New eXperience of
GeoTechnical analysis System

Verification &Application Manual

TABLE OF CONTENTS

Chapter 1
Construction Stage

Chapter 2
Seepage

Chapter 3
Fully-Coupled Analysis

Chapter 4
Consolidation

Chapter 5
Slope Stability

Chapter 6
Dynamics

Verification & Application

Introduction

Thank you for your interest and support in MIDAS.

This manual has been created to help our users develop an understanding of the different analysis methods available to them GTS NX. This manual also provides a review of the reliability of results by classifying the various numerical analyses used in practical geotechnical engineering into each analysis method and field.
In addition, this manual specifies the causes and comparative methods for differences in results that can occur in GTS NX. Additional advanced analysis methods and options for approximate solutions in numerical analysis, as well as various tips and technical data, are also included to help geotechnical engineers develop greater trust in GTS NX.

This manual is composed of 6 chapters, as follows:

Chapter 1 : Construction Stage

Chapter 2 : Seepage

Chapter 3 : Fully Coupled Analysis

Chapter 4 : Consolidation

Chapter 5 : Slope Stability

Chapter 6 : Dynamics

GTS NX

GTS NX is a finite element solution for general geotechnical analysis, with a next generation platform and a 64bit integrated solver that can cope with rapidly advancing computer hardware and OS changes.
GTS NX is based on a convenient pre-post function and provides various material models and advanced analysis features for novice practitioners to research advanced analysis.
In particular, an intuitive user-friendly menu system and various optimized practical features are included and provide reliable finite element results.

New eXperience of
GeoTechnical analysis System

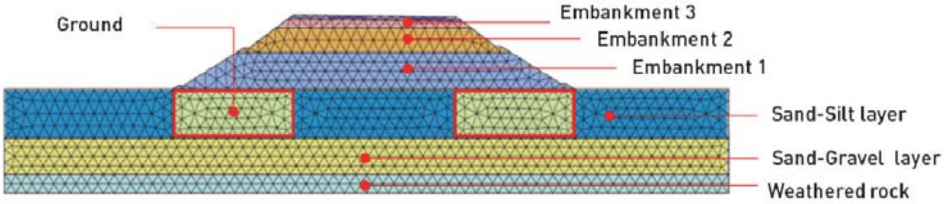
Chapter 1
Construction Stage

TABLE OF CONTENTS

Section 1 Embankment Construction Stage (2D)

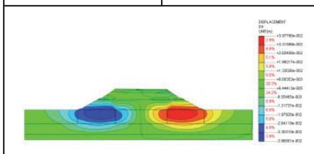
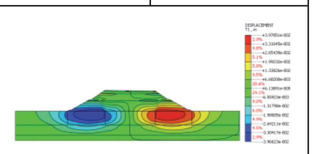
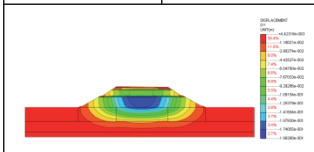
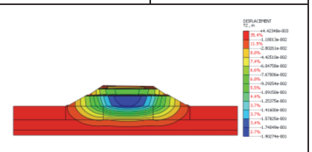
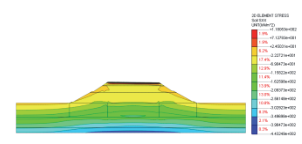
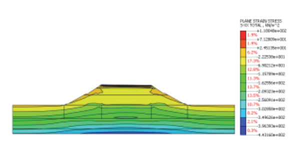
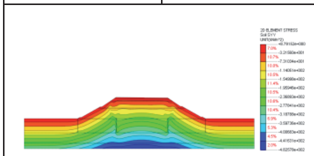
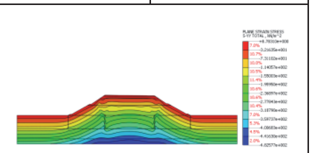
Element characteristics	Ground - Plane Strain
Number of nodes, elements	3,801 nodes, 1,822 elements
Construction stage	6 stages

This embankment model assesses the stability by loading an embankment on soft ground.
[Construction stage (6 stages)] in–situ state → ground reinforcement → 1st stage embankment → 2nd stage embankment → 3rd stage embankment → loading
The stress and displacement of the ground at the final loading stage is identified.



[Units : KN, m]

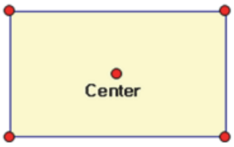
▶ Result comparison(GTS–GTS NX)

	GTS440		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
DX	-3.9650e-2	3.9779e-2	-3.9662e-2	3.97851e-2	0.03% (-1.22e-5)	0.02% (6.10e-6)
						
DY	-1.9028e-1	4.42316e-3	-1.9027e-1	4.42348e-3	0.00% (6.00e-6)	0.01% (3.20e-7)
						
SXX	-4.43249e2	1.18053e2	-4.43160e2	1.18048e2	0.02% (8.90e-2)	0.00% (-5.00e-3)
						
SYY	-4.82578e2	8.79152e0	-4.82577e2	8.78310e0	0.00% (1.00e-3)	0.10% (-8.42e-3)
						

Note Setting the location of stress–strain calculation

The stress–strain can be calculated at the [Center] or [Center+Node].

- ✓ When [Center] is set, the results are calculated using the result value at the element center. When [Center+Node] is set, the results are calculated using the result values at the element center and nodes.



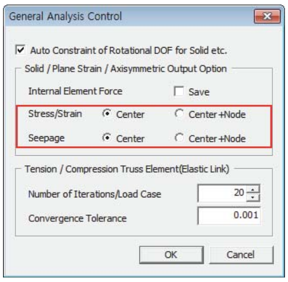
- ✓ [Center+Node] calculates a result value closer to the correct solution and setting this option to be identical is necessary when comparing the analysis results between two programs.

In GTS NX, [Center+Node] is set as the default value to calculate results close to the correct solution.

Class	Default value	Note
GTS NX	[Center+Node]	Analysis 〉 Analysis case 〉 Add 〉 Result control 〉 Element result output location
GTS440	[Center]	Analysis 〉 General analysis control

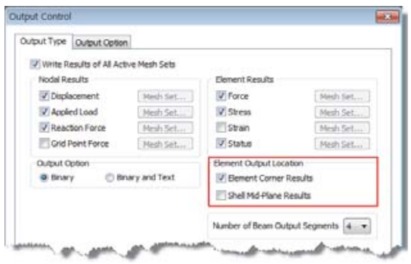
GTS440

- ➢ Location : Analysis 〉 General analysis control
  - ✓ The default value is set at the center. To compare with the GTS NX results, the analysis needs to be re-executed with [Center+Node].



GTS NX

- ➢ Location: Analysis 〉 Analysis case 〉 Add 〉 Result control 〉 Element result output location
- ✓ Default value: The element node result is checked (represents center+node). To compare with the existing GTS results, [Element node result] needs to be unchecked before executing the analysis.



Section 2

Tunnel Construction Stage (2D)

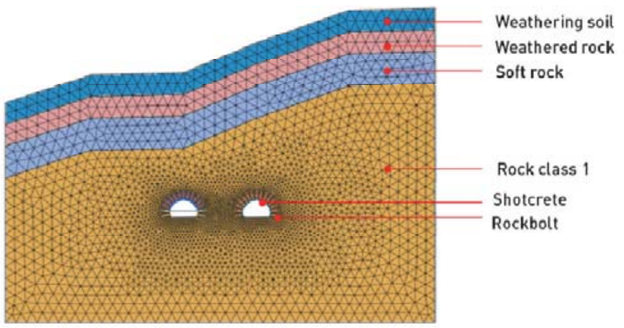
Element characteristics	Ground –Plane-Strain, Shotcrete –Beam, Rockbolt – Truss
Number of nodes, elements	3,284 nodes, 6,618 elements
Construction stage	11 stages

This is a construction stage model for tunnels installed in a sloped section.

[Construction stage (11 stages)] in–situ analysis → initialization → left–top excavation → left girder installation → left–top girder hardening → left–bottom excavation → left–bottom excavation → left–bottom girder hardening → right tunnel excavation → right girder installation → right girder hardening

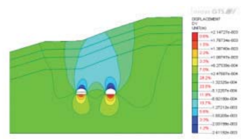
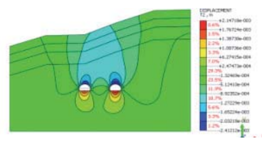
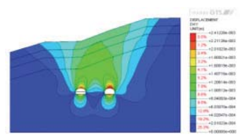
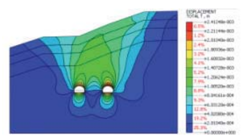
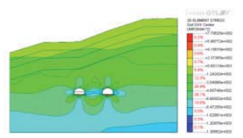
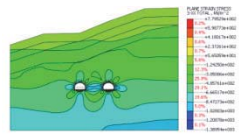
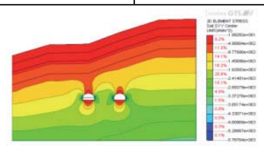
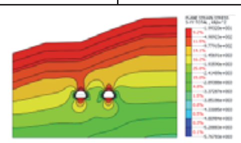
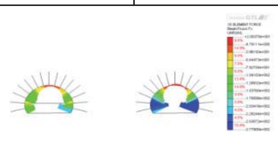
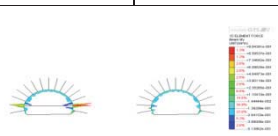
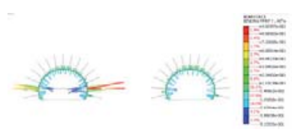
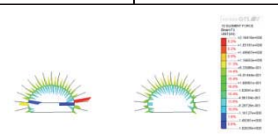
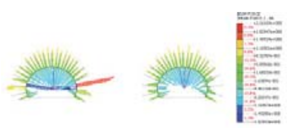
To consider the arching effects using a 2D model, the load distribution factor at the excavation–girder installation–girder reinforcement stages were applied as 0.4, 0.3, and 0.3 respectively.

The generated ground displacement and member force trends of the girders are identified at each analysis stage.



[Units : KN, m]

▶Result comparison(GTS–GTS NX)

	GTS440		GTS NX		Error Ratio (Value)	
	Min	max	min	max	min	max
DY	-2.41192e-3	2.14727e-3	-2.41212e-3	2.14718e-3	0.01% (-2.00e-7)	0.00% (-9.00e-8)
						
DXY	0	2.41228e-3	0	2.41248e-3	0.00% (0)	0.01% (2.00e-7)
						
SXX	-1.38952e3	7.79525e2	-1.38954e3	7.79529e2	0.00% (-2.00e-2)	0.00% (4.00e-3)
						
SYY	-5.76764e3	-1.99282e2	-5.76783e3	-1.99320e2	0.00% (-1.90e-1)	0.02% (-3.80e-2)
						
Beam Fx	-2.7790e2	-2.0037e1	-2.7759e2	-1.99448e1	0.11% (3.10e-1)	0.46% (9.22e-2)
						
Beam My	-5.13862e-1	9.84391e-1	-5.1333e-1	9.82997e-1	0.10% (5.32e-4)	0.14% (-1.39e-3)
						
Beam Fz	-1.82635e0	2.16416e0	-1.82503e0	2.16169e0	0.07% (1.32e-3)	0.11% (-2.47e-3)
						
Truss Fx	-1.58588e0	9.79546e0	-1.5888e0	9.76705e0	0.18% (-2.92e-3)	0.29% (-2.84e-2)
						

Note In–situ stress simulation method

The in–situ stress is set as the stress state obtained from self–weight analysis in the general construction stage analysis process. GTS calculates the in–situ stress using the K0 method or the self–weight analysis method.

K0 method

This method sets the in–situ stress by calculating the horizontal stress from the vertical stress using the constant K0 value from K0=σ h/ σv. When using this method, the vertical stress σv is obtained first from self–weight analysis, and this value is used to calculate the horizontal stress using σ h = K0 x σ v. Here, the shear stress is assumed to be 0.

The general cases where the K0 method is defined is as follows:

- ✓ When the ground shape change in the horizontal direction is insignificant
- ✓ When the pore pressure distribution does not change in the horizontal direction
- ✓ When horizontal stress is generated by the horizontal free edge/surface boundary condition
- ✓ When the material axis is perpendicular or identical to the horizontal axis, when a transversely isotropic material is used

and precautions need to be taken when applying K0 to other cases.

In particular, if the ground surface is not horizontal (sloped), the obtained stress state is not in equilibrium with the self– weight. Hence, when using the KO method to calculate the in–situ stress for these cases, analysis needs to be performed using the unbalanced internal forces between the self–weight and calculated stress state to create equilibrium. This stage can be performed by introducing a NULL Stage, where no conditions change.

Self–weight analysis method

When the ground surface is horizontal, this method is equal to the K0 method where K0=υ /(1– υ). If not, a horizontal strain exists and a different result from the K0 method is deduced along with shear stress. Hence, it is generally recommended that the self–weight analysis method be used for sloped ground. However, it is impossible to use a K0 value larger than 1 and the KO method needs to be used when using a large K0 value.

Note In–situ state setting method

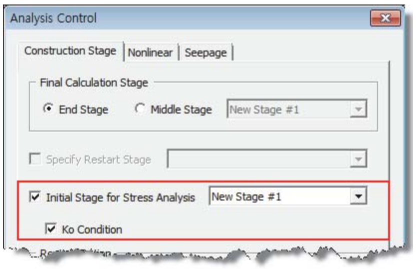
Plasticity generated in the initial state signifies that the ground cannot withstand the self–weight itself and that a long time has passed in the failed state. Because analysis is conducted with the assumption that it is stable for the in–situ state, it is valid to assume the initial stress as the K0 state.
However if the ground is not even, unbalanced forces due to nonlinear material models occur in finite element analysis and the [Null Stage + Displacement initialization] stage, where no conditions change, needs to be set after the in–situ stage to solve this problem..
In other words, it is desirable to perform analysis after setting the strain due to unbalanced forces to be '0' to assume that the initial ground state is stable in K0 analysis.

When comparing the results from GTS NX and GTS440, the following methods are recommended for review when the ground is not even:

Class	Self weight analysis method	K0 method
GTS NX	Step 1 : K0 uncheck	Step 1 : K0 check Step 2 : Null Stage + Displacement initialization
GTS 440	Step 1 : K0 uncheck	Step 1 : K0 check

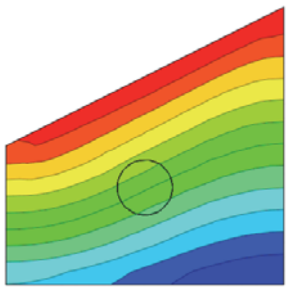
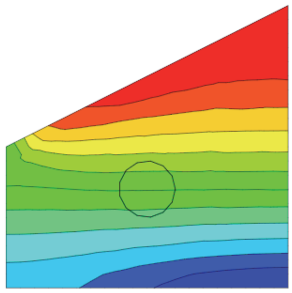
Note Understanding of K0 option – GTS 440

➢ Location : The settings are available after checking the stress analysis initial stage in the Analysis 〉 Analysis case 〉 Analysis control 〉 Construction stage tab.



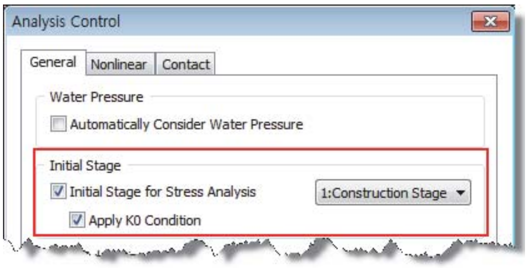
The converging solution is found with K0 stress as the initial condition. (The vertical stress is calculated first and then the K0 condition is added for convergence within one step → simulate stress state with converged unbalanced forces)

Hence when the ground is not even, the ratio of horizontal stress/vertical stress is not equal to K0 at the initial stress stage.

Vertical Stress	Horizontal Stress
	

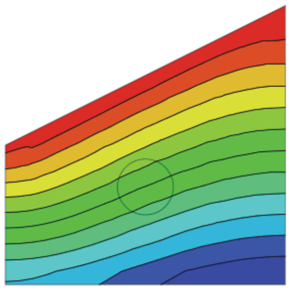
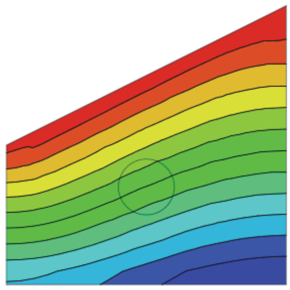
Note Understanding of K0 option – GTS NX

- Location : The settings are available after checking the stress analysis initial stage in the Analysis 〉 Analysis case 〉 Add 〉 Analysis control 〉 Construction stage tab.



Even when the equilibrium state is not reached after applying the K0 condition, analysis is still performed with the unbalanced force state remaining

Hence, the ratio of horizontal stress/vertical stress is equal to K0 at the initial stress stage.

Vertical Stress	Horizontal Stress
	

Note Tunnel reinforcement–Shotcrete : Beam element and results

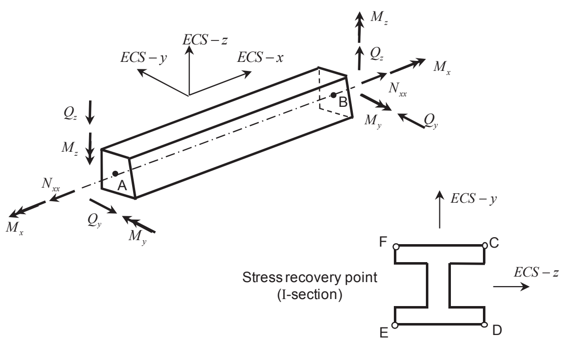
- This element is often used for members which have a relatively longer length than its section size that experience bending deformation. The member forces that occur are axial force, shear force and moment.
- If the ratio between the length and section width or height is larger than 1/5 approximately, the effects of shear deformation become very large and it is appropriate to use the following elements:
  - ✓ For 2D : Beam element → 2D Plane-Strain element
  - ✓ For 3D : Beam element → Shell element or Solid element

Beam element stress results

Class	GTS 440	GTS NX
Bending compressive stress	fiber 1-SXX	Beam Element Stresses S-MIN
	fiber 2-SXX	Beam Element Stresses S-MAX

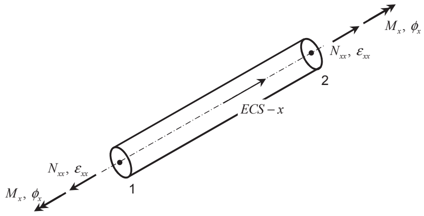
GTS NX stress calculation position setting

GTS NX can additionally express the stress calculated at each position marked on the beam element section. This stress calculation position can be directly controlled by the user.



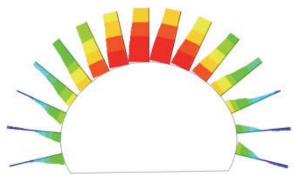
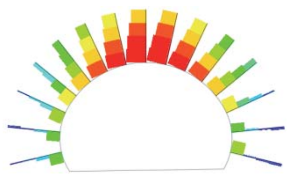
Note Tunnel reinforcement–Rockbolt : Truss element and results

✓ This element is often used to model members with a relatively longer length than its section size. It is often used to model **structural members with negligible bending behavior,** such as anchors, nails or rockbolts. Only axial force occurs.



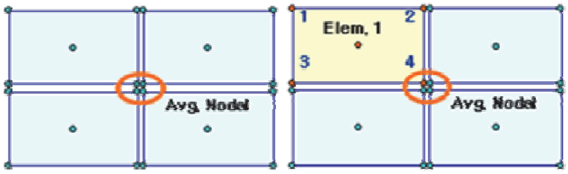
In the existing GTS440, the truss element results were output with the Beam/Truss FX and only the truss mesh set needed to be shown for result verification. However in GTS NX, the results are separated such that the member forces of truss elements can be verified conveniently.

When comparing the results with those of GTS440, the **Result analysis 〉 Show/Hide 〉 Nodal average option needs to be unchecked** in GTS NX before performing the comparison.
(Checking this option in GTS NX outputs the averaged results of the I, J nodes of adjacent elements.)

GTS440	GTS NX(Nodal average)	GTS NX(Node averagal removed)
		

Note Nodal Average

For a rectangular element, 5 results are output for each element and 4 values exist with reference to a single node. The nodal average feature outputs the average of these 4 values when printing the results.

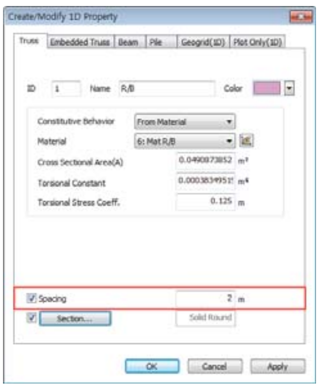


Generally, the element result gives a larger output than the nodal average checked result value. Also, the element result and the nodal average checked result value can differ greatly in a section with large stress change. Designer judgment is important in combining and deciphering analysis result because the result verification methods differ depending on the structure type, modeling method etc. and, consequently, precautions need to be taken when checking the results.
GTS NX provides various combination methods and result verification methods to fit the user's intentions, and the user can check or uncheck this option when outputting the result values to verify the results.
For example, the center of the slab uses the nodal average to compute a continuous analysis result. For the intersection between a slab and wall, using the nodal average can offset the results and so it is appropriate to verify using the element result.
The element results are output at the node or center. Because the result values from result extraction are the same as the results with the nodal average option unchecked, uncheck the nodal average beforehand when verifying the table.

Note Consideration of spacing between members

In 2D analysis, the distance between members for 1D elements (CTC(spacing)) can be entered.

- Location : Static/Slope analysis 〉 Property/Coordinate/Function 〉 Properties



In the existing GTS440, the member forces of Beam, Truss/Embedded Truss elements in a 2D model were output as the generated axial force per unit m. However in GTS NX, the member forces are output as the axial force per member section multiplied by the CTC.

- ✓ GTS440 : $f = \sigma A^* = E\varepsilon(nd)\dfrac{A}{n} = E\varepsilon(d)A$
- ✓ GTS NX : $f^* = nf$

Section 3

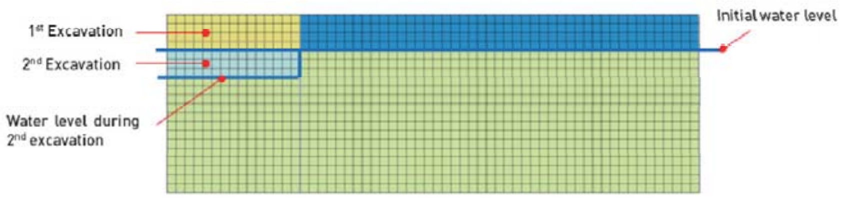
Temporary Structure with changing water level (2D)

Element characteristics	Ground - Plane Strain Wall –Beam element, Strut –Truss element
Number of nodes, elements	1,281 nodes, 1,211elements
Construction stage	5 stages

This is an excavation construction stage model for temporary structures circumjacent to neighboring roads, where road load is applied on the ground surface. The water level is set to change during the 2nd excavation.

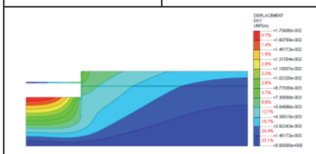
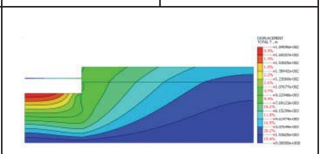
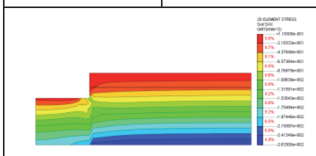
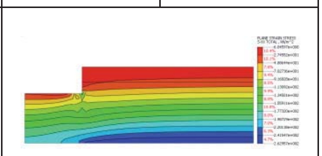
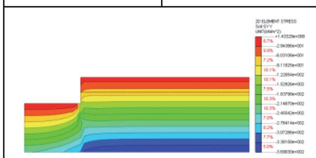
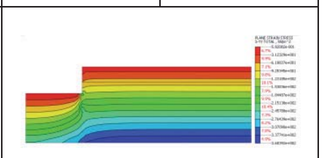
[[Construction stage (5 stages)] in–situ state → retaining wall construction → road load application → 1st excavation → 2nd excavation

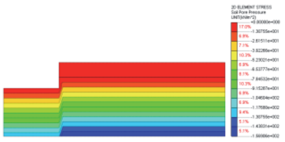
The displacement, stress and pore water pressure within the ground is chekced.



[Units : KN, m]

▶Result comparison(GTS–GTS NX)

	GTS440		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
DXY	0	1.75406e-2	0	1.84590e-2	0.00% (0)	5.24% (9.18e-4)
						
SXX	-2.6330e2	1.183e-1	-2.62957e2	-6.04597e0	0.13% (3.43e-1)	- (6.16e0)
						
SYY	-3.69030e2	1.43329e0	-3.68392e2	-5.82082e-1	0.17% (6.38e-1)	- (-2.02e0)
						

Pore Pressure	-1.56906e2	0	-1.56906e2	0	0.00% (0)	0.00% (0)
						

Note Water level setting

GTS NX can define the water level depending on the construction stage for construction stage analysis. The height of the water level can be input directly in Static/Slope analysis or Seepage/Consolidation analysis 〉 Construction stage 〉 Construction stage set, or the water level can be set using a function with a predefined water level. The water level is applied to the entire construction stage after it is set in its construction stage. When a water level function is used, the input value is multiplied by the function value and applied.

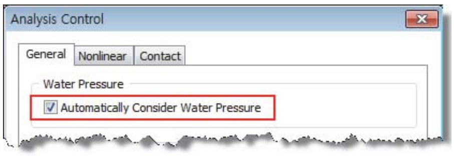
- ✓ If the water level does not change, the GTS NX results are the same as the existing GTS440 results.
- ✓ If the water level changes, the GTS NX results may differ from the existing GTS440 results.

This is because the method for considering stress caused by pore water pressure ($f_p = \int \mathbf{B}^T(\mathbf{I}_3\, p)\, d\Omega$) differs between the two programs.

In GTS440, the stress due to pore water pressure is applied as an internal force and external force simultaneously. However in GTS NX, it is applied as a part of the internal force. Hence when the external force is output, a difference occurs by the amount of stress load due to water pressure. In other words, the seepage force due to the water level difference is reflected in the analysis in GTS NX.

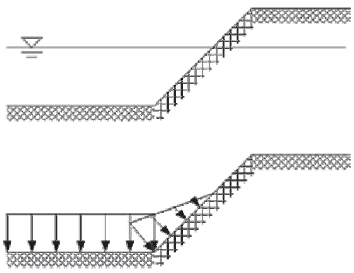
Note Automatic consideration of water pressure

➤ Location : Analysis 〉 Analysis case 〉 Add 〉 Analysis control 〉 General (GTS NX))



In the existing GTS440, when the water level is above the ground surface, an additional pressure load corresponding to the height difference is input to cancel the load due to buoyancy, as shown in the figure below.
In GTS NX, the water pressure acting on the free surfaces/edges of the model can automatically be defined as an external force, and it is calculated with reference to the pore pressure of the target free surface/edge.

- ✓ When the water level is specified, a static water pressure state is assumed with reference to the water level
- ✓ When seepage analysis has been performed, the pore pressure distribution(size) calculated at each node is used
- ✓ When the pore pressure has a negative(–) value, water pressure is not automatically considered



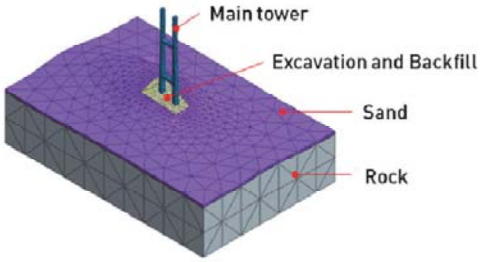
(Caution) This option needs to be turned off for models that do not have an external water pressure corresponding to the pore pressure within it. If stress analysis is conducted by specifying a water level, the pore pressure is calculated depending on the water height at the corresponding node and free nodes. Hence, seepage-stress coupled analysis is recommended to accurately review the impact of the ground water level.

Section 4

General Construction Stage (3D)

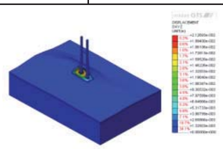
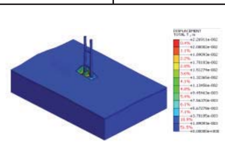
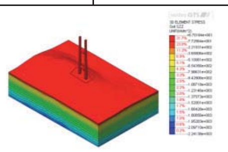
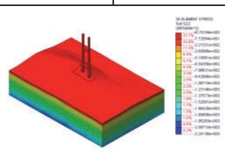
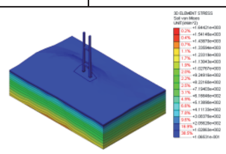
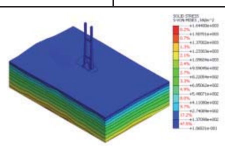
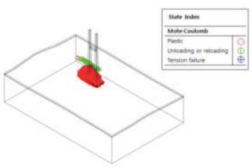
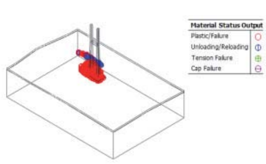
Element characteristics	Ground – Solid
Number of nodes, elements	4,261 nodes, 20,630 elements
Construction stage	5 stages

Large scale loads act on the main span of a bridge and its stability is susceptible to the geography and ground state. Hence, the 3D behavioral properties of the main towers on the main span bridge need to be analyzed. [Construction stage (5 stages)] in-situ state → excavation → foundation installation and loading → main tower installation → backfill. The stress and displacement trends of the ground are examined.



[Units : KN, m]

▶Result comparison(GTS-GTS NX)

	GTS440		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
DXYZ	0	2.12693e-2	0	2.26911e-2	0.00% (0)	6.68% (1.42e-3)
						
SZZ	-2.24138e3	6.70184e1	-2.24138e3	6.70184e1	0.00% (0)	0.00% (0)
						
Von Mises	1.06531e-1	1.64421e3	1.06531e-1	1.6440e3	0.00% (0)	0.01% (-2.10e-1)
						
Plastic Status					The displacement difference between the programs at the final stage is judged to be due to different plastic regions that occur during analysis	

Note Convergence condition

The solution in nonlinear analysis is obtained using the iterative method. When the solution found using the iterative method passes a certain standard, it is regarded as converged. The convergence condition needs to be applied to provide such standards for convergence. The available convergence conditions are the displacement standard (U), member force standard (P), energy standard (W) etc. and the solution is deemed to have converged when these values are smaller than the tolerance value.

The selected standard to judge convergence differs for each analysis type. If the structure can deform freely, the member force standard may be inappropriate. The convergence condition for material softening behavior is more complex than hardening behavior. Appropriate reasoning is needed when selecting the convergence condition for the analysis of any model. If it is difficult to select the convergence condition for nonlinear models, it is recommended that two convergence conditions be applied and their results compared.

Displacement standard (U) = $\dfrac{\sqrt{\delta u_i^T \delta u_i}}{\sqrt{\Delta u_i^T \Delta u_i}}$ Member force standard (P) = $\dfrac{\sqrt{\delta g_i^T \delta g_i}}{\sqrt{\Delta f_{\mathrm{int},i}{}^T \Delta f_{\mathrm{int},i}}}$

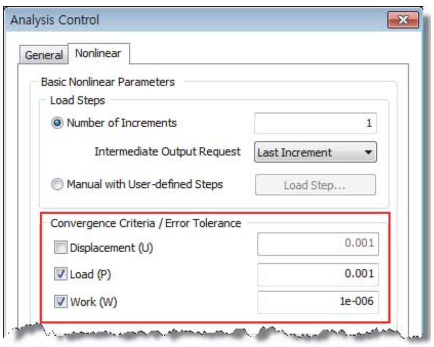
Energy standard (W) = $\left|\dfrac{\delta u_i{}^T g_i}{\Delta u_i{}^T \Delta f_{\mathrm{int},i}}\right|$

Comparison of default values of each convergence standard

Classification	GTS 440	GTS NX
Displacement standard(U)	10^{-3} (UnChecked)	10^{-3} (UnChecked)
Member force standard(P)	10^{-3} (Checked)	10^{-3} (Checked)
Energy standard(W)	10^{-3} (UnChecked)	10^{-6} (Checked)

✓ GTS NX proposes the simultaneous convergence standard (member force standard and energy standard) as its basic setting. The convergence standard solver also converges for more critical conditions in the table above, making it possible to obtain more reliable results.

➢ Location : Analysis 〉 Analysis case 〉 Add 〉 Analysis control 〉 Nonlinearity

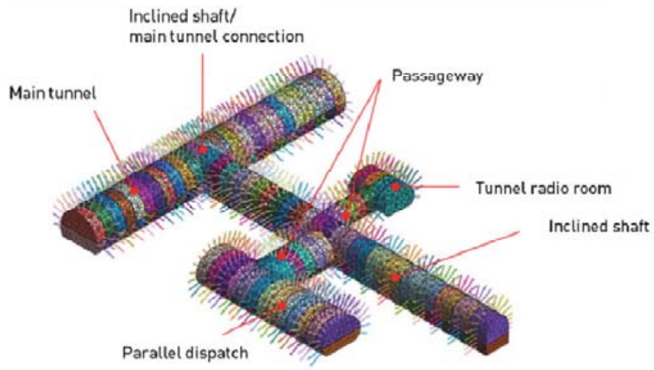


Section 5 Tunnel Construction Stage (3D)

Element characteristics	Ground – Solid. Shotcrete – Plate/Shell Rockbolt – Embedded Truss
Number of nodes, elements	26,257 nodes, 120,931 elements
Construction stage	78 stages

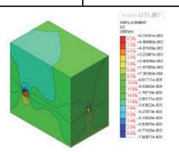
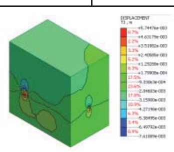
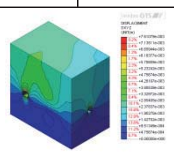
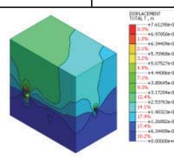
The parallel dispatch center and train radio room, installed additionally at the connection between the inclined shaft and main tunnel, need to be planned considering the stability according to structural interaction and field conditions. These electricity related structures are positioned close to the main tunnel/inclined shaft connection, such that the structure is concentrated in a tight space. Hence, interaction behavior between structures and stress concentration can occur and the stability between structures and the adjacent ground need to be investigated using 3D numerical analysis.

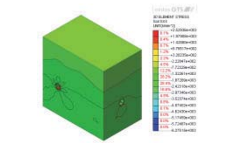
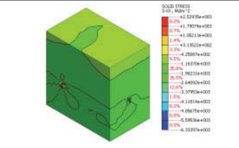
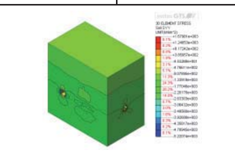
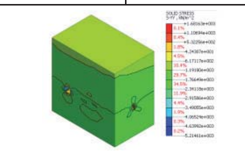
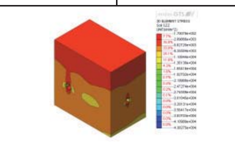
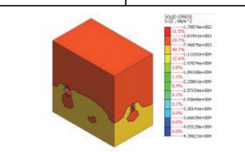
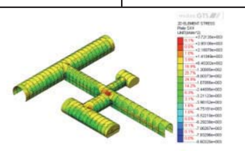
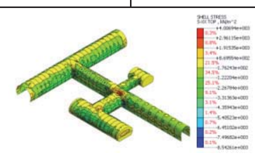
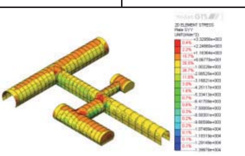
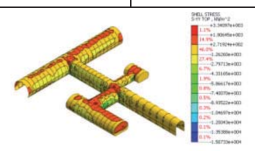
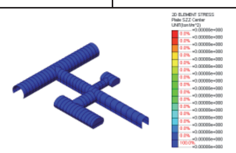
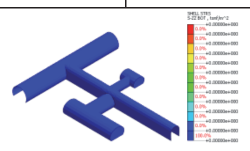
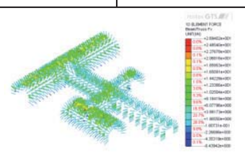
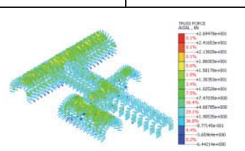
[Construction (78 stages)] This is a large model, comprised in the order of inclined shaft excavation → inclined shaft/main tunnel excavation → main tunnel excavation → parallel dispatch center passageway excavation→ parallel dispatch center excavation → train radio room passageway excavation → train radio room excavation.



[Units : KN, m]

▶Result comparison(GTS–GTS NX)

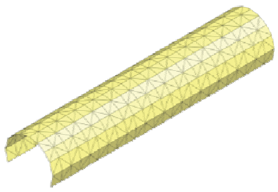
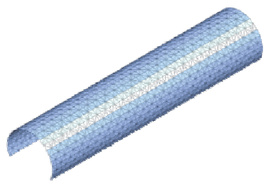
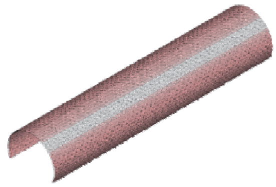
	GTS440		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
DZ	-7.60877e-3	5.74341e-3	-7.61089e-3	5.74476e-3	0.03% (-2.12e-6)	0.02% (1.35e-6)
						
DXYZ	0	2.41228e-3	0	2.41248e-3	0.00% (0)	0.03% (2.00e-7)
						

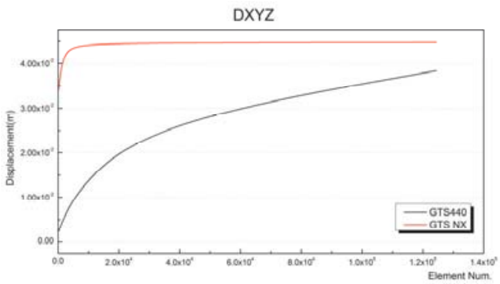
SXX	-6.27515e3	2.52936e3	-6.33397e3	2.52935e3	0.94% (-5.88e1)	0.00% (-1.00e-2)
						
SYY	-5.22074e3	1.67981e3	-5.21461e3	1.68163e3	0.12% (6.13e0)	0.11% (1.82e0)
						
SZZ	-4.38275e4	-1.70078e2	-4.39621e4	-1.70074e2	0.31% (-1.35e2)	0.00% (4.00e-3)
						
Shell SXX-TOP	-8.60325e3	3.72135e3	-8.54261e3	4.00694e3	0.70% (6.06e1)	7.67% (2.86e2)
						
Shell SYY-TOP	-1.3997e4	3.32956e3	-1.50733e4	3.34097e3	7.68% (-1.08e3)	0.34% (1.14e1)
						
Shell SZZ-TOP	0	0	0	0	0.00% (0)	0.00% (0)
						
Truss Fx	-6.43942e0	2.69402e1	-6.44214e0	2.69478e1	0.04% (-2.72e-3)	0.03% (7.60e-3)
						

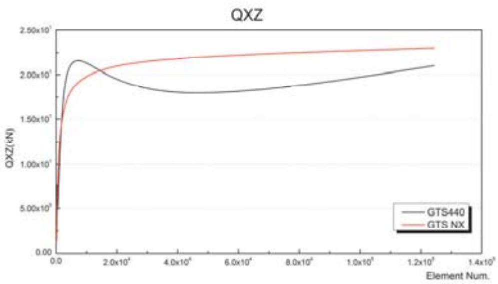
Note Understanding Plate/Shell elements – 1

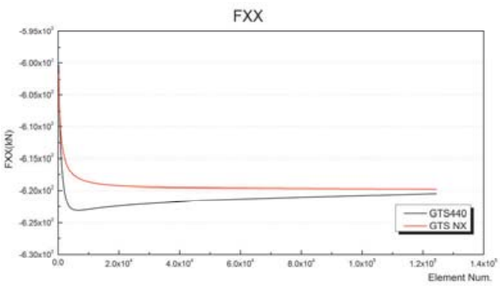
- ✓ GTS NX has improved analysis solver performance and convergence which allows for the accurate calculation of results for even a small number of elements. Hence, the ground/structural member results can differ from that of GTS 440.
- ✓ In GTS440, the result difference can occur depending on the number of elements and degree of convergence. To obtain more accurate solutions for Plate/Shell elements, the number of elements needs to be increased or a higher order element needs to be used.

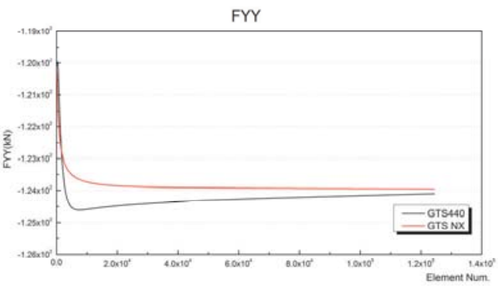
Displacement/member force trends of Plate/Shell elements, depending on the number of elements

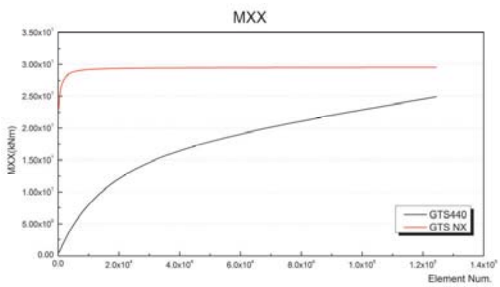
Element 480	Element 1920	Element 7680
		

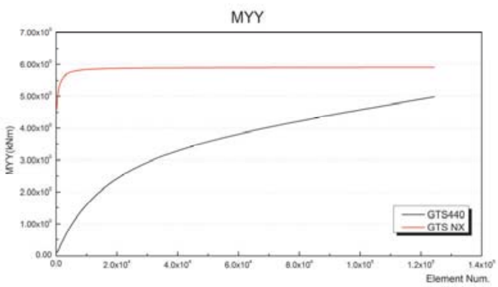








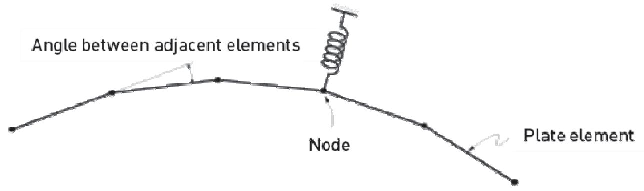




➢ Tip! : The Bottom / Middle / Top in GTS NX corresponds to the layers 1,2,3 of a Plate face in GTS440.

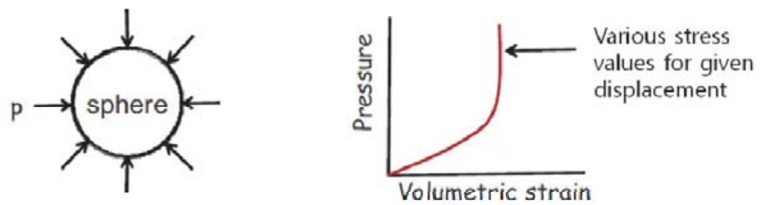
Note Understanding Plate/Shell elements – 2

- ✓ Plate/Shell elements can be used to model concrete lining, shotcrete, earth retaining walls, base plates, etc––all of which can display planar direction behavior and out–plane bending behavior.
- ✓ When modeling a curved structure using Plate/Shell elements, the angle between adjacent elements must not surpass 10° and an angle of 2~3° is recommended for sections that require an exact solution. For sections where the stress change is large or requires an exact solution, subdividing the section into a 4 node element (close to a square) is recommended.

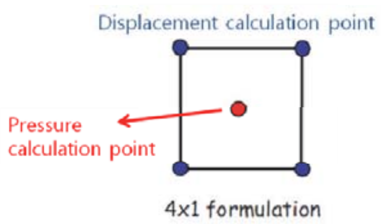


Locking phenomenon

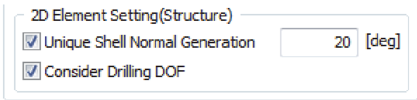
The locking phenomenon is where no deformation occurs for a changing load and it is one the most common reasons for non–converging solutions in nonlinear analysis.
Because this phenomenon cannot be calculated using general methods that calculate the pressure from strain, the existing GTS440 was vulnerable to the locking phenomenon in Plate elements.



GTS NX uses mixed formulation that deals with pressure and displacement independently, allowing the deduction of reasonable results even for the locking phenomenon. For example for linear elements, it can be seen as a method of calculating an additional pressure at a single point per element.



2D element setting: Tools 〉 Option 〉 Analysis/Results 〉 2D element setting (structure)



- Curved surface/Folded surface determining angle
  - ✓ If the angle between normal vectors of adjacent shell elements is larger than the input value, the elements are judged to have different normal directions.
  - ✓ If an uneven curved mesh is created because the element size is relatively larger than the curvature, increasing this value can calculate a smoother contour that takes the curvature of the geometric shape into account.

- In–plane rotational DOF consideration (Drilling DOF)
  - ✓ This option calculates the stiffness of the in–plane deformation by considering the rotation about the out–plane axis (drilling DOF).
  - ✓ The existing GTS440 does not have a separate clause for definition and it is automatically calculated internally. GTS NX on the other hand has additional options that can consider it.
  - ✓ To compare with existing products, the Drilling DOF must be checked.

New eXperience of

GeoTechnical analysis System

Chapter 2
Seepage

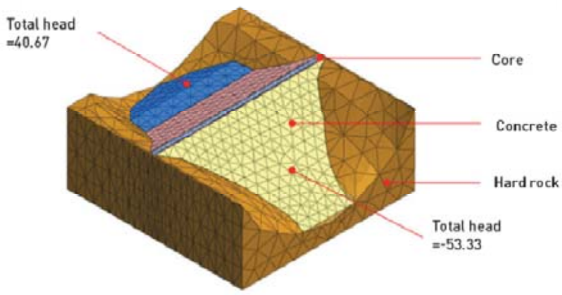
TABLE OF CONTENTS

Section 1

Steady State Analysis (3D)

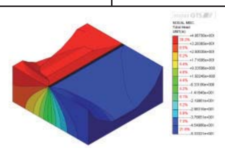
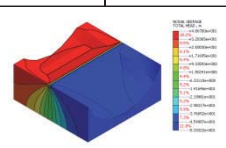
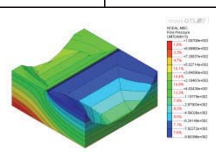
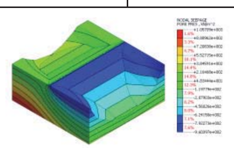
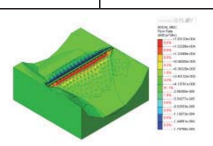
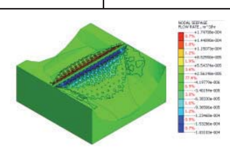
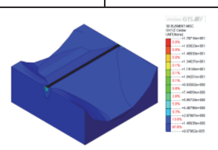
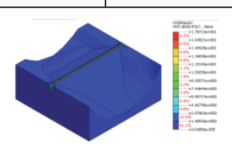
Element characteristics	Ground –Solid
Number of nodes, elements	14,823 nodes, 9,408 elements
Analysis method	Steady State

This 3D steady state seepage model analyzes the water flow in a dam body due to a water level difference. Water is assumed to be filled up to the dam height on the left side, and the water flow phenomenon is simulated with the assumption that there is no water on the right side of the dam. The total head, pressure head, flow rate and hydraulic gradient etc. is verified.



[Units : KN, m, day]

▶Result comparison(GTS–GTS NX)

	GTS440		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
Total Head	-5.33321e1	4.067e1	-5.33322e1	4.067e1	0.00% (-1.00e-4)	0.00% (0)
						
Pore Pressure	-9.60396e2	1.05708e3	-9.60397e2	1.05709e3	0.00% (-1.00e-3)	0.00% (0)
						
Flow Rate	-1.83103e-4	1.74708e-4	-1.83103e-4	1.74708e-4	0.00% (0)	0.00% (0)
						
Gradient	9.07962e-5	1.78716e1	9.04056e-5	1.78713e1	0.43% (-3.91e-7)	0.00% (-3.00e-4)
						

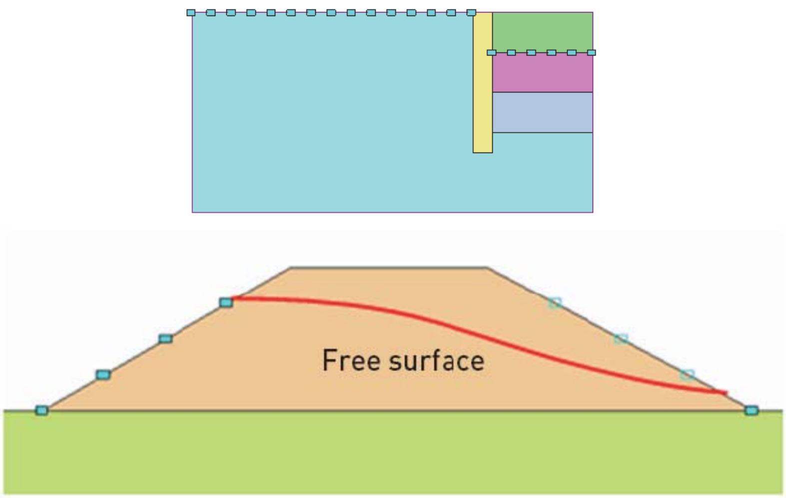
Note Seepage analysis boundary condition

Seepage analysis in GTS can be divided into steady state analysis (internal and external boundary conditions of the ground do not change with time) and transient analysis (internal and external boundary conditions of the ground changes with time).
Like other general purpose FEM analysis program, GTS NX classifies seepage analysis as a boundary value problem. In other words, if the user inputs a value at the boundary surface (exterior) of the model, the internal variables of the model are calculated from that value.

In GTS NX, the model boundary conditions can be applied using the following 3 methods:

Nodal head : Seepage/Consolidation analysis 〉 Boundary condition 〉 Nodal head

- Applicable when the water level position is known
- Model confined flow with no occurring phreatic surface, or unconfined flow with seepage face formation.
- Total head : the head value calculated for the origin is input, regardless of the model position
- Pressure head : ‘0’ is input for the node corresponding to the ground water surface to set the ground water level condition



〈Nodal head application (example)〉

Nodal flow/Surface flow : Seepage/Consolidation analysis 〉 Boundary condition 〉 Nodal flow, Seepage/Consolidation analysis 〉 Boundary condition 〉 Surface flow

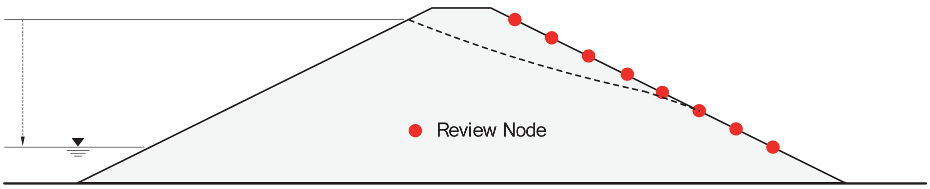
- Used to determine the inflow/outflow that occurs on a node/element surface
  - ✓ Nodal flow : inflow/outflow per unit time is input in volume units
  - ✓ Surface flow : inflow/outflow per unit time is input for unit area

  (+) value : Input when defining the inflow due to heavy rain etc. =

  (–) value : Input when defining the outflow due to excavation or pumping etc.

Boundary review : Seepage/Consolidation analysis 〉 Boundary condition 〉 Boundary review

- Applied when it is hard to find the seepage line accurately
- Assuming that seepage occurs at the downstream face of a homogeneous dam, the location of the seepage line that intersects the downstream face is unknown and the boundary review condition is set to perform iterative calculations



〈 Review node specification (example)〉

The pore pressure P, calculated at the node where the boundary review is set, is considered as the following conditions and the seepage face is automatically defined.

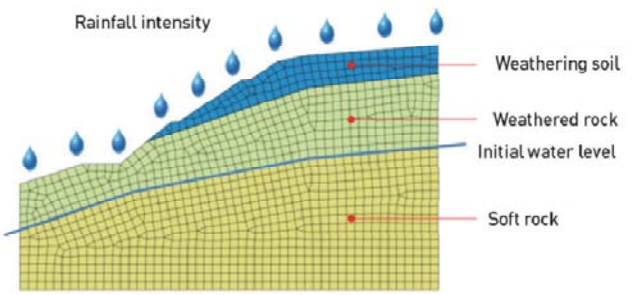
- ✓ P 〉 0 ➔ P = 0
- ✓ P 〈 0 ➔ Delete

Section 2

Transient Analysis (2D)

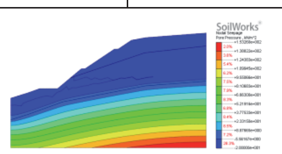
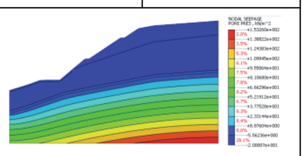
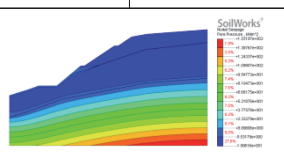
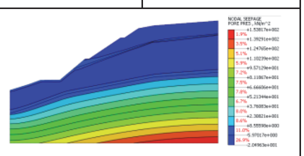
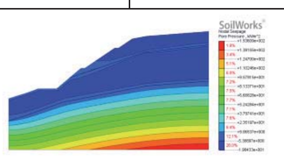
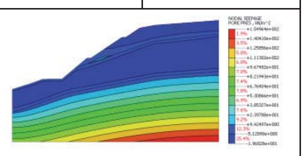
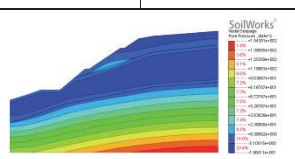
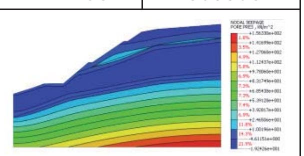
Element characteristics	Ground -Plane-Strain
Number of nodes, elements	1,208 nodes, 1,149 elements
Analysis method	Transient

This seepage model simulates the ground water level change on a sloped surface with a steady rainfall inflow. The unsaturated soil property is applied to the ground and transient analysis is conducted by constant rainfall inflow intensity on the ground surface. Seepage analysis verifies the change in initial water level depending on the rainfall duration.



[Units : KN, m]

▶Result comparison (SoilWorks–GTS NX)

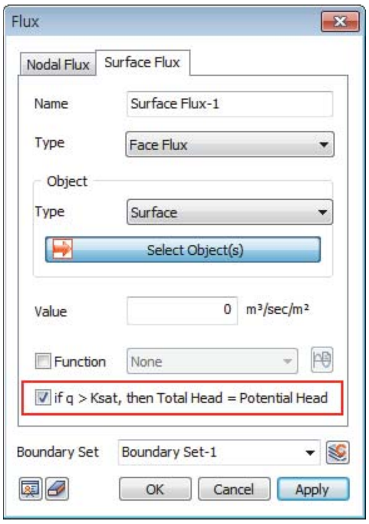
	SoilWorks		GTS NX		Error Ratio(Value)	
	min	max	min	max	min	max
Initial Pore Pressure	-2.00000e1	1.53260e2	-2.00007e1	1.53260e2	0.00% (-7.00e-4)	0.00% (0)
						
Pore Pressure after 1 day	-1.99616e1	1.53197e2	-2.04963e1	1.53817e2	2.68% (-5.35e-1)	0.40% (6.20e-1)
						
Pore Pressure after 2 days	-1.98433e1	1.53609e2	-1.96828e1	1.54964e2	0.81% (1.60e-1)	0.88% (1.35e0)
						
Pore Pressure after 3 days	-1.96011e1	1.54351e2	-1.92426e1	1.56330e2	1.83% (3.58e-1)	1.28% (1.98e0)
						

Note Additional consideration options for seepage analysis boundary – 1

For transient seepage that considers unsaturated characteristics, additional options other than the permeability condition of the ground, head difference or flow needs to be set to appropriately simulate actual seepage phenomena.
These additional seepage boundary consideration conditions can be defined separately in GTS NX.

Flow for rainfall analysis-head conversion boundary: if surface flow 〉 coefficient of permeability then total head = potential head

➢ Location : Seepage/Consolidation analysis 〉 Boundary condition 〉 Surface flow



If the rainfall intensity is input on the ground surface, the surface flow can be used to define the ground surface boundary conditions. This adds a forced inflow on the ground surface, equivalent to the rainfall intensity. If the absorption capacity of the ground surface is larger than the rainfall intensity, then all of the rain is absorbed by the soil layer. If it is smaller, then the soil layer absorbs rain water up to its capacity and the excess water flows along the ground surface.

When the rainfall intensity is greater than the absorption capacity of the ground surface, then the ground surface displays the same phenomenon as a saturated state with a water level through the duration of the rainfall.

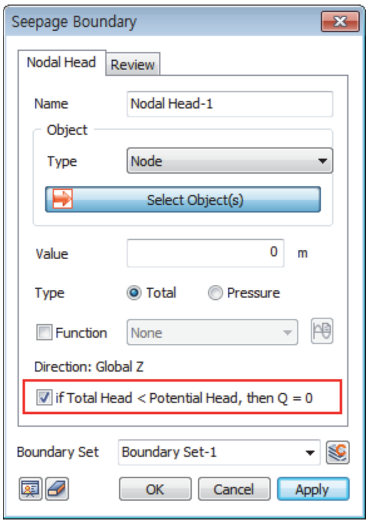
✓ In GTS NX, the "surface flow 〉 coefficient of permeability, total head = potential head" option can be used to automatically change the ground boundary from the existing rainfall intensity inflow condition to a water level condition only when the rainfall intensity, q, acting on the ground surface is larger than the inflow capacity (input initial coefficient of permeability).

✓ However for surface flow, this option cannot be used when an impermeable layer such as asphalt or concrete is applied on the ground surface.

Note Additional consideration options for seepage analysis boundary – 2

Head for water level change analysis – flow conversion boundary (if total head 〈 potential head, flow = 0)

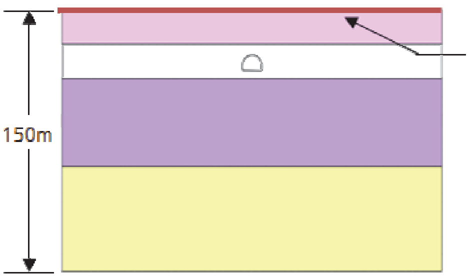
➢ Location : Seepage/Consolidation analysis 〉 Boundary condition 〉 Nodal head



When the water level changes with time, suction can occur and the seepage flow may be reversed. Especially when simulating an embankment or dam, the drop in water level for the boundary condition can create a negative pore pressure. In this case, the negative pore pressure needs to be dissipated to prevent seepage flow due to suction.

✓ This function is convenient to apply in areas where the water level changes periodically, such as areas where the rainfall differs according to the season, and it can be applied simultaneously with a function of change in time.

✓ If the water level height, input in the form of total head, is below the selected area, the boundary condition is automatically eliminated when this option is checked. Hence, this option must be set to [Uncheck] in this case.



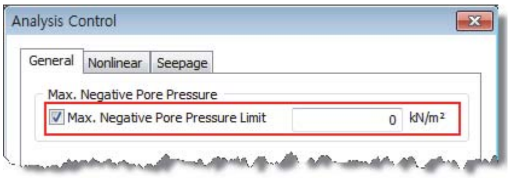
When a total head of 150m is input, the water level height is below the selected area, so,

➔ Uncheck the [If total head 〈potential head, flow = 0] option

Note Additional consideration options for seepage analysis boundary – 3

Constraint on maximum negative pore pressure

- Location : Analysis 〉 Analysis case 〉 Add 〉 Analysis control 〉 General



✓ As evaporation continues within a slope during a long dry season without rainfall, the suction in the ground increases. When rain falls on a slope with highly distributed suction, seepage does not occur because the permeability of the ground has fallen greatly. On the other hand during the wet season, where the slope is preceded by weak rainfall and its surface has sufficient water to lower the suction, the permeability of the ground is large and the rain can seep into the ground easily.

✓ To accurately reflect such phenomena, the initial suction condition of the ground needs to be considered.

✓ From an engineering perspective, the actual natural phenomena can occur up to the range of 500kPa, but changes occur on the actual slope at the range of 200kPa.

✓ GTS NX uses the negative pore pressure constraint option, which makes any value over the maximum negative pore pressure that has been set regardless of the initial ground water level position to have the same uniform negative pore pressure distribution. This is used to simulate field conditions.

Note Comparison of seepage analysis boundary conditions and options

Boundary condition/option	GTS440	GTS NX	Function and use
Nodal head(total/pressure)	O	O	- Set static head/variable head line(surface)
Flow(node/face)	O	O	- Input inflow/outflow
Boundary review	*△	O	- Seepage surface search following head change - Seepage surface search following inflow/outflow
Water level change consideration(option)	X	O	- Prevent seepage flow reversal following head change
Ground surface inflow control (option)	X	O	- Only inflow smaller than the maximum infiltration capacity of the ground surface is reflected in the analysis
Maximum sub pore pressure constraint(option)	X	O	- Set initial ground surface suction

* Only seepage surface search following head change is available

〈Table. Comparison of seepage boundary conditions and options〉

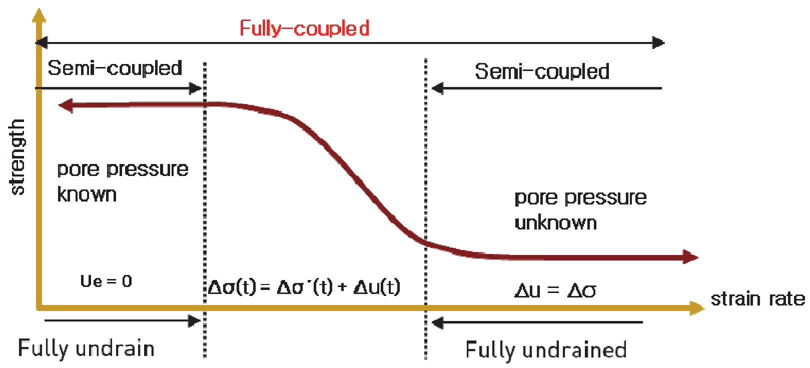
New eXperience of
GeoTechnical analysis System

Chapter 3
Fully-Coupled Analysis

TABLE OF CONTENTS

Section 1 Fully-Coupled Analysis

Numerical analyses for general geotechnical problems assume a fully drained or fully undrained state. However, ground behavior has a close relationship with time and the pore pressure is affected by the permeability of the ground, the loading velocity and numerical boundaries. Fully coupled analysis reflects the water flow within the soil pores and the ground deformation due to loading simultaneously to simulate actual ground behavior.
In principle, stress-seepage problems need to be solved using two-directional coupled analysis which affects each other. However, because this analysis method is complex, a one-directional semi-coupled method is used where stress and water pressure are generally separated for analysis and then combined. This analysis method is often applied to fully drained problems with a pore pressure of 0, or fully undrained problems which have the same pore pressure increment and stress increment.
However if stress and pore pressure change simultaneously with time, such as for transitional sections, Fully-Coupled Analysis with combined displacement and pore pressure is needed and the existing consolidation analysis is applicable.



〈 Outline of coupled analysis〉

Stress seepage fully-coupled analysis uses the seepage analysis results for stress analysis. It is a two-directional analysis, where the ground deformation has an effect on the permeability characteristics of the ground, such that the seepage and stress-consolidation of one stage is calculated simultaneously. Hence, it can be applied to all analyses that consider the water level condition and it can examine the settlement trends and member force changes according to the time flow in one construction stage as well as for the other construction stages.

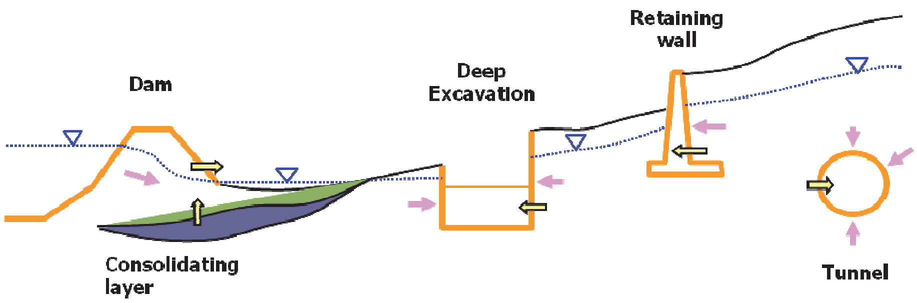
In particular, because the head, flow rate and boundary review conditions that are applied in seepage analysis can also be applied to stress changes and the consolidation process, various previously unavailable special construction methods (Suction Drain, Well-Point etc.) can be applied. It can also analyze the stress and deformation trends of the ground in general construction stage analysis using just the water level condition changes due to ground water outflow or rainfall.

The consolidation drainage condition in stress seepage fully-coupled analysis can be set as total head=0, as shown in the table below, and all load/boundary conditions that were considered separately in existing coupled analysis can be used in combination.

	Semi-Coupled	Consolidation	Fully-Coupled
Nodal head(total head)	O	X	O
Nodal head(pressure head)	O	X	O
Boundary review	O	X	O
Drainage condition	X	O	O (total head=0)
Unconsolidated condition	X	O	O
Node/surface flow	O	X	O

〈Table. List of applicable load/boundary conditions for each coupled analysis type〉

This chapter will compare the analysis results between the existing one-directional seepage-stress coupled (continuous) analysis and stress seepage fully-coupled analysis under the same conditions. The unique methods that can only be applied for fully-coupled analysis are also examined.

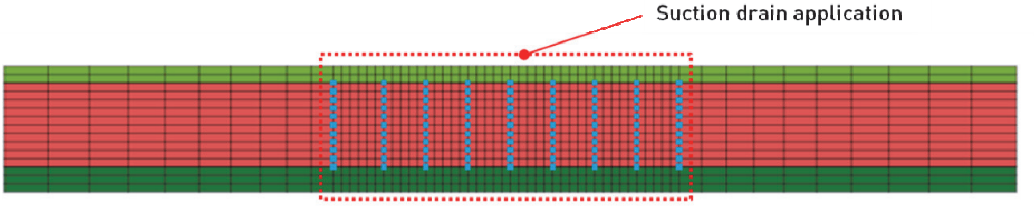


〈Example of ground-ground water couple application〉

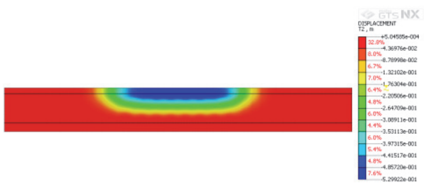
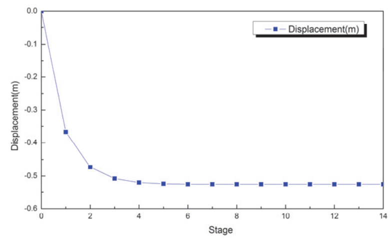
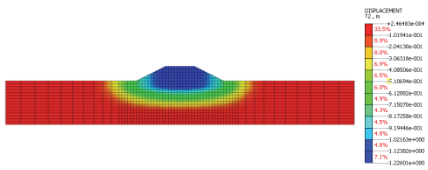
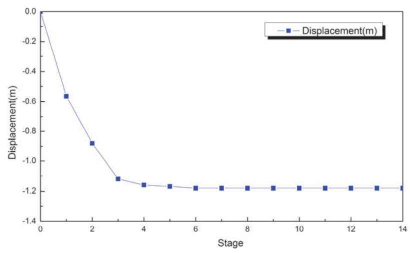
Section 2 Suction-Drain

The suction–drain method, applicable in stress–seepage fully–coupled analysis for soft ground, is examined.

To examine the applicability of special methods, consolidation is performed within the soft ground using the suction drain method, which does not need an embankment, and the settlement and ground stress change according to time is investigated. A uniform atmospheric pressure acts at each depth of the soft layer, such that the loading is applied at once without shear failure due the isotropic consolidation effect. The practical suction pressure is within the range of 50~100 kN/m2, depending on the field condition. By comparing this with the case where an embankment load is also considered, the amount of embankment load that can be replaced by the soft ground condition with applied suction pressure can be found.



[Units : KN, m]

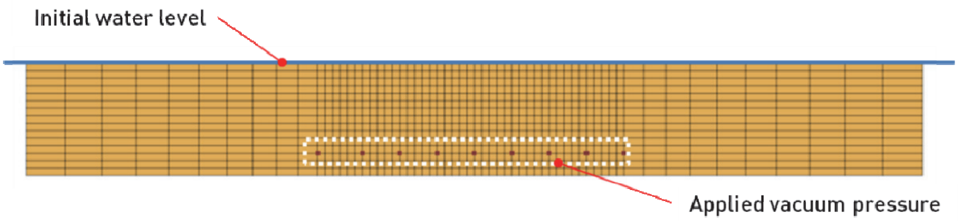
<table>
<tr><th></th><th colspan="2">Suction–Drain application (Pressure = –50 kN/m^2)</th></tr>
<tr><td rowspan="2">DY</td><td></td><td>
</td></tr>
<tr><td colspan="2">Consolidation settlement occurs in the ground due to suction pressure and effective stress increases. The amount of consolidation settlement changes with the size of suction pressure. The converged settlement (approximately 52.5cm) can be verified after a certain period of time.</td></tr>
<tr><th></th><th colspan="2">Embankment load(85kN/m2) + Suction drain (Pressure = –50 kN/m^2)</th></tr>
<tr><td rowspan="2">DY</td><td></td><td>
</td></tr>
<tr><td colspan="2">The final settlement from the simultaneous application of embankment and suction drain is approximately 1.18m. Considering the settlement due to suction, it can be found that approximately 80kN/m2 of the embankment load can be replaced.</td></tr>
</table>

Section 3 Well-Point

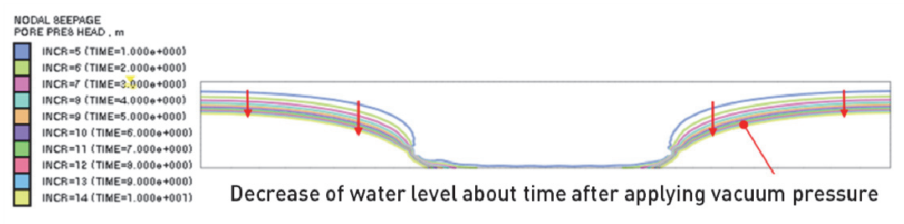
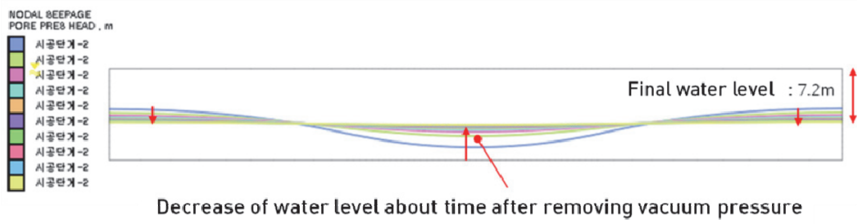
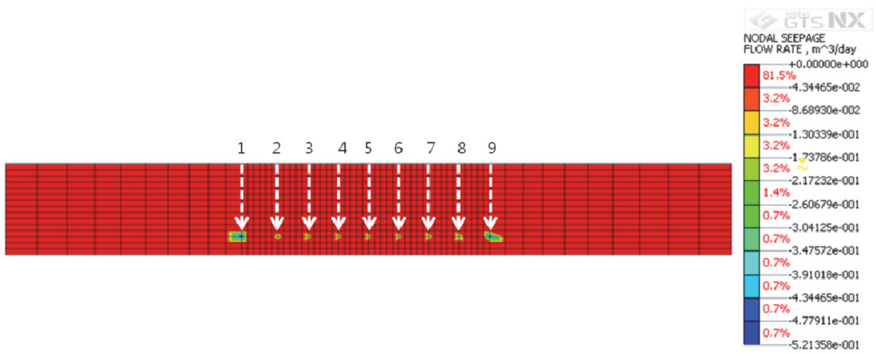
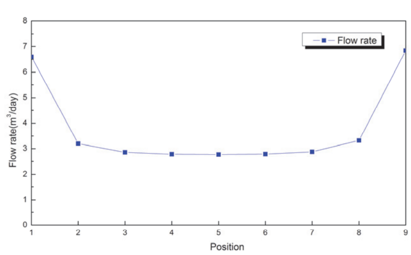
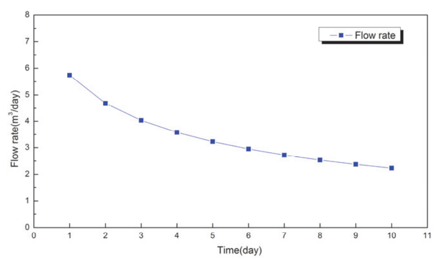
The well-point method, applicable in stress-seepage fully-coupled analysis, is examined.

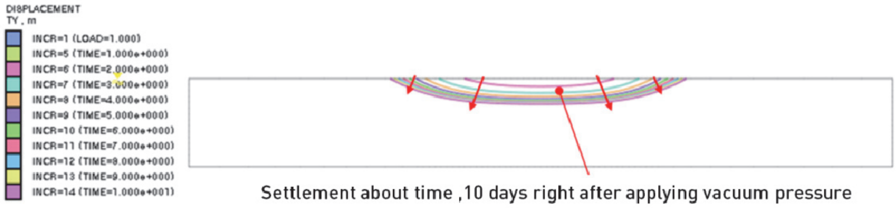
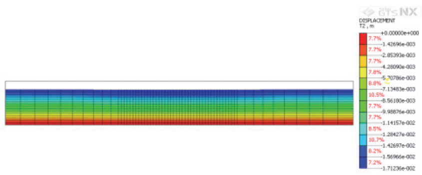
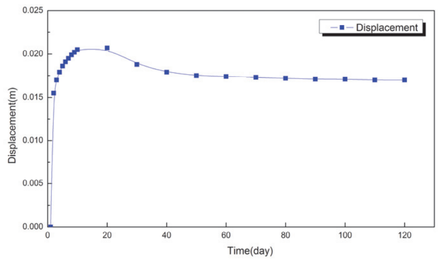
This is a drainage method used to lower the water level when excavating ground with a high ground water level. Well-points are installed at regular intervals around the excavation zone beneath the ground water level, and this is used to increase the effective stress by lowering the ground water level without direct drainage. This method is applied to sand based foundations with high permeability and the water level can be lowered up to 10.3m (atmospheric pressure) theoretically using a vacuum pump. However, considering realistic head loss, the water level is generally lowered by 5~7m.

Because of the forced drainage due to the vacuum pump, the drop in water level with time and settlement trends of the ground can be analyzed simultaneously using the following process:
[Initial water level (in-situ state) → maintain vacuum state for approximately 10 days using the JET PUMP → remove vacuum pressure (review the final water level around the well-point, after the water level has reached equilibrium)]



[Units : KN, m]

<table>
<tr><th></th><th colspan="2">Vacuum pressure (consider head loss) application (Pressure = –70 kN/m^2)</th></tr>
<tr><td rowspan="4">Pore Pressure</td><td colspan="2">
</td></tr>
<tr><td colspan="2">History of water level drop for 10 days after the application of vacuum pressure</td></tr>
<tr><td colspan="2">
</td></tr>
<tr><td colspan="2">Adjacent water level after the removal of vacuum pressure, and water level change with time until equilibrium.
The final drop in water level after equilibrium is approximately 7.2m.</td></tr>
<tr><td rowspan="3">Flow Rate</td><td colspan="2">
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Review of outflow at each well-point position due to vacuum pressure and total outflow change over 10 days for each time interval.
The most outflow occurs at both ends of each position, and the average total outflow at each position is approximately 3.41m3/day.</td></tr>
</table>

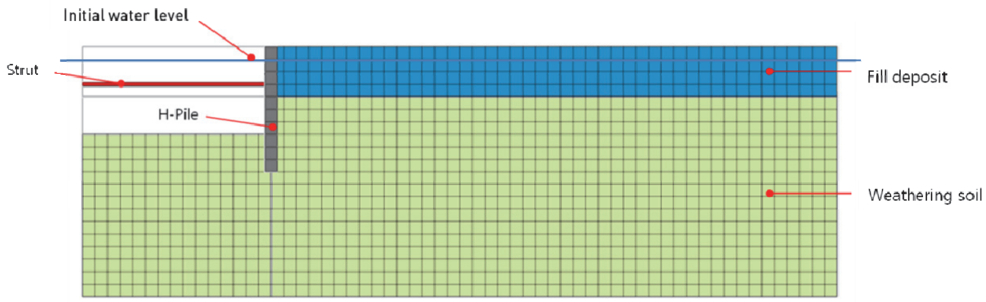
<table>
<tr><td rowspan="2">DY</td><td colspan="2">
</td></tr>
<tr><td></td><td>〈Review position : number 5 〉

</td></tr>
<tr><td></td><td colspan="2">Immediate settlement occurs at the surface of the sand foundation due to force drainage from vacuum pressure.
After a certain time period, the converging settlement for the entire ground surface (approximately 1.71cm) can be verified.</td></tr>
</table>

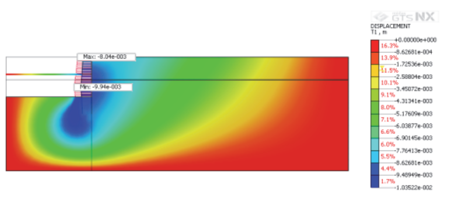
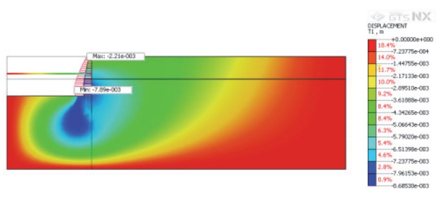
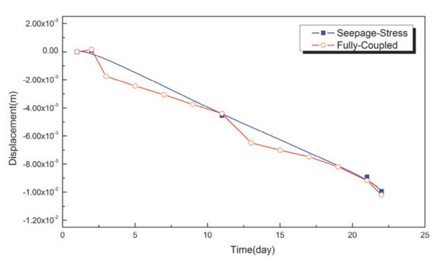
Section 4

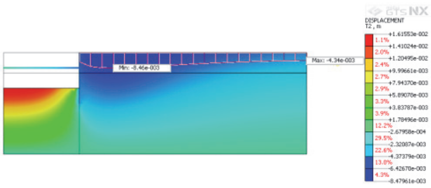
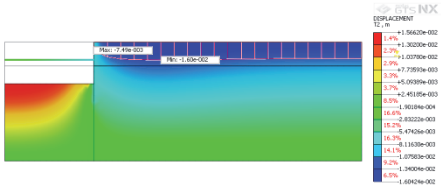
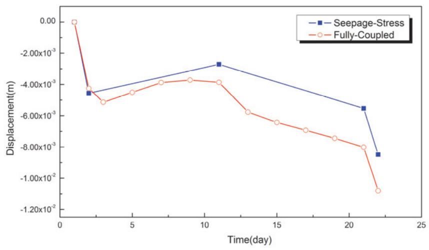
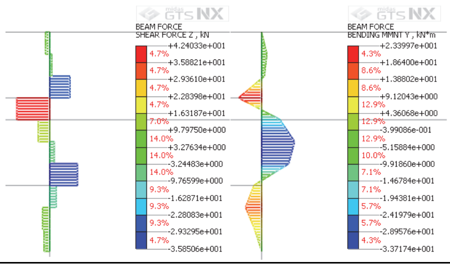
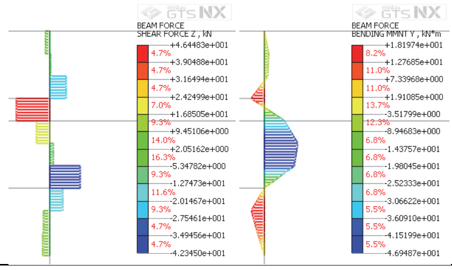
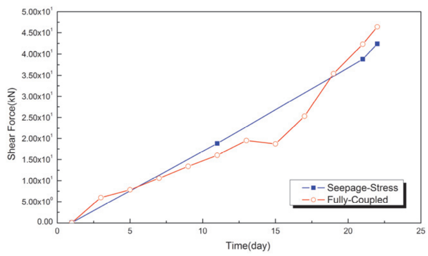
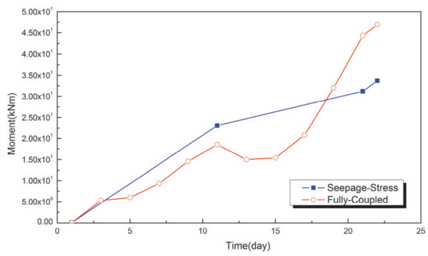
Temporary Structure Excavation Construction Stage
(Ground water level consideration)

Because the existing seepage-stress coupled analysis, which considers the ground water level, is a one-directional semi-coupled analysis where the stress and deformation of the ground does not affect the seepage analysis, the water level and stress change with time within one excavation stage cannot be examined.

This section compares the ground deformation and member force results of the existing one-directional coupled analysis for each excavation stage with the stress seepage fully-coupled analysis under the same conditions and the detailed results for each time step from fully-coupled analysis is verified.



	Seepage stress coupled analysis	Stress seepage fully-coupled analysis
DX (Maximum displacement of wall)		
		Compare maximum displacement of earth retaining wall for each construction stage – Identify the accurate position where maximum displacement occurs – Identify the displacement trends according to time flow within the same excavation stage – Maximum displacement error at final stage : 2.98%

DY (Maximum settlement on back face)		
		Compare settlement trends on back face for each construction stage – Identify the accurate position where maximum settlement occurs – Identify the settlement trends according to excavation stage and time interval – Maximum settlement error at final stage : 21.1%
Shear & Moment (Wall)		
		
	Compare maximum shear force and moment of the wall for each construction stage – Examine the detailed member force changes for each time stage – Maximum member force error at final stage : Shear force (10%), Moment (30%) occurs	

Note Detailed fully-coupled results

Fully-coupled analysis simultaneously considers the initial water level (static head) changes due to drainage and the permeability characteristic changes sue to deformation/consolidation.

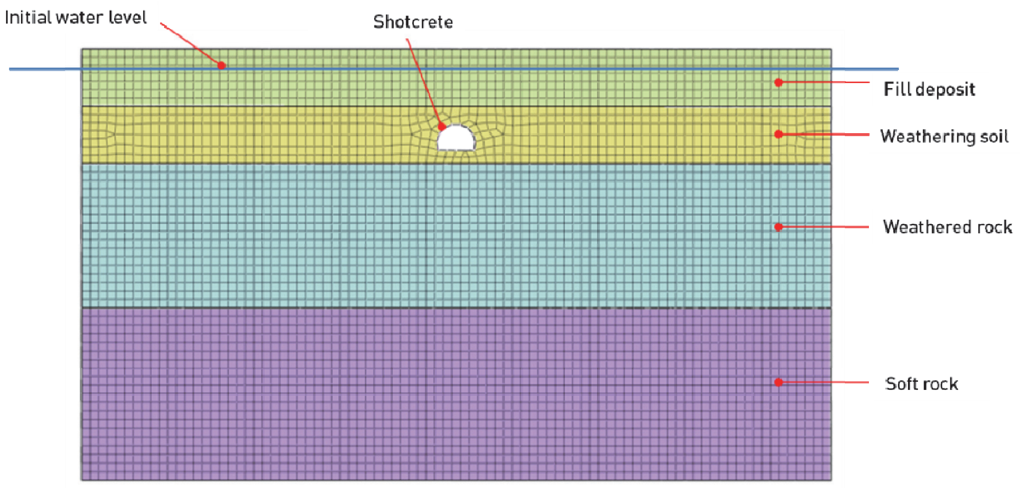
The existing one-directional coupled analysis could only examine one result that is either underestimated or overestimated for each excavation stage, but two-directional coupled analysis can examine the ground deformation with time and changes in structural member forces in detail. In particular, an abandonment stage can be added during or after construction to verify the converged results after water level change and ground deformation is finished. This option was originally only available for consolidation analysis.

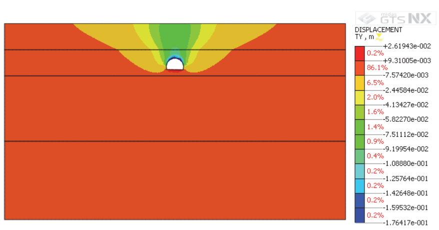
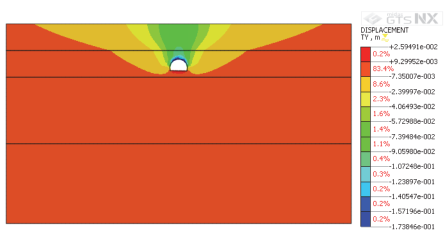
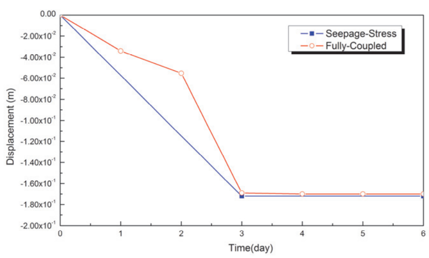
Section 5

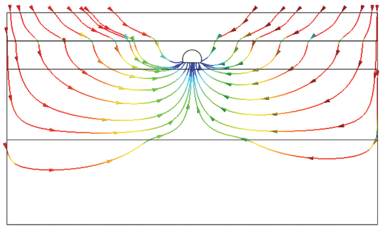
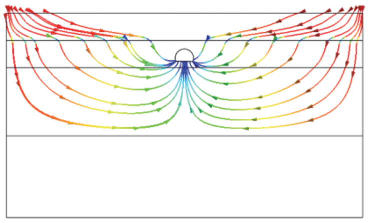
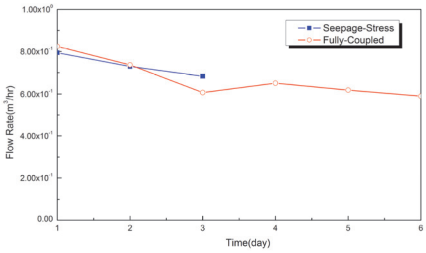
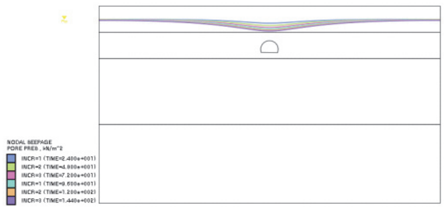
Tunnel Excavation Construction Stage (Ground water level consideration)

This section compares the water level change and tunnel crown settlement results of the existing tunnel excavation stage seepage-stress coupled analysis with the stress seepage fully-coupled analysis under the same conditions and detailed results for each time step from fully-coupled analysis is verified.

In particular, the change in flow rate on the tunnel face during tunnel excavation construction after shotcrete pouring is examined using fully-coupled analysis.



	Seepage-stress coupled analysis	Stress seepage fully-coupled analysis
DY (Crown displacement)		
		Comparison of tunnel crown settlement for each construction stage - Identify the crown displacement history according to time flow within an excavation stage. - Detailed examination of settlement according to water level change after finishing excavation is possible.

<table>
<tr><td rowspan="2">Flow Rate</td><td></td><td></td></tr>
<tr><td>
</td><td>Comparison of tunnel face flow rate for each construction stage
– Reduction in flow rate, due to the decrease in void ratio and permeability characteristics caused by ground deformation during construction.
– Detailed examination of flow rate changes with time after excavation has finished
– Average flow rate per unit time error is approximately 8.7%</td></tr>
<tr><td rowspan="2">Pore Pressure</td><td></td><td></td></tr>
<tr><td colspan="2">Change in initial water level due to tunnel excavation for each construction stage
– Identify detailed history of water level change for each time stage after or during excavation (abandonment period applicable)</td></tr>
</table>

Note Stress seepage fully-coupled characteristics

✓ As the water level changes with time, the degree of saturation within the ground changes and affects the self weight and stiffness. The ground deformation that occurs here affects the void ratio (coefficient of permeability) and redistributes the pore pressure.

✓ It can analyze all phenomena and characteristics of ground behavior due to the water level, and can apply various previously irreproducible special methods.

✓ The construction period, abandonment period etc., which was only mainly considered in consolidation analysis, is now reflected for all stress (construction stage) analyses to examine the detailed stress history of the ground during or after actual construction.

Stress seepage fully-coupled analysis can analyze the effects of the water level on ground behavior and the results that reflect actual field conditions, construction period, state etc. can be applied to design.

New eXperience of
GeoTechnical analysis System

Chapter 4
Consolidation

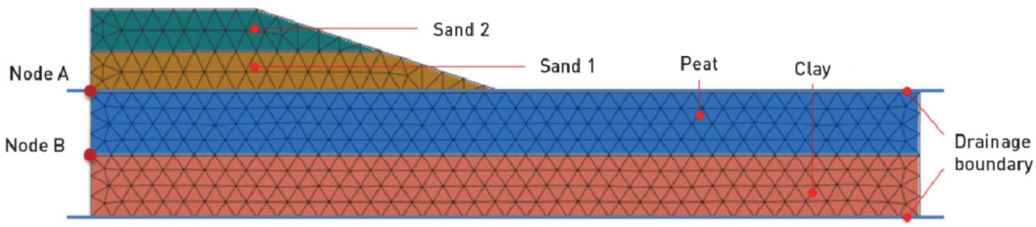
TABLE OF CONTENTS

Section 1

Soft Ground Embankment Consolidation (2D)

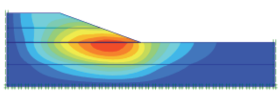
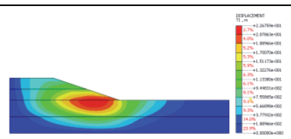
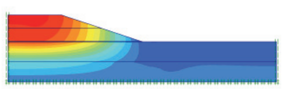
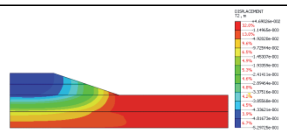
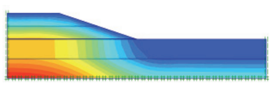
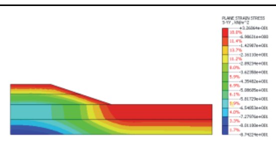
Element characteristics	Ground –Plane Strain
Number of nodes, elements	1,633 nodes, 768 elements
Construction stage	5 stages

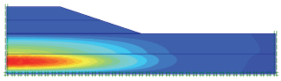
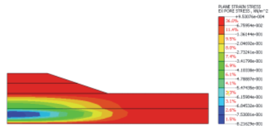
The important items of interest for construction on soft ground are the settlement of the ground and lateral displacement, horizontal displacement distribution with depth and size of excess pore pressure. For embankments that consider the dissipation speed, the safety of the ground against shear failure and the applicability of additional embankments at an arbitrary point are of interest. This model examines the settlement and excess pore pressure distribution of the ground when an embankment is applied on soft ground.

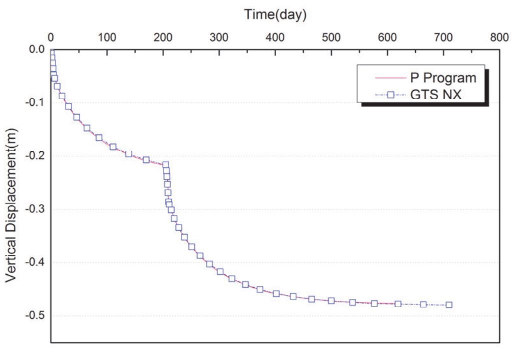


▶Result comparison (P Program–GTS NX)

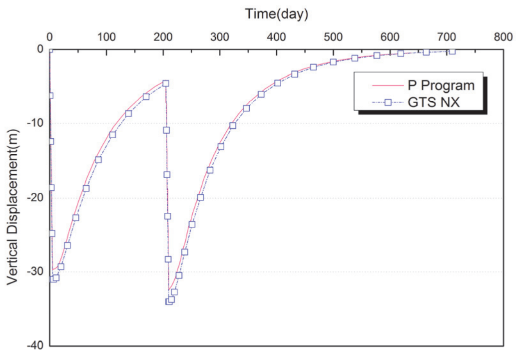
[Units : KN, m]

	P Program	GTS NX	Error Ratio(Value)
	min/max	min/max	min/max
DX	2.1682e-1	2.26759e-1	4.58% (9.49e-3)
			
DY	-5.3246e-1	-5.29725e-1	0.51% (2.74e-3)
			
S'-YY	-8.731e1	-8.74224e1	0.13% (-1.12e-1)
			

Excess. Pore Pressure	-8.4787e-1	-8.21629e-1	3.09% (2.62e-2)
			



[Settlement with time at position A]



[Excess pore pressure with time at position B]

Note Characteristics of consolidation analysis

Consolidation analysis

This analysis method calculates the dissipation of excess pore pressure, caused by instant undrained behavior of the pore water as it resists external loading, with time.

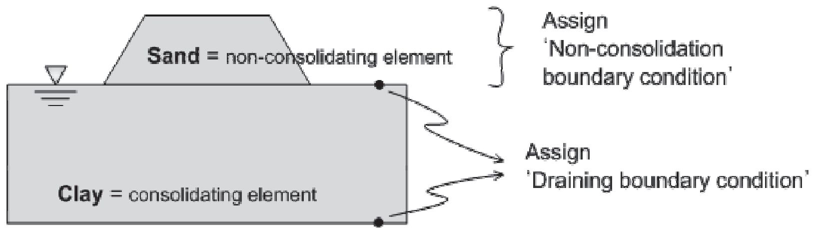
- ✓ Pore water within the ground with a small coefficient of infiltration displays identical behavior as the undraining condition instantaneously
- ✓ Hence, the excess pore pressure bears most of the compression load, depending on the change in load state
- ✓ However, this excess pore pressure is redistributed by the seepage phenomena as time passes. Especially when a drainage boundary is present, the excess pore pressure decreases with time
- ✓ Following this, the load borne by the excess pore pressure is gradually borne by the volume of the soil. This causes a gradual deformation of the soil, which increases the effective stress
- ✓ The ground deformation, due to the increase in effective stress, is accumulated in the gravitational direction and settlement occurs in this direction with time. This increase in deformation causes the settlement of structural foundations

  ➔ The effects of differential settlement etc. that occur at the structural foundation greatly affects the stability of the structure

Consolidation element

This element is a nodal DOF that has an additional excess pore pressure DOF along with the displacement DOF.

- ✓ In consolidation analysis, all elements which are not specified as non–consolidating elements are assumed to have a DOF for pore pressure
- ✓ For elements that do not display direct consolidation behavior, such as embankments, the non–consolidation condition needs to be defined such that they are applied as a general structural elements
- ✓ Draining conditions need to be defined for consolidation elements at the boundary where drainage occurs
- ✓ When the boundary condition is defined and consolidation analysis is conducted correctly, the excess pore pressure at the parts where the non–consolidation conditions and draining conditions are applied is output as '0'



[Boundary conditions of condolidation elements]

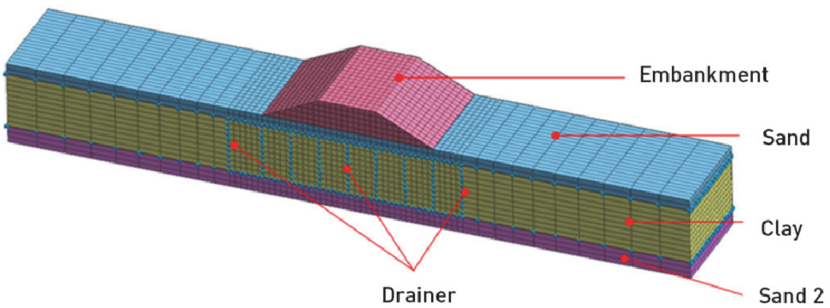
Section 2 Consolidation following Drain Spacing (3D)

Element characteristics	Ground – Solid
Number of nodes, elements	20,843 nodes, 19,475 elements
Construction stage	5 stages

This is a 3D consolidation analysis model with an applied Modified Cam Clay material model.

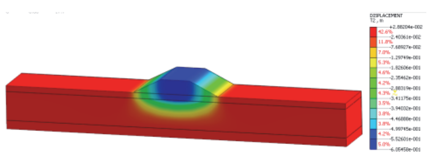
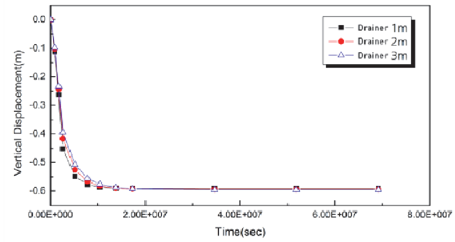
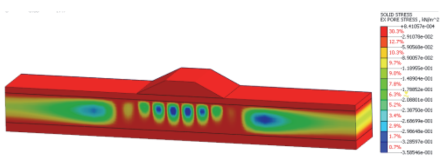
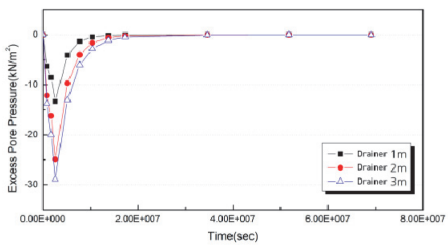
[Construction stage (5 stages)] in-situ state → embankment(30 days) → Abandonment 1(90 days) → Abandonment 2(80 days) → Abandonment 3(1,800 days)

The ground settlement and excess pore pressure distribution with time is examined when the drain spacing in the longitudinal direction is 1m, 2m and 3m.



[Units : KN, m]

▶3D consolidation analysis results

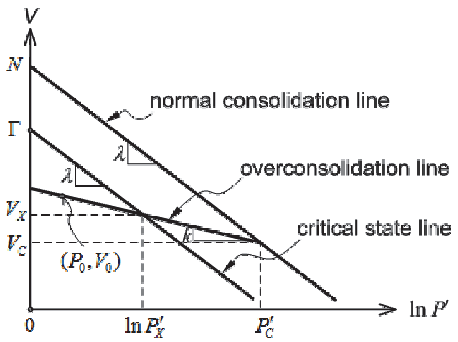
<table>
<tr><th></th><th colspan="2">GTS NX</th></tr>
<tr><td rowspan="2">DY</td><td></td><td>
</td></tr>
<tr><td colspan="2">The time to reach the maximum settlement gets shorter with smaller drain spacing, and the maximum settlement is similar regardless of the drain spacing</td></tr>
<tr><td rowspan="2">Excess. Pore Pressure</td><td></td><td>
</td></tr>
<tr><td colspan="2">The excess pore pressure is dissipated faster with smaller drain spacing, and it is all dissipated at the final stage</td></tr>
</table>

Note Modified Cam Clay

Modified Cam Clay model

Often used to model clay material

- ✓ Specification of initial void ratio, initial stress, pre-consolidation pressure needed
- ✓ In-situ stress distribution calculated from stress distribution due to the current load using the OCR value
- ✓ Because the stress at each depth is calculated from the input OCR, the initial stress can be underestimated than the initial stress of the actual ground and so, the Pc (Pre-consolidation pressure) value can be defined directly
- ✓ When both OCR and Pc are set, the Pc value is reflected in the analysis on a preferential basis
- ✓ When Pc is input, the solver checks whether the input Pc and in-situ stress state satisfy the failure function. If is does satisfy, the input Pc is used; if not, Pc is recalculated



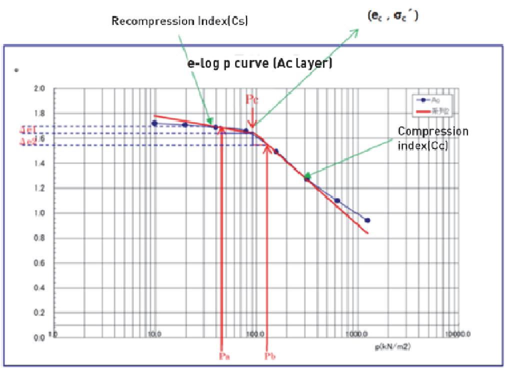
[Volume-pressure relationship]

[MCC input material property window]

Symbol	Significance
κ	Gradient of overconsolidation line
λ	Gradient of normal consolidation line
M	Gradient of critical state line

- ✓ The compression index C_c and recompression index C_s can be obtained from the void ratio (e), found from 1D consolidation test, against $\log_{10}(p)$ graph and the gradients of the normal consolidation line and overconsolidation line are estimated from the following equations.

$$\lambda = \frac{C_c}{2.303} \quad , \quad \kappa = \frac{C_s}{2.303}$$



⟨e–log P curve⟩

✓ The gradient of the critical state line M is estimated from the relationship between the effective shear resistance angle (shear resistance angle from drained experiments)

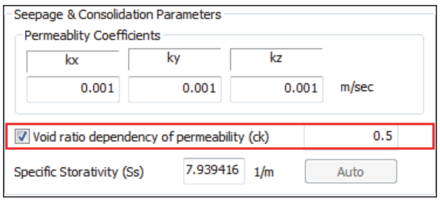
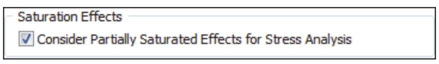
$$M = \frac{6\sin\phi}{3-\sin\phi}$$

here, ϕ : internal friction angle obtained from triaxial compression test

Section 3 Embankment Consolidation (Detailed consideration of ground characteristics (partial saturation))

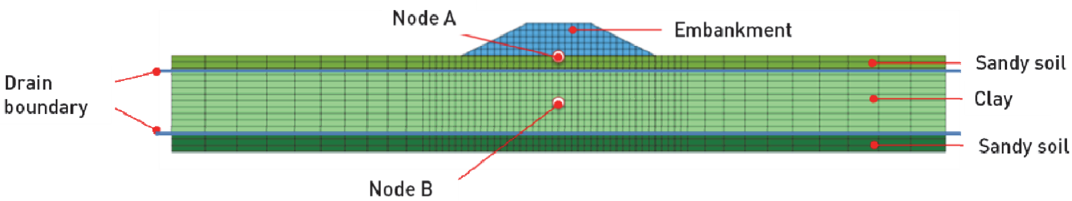
Like the cases in section 1,2, soft ground consolidation analysis uses the initial water level condition and drainage condition to examine the settlement and degree of consolidation due to embankment and loading at each stage, and this corresponds to the seepage-stress two-directional coupled analysis. In other words, like for stress seepage fully-coupled analysis, the ground deformation due to consolidation affects the void ratio and permeability characteristics and this in turn affects the consolidation settlement.

This section compares/examines the consolidation analysis results where the detailed ground characteristics (void ratio dependent permeability ratio, partial saturation consideration option) are applied and not applied.

<table>
<tr><th></th><th>Detailed ground characteristics</th><th>Description</th></tr>
<tr><td rowspan="2">ck
(void ratio dependent permeability ratio)</td><td>
</td><td>Change in coefficient of infiltration due to change in void ratio
– The decrease in void ratio, due to the pressure applied on the ground, decreases the permeability
$k = 10^{(\Delta e / c_k)} k_{sat}$</td></tr>
<tr><td colspan="2">The permeability characteristics decrease with consolidation, and the effects of increased settlement time and consolidation period are considered</td></tr>
<tr><td rowspan="2">Partial degree of saturation</td><td>
</td><td>부분 포화 사용시 지반의 밀도
$\rho = (1-S_e)\rho_{unsat} + S_e\rho_{sat}$
부분 포화 사용하지 않을 경우 지반의 밀도
$\rho = \begin{cases} \rho_{unsat} & (p \le 0) \\ \rho_{sat} & (p > 0) \end{cases}$</td></tr>
<tr><td colspan="2">Detailed consideration of changing ground load, undrained stiffness and stress change</td></tr>
</table>

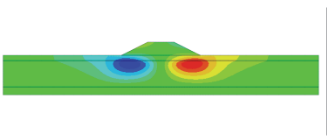
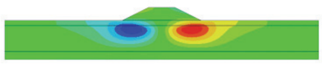
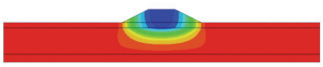
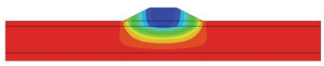
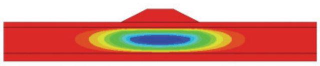
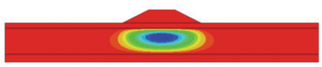
This is a 2D consolidation analysis model with an applied Modified Cam Clay material model.
[Construction stage (5 stages)] in–situ state → embankment(30 days) → Abandonment 1(90 days) → Abandonment 2(80 days) → Abandonment 3(1,800 days)

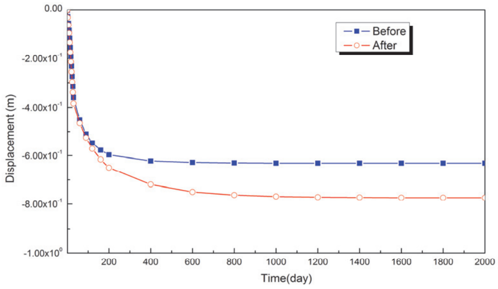
Element characteristics	Ground – Plane Strain
Number of nodes, elements	1,097 nodes, 1025 elements
Construction stage	5 stages



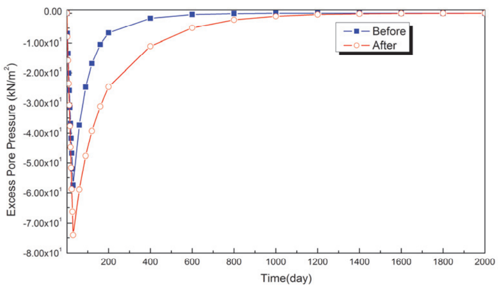
▶ Result comparison after consolidation (before/after applying option)

[Units : KN, m]

	Option application(before)	Option application(after)	Error Ratio(Value)
	max	max	max
DX	1.96775e-1	2.52079e-1	21.9% (5.53e-2)
			
DY	-6.32926e-1	-7.77969e-1	18.6% (-1.45e-1)
			
Excessive Pore Pressure	-1.68e-5	-1.94e-2	- (-1.94e-2)
			



[Settlement with time at position A]



[Excess pore pressure with time at position B]

The water level is one of the most important factors that affect ground behavior analysis. As settlement progresses, the void ratio (permeability characteristics) decrease and this in turn affects the redistribution of excess pore pressure, changing the self weight and ground stiffness with time. All these phenomena are considered in one stage, and a final settlement and consolidation period close to the approximate solution can be computed.

New eXperience of
GeoTechnical analysis System

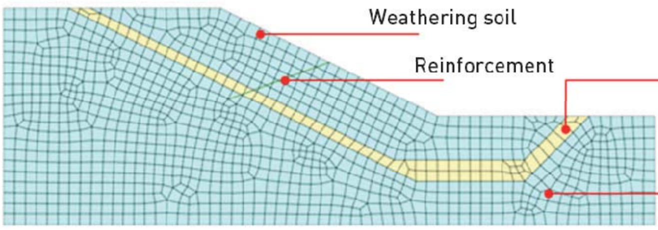
Chapter 5
Slope Stability

TABLE OF CONTENTS

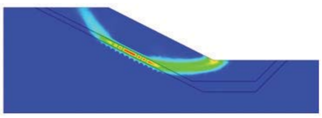
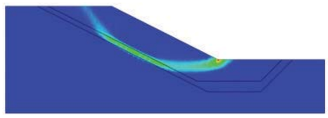
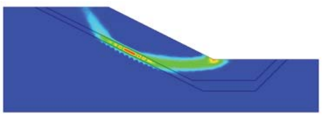
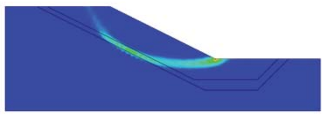
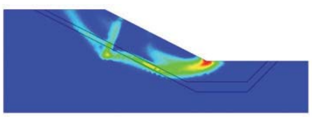
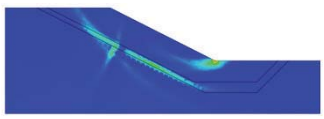
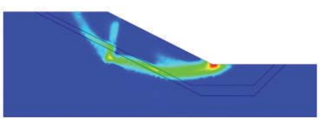
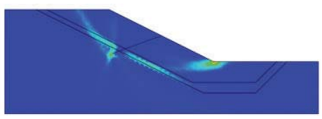
Section 1 Slope Stability (2D)

Element characteristics	Ground –Plane Strain, Reinforcement – Embedded Truss
Number of nodes, elements	3,317 nodes, 1,062 elements
Analysis method	Strength Reduction Method (SRM)

Slope stability analysis requires the analysis of slope failure behavior and the calculation of a minimum safety factor for safety management. This model performs the strength reduction method on a uniform slope that includes a weak layer. The safety factor can be verified from the analysis results and the failure behavior aspect can be checked from the contour for maximum shear strain.



▶Result comparison(GTS–GTS NX)

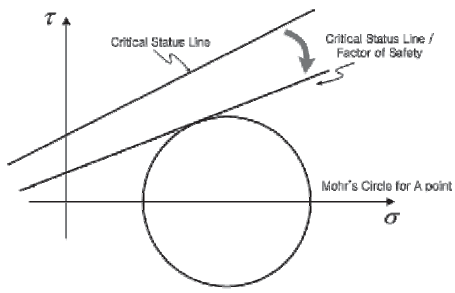
	GTS440 Factor of Safety	GTS NX Factor of Safety	Factor of safety difference
During dry season	1.8750 	1.87695 	0.00195
During wet season	1.0625 	1.07734 	0.01484
During dry season -reinforced	2.0625 	2.07311 	0.01061
During wet season -reinforced	1.1875 	1.19162 	0.00412

Note Slope stability analysis method

GTS NX provides two slope stability analysis methods using the finite element method; the strength reduction method and stress analysis method based on the limit equilibrium method.

Strength Reduction Method (SRM)

The strength reduction method gradually decreases the shear strength (c, ϕ) and the analysis is conducted until the calculation does not converge. At this point, the slope is considered to have failed and the maximum strength decrease rate at this point is assumed to be the minimum safety factor of the slope. This method takes a long time because of nonlinear analysis has to be conducted multiple times, but more accurate results can be obtained and the deformation process from the initial slope to failure can be examined without assuming the failure envelope in advance.



[Strength reduction method]

- ✓ The material models used in SRM are Mohr Coulomb, Drucker Prager and Modified Mohr Coulomb.
- ✓ The input variables used here are assumed to have a uniform value except for the cohesion, friction angle and dilatency angle which determine shear failure, cohesion, friction angle and dilatency angle corresponding to ground elements (plane strain, axisymmetric, solid) are gradually decreased until slope failure. The factor of safety F_s is computed at this point.

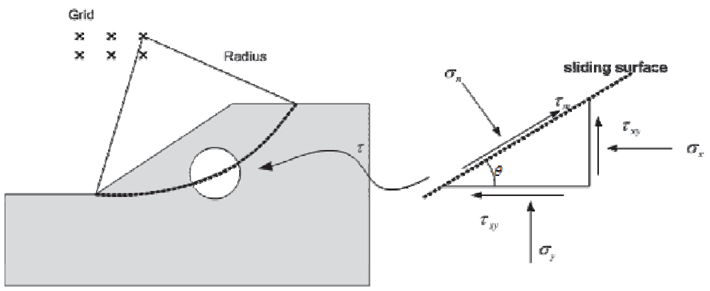
Stress Analysis Method (SAM) based on the limit equilibrium method

The limit equilibrium method is the most widely used slope stability analysis method in actual design; it cannot find the stress history of the actual slope or the change in ground behavior.

On the other hand, slope stability analysis using the finite element method can consider the slope formation process or other ground properties, but it takes longer for the analysis to run because multiple nonlinear analyses need to be performed.

Recently, much research has been conducted to take advantage of both the limit equilibrium method and slope stability analysis based on the finite element method. GTS NX provides a slope stability analysis method that uses the finite element stress analysis results. This method is based on the virtual slip surface concept of the limit equilibrium method and the stress results from stress analysis. In other words, the stress results from stress analysis are used to compute the factor of safety for multiple assumed virtual slip surfaces. The minimum value and the critical surface at this point is computed.

- ✓ The material models used in SAM are the same as that of SRM; Mohr Coulomb, Drucker Prager and Modified Mohr Coulomb.

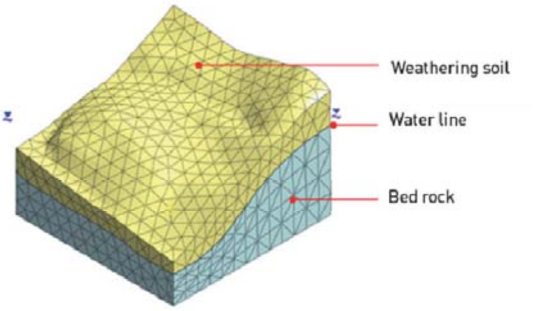


[Stress analysis method]

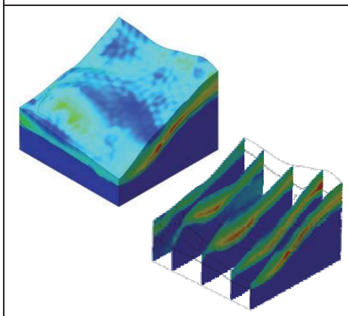
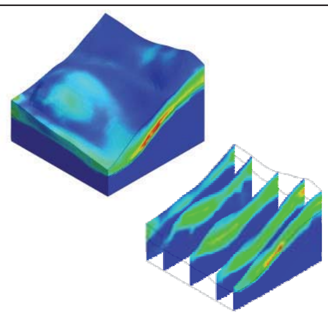
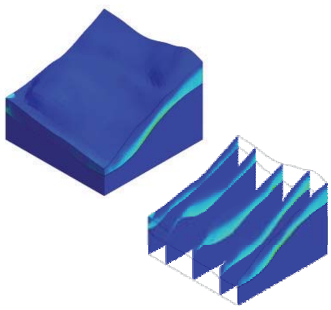
Section 2 Slope Stability (3D)

Element characteristics	Ground - Solid
Number of nodes, elements	8,968 nodes, 5,668 elements
Analysis method	Strength Reduction Method (SRM)

Analyzing a single slope section using 2D analysis has limitations in analyzing the 3D slope behavior. The largest difference between 2D analysis and 3D analysis is the reflection of elements that affect slope activity such as the shape of the ground surface and slip surface, ground material property distribution, strength of slip surface etc. This model uses 3D slope stability analysis to examine the position where the shear strain is concentrated.



▶Result comparison(GTS–GTS NX)

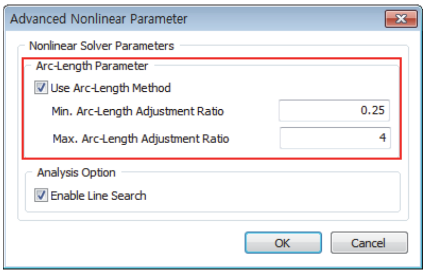
	GTS440 Factor of Safety	GTS NX Factor of Safety	Factor of safety difference
During dry season	1.9875 	2.0423 	0.0548
During wet season	- X	0.9974 	The water level defined by a spatial function could not be applied for 3D slope stability analysis in the existing GTS440

Note Strength reduction method using arc–length

Arc–length method

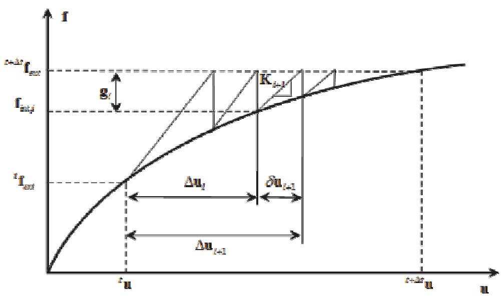
The main difference between SRM using the arc length method and the existing STM is the method for increasing or decreasing the factor of safety.

➢ Location : Analysis 〉 Analysis case 〉 Analysis control 〉 SRM 〉 Advanced nonlinear parameter

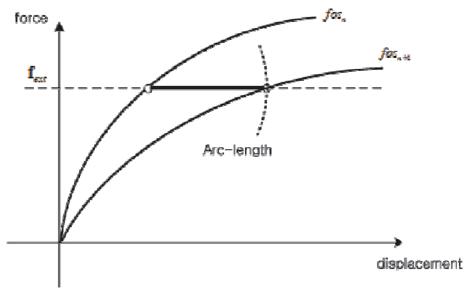


✓ SRM computes the factor of satiety of the next stage by controlling the factor of safety by the user defined increment. Hence, a uniform increment is applied to both very stable models and very unstable models and inefficient calculations are conducted without the engineer's judgment.

✓ However, the arc–length method computes the arc–length based on the convergence speed of the previous stage so that a more appropriate factor of safety increment can be obtained.



[Convergence procedure for nonlinear FEM]



[Factor of safety increment due to arc–length]

New eXperience of
GeoTechnical analysis System

Chapter 6
Dynamic

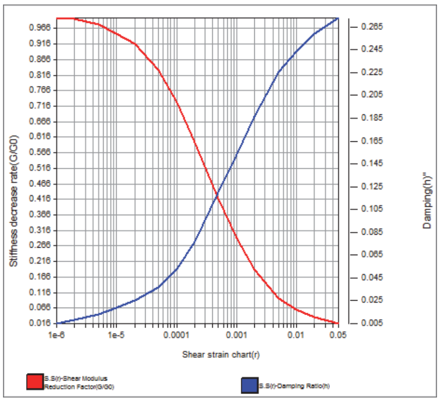
TABLE OF CONTENTS

Section 1 1D Ground Response

Finding the ground response to an input earthquake for the ground state before construction is called free field analysis. Free field analysis is based on the assumption that all strata boundaries are horizontal and that the ground response is ruled by the horizontal shear wave propagated vertically from the bedrock. This model conducts a primary analysis using the bedrock as the entire ground, and then conducts a secondary analysis with the in-situ condition to investigate the maximum acceleration of the ground.

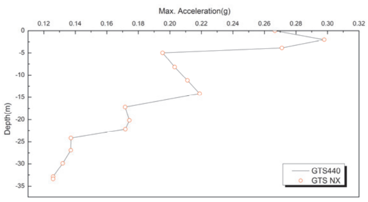
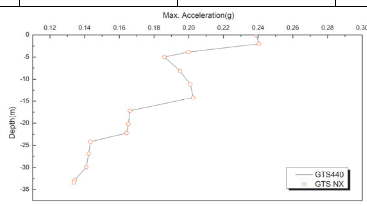
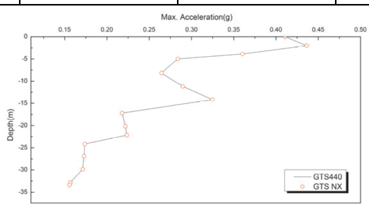
▶Stratum modeling
▶▶ Strain suitability characteristic curve

Layer No.	Name	Thickness (m)	Unit Weight (kN/m³)	Maximum Shear Wave Velocity (m/sec)	Depth (m)
1	Burried layer1	2	18	155	2
2	Burried layer2	1.88	18	155	3.88
3	Clay	1.1	18	155	4.98
4	Weathering soil1	3.2	19	390	8.18
5	Weathering soil2	3	19	390	11.2
6	Weathering soil3	3	19	390	14.2
7	Weathering soil4	3	19	390	17.2
8	Weathered rock1	3	20	640	20.2
9	Weathered rock2	2	20	640	22.2
10	Weathered rock3	2	20	640	24.2
11	Soft rock1	2.7	24	980	26.9
12	Soft rock2	3	24	980	29.9
13	Soft rock3	3	24	980	32.9
14	Hard rock1	0.5	26	1.2e+003	33.4
15	Hard rock2	1e-006	26	1.2e+003	33.4



[Units : g]

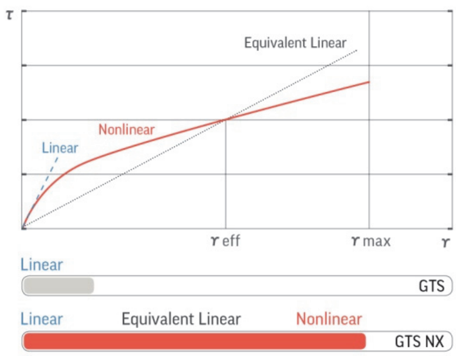
▶ Result comparison of maximum acceleration according to depth (GTS-GTS NX)

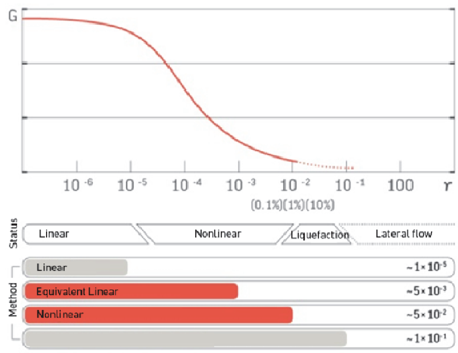
	GTS440		GTS NX		Error Ratio(Value)	
	max	max	min	max	min	max
Long period wave	0.1259	0.2978	0.1259	0.2978	0.00% (0)	0.00% (0)
						
Short period wave	0.2406	0.1339	0.2404	0.1338	0.08% (-2.00e-4)	0.07% (-1.00e-4)
						
Artificial earthquake wave	0.4363	0.1557	0.4363	0.1558	0.00% (0)	0.06% (1.00e-4)
						

Note Understanding dynamic analysis

The methods for performing dynamic analysis can be classified into linear dynamic analysis, equivalent linear analysis and nonlinear dynamic analysis. GTS NX supports linear ~ nonlinear dynamic analysis.

Analysis method		Characteristics
linear dynamic analysis	Response spectrum Time history analysis(linear)	✓ Effective for very small strains when the ground stiffness is very large
equivalent linear analysis	1D free field analysis 2D equivalent linear analysis	✓ Frequency domain analysis method that uses the secant elastic modulus corresponding to effective shear strain and has a fast analysis speed ✓ The ground nonlinearity is small when the ground stiffness is large or the earthquake wave energy is small, so a rational result can be obtained effectively
nonlinear dynamic analysis	Time history analysis (nonlinear)	✓ Can consider the creation and dissipation of pore pressure using time history analysis that can fully simulate the nonlinearity of the ground ✓ If a large response occurs because the ground deformation is very large or due to resonance, nonlinear analysis is needed





▶ Stress-strain relationship
▶▶ Applicable range of strain according to dynamic analysis

The dynamic analyses and applicable fields supported in GTS NX are as follows:

Response spectrum analysis

- Method to obtain the structural response using the response spectrum for the maximum displacement, velocity, acceleration etc. of the structure for each load in the single DOF system
- This is an approximate method and has some error compared to time history analysis, but it can conveniently find the dynamic properties of the structure when large structure/detailed results are not wanted

Linear time history analysis (mode/direct)

- Method uses the dynamic equilibrium equation solution to calculate the structural behavior(displacement, member force etc.) at an arbitrary time, using the dynamic properties of the structure and applied load

- Modal superposition method : the response of each mode according to the input earthquake wave is found, then superpose
- Direct integration : the dynamic equilibrium equation for the total DOF is gradually integrated with time to find the solution

Class	Direct integration	Modal superposition method
Analysis time	Analysis time is long	Analysis time is short
Important points	Selecting the time step is important	Selecting the number of modes is important
Model size	Appropriate for small models	Appropriate for large models
Analysis accuracy	Long analysis time taken, but relatively high accuracy	Accuracy depends on number of selected modes

- For ground seismic analysis, it can be divided into time history analysis which uses long period/short period/artificial earthquake waves and response spectrum analysis which follows the design performance level. The simple response spectrum analysis is used more often than time history analysis because of structure shape/ground condition/importance etc.

1D Free Field Analysis

- Used to find the ground response to an input earthquake for the ground state before construction
- Based on the assumption that all strata boundaries are horizontal and that the ground response is ruled by the horizontal shear wave propagated vertically from the bedrock
- Often used to predict the vibrations on the ground surface to determine the design response spectrum, to compute the dynamic stress and strain for liquefaction evaluation and to determine the earthquake load which causes instability of the ground or earthen structures

2D Equivalent Linear Analysis

- The continuous change in shear strain can be considered depending on the size of vibration load and the radiation damping phenomena due to the infinite ground is simulated
- Soil–structure interaction (SSI) is also supported, not just for the ground
- Often used for detailed seismic evaluation of underground structures(subway etc.)

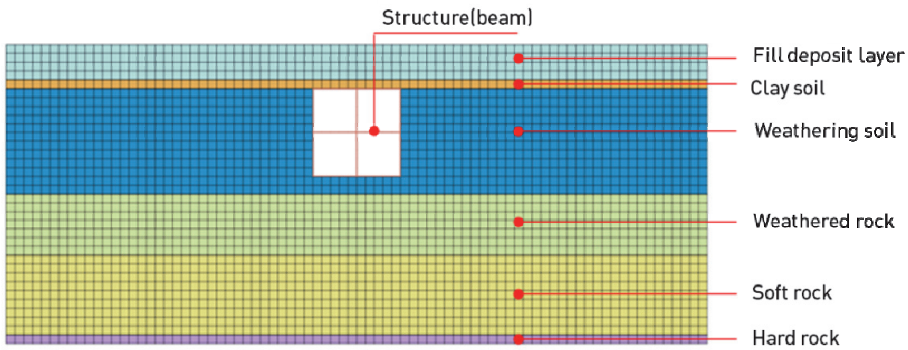
Nonlinear Time History Analysis

- Most grounds have nonlinear material properties and thus the dynamic response of the ground can be accurately simulated using time history analysis
- Used for detailed seismic evaluation of nuclear power plants/plants/civil structures

Section 2 2D Equivalent Linear

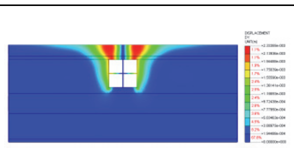
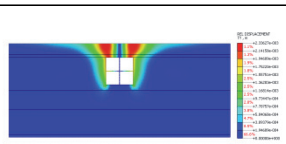
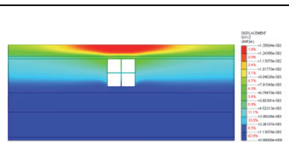
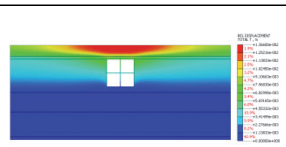
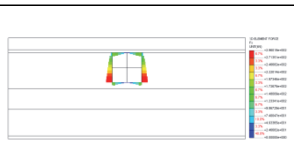
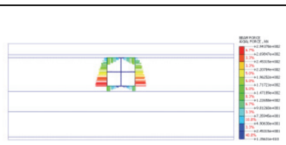
Element characteristics	Ground – 2D equivalent linear, Structure –Beam
Number of nodes, elements	2,835 nodes, 2,780 elements
Analysis method	2D equivalent linear analysis

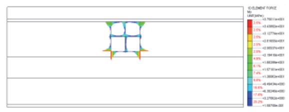
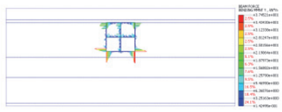
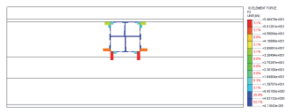
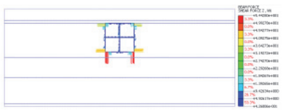
When a structure exists within the ground, 2D equivalent linear analysis needs to be conducted to assess the ground–structure interaction. The shear strain of the ground continuously changes due to time history load such as earthquakes and the increase in shear strain shows decreasing trends in shear modulus and increasing trends in damping ratio. This model adds a concrete structure in the weathered soil layer to analyze the ground/structure seismic response result using 2D equivalent linear analysis to evaluate the structural stability.



▶ Result comparison (GTS–GTS NX)

[Units : KN, m]

	GTS440	GTS NX	Error Ratio(Value)
	min/max	min/max	min/max
DY	2.33385e-3 	2.33627e-3 	0.10% (2.42e-6)
DXY	1.35694e-2 	1.36600e-2 	0.67% (9.06e-5
Beam Fx	2.96019e2 	2.94378e2 	0.55% (-1.64e0)

Beam My	3.75011e1 	3.74521e1 	0.13% (-4.90e-2)
Beam Fz	5.46478e1 	5.44280e1 	0.40% (-2.20e-1))

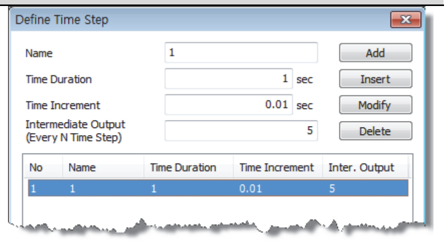
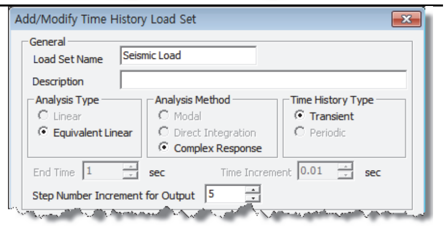
Note 2D equivalent linear analysis setting

✓ To perform 2D equivalent linear analysis, the analysis method needs to be set as [2D equivalent linear]. Also, the ground material properties also need to be defined using 2D equivalent material properties (unit weight, shear modulus, Poisson's ratio, damping ratio, strain suitability characteristics).

Class	Analysis method setting	Description
GTS NX	2D equivalent linear	1. Analysis 〉 Analysis setting 〉 Model type is set as a 2D model 2. Ground material properties defined as Dynamic analysis 〉 Material 〉 2D equivalent 3. Select Analysis 〉 Analysis case 〉 Add 〉 2D equivalent linear analysis
GTS440	2D equivalent linear	1. File 〉 Project setting 〉 Model type is set as 2D equivalent linear analysis 2. Ground material properties defined as Material 〉 Properties 〉 Properties 〉 Plane 2. Select Analysis 〉 Analysis case 〉 Add 〉 2D equivalent linear analysis

Note Time step setting

✓ The dynamic analysis results are output by the user defined time increment, end time, number of output increment steps.
This can be checked at Result analysis 〉 Special post-processing 〉 Dynamic analysis 〉 Graph.

Class	Time step setting	
GTS NX	Define target time step in Analysis 〉 Analysis case 〉 Add 〉 Time step	
GTS440	Set the number of output increment steps in Model 〉 Load 〉 Time history data 〉 Time history load set (The time increment is fixed as 0.01 seconds, and the end time is output by internal calculations)	

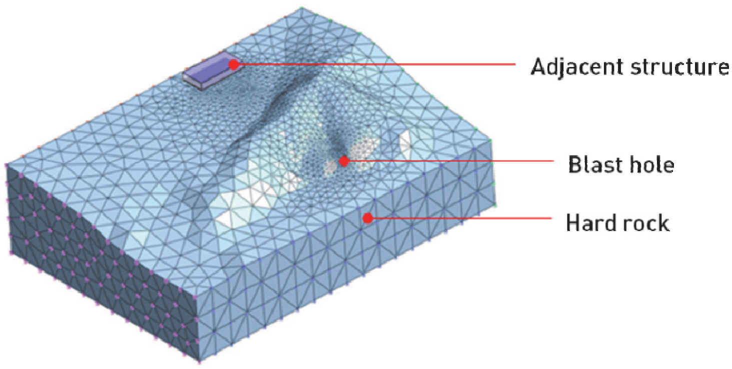
✓ Because 2D equivalent linear analysis transforms the history function into the frequency domain for analysis, and this is re-expressed in the time domain, it is recommended that a uniform time increment be set when defining the time history function (earthquake data) and that this increment be the same as the time defined in the time history function.

Section 3

Blasting Dynamics (3D)

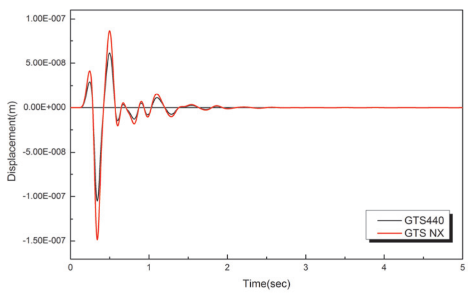
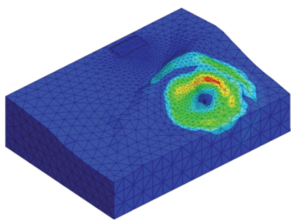
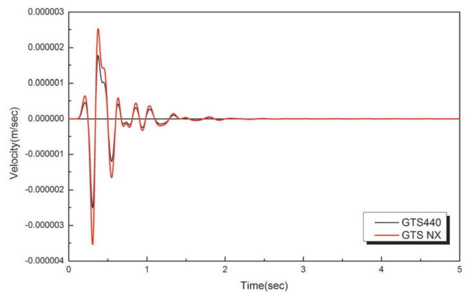
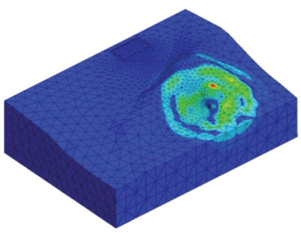
Element characteristics	Ground – Solid
Number of nodes, elements	3,621 nodes, 17,286 elements
Analysis method	Time history analysis (direct)

Because blasting the ground is accompanied by pollution such as vibration and noise, the stability evaluation against blast vibration is very important when a structure is located nearby. This model applies a mesoscale vibration control blast according to the standard blasting method on a ground composed of hard rock. The displacement and velocity of adjacent structures are examined with respect to time.



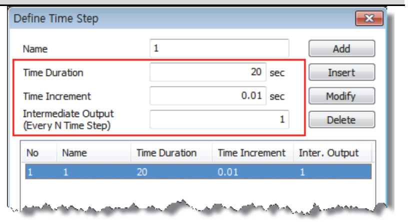
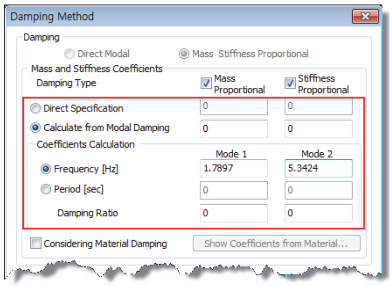
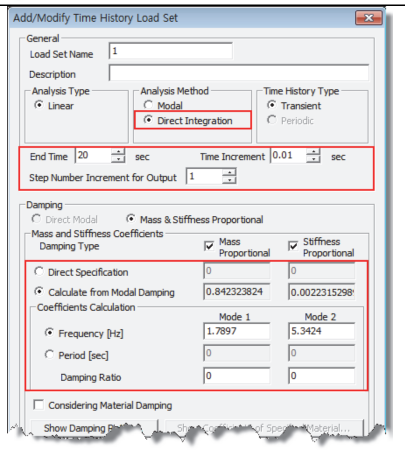
[Units : KN, m, sec]

▶Time History Graph (GTS–GTS NX)

<table>
<tr><th rowspan="2"></th><th>GTS NX</th><th rowspan="2">Time History Graph</th></tr>
<tr><th>min/max</th></tr>
<tr><td rowspan="2">Displace -ment</td><td>-1.48548e-7</td><td rowspan="2">
</td></tr>
<tr><td></td></tr>
<tr><td rowspan="2">Velocity</td><td>-3.54105e-6</td><td rowspan="2">
</td></tr>
<tr><td></td></tr>
</table>

Note Eigenmode input

- ✓ Eigenvalue analysis is used to analyze the inherent dynamic characteristics of the structure, and the important dynamic properties found in eigenvalue analysis are eigenmode(or mode shape), natural period(or natural frequency),modal participation factor etc. These are determined by the mass and stiffness of the structure.
- ✓ In time history analysis (direct), eigenvalue analysis is conducted and the two mode values with the highest mass participation rate is input.

Class	Mode input	
GTS NX	Analysis 〉 Analysis case 〉 Add 〉 Linear time history analysis (direct) 1) Define the target time step in time step : end time, time increment, number of output increment steps 2) Calculate and select modal damping in Analysis control 〉 Dynamic analysis 〉 Define damping 3) Input frequency or period and damping ratio	
GTS440	Model 〉 load 〉 Time history data 〉 Time history load set 1) Select direct integration 2) Define output time step : end time, time increment, number of output increment steps 3) Select modal damping calculation 4) Input frequency or period and damping ratio	

Note Time history load function definition/application

✓ Time history analysis uses the dynamic properties of the structure and applied load to calculate the structural behavior (displacement, member force etc.) at an arbitrary time. These should be input as a function of time.

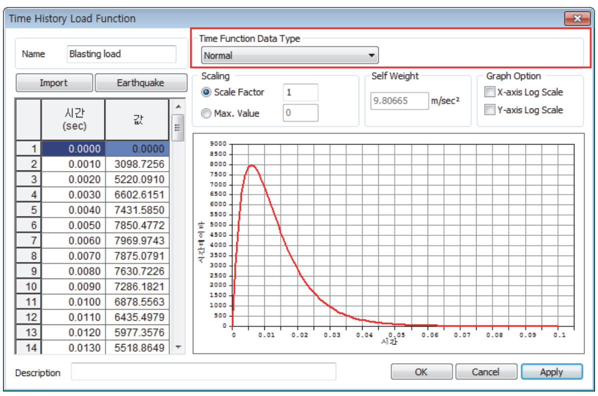
Class	Time history load function definition/application
GTS NX	[Definition] Property/Coordinate/Function 〉 Function 〉 Time history function Data type : classified as normalized acceleration, acceleration, force, moment, normalization [Application] Force/moment : Used in Dynamic analysis 〉 Load 〉 Dynamic nodal load Acceleration/normalized acceleration : Used in Dynamic analysis 〉 Load 〉 Ground acceleration Normalization : Used in Dynamic analysis 〉 Load 〉 Time history static load
GTS440	[Definition] Model 〉 Load 〉 Time history data 〉 Time history function Data type : classified as normalized acceleration, acceleration, force, moment, normalization [Application] Force/moment: Used in Model 〉 Load 〉 Time history data 〉 Dynamic nodal load, dynamic face load Acceleration/normalized acceleration: Used in Model 〉 Load 〉 Time history data 〉 Ground acceleration Normalization : Used in Model 〉 Load 〉 Time history data 〉 Time variant static load

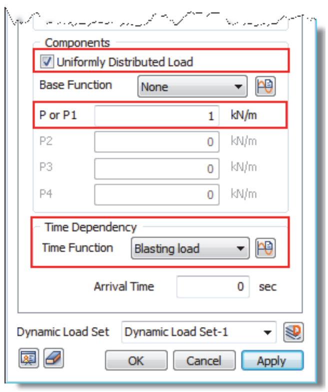
✓ When entering a load acting on a surface (such as the dynamic blast load) in GTS440, the time function data type is input as a 'force'. If the force acting on a unit area is input, it is automatically converted for analysis. (ex: when a force of 1kN is input, it is calculated as 1kN/m2).

✓ When a load is input in pressure form in GTS NX, changing the time function data to [General] and entering the unit load in P or P1 at Dynamic analysis 〉 Load 〉 Dynamic face load gives the same results as GTS440.

▶ Time history load function definition

▶▶ Dynamic face load function definition





Note History result search

- ✓ To output the results at a particular position in a graph for analysis cases with time (transient seepage, consolidation, stress seepage fully-coupled, linear/nonlinear time history analysis, 2D equivalent linear), the [Before analysis] history result search node needs to be registered.
- ✓ GTS440 outputs the entire analysis results on the analysis tree, but this was inefficient in terms of usability or performance. In GTS NX, this was modified such that the user can efficiently define the target position to be reflected in the analysis.

Class	History result search
GTS NX	1. Specify result articles and result check positions in Analysis 〉 History 〉 History result search 2. Register the histories to check in Analysis 〉 Analysis case 〉 Add 〉 Result control 〉 History 3. Check the history results using Result analysis 〉 Special post-processing 〉 History result 〉 Graph
	Output the history results for the entire time step (A more rational history curve can be extracted compared to GTS440))
GTS440	1. Specify the position in Model 〉 Load 〉 Time history data 〉 Time history result function 2. Check history results using Result 〉 Time history analysis 〉 Time history graph
	Output the history results for only the user defined steps (ex. When end time : 10 seconds, time increment 0.01 seconds, output step 100 is defined the results are only output for 1,2,…,10 seconds)