Section 1 Analysis Case

Ground analysis can be explained by comparing it to general structural analysis. Structural analysis gives importance to the uncertainty of load, acting on the structure. Hence, it conducts the member design for the largest member force obtained by combining various results systematically. In contrast, ground analysis focuses more on the uncertainty of construction step and material itself, rather than the load and so, it is important to understand physical state inside the ground. As a result, for ground analysis, solid elements need to be used in modeling to reflect the ground shape and construction situation; and non-linear anisotropic materials and in-situ stress states need to be considered to reflect the actual on-site conditions as much as possible.

The program for ground analysis can be used to simulate in-situ conditions and determine whether design or construction condition is appropriate. For ground analysis, various types of analysis are covered, from General static analysis to Seepage analysis, Stress-seepage coupled analysis, Consolidation analysis, Construction stage analysis, Slope stability analysis etc.

This section provides a brief overview of the analysis methods and descriptions for the analysis options. Refer to the Ch. 5 of the Analysis manual for more detailed information on analysis algorithms.

Provided ground analysis features are as follows:

Static Analysis

- Linear static analysis
- Non-linear static analysis (Non-linear elastic or Elasto-plastic analysis)

Construction Stage Analysis

- Stress Analysis
- Seepage Analysis
- Stress-Seepage-Slope Analysis
- Consolidation Analysis
- Fully Coupled Stress Seepage Analysis
- Stress Nonlinear Time History Analysis
- Heat Transfer Analysis
- Seepage-Thermal Stress Analysis
- Thermal Stress Analysis

Seepage Analysis

- Steady State Seepage Analysis
- Transient Seepage Analysis

Coupled Seepage-Stress Analysis

- Seepage-Stress Sequential Analysis
- Consolidation Analysis
- Fully-coupled Seepage-Stress Analysis

Dynamic Analysis

- Eigenvalue Analysis
- Response Spectrum Analysis
- Linear Time History(Modal) Analysis
- Linear Time History(Direct) Analysis
- Nonlinear Time History Analysis
- 2D Equivalent Linear Analysis

Slope Stability Analysis

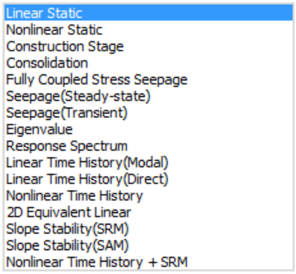
- Slope Stability (Strength Reduction Method, SRM)
- Slope Stability (Stress Analysis Method, SAM)
- Dynamic-Slope Coupled Analysis

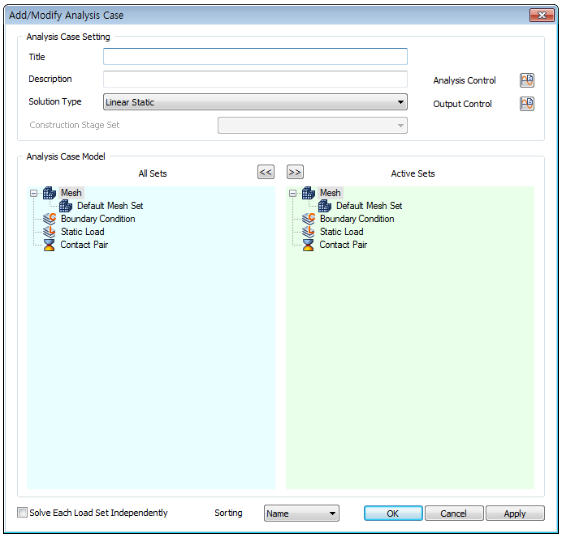
1.1 General

Overview

Create an ‘Analysis Case’ to perform analysis. This step sets the analysis method and the analysis conditions (Mesh set, Boundary condition, Load condition etc.) for each method. In addition, the detailed analysis option and result output option can be adjusted according to each analysis method. For Construction stage analysis in particular, the analysis can be set using the following five different analysis methods and the data used for analysis is specified. The detailed analysis option and result output option can be adjusted and multiple construction stage sets can be set for recursive analysis on a single model with changing analysis conditions.

▸Construction state-supported Analysis methods





Methodology

Input the Title (and Description) to differentiate each Analysis case and select the Solution (analysis) type. The Analysis Case Model should be defined according to the selected Solution (analysis) type. Only the activated analysis conditions (Mesh, Boundary condition, Load etc.) moved to the Active Sets are reflected in the analysis, not All Sets included in the analysis model. Therefore, it is possible to create several Analysis cases by adjusting the analysis condition on the same model and comparing the results. Especially for Linear static analysis, the [Solve Each Load Set Independently] option can be used to apply all the activated Load Sets individually for recursive analysis and comparison of results.

Particular detailed analysis settings (Time step, Analysis control, Output control) can be added, according to each Solution type (analysis method). Each analysis method has different settings and the settings need to be checked before creating an Analysis Case.

If all sets are reflected in the analysis, use the >> button to move all conditions to the Active Set. If only several particular conditions are reflected, use drag & drop to move the selected sets to the Active Set. For Construction stage analysis, because the data used for stage-by-stage analysis is already set, select only the sets used to perform the analysis.

1.2 Solution Type

Linear Static Analysis

Linear static analysis assumes ground and rock materials to be linear-elastic materials and uses static analysis to determine the behavioral characteristics when loading is applied. Ground materials only display linear-elastic properties in the early stages of loading, where only a small strain is generated. However, because Linear-elastic analysis does not consider failure and idealizes the stress-strain relationship linearly, it is widely used for simple analysis, such as identifying the stress distribution or concentration in the in-situ ground.
In a broad sense, linear behavior can be viewed as a special form of nonlinear behavior but because linear analysis is convenient and intuitive, linear static analysis is categorized as a separate analysis type. Linear analysis, including Linear dynamic analysis, utilizes the elastic behavior of a defined material for analysis. Also, the non-linear behavior (tension only, compression only and non-linear elastic behavior) of elastic link elements or truss elements are ignored and the elements are used in the analysis as elastic bodies. Consequently, it is useful to use Linear static analysis to observe the approximate behavior of the ground, to organize the initial stage conditions of the construction stage or to conduct tunnel lining analysis.

Particularly, Linear static analysis does not require recursive calculations and hence, it has a short calculation time. FEA NX considers pore pressure in Linear-static analysis and drained and undrained conditions can be assigned to elastic materials for analysis.

Non-linear static analysis

All physical phenomena includes non-linearity. Ground or structural behaviors are not an exception. Non-linear static analysis is used to simulate the behavior of ground considering such non-linearity, when the change with time is small and can be ignored. FEA NX considers the following non-linearity

- Non-linearity of material : This occurs when the stress-strain relationship is non-linear. Most ground materials have this non-linearity.
- Geometric non-linearity: If the displacement-strain relationship is non-linear, the linear assumption is no longer applicable when the displacement is large, or the rotational deformation is large.
- Non-linearity of load and boundary: Non-linearity that includes the non-linear behavior at an interface, or non-linearity caused by the direction change of a load due to strain, caused by forces such as the follower force.

FEA NX can consider all non-linearity mentioned above in analysis. Non-linear analysis can take a long time for complex non-linear systems because repeated calculations are conducted. Hence for the practicality, considering appropriate non-linearity can result in analysis results that simulate non-linear behavior, while maintaining the accuracy with little computational cost.

Construction Stage Analysis

Construction stage analysis can be used to simulate the construction process of the ground using numerical analysis. Construction stage analysis consists of multiple stages and loading/ boundary conditions, as well as elements, can be added or removed at each stage. This loading/ boundary or element change is applied at the start of each stage. FEA NX can use following types of analysis features to conduct Construction stage analysis.

Stress-Slope Analysis
Analysis of stress and slope stability during the construction process

Seepage Analysis

Stage by stage Steady state seepage analysis, Stage by stage Transient seepage analysis,

Stress-Seepage-Slope coupled analysis

Sequential Seepage-stress analysis and Slope stability analysis during the construction process

Consolidation analysis

Consolidation analysis for environment change and construction process of embankment

Fully-coupled Stress-Seepage analysis

Stress analysis fully coupled with Transient seepage phenomenon

Stress - Nonlinear Time History analysis

Users can perform nonlinear dynamic analysis considering stress status of ground resulted from not only self weight but also construction stage (the history of stress).
Nonlinear time history stage must be set at the final stage.

Heat Transfer

Stage by stage of steady state thermal analysis, stage by stage of transient thermal analysis can be conducted.

Seepage-Thermal Stress

Thermo-hydro-mechanical analysis can be conducted.

Thermal Stress

Thermo-mechanical simulation can be conducted.

When conducting Construction stage analysis, the following should be considered.

- Addition/Removal of element
- Loading/Unloading of weight
- Change in boundary condition
- Change in rock material property
- Definition of load distribution factor
- Step by step underground water level
- Drained-Undrained analysis
- Initialization of displacement
- Stress Analysis for initial construction stage (Consider Ko condition)
- Restart

For example, the construction stages for a tunnel are as follows.

1 Stage: Initial ground stress
2 Stage: 1st face excavation
3 Stage: 1st reinforcement + 2nd face excavation
4 Stage: 2nd reinforcement + 3rd face excavation
5 Stage: 3rd reinforcement + 4th face excavation
…… (Repeat) ……

The first stage is the used to calculate the in-situ stress of soil strata. Because stress analysis of the ground assumes the in-situ state to be the initial state, the in-situ stress state needs to be calculated.
FEA NX uses self-weight analysis to calculate the initial in-situ stress.

Activate / Deactivate element

An element activated during Construction stage analysis has a default in-situ stress value of '0 (zero)'. But if a prestress is defined on the element, the element has the defined prestress value as its in-situ stress. If the self-weight is defined, the activated element has a body force due to its self-load. If the activated element uses the Modified Cam-Clay material model, it has an initial linear-elastic property that is determined by the loading/ boundary conditions of the corresponding stage.
New nodes will be activated as the element appears, and the initial displacement of the node is also '0 (zero)'.

If an element is deactivated at a construction stage, and an additional load distribution factor is not defined, the internal forces of the deactivated element are no longer considered. The total stress state is re-distributed according to this condition.

Loading / Unloading of weight

The addition and removal of load at each construction stage is possible and the load from the previous stage is maintained, except for following cases.

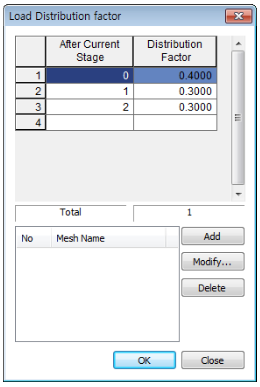
- When an element subjected to the load is deactivated at a stage, its self-weight and external load applied to it are also removed.
- When a node subjected to the load is deactivated, the external load applied to it is also removed.

Added load is cumulated to the already applied load from the previous construction stage.
Boundary conditions can also be modified in the same way and the same exceptions hold true.

Load distribution factor

The Load distribution factor is used during Construction stage analysis to simplify the modeling. The Load distribution factor is a numerical analysis method that uses the load distribution factor to apply the effects of element removal sequentially in the following stages. The Load distribution factor can be used to simplify a 3D problem in 2D, or to downscale the construction stage of a 3D model during analysis.

For example, if stress relaxations of 40%, 30% and 40% are assumed to occur in three consecutive stages, starting from the excavation stage, define the excavation stage and activate the load distribution factor option for that step. Input 0.4, 0.3 and 0.3 for the load distribution factor in option window for 'After Current Stage 0, 1, 2 stage' respectively.



Material property conversion

During Construction step analysis, the ground material properties can change to model time dependent ground disturbance, soil improvement or hardening. There are also cases where the structural material properties need to be change in the middle of the stage, such as hardening of lining concrete or changes in lining thickness. For this purpose, the material properties of a specific element can be changed without any number limits. The changed material property is continued onto the element results(displacement, stress, strain etc.) of the previous stage for analysis.

Caution: The Material property conversion feature needs to be used carefully. Changing the infill material after excavation in Construction stage analysis can be simulated using just the property conversions, without activating or deactivating any elements. Here, the stress conditions from the previous stage is applied to the following stage and so, physically inappropriate behavior can be observed due to the material property conversion. Hence, the material property conversion needs to be conducted at a stage where the element is deactivated and re-activated to obtain the intended results. If a new element is activated, the internal element has an in-situ stress, strain and interior state variable of ‘0(zero)’.

Undrained Analysis

Undrained analysis can be conducted for selected elements and selected construction stages. Two conditions should be satisfied beforehand to apply undrained analysis. Firstly, the drainage parameter should be set as undrained type on material model. Secondly, the undrained condition should be checked on the Analysis control of construction stage. If only one of the conditions is met, the material conducts drainage analysis in the corresponding stage.

For singular analysis cases such as static linear/nonlinear analysis, or slope stability examination, check the Analysis Control > Undrained Condition > Allow Undrained Material Behavior.

Consolidation analysis

Consolidation analysis is an analytical method that calculates the behavior of pore water pressure when it resists external loading, when excess pore water pressure occurs and as the excess pore water pressure reduces with time for an undrained condition.

Pore water pressure in the ground with a small osmotic coefficient instantaneously displays the same behavior as the undrained condition. Hence, it bears most of the compressive load by the created excess pore water pressure, according to the change in load state. However, as time goes by, excess pore water pressure is re-distributed and if there is a drainage boundary, the excess pore water pressure decreases gradually. Because of this, the load previously resisted by the excess pore water pressure is gradually resisted by the soil frame, causing gradual deformation of the soil frame and increasing effective stress within the frame.

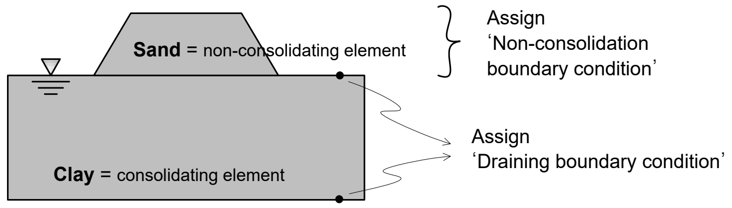
The increase in effective stress leads to the deformation of soil structure and this deformation is accumulated in the gravitational direction, eventually displaying settlement behavior in the gravitational direction with time elapse.

This gradual increase in deformation creates settlement at the base of structural foundation and differential settlement at the base foundation greatly affects the stability and safety of the structure.

Characteristics of Consolidation element

In Consolidation analysis, the elements have an additional pore water pressure degree of freedom, as well as displacement degree of freedom, at the nodes. It is assumed that all elements have a degree of freedom for pore water pressure, unless the two boundary conditions (Non-consolidating condition, Drainage condition) are specified, for Consolidation analysis. Hence, for bank materials that do not express consolidated behavior directly, the non-consolidating element conditions need to be defined to apply it as a general structural element. Also, the drainage conditions needs to be defined for drainage boundaries in consolidating elements. If the boundary conditions are properly defined and Consolidation analysis is conducted, the excess pore water pressure is 0 (zero) where the non-consolidation conditions and drainage conditions are applied.

▸Consolidation element boundary conditions



Fully-coupled Stress-Seepage analysis

Analysis that couples the seepage phenomenon and ground stress analysis can be classified in various ways, depending on the coupling.

The simplest way is to obtain the pore water pressure distribution by conducting seepage analysis beforehand, and reflecting it in the total stress/effective stress relationship equation of the stress analysis conducted in the following step. Such analysis is called sequential analysis. This method can be used to understand the static stress state of the given steady groundwater flow. However, since deformation due to stress analysis does not influence the seepage phenomenon inversely, there is no two-way coupling.

Fully-coupled Stress-Seepage analysis is the two-way coupled analysis between seepage analysis and stress analysis. Both analyses are used to solve the coupled equation. It can display the pore water pressure, stress or deformation changes with time.

The consolidation analysis begins with the assumption that steady state pore water pressure can be maintained, and is used to see the changes in excess pore water pressure. In other words, this analysis is used to simulate the phenomenon of how excess pore water pressure changes with the changes in load/boundary conditions.

Fully-coupled Stress-Seepage analysis does not follow assumption that steady state pore water pressure is maintained. Hence, it is suitable for simulating the transient seepage phenomenon, stress analysis and stability in abnormal condition in a fully coupled form. Unlike the consolidation analysis, it is possible to define the changes in seepage boundary conditions with time, boundary flow rate etc.

In other words, for Fully-coupled Stress-Seepage analysis, it is possible to use all the transient seepage boundary conditions, structural load and boundary conditions.

This analysis can be applied to the ground stability analysis for rainfall or the large-scale dam stability analysis for water level change. The seepage boundary conditions (Head/Flux) can all be used to analyze not only the changes in excess pore water pressure, but also the consolidation analysis that considers the total change in pore water pressure.

Seepage analysis

Seepage analysis can be largely divided into two; the steady state analysis and transient analysis.

Steady state flow analysis is where the boundary conditions inside and outside of the ground does not change with time. Therefore, the inflow is always equal to the outflow within the analysis range. Transient analysis on the other hand, can display different inflow and outflow with time, even when the same boundary conditions as the steady state analysis are applied.

A permeable ground layer (aquifer) exists where the groundwater can seep through, and if the head difference or flux exists at the boundary, the seepage phenomenon occurs.

Seepage flow occurs along the waterway that connects through the empty pores between soil particles. This flow complies with Darcy's law. According to this law, the seepage quantity through the soil volume is equal to the multiplication of permeability coefficient, hydraulic gradient and cross sectional area. Darcy's law originally started from the saturated domain, but can also be applied to the unsaturated domain.

The unsaturated domain includes all non-saturated domains, from the fully dried condition to the almost saturated condition. As the degree of saturation falls below 100% , air bubbles will also exist in the pores and if the saturation is very low, the water particles will attached between soil particles in a concave form.

Negative pore pressure is referred to as suction pressure. In most cases, suction pressure increases as the degree of saturation decreases.

Transient analysis is used when the boundary conditions inside or outside of the ground changes according to time.

The main differences between transient analysis and steady state analysis are that the boundary conditions change as time passes and the fact that the transient analysis requires volumetric water content. When the underground water level goes up or down, the influence factors such as the water content in the unsaturated domain and porosity are needed.

Comparing the water filling of a reservoir between the fully dried initial state and the partially saturated state, there is significant difference in the time it takes for the seepage in the reservoir body to reach a steady state. Hence, transient analysis can be used to estimate the time it takes to saturate the interior of the body, or deduce a more economic design variable by comparing it with the saturated case.

Eigenvalue analysis (Mode analysis)

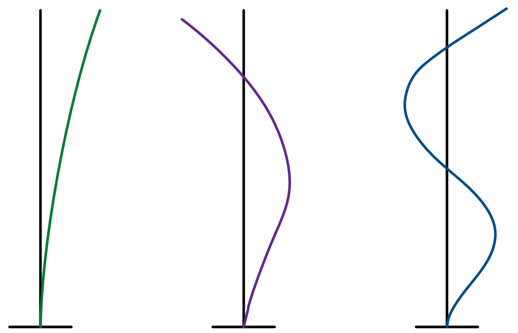
Eigenvalue analysis is used to analyze the inherent dynamic properties of the ground/structure, and this can be used to obtain the natural mode(mode shape), natural period (natural frequency), modal participation factor etc. of the ground/structure. These properties are determined by the mass and stiffness of the structure. In other words, if a structure is determined, the natural frequency and vibration mode (natural mode) are also determined and the number of properties are the same as the degree of freedom of the structure. For real cases, the structure does not vibrate at a single mode shape and multiple modes overlap to display a complex vibration shape.

Here, the Mass participation factor is a mass percentage factor that represents how much of the structure participates in the vibration for each vibration mode when the structure is vibrated at a complex vibration mode. For example, if the first mode mass participation factor is 60%, 60% of the total mass of the structure participates in the first mode. Hence, the a mode with a high mass participation factor is considered in the earthquake wave for analysis.

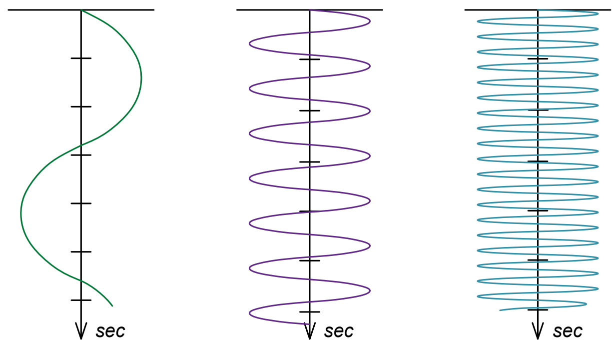
For general structure, considering only vibration modes with a mass participation factor sum of around 90% is still regarded as a sufficiently accurate analysis. However, the ground material properties are relatively smaller that structural properties and it is hard to have a mass participation factor of 90% in Eigenvalue analysis. The period is also relatively smaller and no specific standard exists.

Natural periods are defined as the time taken for a structure to vibrate from its natural vibration state to the particular mode shape using a natural value that 1:1 corresponds to the natural mode.

▸Natural mode shapes



▸Natural frequency



$\lambda_1 = 1.87510407$ $T_1 = 1.78702$ sec

$\lambda_2 = 4.69409113$ $T_2 = 0.28515$ sec

$\lambda_3 = 7.85475744$ $T_3 = 0.10184$ sec

The General seismic design criteria requires that each mode's effective model mass included in the analysis should retain more than 90% of the total mass. This is to include most of the major modes that influence the result.

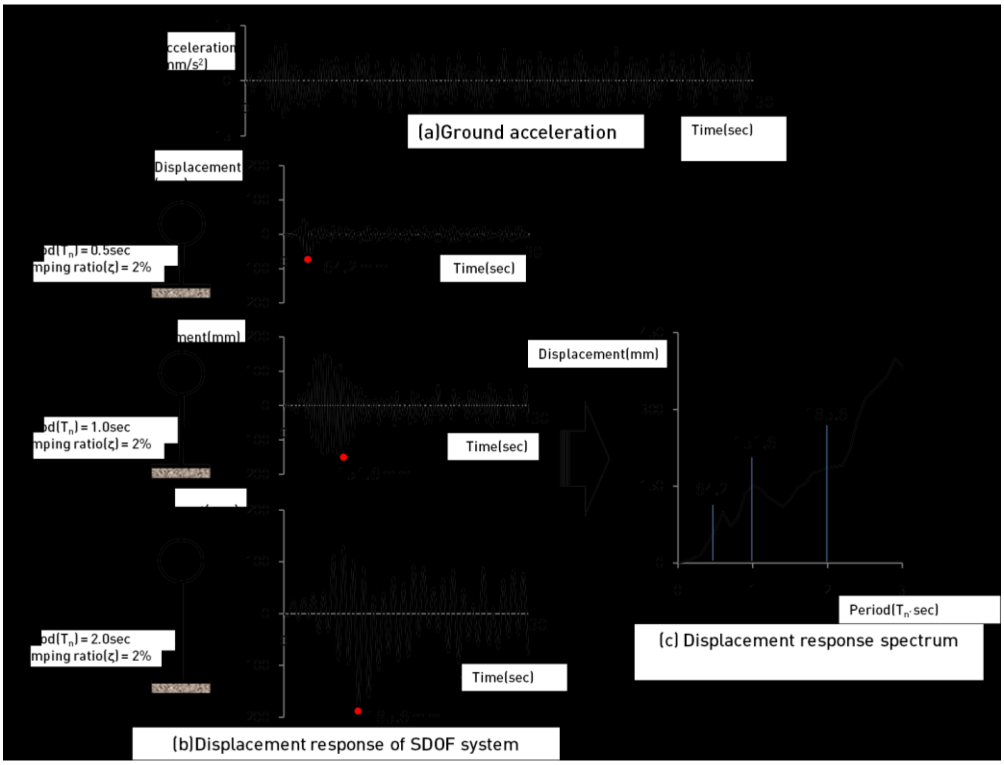
Response Spectrum Analysis

The Response spectrum analysis uses and combines the spectrum data, corresponding to the absolute maximum values of the time response for each mode, using the principle of superposition. Because the simultaneity of the maximum value occurrence for each mode is not considered and only the absolute maximum values are combined, it is considered as an approximate solution to the Modal linear time history analysis. However, the correlation between modes is considered in the mode combination to correct for simultaneity errors.

Spectrum data

A spectrum data is the absolute maximum displacement (or velocity, acceleration) values found by fixing the damping ratio and changing the period (mass, stiffness) of a single DOF system for a time history data. Because the damping ratio for each mode of the structure can be different, instead of using a spectrum data for one damping ratio, multiple spectrum data for different damping ratios are created and interpolated for the damping ratio. The period spacing used to create the spectrum data can be different to the natural period of the structure and so, interpolation for the natural period is also needed. Both linear interpolation and logarithmic interpolation are supported. Also, the spectrum data generally uses the design response spectrum created from the statistical historic earthquake data in the target region, rather than converting a particular given data. Because a design response spectrum with one damping ratio is generally used, correction instead of interpolation is applied for this case.

▸Create displacement response spectrum of SDOF system

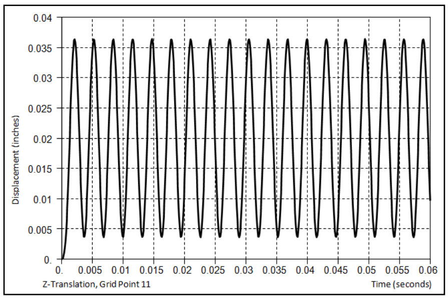


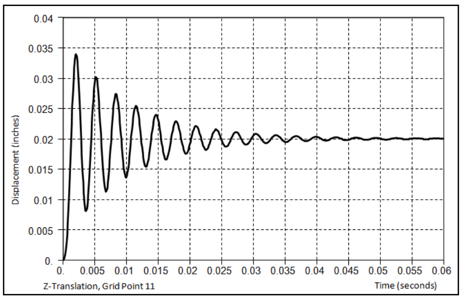
Damping consideration

Damping consideration is essential for accurate dynamic analysis. All real structures cannot vibrate infinitely and the energy loss at the molecular level, or energy loss in the structure due to interaction between component elements, applies damping to the system and gradually decreases the vibrations of the structure.
In particular, for stimulation near the natural frequency region (resonance effect) of the structure, damping has a dominant effect on the analysis results.

▸Without damping

▸▸With damping effects





The accurate damping ratio of the structure needs to be determined through experimentation, and the generally used damping ratios used are as follows.

System	Damping ratio
Steel (within elastic region)	< 1 %
Steel structure with joint	3 % ~ 7 %
Pipe system with small diameter	1 % ~ 2 %
Pipe system with large diameter	2 % ~ 3 %
Rubber material	~ 5 %
Prestressed concrete structure	2 % ~ 3 %
RC structure	4 % ~ 7 %

The modal damping is most frequently used to express the structural damping for general numerical analysis. The modal damping determines the damping for each natural frequency in the system and can be largely divided into proportional and non-proportional damping. Proportional damping can use the mass proportional type, stiffness proportional type or Rayleigh type damping.
Mass proportional damping expresses the viscous damping due to external factors such as air resistance and assumes that the damping matrix is proportional to the mass. For stiffness proportional damping, the dissipation damping effect (vibration energy dissipated to the ground) is hard to express directly and is assumed to be proportional to the damping stiffness. Hence, the damping can be overestimated at higher modes.

Rayleigh damping is the stiffness proportional type corrected for the damping constant at higher modes and can be expressed as the sum of the mass proportional type and stiffness proportional type.

Linear time history analysis (Modal/Direct)

Linear time history analysis calculates the solution to the dynamic equilibrium equation for the structural behavior (displacement, member force etc.) at an arbitrary time using the dynamic properties of the structure and applied loading when a dynamic load is applied. The Modal superposition method and Direct method are used for linear time history analysis.
Because of linear analysis characteristics, nonlinearity is not considered. When using a nonlinear material, the material is converted to an equivalent linear elastic material for analysis.

The water level can be defined for the linear time history analysis and the effective stress results can be viewed. Also the drained/undrained effects of the material can be included in the analysis.

Mode superposition method

The mode superposition method assumes the structural displacement as a linear combination of orthogonal displacements. Using this, a more simplified time integral function can be used to calculate the dynamic response for a selected mode. The mode superposition method is used in many structural analysis programs and is an effective way to calculate the dynamic response for the linear dynamic analysis of large structures with little computational cost. However, the accuracy of the total response depends on the number of used natural modes and so, the number of modes used in the calculation need to be selected appropriately.

Direct method

The direct method is a time history analysis that uses the DOF of the total analysis area as a variable. The dynamic equilibrium equation for the total DOF can be integrated gradually with time to find the solution. The solution is found for each time stage without any form change to the equilibrium equation and various integration methods can be used. The direct integration method conducts the analysis for all time stages and the number or time stages is proportional to the analysis time.

Loading in Linear time history analysis

Dynamic loads that change with time can be used in linear time history analysis.

Define time step

The time step for time history analysis is different for the direct method and mode method.
The direct method uses the defined time step to conduct time integration implicitly. Hence, accuracy difference can occur depending on the time step size. Generally, accurate results can be obtained when a time step that is smaller than 10% of the minimum period is used. Using large steps create errors in the time integration and using too small steps create unnecessary computation cost.

The direct method conducts the time integration analytically. Hence, the time step does not affect the accuracy of the calculated result. The Mode method time step is used to set the time to view the interim results of the time history.

Tip

Comparison between the Direct method and Mode method
The direct method generally takes longer than the mode method for analysis. Hence, if many time steps are needed or if the model size is large, it is effective to use the mode method. If many natural frequencies are calculated near the analysis load frequency (for example, the problem of too many active frequencies), it is better to use the direct method to obtain more accurate results.

Element	Direct method	Mode method
Analysis time	Long analysis time	Short analysis time
Details	Time step setting is important	Number of modes setting is important
Model size	Fit for small models	Fit for large models
Analysis accuracy	Long analysis time, but highly accurate results	Accuracy errors can occur depending on the selected number of modes

Nonlinear time history analysis

Nonlinearity can also be considered for ground and civil structures in time history analysis. Like the nonlinear static analysis, the material nonlinearity, geometric nonlinearity, load and boundary nonlinearity can all or selectively be considered in analysis.

Generally, most ground materials have nonlinear properties and so, the dynamic response of the ground can be accurately simulated using nonlinear time history analysis.

Water level
Nonlinear time history analysis is used to identify the dynamic response in a relatively short time and the water level for dynamic analysis is assumed to be constant.

Loading in Nonlinear time history analysis
Dynamic loads that change with time can be used in nonlinear time history analysis.

Define time step
Nonlinear time history analysis conducts time integration implicitly. Hence, accuracy difference can occur depending on the time step size. Generally, accurate results can be obtained when a time step that is smaller than 10% of the minimum period is used. Using large steps create errors in the time integration and using too small steps create unnecessary computation cost. In particular for nonlinear analysis, the converging solution is found using iterative calculations and hence, the time step must be computed carefully.

2D equivalent linear analysis

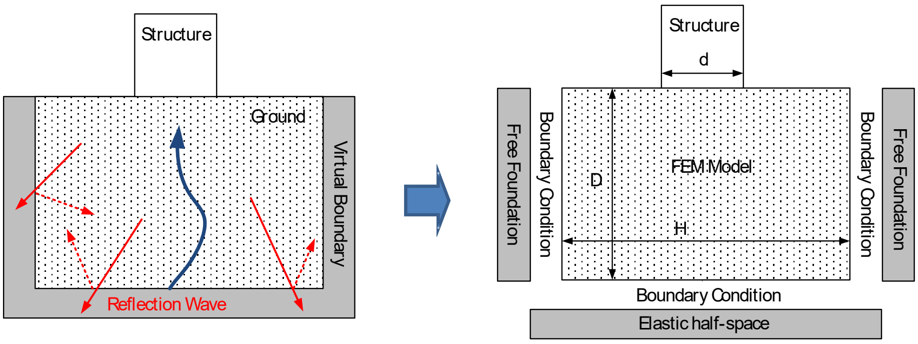
The equivalent linear method on the FEA NX is applied to free field analysis and 2D equivalent linear analysis. The equivalent linear method approximates the ground material nonlinearity as an equivalent linear material property for linear iterative analysis. It is generally valid for strain sizes of $10^{-5} \sim 10^{-3}$.

For the equivalent linear method, the initial element shear modulus G and initial damping ratio h need to be set for each layer. Generally, the values when the strain is minimum are used, and the frequency domain analysis is conducted to calculate the maximum shear strain $\gamma_{\max}$ of each layer or element. The effective shear strain γ_{eff} is calculated as 0.65 or using the earthquake magnitude M by multiplying $(M-1)/10$ to that value and the shear modulus G and damping ratio h are calculated using the dynamic material function curve, created from the effective shear strain γ_{eff} of each layer or element. This analytical process is repeated until the G and values h converge, generally determined by the relative error of 5% or less, then the analysis is complete and the calculated results are output.

Free field analysis finds the ground response to an input earthquake at the in-situ ground stage before structure construction. Free field analysis is often used to for the determination of design response spectrum using ground surface vibration estimation, liquefaction assessment using dynamic stress-strain computation and the determination of earthquake load that causes instability of ground or structures.

2D equivalent linear analysis not only provides analysis for the ground, but also for soil-structure interaction. To minimize the earthquake damage, the seismic design for underground structures is performed, and the stability examination needs to be done to the structure that considered seismic safety. If the structure is built on a soft ground layer such as clay or silt layers, the vibrations in the bedrock due to earthquakes can be greatly amplified at the surface and hence, the effects on the structure by the soil-structure interaction due to earthquake vibrations need to be assessed in detail. Because underground structure are different from ground structures in that the structural response to earthquakes is mostly controlled by the ground displacement, the dynamic material properties of the ground and modeling method determines the analysis results. The figure below displays the general information on modeling the actual analysis domain using the finite element method (FEM) for soil-structure interaction analysis.

▸Schematic diagram of analysis domain and FEM modeling



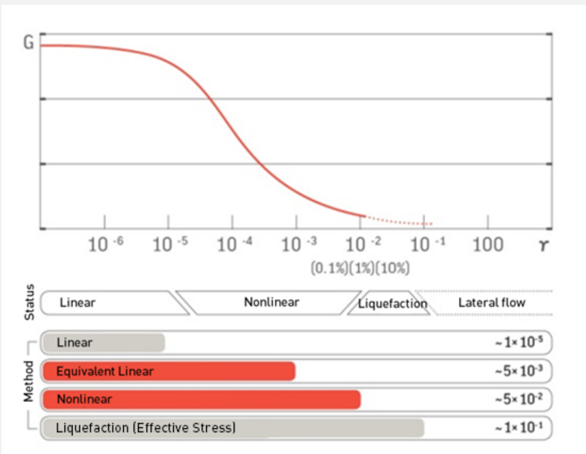
▸Table. FEM model size to minimize analysis result interference cause by the boundary

Boundary conditions	Analysis method	Model depth	Model width
Transfer	Frequency domain	-	$D \geq 2d$
Viscous	Time domain	$H \geq d$	$D \geq 5d$
Symmetric	Effective stress		$D \geq 10d$

The main difference between soil-structure interaction problems and normal structural dynamic problems is the radiation damping effect due to the infinite property of the ground. Normal damping properties come from material friction etc. that dampens the structural movement, but radiation damping radiates the wave energy into the infinite ground space to dampen the structural energy. Hence, seismic analysis conducted using the equivalent linear method to consider the material nonlinearity and use the frequency domain analysis, that makes the modeling of radiation damping easier.

Tip

The analysis method depending on the shear modulus G and shear strain γ relationship is shown in the figure below.



Slope stability analysis (SRM/SAM)

Slope stability for an embankment or excavation is one of the most frequently dealt problems in geotechnical engineering. The slope always has a self-weight potential energy due to gravity and if external forces such as pore water pressure, applied load, earthquake, wave force etc. act on the slope, its stability is greatly affected. Here, slope failure can occur if the internal shear stress due to the self-weight and external forces is greater than the shear strength of the slope soil. Calculating the safety for this slope failure due to shear stress and shear force is called Slope stability analysis.

The following slope stability analysis methods can be used on the FEA NX.

- Strength Reduction Method (SRM): Nonlinear FEM-coupled strength reduction method
- Stress Analysis Method (SAM): Method based on Nonlinear FEM and limit equilibrium theory

Strength Reduction Method (SRM)

Slope stability analysis using the finite element method is a numerical analysis method that analyzes the minimum safety factor and failure behavior using various shapes, loads and boundary conditions. In particular, the strength reduction method can be used to simulate the failure process without any previous assumptions(Griffith et. al. 1999; Matsui, 1990). It can also be applied to 3D axis symmetric problems.
The strength reduction method gradually decreases the shear strength and friction angle (c, ϕ) until the calculation does not converge, and that point is considered to be the failure point of the slope. The maximum strength reduction ratio at that point is used to calculate the minimum safety factor of the slope.

Stress Analysis Method based on limit equilibrium theory (SAM)

This method first uses the finite element method to perform stress analysis on the slope and the safety factor for each various virtual slip surface, created from the assumptions of the limit equilibrium theory, is calculated based on the stress analysis results. Here, the calculated minimum safety factor of the various virtual slip surfaces becomes the safety factor, and the critical section is computed. The SAM method can only be used on the 2D environment.

Nonlinear time history analysis + SRM

Slope stability analysis using the general SRM cannot be used as a factor of safety for the dynamic state. Since slopes are more vulnerable to dynamic loads such as earthquakes. In a dynamic equilibrium state, the ground receives stress from not only its self-weight, but also from the inertial force due to vibrations.

FEA NX can conduct such slope stability analysis for the dynamic equilibrium state. The slope stability analysis is based on the SRM and can be applied to 2D, axis symmetric and 3D problems.
The input time during nonlinear time history analysis can use the stress state of the ground at that point as the initial values to calculate the slope stability.

Heat Transfer analysis

Heat Transfer analysis can be divided into two: the steady state analysis (state of the model: the temperature and heat flow rates, is steady) and transient analysis (time dependent). This analysis type can be used to model the thermal changes in the ground due to environmental changes, or due to the construction of facilities, such as buildings or pipelines. Heat Transfer is transmission of thermal energy due to a gradient in temperature.

Thermal energy is exchanged based on following phenomenological ways:

- Conduction: determines heat transfer from a hot to a cold object, that are in direct contact to each other. The thermal conductivity of the different objects decides how much heat in which time is being transferred.

- Convection: determines transfer of heat between two areas without physical contact. Convective heat transfer is the process that removes heat from a surface when that surface is exposed to fluid (liquid or gas) of a different temperature flowing over it.

After the heat transfer analysis, results such as temperature distribution, temperature gradient, heat flow direction / size are output. Available under Construction Stage analysis.

Thermal Stress analysis (thermo-mechanical)

Thermal stress analysis can be divided into two: the steady state analysis (state of the model: the temperature and heat flow rates, is steady) and transient analysis (time dependent). Thermal stress and thermal deformation due to heat will be calculated as additional to Heat Transfer analysis output. Available under Construction Stage analysis.

Seepage-Thermal Stress analysis (hydro- thermo-mechanical)

Seepage-Thermal Stress analysis is an analytical type that can use seepage boundary condition. It can select steady state and transient state type as same as thermal stress analysis. Both structural analysis results and seepage / heat transfer analysis results are output.

The initial condition used for transient seepage-thermal stress analysis can be interpreted as a steady state at time = 0 from the seepage boundary condition, or it can be calculated by specifying the water level.

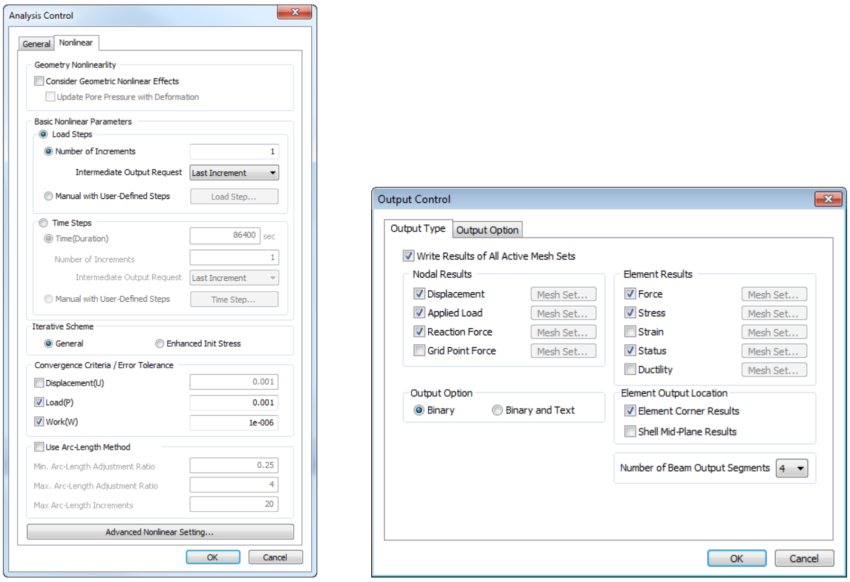
Available under Construction Stage analysis.

1.3 Analysis Control (Option)

Overview

The basic options, automatic settings and various advanced analysis options can be checked and changed depending on the selected analysis type. For construction stage analysis, the options can be defined for each separate stage. For analysis results, the output result list can be set in terms of element type to effectively decrease the size and output time of the result file.
For time history dependent analysis, such as transient seepage, consolidation, time history analysis, the time step for result check and print can be set separately.

Methodology



The figure above is the Analysis control and Result control setting windows.

The additional setting control options for each analysis type is shown, and the detailed inputs are listed in the table below.

▸Table. Static analysis - Analysis control options for each analysis type

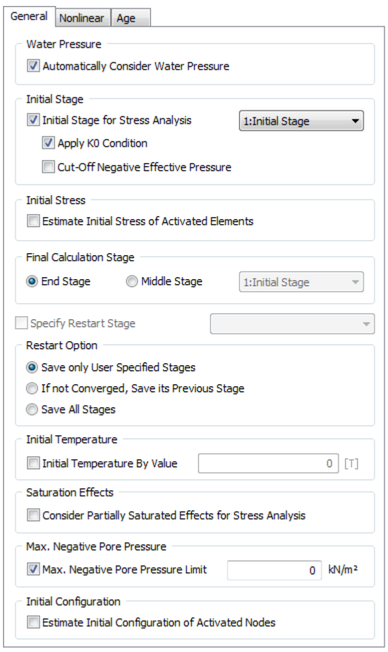
<table>
<tr><th>Tab</th><th>Linear/Nonlinear Static analysis</th><th>Construction stage</th><th>*Consolidation analysis, *Fully coupled seepage stress</th><th>Seepage (Steady/*Transient)</th><th>Slope stability (SRM/SAM)</th></tr>
<tr><td rowspan="9">General</td><td>Water pressure (Automatic)</td><td>Water pressure (Automatic)</td><td>Water pressure (Automatic)</td><td>Maximum negative pore pressure</td><td>Water pressure (Automatic)</td></tr>
<tr><td>In-situ analysis</td><td rowspan="4">Initial stage(ko)
Final calculation stage
Specify restart stage
Restart option</td><td>In-situ analysis</td><td rowspan="8">-</td><td>Water level</td></tr>
<tr><td>Initial temperature</td><td>Water level</td><td>Saturation Effects</td></tr>
<tr><td>Water level</td><td>Saturation Effects</td><td>Maximum negative pore pressure</td></tr>
<tr><td>Saturation Effects</td><td>Maximum negative pore pressure</td><td>Undrained Condition</td></tr>
<tr><td>Maximum negative pore pressure</td><td>Initial temperature</td><td>-</td><td>-</td></tr>
<tr><td rowspan="3">Undrained Condition</td><td>Saturation Effects</td><td rowspan="3">-</td><td rowspan="3">-</td></tr>
<tr><td>Maximum negative pore pressure</td></tr>
<tr><td>Initial Configuration</td></tr>
<tr><td rowspan="5">Nonlinear</td><td>Geometry Nonlinearity</td><td>Geometry Nonlinearity</td><td>Geometry Nonlinearity</td><td>Load steps (or Time Steps)</td><td>Load steps (or Time Steps)</td></tr>
<tr><td>Load steps (or Time Steps)</td><td>Load steps (or Time Steps)</td><td>Convergence Criteria</td><td>Convergence Criteria</td><td>Convergence Criteria</td></tr>
<tr><td>Convergence Criteria</td><td>Convergence Criteria</td><td>Advanced nonlinear setting</td><td>Advanced nonlinear setting</td><td>Use arc-length method</td></tr>
<tr><td>Use arc-length method</td><td>Advanced nonlinear setting</td><td rowspan="2">-</td><td>-</td><td>Advanced nonlinear setting</td></tr>
<tr><td>Advanced nonlinear setting</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>Age</td><td>-</td><td>Age</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>Seepage</td><td>-</td><td>-</td><td>-</td><td>Initial condition</td><td>-</td></tr>
<tr><td rowspan="4">Slope stability (SRM)</td><td rowspan="4">-</td><td rowspan="4">-</td><td rowspan="4">-</td><td rowspan="4">-</td><td>Geometry Nonlinearity</td></tr>
<tr><td>Nonlinear parameter</td></tr>
<tr><td>Safety factor</td></tr>
<tr><td>Advanced nonlinear setting (Use arc-length method)</td></tr>
</table>

<* : Time step setting analysis type >

Tab	Construction Stage Heat Transfer	Construction stage Thermal Stress	Construction Stage Seepage Thermal Stress
General		Water Pressue(Automatic)	Water Pressue(Automatic)
		Initial stage(K0)	Initial stage(K0)
		Initial Stress	Initial Stress
	Initial Temperature	Initial Temperature	Initial Temperature
		Final calculation stage	Final calculation stage
		Specify restart stage	Specify restart stage
		Restart option	Restart option
		Saturation Effects	Saturation Effects
		Maximum negative pore pressure	Maximum negative pore pressure
		Initial Configuration	Initial Configuration
Nonlinear		Geometry Nonlinearity	Geometry Nonlinearity
	Load steps (or Time Steps)	Load steps (or Time Steps)	Load steps (or Time Steps)
	Convergence Criteria	Convergence Criteria	Convergence Criteria
	Advanced nonlinear setting	Advanced nonlinear setting	Advanced nonlinear setting

<* : Time step setting analysis type >

Water Pressure (Automatically consider water pressure)

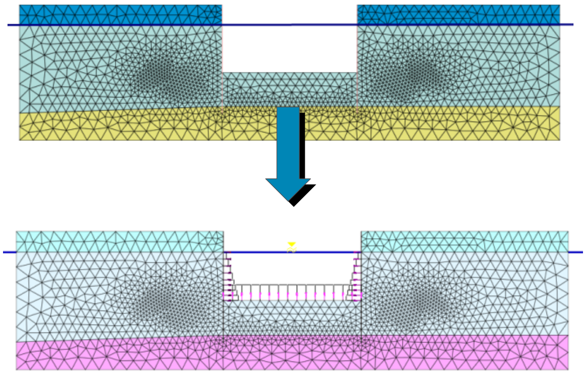


This option considers all free surfaces/edges of the model as an external water pressure. The water pressure is calculated with reference to the pore pressure acting on the free surface/edge.

- If water level is set, assume constant water pressure with reference to the water pressure.
- If seepage analysis was conducted previously, use the pore water distribution (size) calculated for each node.
- If the pore pressure is a negative (-) value, water pressure is not considered automatically.

Caution: If modeling is done for the case where the external water pressure corresponding to pore water pressure in the model does not exist, this option needs to be canceled. When conducting stress analysis by specifying the water level line, the pore pressure is calculated by the water level difference between the free node and corresponding load. Hence, to accurately examine the influence lines of the underground water level, Seepage-stress coupled analysis is recommended.

▸Application of auto-water pressure on excavation surface when excavating below the water level



In-situ analysis

Include in-situ Analysis with Self-weight

This option resets the stress state of the singular analysis ground. The calculated in-situ stress is in equilibrium with the self-weight and the same boundary conditions used in singular analysis for analysis. When considering self-weight in time history analysis, the initial in-situ stress needs to be calculated. If not, vibrations can occur due to the load addition. In particular, the self-load must be included for nonlinear time history analysis.

Ko condition consideration

The K_0 method uses the constant K_0 defined by $K_0 = \sigma_h / \sigma_v$ to calculated the horizontal stress from the vertical stress to set it as the in-situ stress.

Using this method, the vertical stress σ_v needs to be found first using self-weight analysis and that value can be used to compute the horizontal stress using $\sigma_h = K_0 \sigma_v$. Here, the shear stress maintains its value, calculated from the analysis result.

If the ground surface is horizontal, there are no problems in using this method, but if not, the calculated stress state and self-weight are not in equilibrium.

If the stress is adjusted without maintaining the equilibrium state, the stress can change to fit the equilibrium with the external force in future stress analysis, even if there are no external force changes, causing deformation. Hence, the K_0 method can be applied if the additional stress changes are relatively small. Generally, the conditions when the stress modification due to the K_0 method can be used are as follows.

- When the ground shape change in the horizontal direction is small
- When the pore pressure distribution shows no change in the horizontal direction

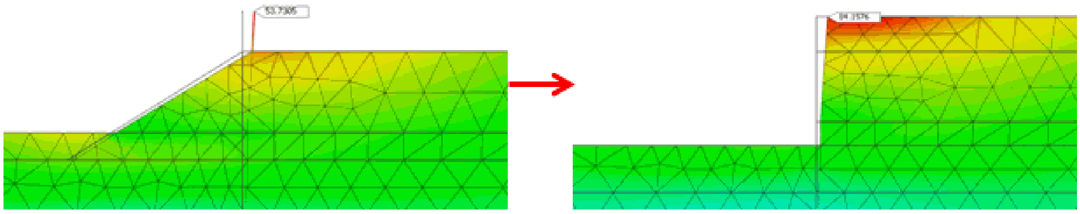
- When the horizontal stress can occur due to the horizontal boundary condition of the free line/face
- When using transversely isotropic materials that have the same material vertical/horizontal axis

If the K_0 condition is not considered, the stress state obtained from the self-weight analysis is set as the in-situ stress. If the ground surface is horizontal, this method is the same as the K_0 method when $K_0 = \nu / (1-\nu)$. If not, a horizontal strain exists and different results than the K_0 method results can be obtained. Shear stress also occurs.
This method is generally recommended when the ground is sloped. However, because a value larger than 1 cannot be set for the K_0 value, a null stage can be added for re-analysis after using the K_0 method to calculate the equilibrium stage, without adding extra external conditions when a K_0 value larger than 1 is needed. However in this case, the final equilibrium state does not satisfy the K_0 condition. Also, the modified stress is vastly different from the equilibrium point, it can be hard to calculated a converging solution using nonlinearity.

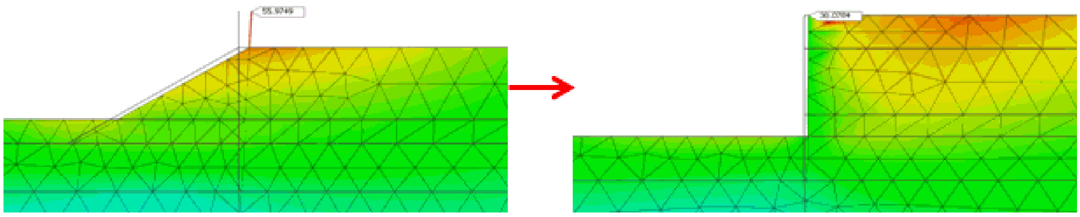
Estimate Initial Stress of Activated Elements

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

▸Engineering example



[Without Initial Stress Option : Horizontal Displacement : 84mm]



[With Initial Stress Option : Horizontal Displacement : 30mm]

Clear Displacement/Strain

The displacement reset condition may be needed during analysis. For example, when the displacement and strain due to self-weight need not be considered in the initial analysis stage, the reset option can be used to reset the in-situ state displacement and strain to '0(zero)'.
Also, the reset can be performed at an arbitrary construction stage, such that the middle stage after analysis of several stages can be set as the reference state. Displacement/Strain reset is applied at the end of the specified stage, after the analysis has finished.

Caution: When conducting nonlinear analysis by considering geometry nonlinearity, arbitrarily modifying the deformation does not guarantee the continuity. Hence, this option is not recommended for geometric nonlinear analysis of construction stages.

Cut-Off Negative Effective Pressure

When conducting linear static analysis for initial stress of ground, tensile stress can be generated especially at the ground surface according to the geometry and stiffness differences. In this case, this tensile stress can take effect on the convergence for the following stage (nonlinear analysis) significantly. If there is tensile stress generated in in-situ state, software will make it close to Zero to ignore the abnormal stress distribution. Since this is the basic concept of initial stress of ground, strongly recommended to use for all staged analysis.

Initial Temperature

This option sets the initial temperature of the single analysis model. If not checked, the initial temperature defined in the [Analysis Control] is considered. The temperature is used to assess the effects of thermal load, and the temperature difference with the input temperature load is considered in the analysis.

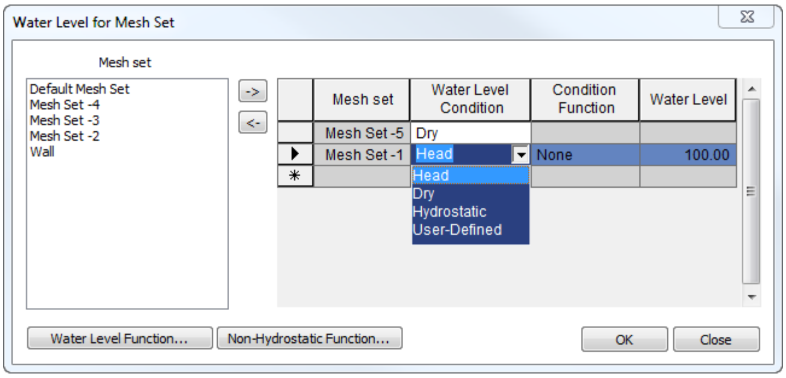
Water Level

Define water level

Directly input the water level height, or select a water level function that already has a specified water level to set the water level. The set water level is applied to the total model. When using the water level function, the input value is multiplied to the function value and applied.

Define water level for mesh set

Define the water level for each mesh set.
If the groundwater layer is surrounded by rocks or an impermeable clay layer (confined aquifer), the presence/absence of the groundwater level for each ground layer can be set for analysis.
If the total groundwater level is input and a mesh set has a defined groundwater level, the mesh set groundwater level has priority and the total groundwater level is applied to mesh sets that do not have a defined level.
If the water level and the function are specified at the same time, the input water level and the function are multiplied and reflected in the analysis.



Mesh Set – select the mesh set to apply the water level condition.

Water Level Condition – select between Head, Dry, Hydrostatic and User-Defined for applying water pressure

Head – compute head according to the water level assigned to the mesh set.
Dry – assume there is no pore water pressure applied to the mesh set.
Hydrostatic – assign a non-hydrostatic water pressure to a mesh set.
User-Defined – apply a user-defined pressure gradient to a mesh set

Condition Function – select a condition function for Head, Hydrostatic & User-Defined

- Head
  - None – set a single water level for water pressure calculation.
  - Water Level Function – assign a function which describes the water level using General Function in 2D and Surface Function in 3D.
- Hydrostatic – not available
  - Water Level Function – assign a Non-Hydrostatic Water Pressure function type Hydrostatic to define a pressure profile to be used to compute the water pressure.

- User Defined
  - Water Level Function – assign a Non-Hydrostatic Water Pressure function type User-Defined to apply a linear pressure profile

Water Level – input water level to be considered for the selected mesh set (only for Head).

Saturation Effects

This option is to conduct accurate analysis when the saturation has a value between the unsaturated state (S_e=0) and the saturated state (S_e=1). The partial saturation can be applied in the following two cases.

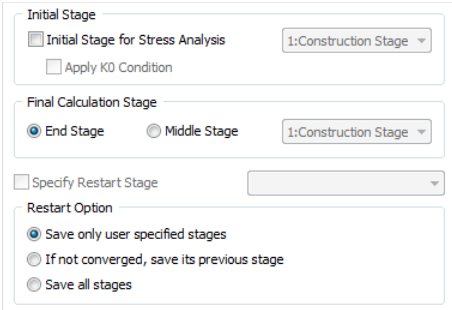
- Applying the partial saturation to calculated the effective stress-total stress relationship (Use Bishop's effective stress relationship equation)
- Consider the partially saturated state in the unit weight calculations for a material, such that the unit weight when partially saturated has a value between the saturated unit weight and unsaturated unit weight.

If partial saturation is not considered, Terzaghi's effective stress formula is used and the unit weight is set as either the saturated unit weight or the unsaturated unit weight, depending on the pore water pressure distribution (a value in between is not used.). The saturation is defined as a function of pore water pressure and if partial saturation is considered, the unsaturated properties of the material need to be defined to define the saturation function for pore water pressure.

Maximum negative pore water pressure limit

This option limits the maximum negative pore pressure by the input number. If partial saturation is not considered, Terzaghi's effective stress formula is used and the pore stress of the unsaturated state can be overly reflected in the calculation. Hence, when not considering partial saturation, the negative pore water pressure needs to be limited to a certain value. Reversely, if the partial saturation is considered, Bishop's equation is used and there is no such danger. In other words, the pore stress is limited by the unsaturated property function and there is no need for a particular limit on the negative pore water pressure.

Construction stage general setting



Initial stage

Specify the construction stage that will be considered as the in-situ condition and check the Ko consideration. Refer to the 'Linear analysis' option for more information on the Ko. The displacement and strain for the construction stage specified as the initial stage, is reset.

Final calculation stage

The default setting is calculation up to the final stage, but a separate Final calculation stage can be set when stopping the analysis to check the interim results.

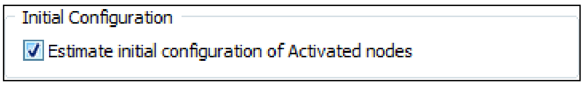
Specify restart stage

When specifying the construction stage, the [Specify restart stage] option can be checked on the Analysis control for each stage. The checked stage is automatically saved on a separate result file and when the same model is used for re-analysis, the re-analysis can be performed starting from the next stage of the result file. It is useful when many construction stages are specified.

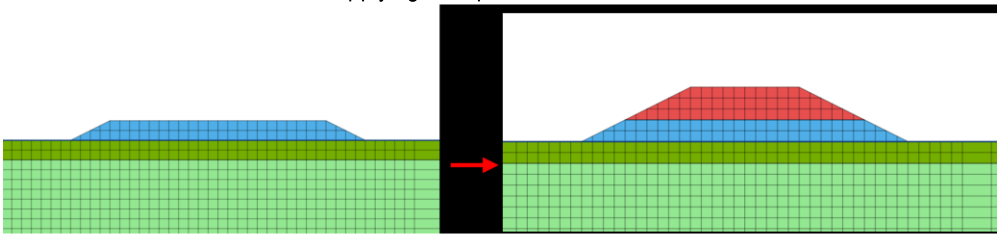
Restart option

If the converge standard is not satisfied for non-linear analysis, the reliability can be in question and so, it is important to check whether the converge standard is satisfied for each stage during construction stage analysis. In particular, because construction stage analysis can take longer time than single analysis, the [If not converged, save its previous stage] option is available when a stage does not satisfy the standard. This option saves the stage before as a result file and the model can be review and modified before restarting. Also, the [Save all stages] option is available for when the analysis is terminated forcefully, due to the computer system instability or to check the interim results. However, because saving all analysis results takes up a large size, the save capacity needs to be secured on the computer.

Initial Configuration

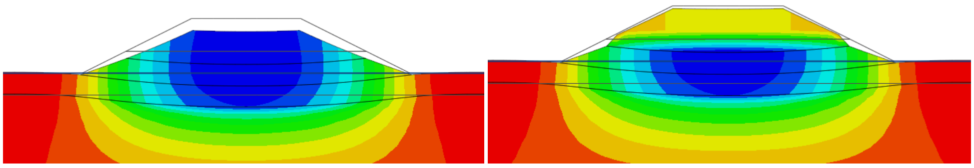


During construction, the newly activated nodes(elements) can be set to the position considering deformed shape in the previous stage. Following is the example of staged embankment to compare the settlement distribution between with and without applying the option.

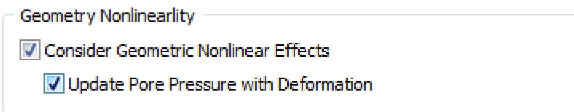


▸With option

▸▸Without option

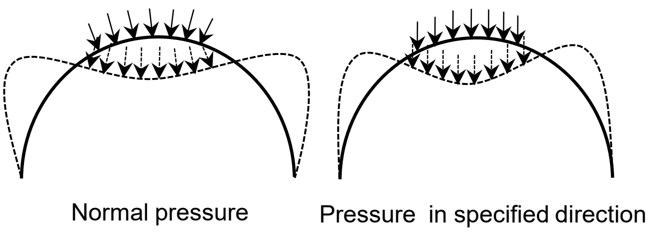


Geometry Nonlinearity (Nonlinear static / time-history analysis)

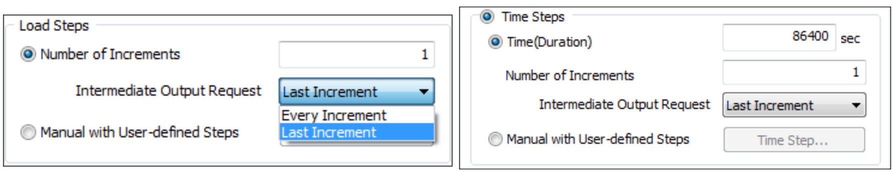


In case of large deformation analysis, the user can check more reasonable behavior with this option. This is to consider geometric nonlinear effects in stress, fully coupled and slope stability analysis. Analysis can take into account load nonlinearity which is reflecting the effects of follower loads, where the load direction changes with the deformation. Depending on the deformed shape, the pore water pressure can be updated automatically.

▸Directional change of pressure load due to the large deformation



Load steps (or Time Steps)

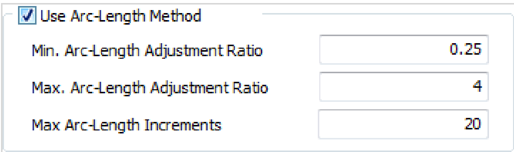


A static load can be used for nonlinear static analysis. The defined load sum can be applied at once or in stages, as an increment, cumulatively. If the load increment is too large, it may be hard to calculate the converging solution and if the load increment is too small, unnecessary is spent on calculations. In case of considering time-dependent material, the user can define Time steps to check the results with time elapse.

Convergence Criteria

Because nonlinear analysis uses iteration methods, the converge condition can be used to determine whether the solution has converged. The convergence is determined by comparing the displacement, member force or energy change in the previous calculation with the reference values. If all selected conditions are satisfied, the iteration is determined to have converged.

Use Arc-Length Method



FEA NX uses the Newton-Raphson method, where the increments are calculated to minimize the error repeatedly, as a base for calculating the nonlinear analysis solution. The Full Newton-Raphson, which renews the stiffness matrix for each repeated calculation, is basically used and the Newton-Raphson method or Initial stiffness method can be used at the renewal point. Also, other various options such as the line search method to improve the convergence, or arc length method, to calculate the unstable equilibrium state, can be used (Refer to Chp.5-5 of the Analysis manual for more details). The iterated calculation method repeats the calculation until a satisfactory solution is obtained. If there is no accurate numerical basis, the initial setting value is recommended.

Minimum arc-length adjustment ratio

Input the minimum change to the initial arc length to current increment arc length ratio. This prevents the arc length from becoming infinitely small.

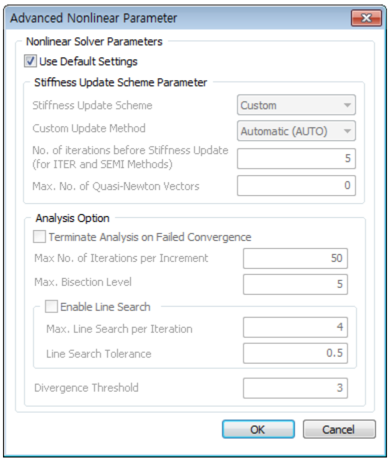
Maximum arc-length adjustment ratio

Input the maximum change to the initial arc length to current increment arc length ratio. This prevents the arc length from becoming infinitely large.

Maximum arc-length increments

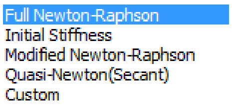
Input the maximum number of increments. Nonlinear analysis using the explicit arc length method is conducted until the load factor is larger than 1, or when the number of increments reaches the maximum value. The explicit arc length method may not work, according to the load in the problem, and the number of maximum allowable load increments is input to prepare for this.

Advanced nonlinear setting



The basic settings use the nonlinear analysis parameters and the [Use default settings] option is selected for most problems. The detailed settings are as follows.

Stiffness update scheme parameter

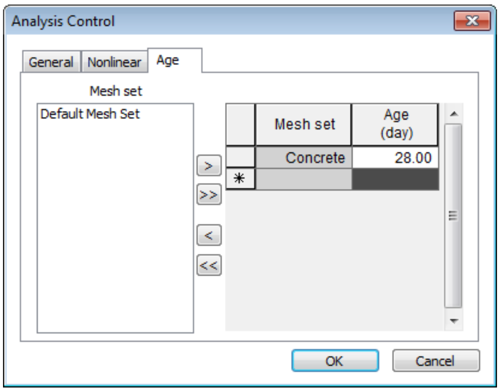


The Full Newton-Raphson, which renews the stiffness matrix for each repeated calculation, and the Initial stiffness method, which maintains the initial stiffness matrix and has very weak nonlinearity, are available. Other options such as the Modified Newton-Raphson method or Secant method, which increases the convergence and efficiency of the Newton-Raphson material properties, can be selected. Refer to Chp.5 of the Analysis manual for more details on the algorithms. The user can also specify a method to recompose the stiffness matrix by selecting repetition, semiautomatic and automatic.

Analysis option

[Terminate Analysis on Failed Convergence] : Close analysis when convergence fails. If the option is not selected, the analysis is continuously conducted even when the values to not converge.
[Max number of Iterations per Increment] : Input the maximum number of iterations for one increment.
[Maximum Bisection level] : Specify the maximum division level.
[Enable Line Search] : Use the line search feature. This feature is helpful for problems with flexible structures, where the stiffness increases with the load, or if the nonlinear analysis solution converges while vibrating. It may only increase the analysis time when used on an ineffective problem.
[Max Line Search per Iteration] : Input the maximum number of line search per repeated calculation.
[Line Search Tolerance] : Input the line search tolerance.
[Divergence Threshold] : Specify the number of allowable diversions if the value does not converge. The modified Newton-Raphson method renews the stiffness matrix at the start of each load increment.

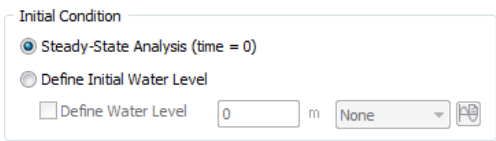
Age



In case of construction stage analysis, the user can take Age into account to consider creep / shrinkage effect generated in the previous stage. For the time-dependent material, the user, in general, can enter the curing period of concrete.

Initial condition (Seepage)

This option specifies the initial pore water pressure distribution in the ground for transient seepage analysis. The initial conditions must be set for the transient analysis. The initial condition can be selected by using the values at time '0(zero)' of the transient time step, using an arbitrarily set water level height, or using the water level function.



Safety factor (SRM)

Input the initial safety factor and the safety factor increment for each repeated calculation step. The resolution of safety factor can also be set.
[Resolution of Safety Factor] - Slope analysis using SRM uses the strength reduction method, and the resolution of safety factor value can be input to specify the accuracy of the safety factor calculation. The resolution of safety factor is used as a convergence standard for stability analysis. However, if the resolution of safety factor is entered too low, the analysis time increases greatly and so, the following guideline needs to be used to input an appropriate value.

Safety factor accuracy	Applicability
0.05	Low(Use as initial review)
0.01	Average
0.005	High

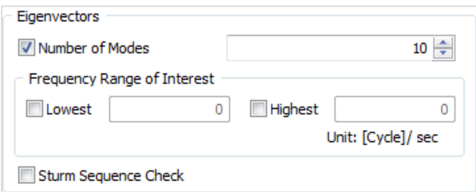
▸Table. Dynamic analysis-Analysis control options for each analysis type

Tab	Eigenvalue, Response spectrum	*Linear time history (Modal/Direct)	*Nonlinear time history, * Nonlinear time history +SRM	*2D equivalent linear
General	Initial temperature	Water pressure (Automatic)	Water pressure (Automatic)	-
	Water level	In-situ analysis	In-situ analysis	

	Eigenvectors	Water level	Water level	
	Saturation effects	Eigenvectors	Saturation effects	
	Max negative pore pressure	Saturation effects	Max negative pore pressure	
	Undrained condition	Max negative pore pressure	Undrained condition	
	Mass parameters	Undrained condition	Mass parameter	
	-	Mass parameter	-	
	-	-	-	
Nonlinear	-	-	Geometry Nonlinearity	-
			Converge standard	
			Advanced nonlinear setting	
Dynamic analysis	Modal combination type	Damping definition	Damping definition	Effective shear strain
	Damping definition	-	-	Convergence
	Interpolation of spectral data			Interpolation control
	-			Mass parameters
Slope stability (SRM)	-	-	Define time	-
			Nonlinear parameters	
			Convergence criteria	
			Safety factor	
			Advanced nonlinear parameters (Use arc - length method)	

<* : Time step setting analysis type >

Eigenvectors



Input the number of natural frequency modes (number of modes) to input and specify the range to search. The option to check for any omitted eigenvalues can be applied.

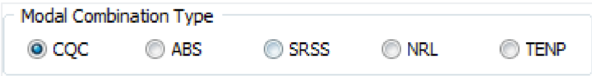
Mass parameters

[Coupled Mass Calculation]: Use a mass matrix that considers the coupling between modes. Check to use a consistent mass matrix, and uncheck to use a lumped mass matrix. It is hard to determine which is more accurate, but for eigenvalue analysis, using a lumped mass matrix displays a more flexible behavior than using a consistent mass matrix.

Modal combination type

If the maximum actual physical quantity is assumed to be the maximum physical quantities (maximum values for displacement, stress, member force, reaction force etc.) of each mode, the maximum values of each mode can simply be added. But because there is no guarantee that the maximum values of each mode occur on the same time step, it is difficult to express the maximum actual physical quantity through simple linear super positioning.

Hence, a mode combination method to approximate the maximum value is needed. Various mode methods are suggested, but because no one combination can give the appropriate approximation for all cases, the characteristics of each mode combination needs to be understood. The modal combination types are as follows, and refer to Ch.5 of the Analysis manual for more detailed algorithms.



ABS(summation of the Absolute value)
This method assumes that all mode responses occur on the same phase and the maximum value for each mode is judged to occur on the same time step, giving the most conservative results.

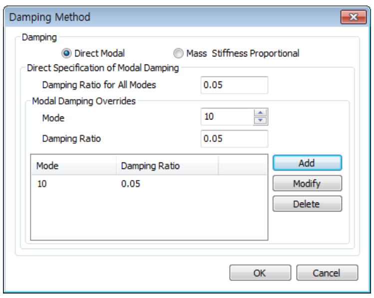
SRSS(Square Root of the Summation of the Squares)
This method provides appropriate results when each mode is sufficiently separated.

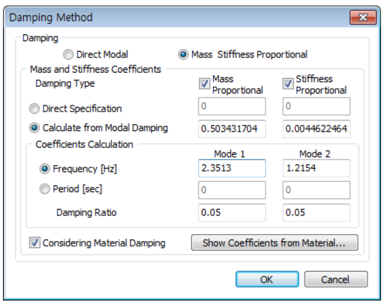
NRL(Naval Research Laboratory method)
This method removes one mode(m) that has the maximum absolute value from the SRSS method, and like the SRSS method, this method provides appropriate results when each mode is sufficiently separated.

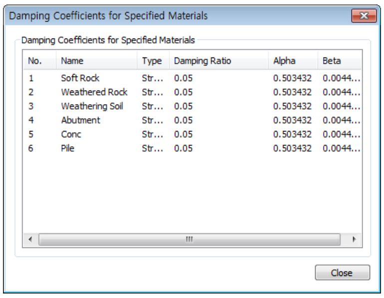
TENP(TEN Percent method)
This method includes effect of adjacent frequency modes in the SRSS. In other words, if two mode frequencies satisfy the following, the two modes are determined to be adjacent, within 10% of the frequency.

CQC(Complete Quadratic Combination method)
If the cross-correlation coefficient between modes is 1, it displays the same results as the SRSS method.

Damping definition







Direct modal

The user directly defines the damping ratio of each mode, and the mode response is calculated using that ratio. The direct modal method is only activated for Response spectrum / Time history (Modal) analysis.

[Damping Ratio for All Modes]
Define the default damping ratio that is applied to all modes, except for the ones defined by the user. The default damping ratio is applied to all modes that have a lower priority than the specified mode. If the input damping ratio is different from the damping ratio of the response spectrum function, the spectrum data is adjusted with reference to the input damping ratio and used for analysis.

[Modal Damping Overrides]
It is used to directly input the damping ratio for each mode. The mode number and mode damping ratio are input separately and then added.

Mass Stiffness Proportional

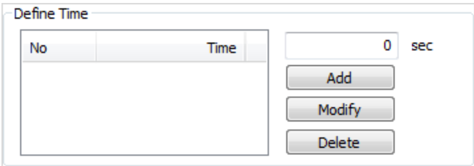
Compute the damping constant for mass proportional attenuation and stiffness proportional attenuation. The proportional coefficient can be directly input, or automatically calculated from the mode attenuation, for checked items on the attenuation type.
Input the mode frequency or period and specify the damping ratio to automatically calculate the proportionality coefficient.
Here, the attenuation for each material, when calculating the mass & stiffness coefficients from the modal damping, can be reflected in the analysis. The damping ratio of each material, input in the [Show Coefficients from Material], and the damping coefficient (alpha, beta) of the damping matrix, calculated using that value, can be checked.

Interpolation of Spectral Ratio

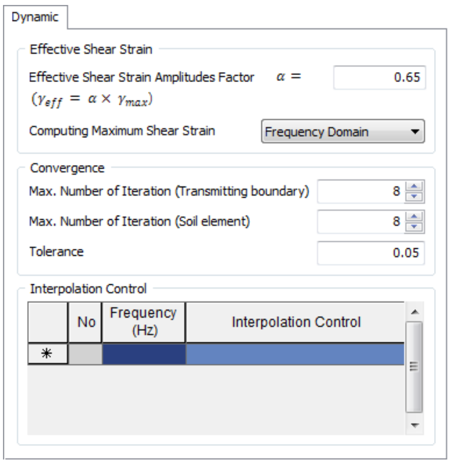
Select the interpolation method for the response spectrum load data. Both linear interpolation or logarithmic interpolation can be used for the spectrum data period and the default setting is the logarithmic interpolation method. If multiple damping ratios are in the spectrum data, interpolation of the damping ratio also follows this option. Spectrum data with one damping ratio cannot be interpolated and is corrected using the following equation. (1.5/(40xAttenuation+1) + 0.5)

Define Time (Nonlinear time history + SRM)

Specify the time to view the SRM analysis results. Multiple time steps can be specified. The SRM stability assessment is conducted using the nonlinear time history stress results from the specified time period.



Effective shear strain (2D equivalent linear analysis)



The shear strain of the ground changes with the input seismic motion or vibration load. To apply equivalent linear analysis, the concept of effective shear strain is introduced, and the material properties are simplified to have equivalent linear values for calculation.

Frequency domain analysis is analyzed to have a certain shear modulus and damping for each frequency, and the material nonlinearity cannot be considered. Hence, the 2D equivalent linear analysis uses iterated calculations, using the changing ground stiffness and damping ratio due to the shear strain calculated in the previous stage, to consider the nonlinear behavior of the ground. Here, the maximum shear strain used in the previous stage is multiplied by a certain value(50%~70%) smaller than 1 to define the effective shear strain. The effective shear strain is used because the maximum shear strain generates a larger strain energy than the actual behavior.

Generally, an effective shear strain coefficient of 0.65 (65%), or the $(M-1)/10$ value that uses the earthquake magnitude M is used. Also, a maximum shear strain calculation method in the time domain is supported to calculate shear strain more precisely than the maximum shear strain found using the RMS(root mean square) in the frequency domain.

$$\gamma_{\max} = \max_t \{\gamma_{oct}(t)\}$$

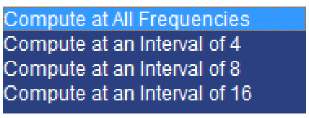
$$\gamma_{oct}(t) = \frac{1}{3}\sqrt{\varepsilon_x^2(t) + \varepsilon_z^2(t) + \{\varepsilon_x(t) - \varepsilon_z(t)\}^2 + 6\gamma_{yz}^2(t) + 6\gamma_{xz}^2(t)}$$

▸Difference between maximum and effective strain

There are two methods to calculating the maximum shear strain; the time domain and frequency domain. The time domain method defines the load (acceleration, force etc.) changes according to time and composes the structural state as a differential equation. Hence, the structural response (displacement, velocity, acceleration response) can be calculated by performing the integration for every time interval. The frequency domain method is useful when determining the relationship and ratio between the load response and frequency characteristics. Because it is hard to determine this relationship and ratio for irregular waves such as earthquake response, the wave in the time domain is converted to the frequency domain and used for analysis.

Interpolation control

Input the frequency range for frequency domain analysis. Interpolation methods are used to efficient frequency domain analysis and one of the four methods can be selected.



Select [Coupled Mass Calculation] to conduct the analysis for all frequencies and if the interval is specified, the analysis frequency interval in the frequency domain becomes the set interval.

1.4 Time Step

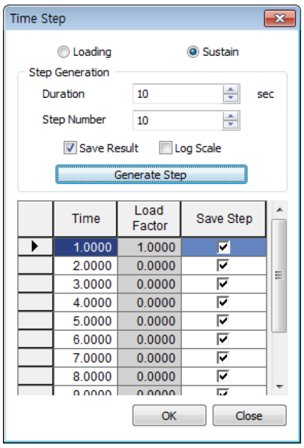
Overview

The time step is specified for result examination with time in Consolidation, Seepage (transient), Dynamic (linear/nonlinear time history) analysis. The analysis result can be checked at every specified time interval and for dynamic analysis, the time history result graph is provided.

Methodology

Consolidation (singular) analysis, Stress seepage-Fully coupled – Loading/Sustain

Specify the applied time step and load factor. In consolidation analysis, the analysis can be simulated using 'Sustain' option without creating the construction stage. User can define loading and leave time separately. All applied loads including self-weight is used for analysis, according to the input time and load factor. The analysis is conducted by simulating the dissipation of excess pore water pressure in the lead load stage, according to the defined time information, after the [Loading] analysis has finished. Because of this, the load factor in the [Sustain] cannot be input and the time information is added to the final time of the [Loading].



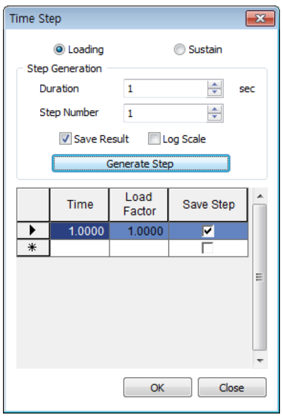
Construction stage consolidation analysis - Input time and load

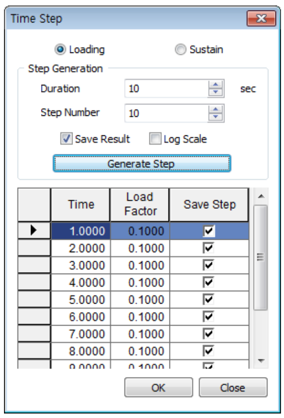
For consolidation analysis, the appropriate load application time and load distribution of an added load can be done by inputting the time and load factor.

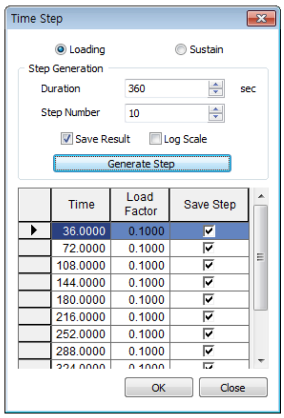
▸Initial state

▸▸Embankment load application

▸▸▸Sustained period



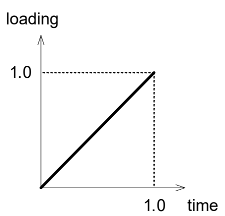


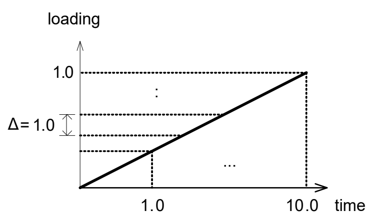


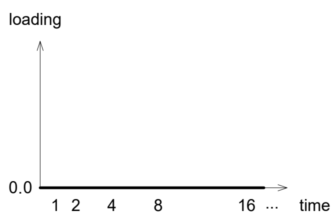
▸Initial state

▸▸Embankment load application

▸▸▸Sustained period





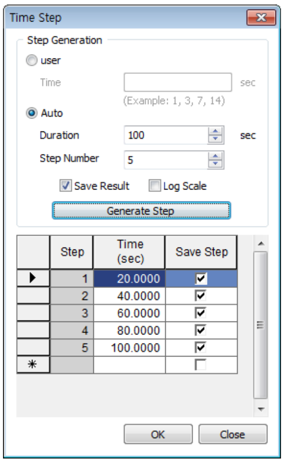


When using a Modified Cam-clay material model, the in-situ stress needs to be defined. Hence, drainage analysis is needed to calculate in-situ stress and in this case, the application time is meaningless.

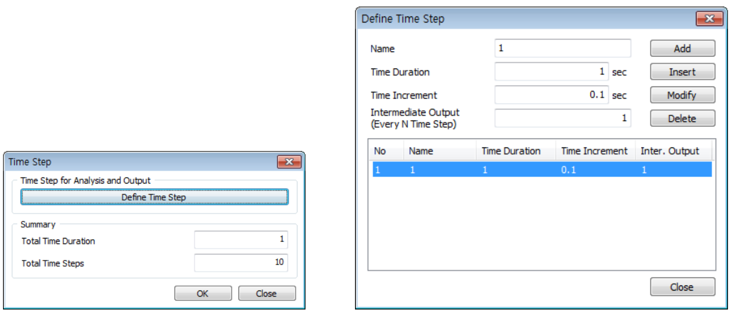
If an additional load is applied, the load is applied over an appropriate total time, as shown in the 'Embankment load application' figure above. The figure displays the additional load applied in 10 equal stages over 10 days. Here, the sum of the load factors is 1.0.
After the embankment has finished, the leave analysis can be conducted where the load is left continuously without any additional increase. The leave time is defined as shown in the 'leave period' figure above. Because of the characteristics of consolidation analysis, the initial stress change happens rapidly. Hence, the first time step can start as small and gradually increase with time to pursue both convergence and analysis speed.

Seepage(Transient analysis) - Define time step

Specify the time step for performing analysis and saving results. The seepage boundary time function value (heat, flow rate etc.) that corresponds to the set time step is used for analysis, and the results can be examined. Values outside the time range of the seepage boundary time function can be automatically applied using linear interpolation.



Dynamic analysis – Define time step



It is possible to assign time step using irregular intervals and so, the target time step is assigned a name.

- Time Duration : Insert the time interval
- Time Increment : Input the time interval that will be used for calculation for the entered time range.
- Intermediate Output : Input the time interval for the output of analysis results. The results are printed for every time interval x print interval for interim results.

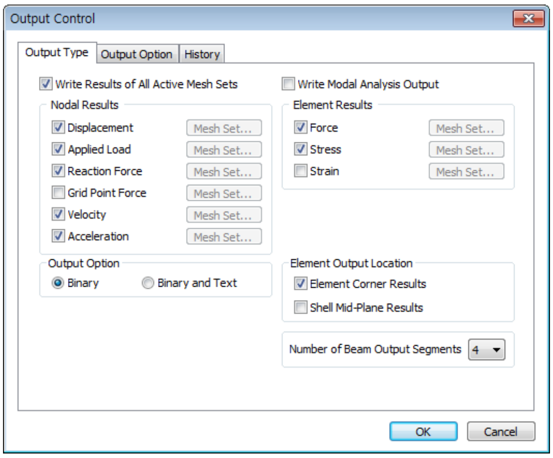
※ At every assigned interval, a result is printed on the result tree. If you need to print out any amount of data, it might take a considerable amount of time. In case your PC is lack of RAM, the result file might not be created.

According to the assigned time step, it shows total amount of time elapsed (total amount of time used for analysis according to the time step), the total number of time step (The total number of time steps used during the analysis).

1.5 Output Control

Overview

Specify the Output type/print option (element result print coordinate system) and for dynamic analysis, the time history to print. The node result and element results are set as default, depending on the analysis type, and the desired results in each result column can be selectively saved and printed, allowing efficient management of analysis result file size and print time etc.



Methodology

It is possible to print out every mesh set used in the analysis, or select only the main mesh set that needs result confirmation. Check the [Element Corner Results] option to output the calculated results for each element node on the element result directly. Uncheck this option to view the average output of each node. The checked result is the same as the legend results when viewed using contours.

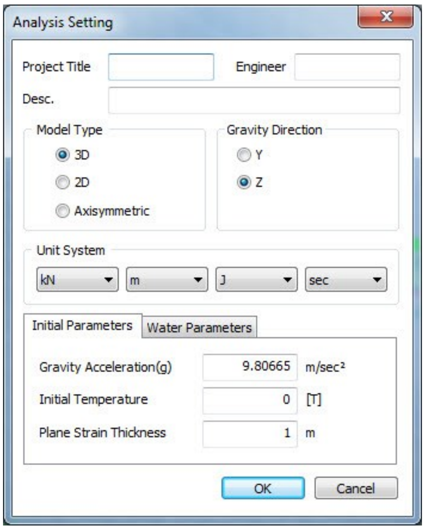
If the neutral plane result of shell element is needed, check the [Shell Mid-Plane Results] option.
For a beam element, the number of printed segments can be adjusted.
The results can be printed out at points 1(i, j), 2(i, 1/2, j), 3(i, 1/3, 2/3, j), 4(i, 1/4, 1/2, 3/4, j).
The analyzed result are arranged as a separate text file in the folder in which the model is saved.

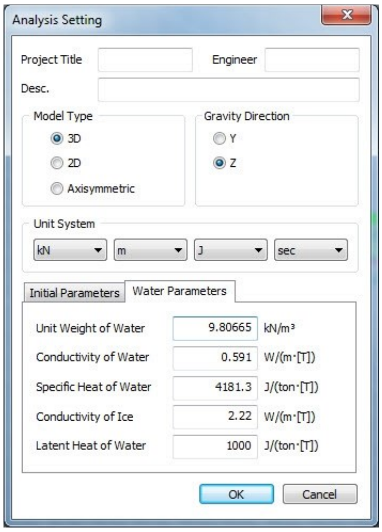
1.6 Analysis Setting

Overview

Specify the model type (2D/3D) and setup the working environment. The sub menus, such as available features or elements, are determined by the set environment. Here, the unit system and initial variables of the analysis model is set, and these values can be modified during the modeling process.

Methodology



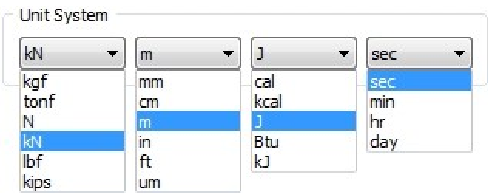


[Model Type]
Select the model type. For a 3D model, the gravitational direction can be set as Y or Z and for the 2D or axis symmetric model, the gravitational direction is fixed in the Y direction.

[Unit System]
Setup project unit for force/length/energy/time. Selected system of unit can be changed anytime during the modeling process and on the status bar in the lower right-hand corner.

The provided unit systems are as follows.

▸Force/length/energy/time



When importing a geometric shape from an DXF(2D/3D) file, AutoCAD does not have a length unit so, the imported shape follows the units set on the program. When importing a CAD geometry, the length unit of the object model can be separately specified by importing the length unit file.

[Initial Parameters]
When converting the mass of dynamic loading and self-load by material’s unit weight or calculating pore water pressure, unit weight of water and acceleration of gravity is set as variables. Initial temperature is a variable that is assigned when thermal load is inserted. The difference between set initial temperature and inserted thermal load would be converted into weight and assigned.

Additional input for thermal analysis is provided:

[Specific Heat of Water] : determines amount of heat (energy) can be stored in the pore water per unit of mass.
[Conductivity of Water] : determines rate of heat (energy) can be transported in the pore water.
[Conductivity of Ice]: determines rate of heat (energy) can be transported through ice.
[Latent Heat of Water] : amount of energy required for water phase change (from liquid to solid state and vice versa).

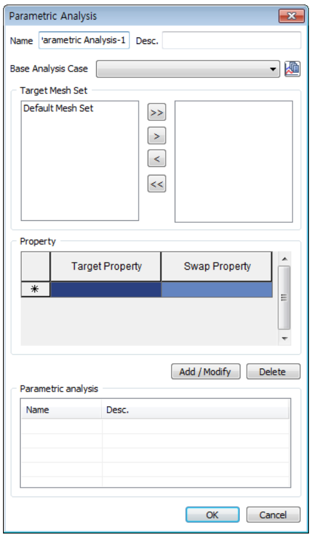
* For 2D analysis, the planar deformation thickness of the ground needs to be directly input to be applied to the analysis. Generally, the unit width is input, with respect to the set length unit system.

1.7 Parametric Analysis

Overview

This feature is useful when changing the properties (material) of a particular element in a model and conducting the analysis to compare results. The notable example is the Ko parameter analysis. For example, when conducting the tunnel excavation construction step analysis, comparing the analysis results by changing the coefficient of earth pressure (Ko) value as 0.5, 1, 1.5 in stages, the model is not separated and only the property is added for analysis. Changing various conditions, not only ground materials but also structural characteristics (section characteristics), is possible for iterative calculations and all outcomes can be compared at once.

Methodology



First, select the base analysis case. Automatically create the parametric analysis case by assigning the changed characteristic conditions below.

Select the target mesh set and select the target property change (current properties of the selected element) and changed property sequentially. Select the Add/Edit button to create a parametric analysis case. It is good to add properties in advance to the parameters for the change property setting.

Section 2 Analysis

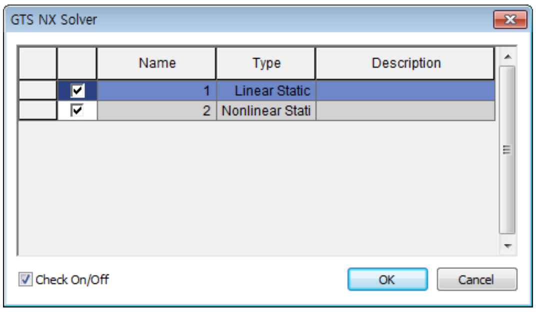
2.1 Perform/ Batch Analysis

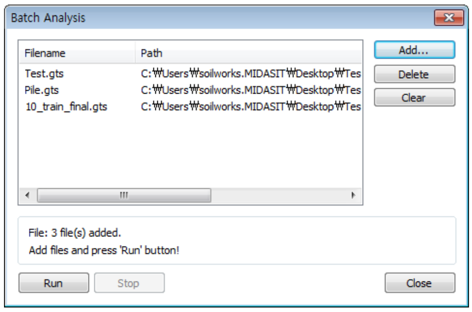
Overview

Conduct analysis on the created analysis case. If multiple analytical cases have been created, select the cases to conduct analysis upon and conduct arrangement analysis to analyze multiple models simultaneously.

Methodology

▸Perform/Batch Analysis





Select the analysis cases to conduct analysis upon. Arrangement analysis is useful when performing analysis on multiple models. During analysis, the convergence, warning/error can be checked on the output window. After analysis, the automatically created text file in the saved folder can be used to review the analysis results, convergence and the warning messages that occurred during analysis.

Section 3 History

3.1 History Output Probes

Overview

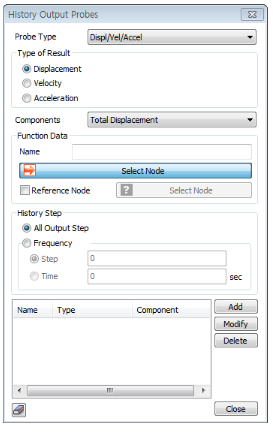
This operation specifies the function that outputs the results at a particular location in graph form for time variant analysis cases (Transient seepage, Consolidation, Fully-coupled stress-seepage analysis, Linear/nonlinear time history analysis, 2D Equivalent linear). Because printing the entire analysis results on the analysis tree is inefficient in terms of both usability and performance, the user can specify the desired location effectively and apply it to analysis. In other words, the history result search node must be registered before conducting analysis.

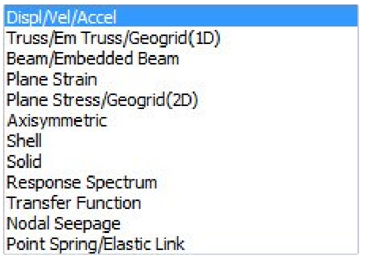
Methodology

The following process can be performed to output the history result graphs for all time variant analysis methods. The printable history results include ground deformation, member force and seepage results.

1. Select the Probe type (Analysis > History > History Output Probes > Probe Type) and select the Object to check the result.

▸History result search and search type





Probe Type	Type Of Result
Displ/Vel/Accel	Displacement, Velocity, Acceleration,
Truss/Em Truss/Geogrid(1D)	Strain, Stress, Force, Seepage,Hinge Force, Hinge Deform
Beam/Embedded Beam	Displacement, Velocity, Acceleration,
Plane Strain	Strain, Stress, Force, Seepage
Plane Stress/Geogrid (2D)	Strain, Stress, Force, Seepage
Axisymmetric	Strain, Stress, Force
Shell	Strain, Stress, Force, Seepage
Solid	Strain, Stress, Seepage
Response Spectrum	Rel Displacement, Rel Velocity, Abs Acceleration, Pseudo Rel Velocity, Pseudo Abs Acceleration
Transfer Function	Displacement, Velocity, Acceleration,
Nodal Seepage	-
Point Spring/Elastic Link	Displacement, Velocity, Acceleration,

This feature can be specified for each search type and the detailed components are selected.

For element results, the results can be output for the selected node center or for each node position. However, the middle nodes of higher order elements cannot output results. For the 'Displacement/Velocity/Acceleration' type, the results calculated at the [Reference Node] can be used to deduct relative results.

In case of 'Transfer Function', it has always a relative result, the user must specify the [Reference Node].

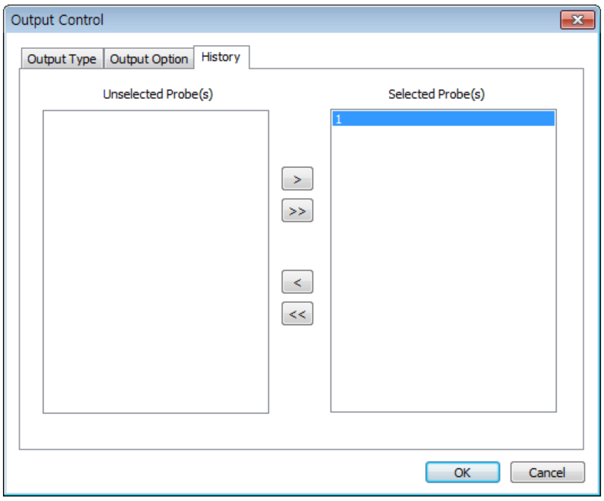
[History step] specifies the step at which the results will be output. Select [All Output Step] to output results at all the time intervals, not just at the output time steps specified in the Analysis case.

Specify the [Frequency] option when printing the results at analysis conducting step/time intervals.
For an example analysis case with an analysis time increment as 0.1,0.2,0.3 sec... and an interim result output time as 0.3,0.6sec..., specifying 'Step : 2' outputs the results at 0.2, 0.4, 0.6sec... . Entering '0.2sec' in the 'Time' option can output the results at 0.2, 0.4, 0.6sec... .

※ Be careful not to reflect search types that are not printed in the Analysis case.

2. Analysis > Analysis > General > Output control > Register the history items for result checking, added from the history tab

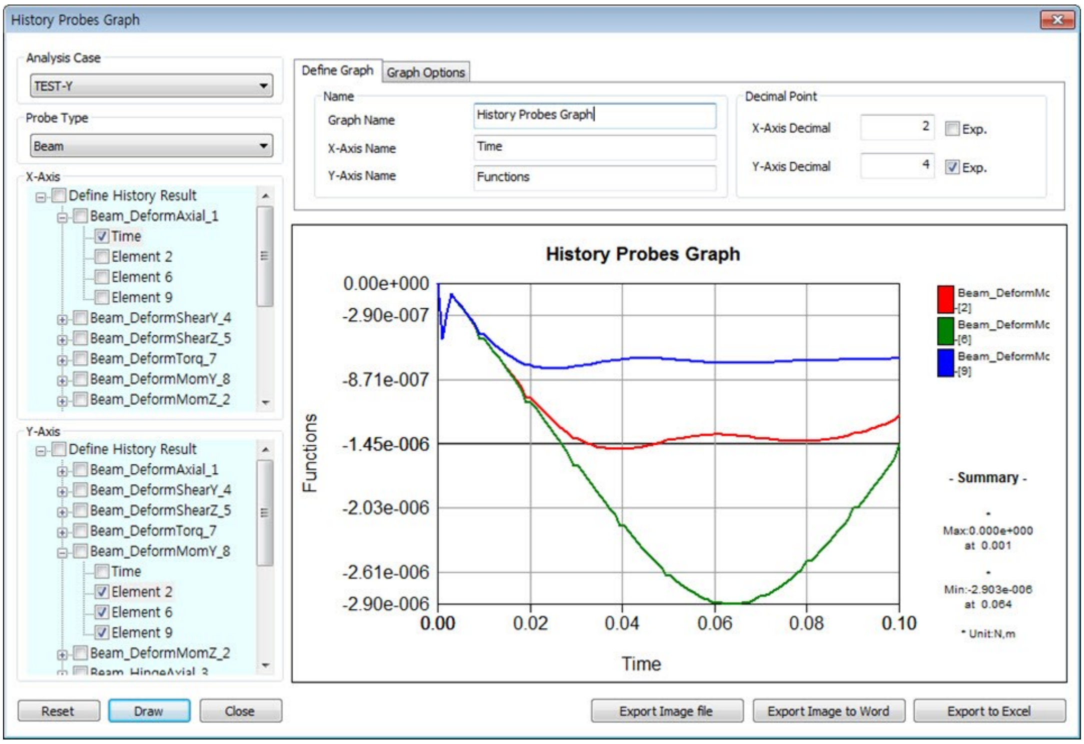
▸History result output setting



<u>3. Result > Special Post > History > Check time history result using graph function</u>

History Probes Graph allows for customizing displayed data for both X and Y axes.

▸History result search graph



<History result search graph>