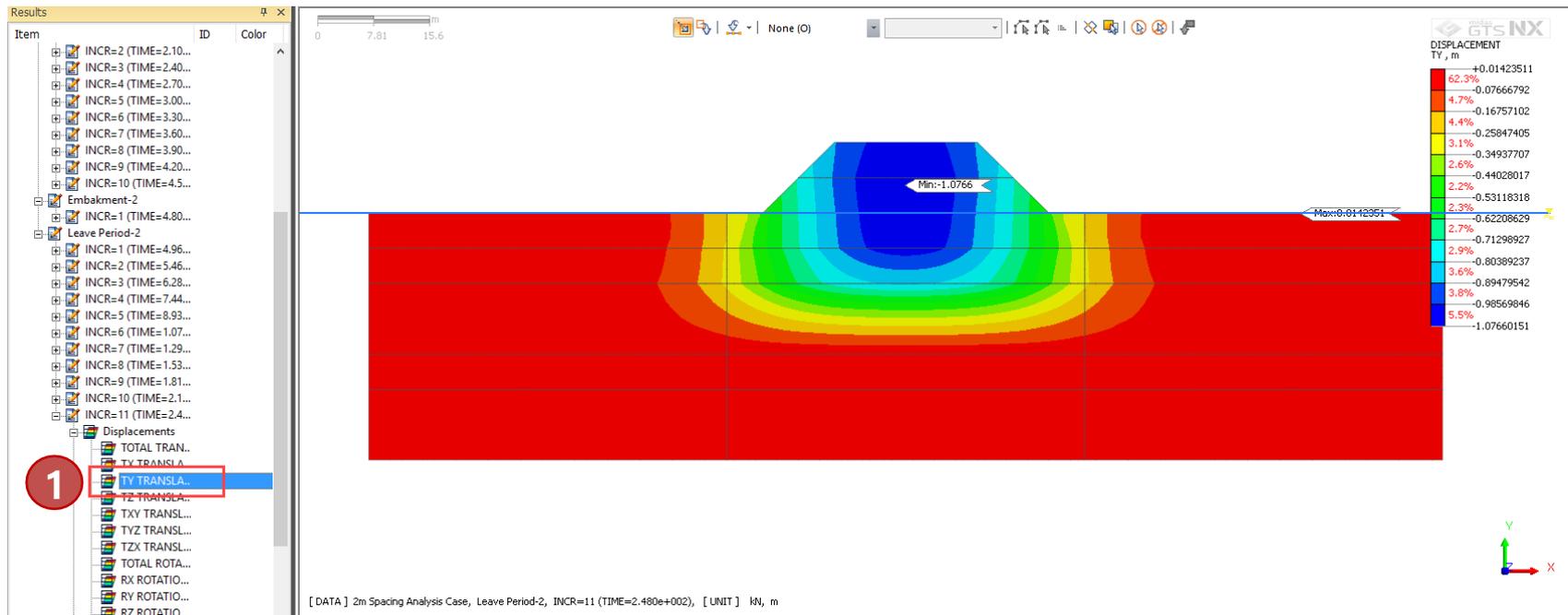
	Name	Type	Description
<input checked="" type="checkbox"/>	2m Spacing Analysis ...	Construction Stage	

The 'Check On/Off' checkbox is checked. The 'OK' button is highlighted with a red box and a circled '3'. A circled '2' is also present near the table.

Step 12-1 Settlement

Procedure

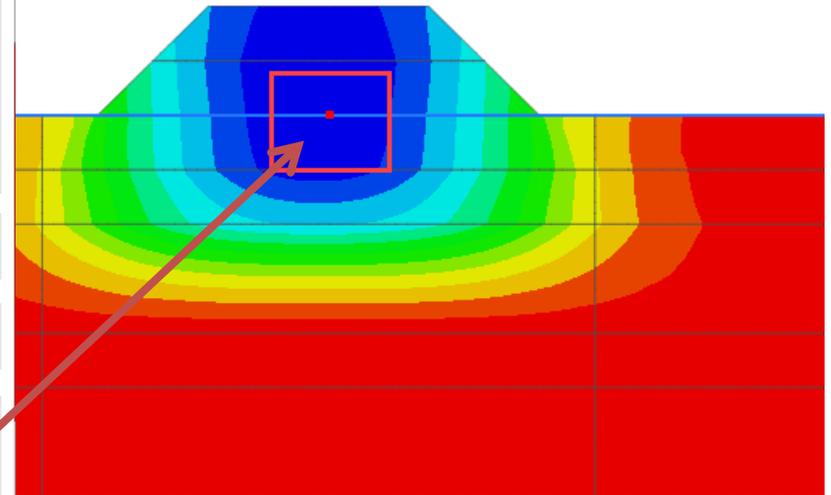
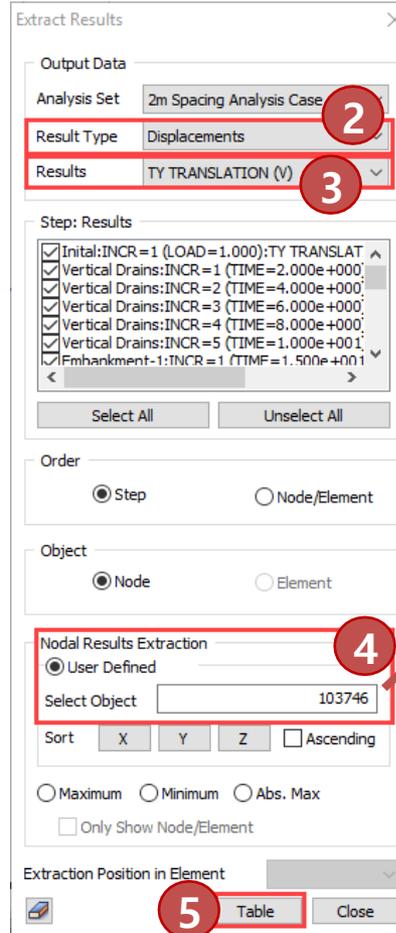
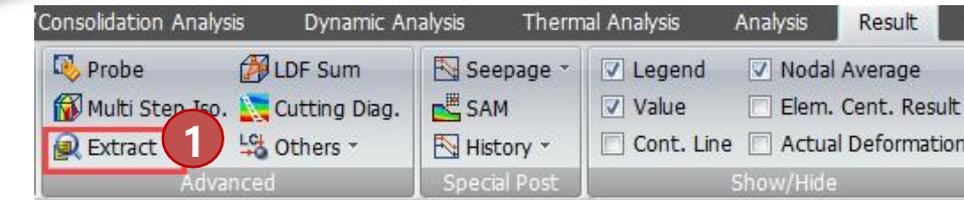
1. Post Works Tree > Leave Period-2 (last Step)> Displacements > Ty Translation



Step 12-2 Settlement vs Time Graph

Procedure

1. Results > Advanced Extract
2. Select the result type as 'Displacement'
3. Select Ty translation
4. Select the node using the cursor at the ground surface (below the embankment)
5. Click Table



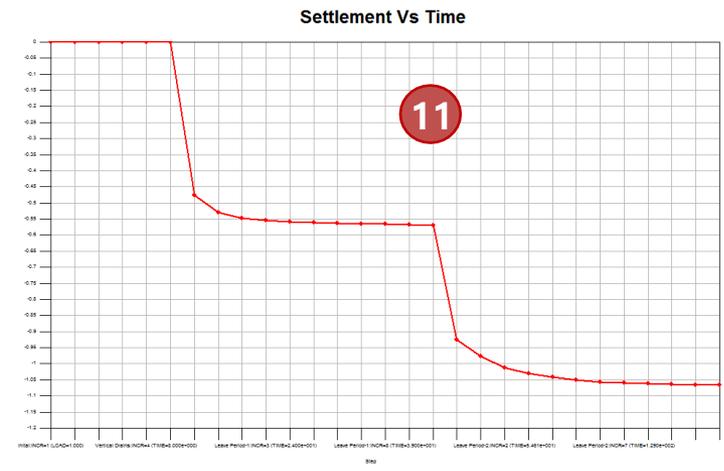
Step 12-2 Settlement vs Time Graph

Procedure

Steps Contd..

6. Select the 'Step' and 'Ty translation' columns from the table.
7. Right click using the cursor and select the 'Show graph'.
8. Enter the 'Settlement' in the Y-label.
9. Enter the graph title as 'Settlement vs time'
10. Click 'OK'
11. A graph can be seen on your screen.

No	Step	Step Value	TY TRANSLATION (V) (m)
1	Intal.INCR=1 (LOAD=1.000)	1.000000e+000	0.000000e+000
2	Vertical Drains.INCR=1 (TIME=2.000)	1.728000e+005	-2.199384e-008
3	Vertical Drains.INCR=2 (TIME=4.000)	3.456000e+005	-6.885588e-008
4	Vertical Drains.INCR=3 (TIME=6.000)	5.184000e+005	-1.049244e-007
5	Vertical Drains.INCR=4 (TIME=8.000)	6.912000e+005	-1.438548e-007
6	Vertical Drains.INCR=5 (TIME=1.000)	8.640000e+005	-1.782884e-007
7	Embankment-1.INCR=1 (TIME=1.500)	1.296000e+006	-4.752946e-001
8	Leave Period-1.INCR=1 (TIME=1.80)	1.555200e+008	-5.300366e-001
9	Leave Period-1.INCR=2 (TIME=2.10)	1.814400e+006	-5.468706e-001
10	Leave Period-1.INCR=3 (TIME=2.40)	2.073600e+006	-5.539953e-001
11	Leave Period-1.INCR=4 (TIME=2.70)	2.332800e+006	-5.580762e-001
12	Leave Period-1.INCR=5 (TIME=3.00)	2.592000e+008	-5.60944
13	Leave Period-1.INCR=6 (TIME=3.30)	2.851200e+006	-5.63161
14	Leave Period-1.INCR=7 (TIME=3.60)	3.110400e+006	-5.64961
15	Leave Period-1.INCR=8 (TIME=3.90)	3.369600e+006	-5.66461
16	Leave Period-1.INCR=9 (TIME=4.20)	3.628800e+006	-5.67733
17	Leave Period-1.INCR=10 (TIME=4.5)	3.888000e+006	-5.68824
18	Embankment-2.INCR=1 (TIME=4.800e)	4.147200e+006	-9.24962
19	Leave Period-2.INCR=1 (TIME=4.96)	4.290020e+006	-9.77446
20	Leave Period-2.INCR=2 (TIME=5.12)	4.432040e+006	-1.01276
21	Leave Period-2.INCR=3 (TIME=5.28)	4.574060e+006	-1.03063
22	Leave Period-2.INCR=4 (TIME=5.44)	4.716080e+006	-1.04235
23	Leave Period-2.INCR=5 (TIME=5.60)	4.858100e+006	-1.05066
24	Leave Period-2.INCR=6 (TIME=5.76)	4.999920e+006	-1.05639
25	Leave Period-2.INCR=7 (TIME=5.92)	5.141740e+006	-1.06016
26	Leave Period-2.INCR=8 (TIME=6.08)	5.283560e+006	-1.06254
27	Leave Period-2.INCR=9 (TIME=1.81)	1.571480e+007	-1.06398
28	Leave Period-2.INCR=10 (TIME=2.1)	1.842820e+007	-1.06483
29	Leave Period-2.INCR=11 (TIME=2.4)	2.142720e+007	-1.065327e+000



Step 13

Excess Pore Pressure vs Time Graph

Procedure

1. Select the result type as 'Plane Strain Stresses'
2. Select 'Excess Pore Pressure'
3. Let's get maximum excess pore pressure in model at each time increment. Hence select 'Abs Max'
4. Click Table
5. Select the 'Step' and 'Abs Max' columns from the table.
6. Right click using the cursor and select the 'Show graph'.
7. Enter the 'Excess pore pressure' in the Y-label.
8. Enter the graph title as 'Excess Pore Pressure vs Time'
9. Click 'OK'
10. A graph can be seen on your screen.

Extract Results

Output Data

Analysis Set: 2m Spacing Analysis C

Result Type: Plane Strain Stresses

Results: EXCESSIVE PORE STRESS

Step: Results

Initial: INCR=1 (LOAD=1.000): EXCESSIVE PC

Vertical Drains: INCR=1 (TIME=2.000e+000)

Vertical Drains: INCR=2 (TIME=4.000e+000)

Vertical Drains: INCR=3 (TIME=6.000e+000)

Vertical Drains: INCR=4 (TIME=8.000e+000)

Vertical Drains: INCR=5 (TIME=1.000e+001)

Embankment-1: INCR=1 (TIME=1.500e+001)

Select All Unselect All

Order: Step Node/Element

Object: Node Element

Nodal Results Extraction

User Defined

Select Object

Sort: X Y Z As

Maximum Minimum Abs. Max

Only Show Node/Element

Extraction Position In

Table Close

No	Step	Step Value	Node	Abs. Max (kN/m ²)
1	Initial: INCR=1 (LOAD=1.000)	1.000000e+000	84735	0.000000e+000
2	Vertical Drains: INCR=1 (TIME=2.000)	1.728000e+005	85254	1.034894e-004
3	Vertical Drains: INCR=2 (TIME=4.000)	3.456000e+005	85305	1.354766e-004
4	Vertical Drains: INCR=3 (TIME=6.000)	5.184000e+005	85254	1.225893e-004
5	Vertical Drains: INCR=4 (TIME=8.000)	6.912000e+005	85254	1.209371e-004
6	Vertical Drains: INCR=5 (TIME=1.000)	8.640000e+005	85254	1.199009e-004
7	Embankment-1: INCR=1 (TIME=1.500)	1.296000e+006	87146	4.127338e+001
8	Leave Period-1: INCR=1 (TIME=1.80)	1.555200e+006	87316	3.714425e+001
9	Leave Period-1: INCR=2 (TIME=2.10)	1.814400e+006	87176	3.191625e+001
10	Leave Period-1: INCR=3 (TIME=2.40)	2.073600e+006	87325	2.712829e+001
11	Leave Period-1: INCR=4 (TIME=2.70)	2.332800e+006	87176	2.331890e+001
12	Leave Period-1: INCR=5 (TIME=3.00)	2.592000e+006	87325	2.036735e+001
13	Leave Period-1: INCR=6 (TIME=3.30)	2.851200e+006	87176	1.783170e+001
14	Leave Period-1: INCR=7 (TIME=3.60)	3.110400e+006	87325	1.565611e+001
15	Leave Period-1: INCR=8 (TIME=3.90)	3.369600e+006	87176	1.384342e+001
16	Leave Period-1: INCR=9 (TIME=4.20)	3.628800e+006	87325	1.232999e+001
17	Leave Period-1: INCR=10 (TIME=4.50)	3.888000e+006	87176	1.099679e+001
18	Embankment-2: INCR=1 (TIME=4.800e)	4.147200e+006	87325	4.683585e+001
19	Leave Period-2: INCR=1 (TIME=4.96)	4.306400e+006	87176	4.648411e+001
20	Leave Period-2: INCR=2 (TIME=5.12)	4.465600e+006	87325	3.761736e+001
21	Leave Period-2: INCR=3 (TIME=5.28)	4.624800e+006	87176	2.893970e+001
22	Leave Period-2: INCR=4 (TIME=5.44)	4.784000e+006	87325	1.811727e+001
23	Leave Period-2: INCR=5 (TIME=5.60)	4.943200e+006	87176	1.167407e+001
24	Leave Period-2: INCR=6 (TIME=5.76)	5.102400e+006	87325	7.200054e+000
25	Leave Period-2: INCR=7 (TIME=5.92)	5.261600e+006	87176	4.254546e+000
26	Leave Period-2: INCR=8 (TIME=6.08)	5.420800e+006	87325	2.450802e+000
27	Leave Period-2: INCR=9 (TIME=6.24)	5.580000e+006	87176	1.365875e+000
28	Leave Period-2: INCR=10 (TIME=6.40)	5.739200e+006	87304	7.428776e-001
29	Leave Period-2: INCR=11 (TIME=6.56)	5.898400e+006	87281	3.949352e-001

Sorting Dialog...

Style Dialog...

Show Graph...

Export to Excel

Graph View

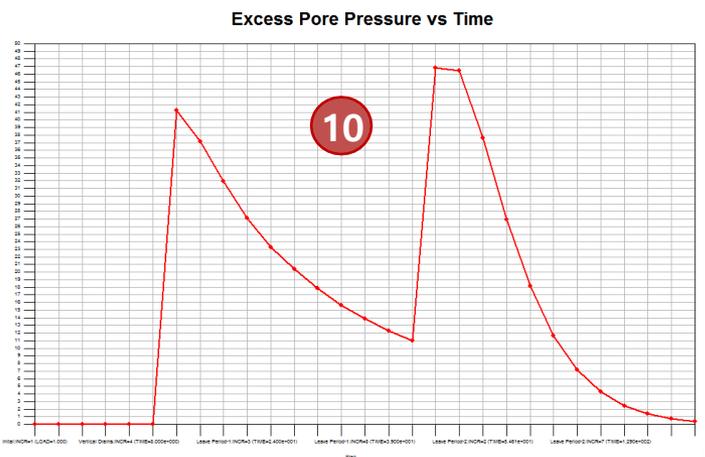
Graph Type: Simple line (x-axis: index)

X Label (Index): Step

Y Label (Value): Excess Pore Pressure

Graph Title: Excess Pore Pressure vs Time

OK





Happy Modeling



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

- Generating the model as shown in tutorial.
- Settlement vs Time – Graph Picture
- Excess Pore Pressure vs Time – Graph Picture
- Short summary of model creation, and results.

Note: Please enter the name and the country same as you entered in the previous submissions. In case of any name discrepancies among the submissions, the participants will be awarded Zero.

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

The name of the word file should follow **“YOUR NAME_COUNTRY.docx/.pdf”** format.