Section 1 Load

1.1 Structural Load Types

The usable structural loads in GTS NX can be largely classified into force, gravity, displacement and thermal expansion due to temperature, as shown in table 6.1.1

Table 6.1.1 Usable loads in GTS NX

Types	Applicable range
Nodal force/moment	node
Surface/edge pressure load	1D element, 2D element, 3D element
Surface/edge water pressure load	2D element, 3D element
Beam load	Beam element
Gravity	All elements with mass
Specified displacement/velocity/acceleration	node
Temperature load	node, 1D element, 2D element, 3D element
Temperature gradient	Beam element, shell element
Prestress and initial equilibrium force	truss element, beam element, plane stress element, plane strain element, axisymmetric solid element, solid element

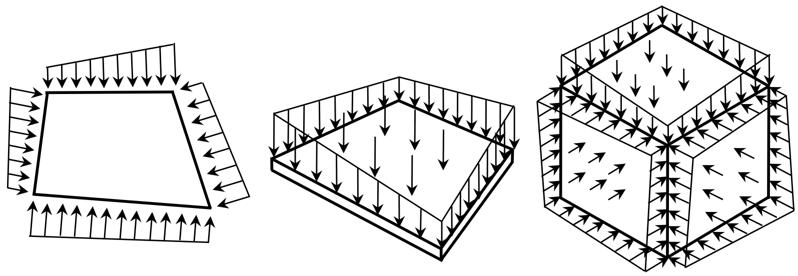
Nodal force

Nodal forces are the most basic loads and have 3 force component inputs and 3 moment component inputs for each node. The direction can be defined about an arbitrary coordinate system.

Pressure load

The pressure load is input as a distributed force form for an element face or edge. The surface pressure load is applicable for 2D or 3D elements and the edge pressure load is applicable for 1D or 2D elements. The input direction can be specified as an arbitrary coordinate axis direction, arbitrary vector direction or normal direction. Figure 6.1.1 displays the pressure load acting on various elements.

Figure 6.1.1 Pressure load acting on various elements



Water pressure load

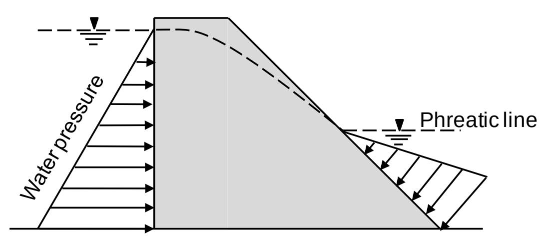
GTS NX can consider water pressure that occurs from the defined water level of a model. Water pressure is added as an edge pressure load for 2D problems and as a surface pressure load for 3D problems.

The pressure size needed for calculating the water pressure is determined as follows:

- ▶ Same size as the pore pressure defined on the target edge/surface
- ▶ Hydrostatic pressure due to user input water level position, $z_{phreatic}$:
  $-z$ for gravitational direction $p = \rho_f \, g \, (z_{phreatic} - z)$

GTS NX supports the automatic water pressure method, which applies an automatic water pressure to all free edges/surfaces. In this case, only the parts below the water level receive a pressure load automatically. On the other hand, to respond to the modeling needs for general water pressure conditions, the manual setting of water pressure for necessary edges/surfaces is also provided.

Figure 6.1.2 Water pressure generated by to water level



Self weight due to gravity

The gravity is used to model the self weight or inertial force of a structure and can be applied to all elements with mass.

Gravity is applied to all activated elements. It is also automatically applied to elements added in the construction stage. Likewise, it is automatically removed when the element becomes inactive. Analysis considering pore pressure has the self-weight due to degree of saturation as an additional external force.

Specified displacement/velocity/acceleration

Specified displacement is assigned to the displacement of particular nodes and is used when the nodal position after deformation is known. Specified displacements generate structural deformation and are classified as loads, but they have similar characteristics as boundary conditions such as the generation of confining force.

Setting the total DOF of the target problem as $\mathbf{u}_A$, it can be divided into DOF with/without assigned specified displacement as follows:

$$\mathbf{u}_A = \begin{Bmatrix} \mathbf{u}_F \\ \mathbf{u}_S \end{Bmatrix} \tag{6.1.1}$$

$\mathbf{u}_F$: DOF without assigned specified displacement

$\mathbf{u}_S$: DOF with assigned specified displacement

The stiffness matrix can also be classified and expressed using the same principle:

$$\mathbf{K}_{AA}\mathbf{u}_A = \begin{bmatrix} \mathbf{K}_{FF} & \mathbf{K}_{FS} \\ \mathbf{K}_{SF} & \mathbf{K}_{SS} \end{bmatrix} \begin{Bmatrix} \mathbf{u}_F \\ \mathbf{u}_S \end{Bmatrix} = \mathbf{f}_A = \begin{Bmatrix} \mathbf{f}_F \\ \mathbf{f}_S \end{Bmatrix} \tag{6.1.2}$$

$\mathbf{u}_S$ in the equation above is a determined value and thus the 2nd row of the stiffness matrix does not have meaning. Using $\mathbf{u}_S$ to rearrange the 1st row, the load due to specified displacement can be calculated as follows:

$$\mathbf{K}_{FF}\mathbf{u}_F = \mathbf{f}_F - \mathbf{K}_{FS}\mathbf{u}_S \tag{6.1.3}$$

Specified velocity and acceleration loads can be assigned in dynamic analysis. Specified velocity and acceleration conditions are coupled with time integration and can be converted to the specified displacement condition. The converted condition is applied to the matrix operation as explained above.

Temperature load

Temperature load can be expressed as the difference ΔT between the initial temperature and final temperature, and the strain $\boldsymbol{\varepsilon} = \boldsymbol{\alpha}(T_m)\Delta T$ is generated by the coefficient of thermal expansion $\boldsymbol{\alpha}(T_m)$ determined by the material temperature T_m . Hence the initial and final temperature of each structure needs to be defined. The method of defining the temperature is listed in table 6.1.2.

Table 6.1.2 Temperature types that can be defined on structures

Types	Applicable range
Nodal temperature	node
Element temperature	truss, embedded truss, beam, geogrid, plane stress, plane strain, shell, solid

Temperature gradient

Temperature gradient load can only be applied to beam elements and shell elements, which can consider bending stiffness. For beam elements, the temperature difference and distance of the outermost portion is input with reference to the y axis and z axis of the element coordinate system. For shell elements, the temperature difference between the top and bottom faces and the plate thickness is input to consider the temperature gradient load.

Prestress and initial equilibrium force

GTS NX uses the prestress function to apply the resultant force or stress as an initial condition, depending on the element type. If the inital stress is given as such, an internal force corresponding to the initial stress is generated. If the final external stress is not in equilibrium with the initial internal force, changes in strain and stress occur according to that difference.

The initial equilibrium force is a function that generates the internal force, corresponding to the initial stress applied by prestress, as an external force. The initial equilibrium force for the element e, with the reflected initial stress from prestress, is expressed as follows:

$$\mathbf{f}^{e}_{ext,eq} = \int_{\Omega_e} \mathbf{B}^T \ \boldsymbol{\sigma}_0 d\Omega \tag{6.1.4}$$

If additional external forces do not exist, the initial equilibrium force and initial stress form equilibrium and so, the initial state is maintained. Also, GTS NX can assume the initial stress applied on an element to be the stress generated by gravity. When this assumption is applied, the load distribution factor considering self weight is applied when the element is removed from the construction stage.

Pretension can be used for 1D elements to maintain the given axial direction resultant force. The axial resultant force of the element in the load applied step and the corresponding element internal force does not change.

1.2 Static/Dynamic Load Definition

Static load

Static loads in GTS NX are used for linear/nonlinear static analysis and consolidation analysis, and are composed of combinations of the load types introduced above. The load is applied differently for linear static analysis and nonlinear analysis.

► Linear static analysis

Linear static analysis is performed in one stage for the total load and so, the total static load is applied.

► Nonlinear static analysis/construction stage analysis

The load is applied proportional to a load scale $0 \le \lambda_I \le 1$ defined by each construction stage or as a unit of singular analysis. The total load vector in nonlinear analysis is defined in the following form.

$$\mathbf{f}_{ext}(\lambda_I) = (1-\lambda_I)\,\mathbf{f}_{int,0} + \lambda_I \sum_j \mathbf{f}_{ext}^j \tag{6.1.5}$$

Here, $\mathbf{f}_{ext}^j$ represents the external force vector due to the j th load, and $\mathbf{f}_{int,0}$ is the internal force corresponding to the initial state of nonlinear analysis or each stage of construction stage analysis, which can be calculated as the sum of initial internal forces for each element ($\mathbf{f}_{int}^e$):

$$\mathbf{f}_{int,0} = \sum_e \mathbf{f}_{int}^e(\boldsymbol{\sigma}_0, \mathbf{u}_0, \cdots) \tag{6.1.6}$$

When the load scale is $\lambda_0 = 0$, the total external forces and initial internal forces are in an equilibrium state and when the load scale is $\lambda_I = 1$ for the final state of the stage, the total external force is the total sum of the given load combinations. Particularly, the continuity of the external force is guaranteed when an element is added or removed from the construction stage and so, this can prevent deterioration of convergence due to discontinuous load properties in nonlinear analysis.

The load scale corresponding to the I th load increment, λ_I can be set to have an even interval or an arbitrary value between '0' and'1'.

Dynamic load

Dynamic loads are used to apply various changing load conditions for linear/nonlinear time history analysis. The total load vector in time history analysis is defined as the following function of time, t.

$$\mathbf{f}_{ext}(t) = \sum_j T_j(t)\,\mathbf{f}_{ext}^j \tag{6.1.7}$$

Here, $T_j(t)$ is the j th load scale and it is generally defined as a tabular data form for time, or by inputting coefficients for a particular function (equation 6.1.8):

$$T_j(t) = (A + C\,t)e^{-Dt}\sin(2\pi f\,t + \bar{P}) \tag{6.1.8}$$

1.3
Construction Stage Analysis Load

Static loads are used for construction stage stress analysis, and the body force due to self weight is automatically reflected in the total external force for the addition and removal of elements in each stage.

Load distribution factor

When partial elements are removed in a construction stage due to excavation, GTS NX supports construction stage analysis that uses the load distribution factor (LDF) such that the effects of element removal are reflected gradually for multiple construction stages.

The internal forces and body forces from the removed elements in a construction stage can be removed gradually in later stages and the load distribution factor s_{LDF} controls the size of the force the removed element once shared. Ultimately when s_{LDF} is '0', the effects of the removed element completely disappear (figure 6.1.3). The LDF of each stage is defined by the user:

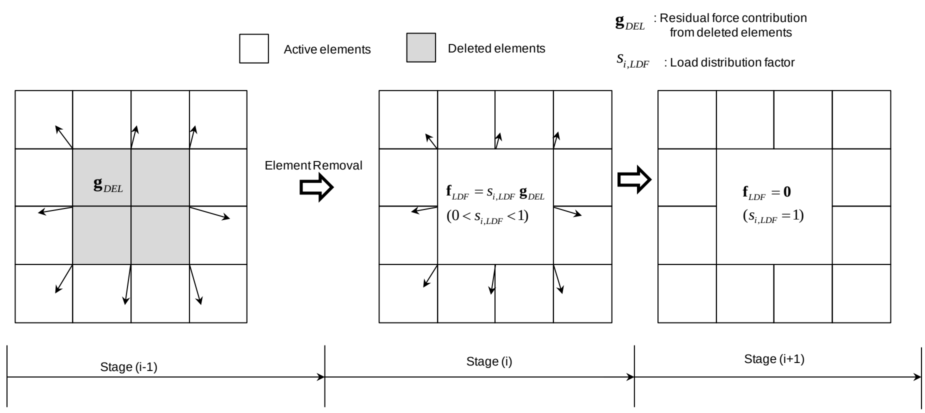


Figure 6.1.3 Construction stage load from load distribution factor (LDF)

The loads which were shared by the removed elements act as external forces in later stages, and can be expressed as follows:

$$\mathbf{f}_{LDF} = s_{LDF}\,\mathbf{g}_{DEL}$$
$$\mathbf{g}_{DEL} = \sum_{e}^{N_D}\int_{\Omega_e}\mathbf{B}^T\boldsymbol{\sigma}\,d\Omega - \mathbf{W}_G^e \qquad (6.1.9)$$

$\mathbf{f}_{LDF}$: Load distribution external force vector

$\mathbf{g}_{DEL}$: Load sharing vector for the unbalanced force due to the removed element

$\mathbf{W}_G^e$: Body force due to element self-weight

1.4 Nonlinearity of Loads

The nonlinearity of the system often results from material nonlinearity or geometric nonlinearity. In addition, there are cases where nonlinear analysis is needed for systems with load nonlinearity. For example, when the load direction changes with the structural displacement or when the load size is dependent on the structural behavior. GTS NXL can reflect the effects of follower loads, where the load direction changes with the structural displacement when performing geometric nonlinear analysis.

Follower force

When the nodal force direction is determined by the relative position of two different points, its size and direction are as follows:

$$f_\beta = Fn_\beta = F\frac{(x_{k\beta} - x_{j\beta})}{\left\|\mathbf{x}_k - \mathbf{x}_j\right\|} \tag{6.1.10}$$

j, k : Nodes that determine the load direction

β : Load component (x, y, z)

Differentiating f_β for $x_{j\alpha}$ is as follows:

$$\frac{\partial f_\beta}{\partial x_{j\alpha}} = F\left(\frac{(x_{k\alpha} - x_{j\alpha})(x_{k\beta} - x_{j\beta})}{\left\|\mathbf{x}_k - \mathbf{x}_j\right\|^3} - \frac{\delta_{\alpha\beta}}{\left\|\mathbf{x}_k - \mathbf{x}_j\right\|}\right) \tag{6.1.11}$$

Using the same process, the derivative of $x_{k\alpha}$ can be calculated and as a result, an asymmetric stiffness matrix is composed. This stiffness due to the nonlinearity of load is called the load stiffness, and it is generally a matrix with a strong asymmetry.

The direction of the pressure acting normal to a particular face of the structure changes as the direction of that face changes. Figure 6.1.4 displays the large deformation effects when the forces acting on a structure face are normal or in a specified direction.

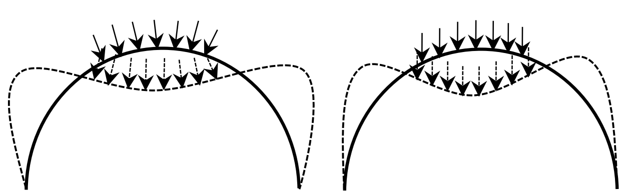


Figure 6.1.4 Directional change of pressure load due to large deformation

The load stiffness calculation process[1] for pressure acting normal to the element face is as follows. The nodal force of the I th node is integrated and calculated using the pressure $p(\xi,\eta)=p^J N^J(\xi,\eta)$ as follows:

$$\mathbf{f}^I=\iint p(\xi,\eta)\frac{\partial \mathbf{x}}{\partial \xi}\times\frac{\partial \mathbf{x}}{\partial \eta}N^I(\xi,\eta)d\xi d\eta \tag{6.1.12}$$

The geometric stiffness $\mathbf{K}^{IJ}$ can be calculated by differentiating the equation above at a nodal position:

$$\mathbf{K}^{IJ}=\frac{\partial \mathbf{f}^I}{\partial \mathbf{x}^J}=\begin{bmatrix}\dfrac{\partial \mathbf{f}^I}{\partial x_1^J} & \dfrac{\partial \mathbf{f}^I}{\partial x_2^J} & \dfrac{\partial \mathbf{f}^I}{\partial x_3^J}\end{bmatrix} \tag{6.1.13}$$

Each column composing the stiffness can be found from the derivative of $\mathbf{x}$ in equation (6.1.12), which changes with the displacement:

$$\frac{\partial \mathbf{f}^I}{\partial x_j^J}=\iint p\left[\left(\frac{\partial}{\partial x_j^J}\frac{\partial \mathbf{x}}{\partial \xi}\right)\times\frac{\partial \mathbf{x}}{\partial \eta}+\frac{\partial \mathbf{x}}{\partial \xi}\times\left(\frac{\partial}{\partial x_j^J}\frac{\partial \mathbf{x}}{\partial \eta}\right)\right]N^I d\xi d\eta \tag{6.1.14}$$

Apart from this, beam element load, gravity, rotational inertial force etc. have follower force effects, and the changing effects according to the displacement for each load can be selected for consideration. Table 6.1.3 displays the load types that can consider the follower force effects:

Table 6.1.3 Loads that can consider the follower force effects in GTS NX

Type	Applicable range/method
Nodal force	When direction is defined by two nodes / Use load stiffness
(Water)pressure load	When load acts normal to an element face / Use load stiffness
Beam load	When defined with reference to ECS / Use load stiffness
Gravity	Use load stiffness

[1] Hibbit, H.D., “Some follower forces and load stiffness”, International Journal for Numerical Methods in Engineering, Vol. 4, 1979

Section 2 Boundary and Constraint Conditions

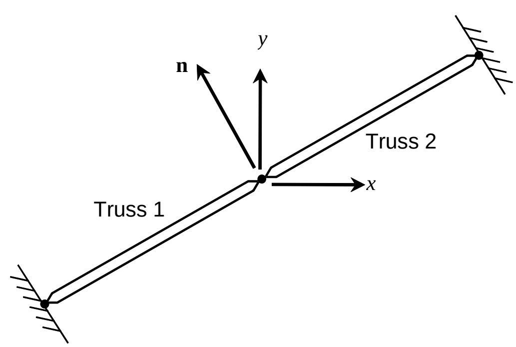
Constraint conditions can be largely divided into single-point constraints and multi-point constraints. Single-point constraints represent constraint conditions that are applied to a single node, and multi-point constraints represent a constraint that is applied between multiple DOFs to form a particular relationship.

2.1 Single Node Constraint

Single node constraints selectively fix the DOF components of individual target nodes, resulting in the removal of that DOF. Single node constraints are generally applied where actual displacement does not occur or is used to set the symmetry condition.

Apart from this, single node constraints can also be used to remove DOFs that should not be reflected in the analysis. This method is the same as removing the singularity of the stiffness matrix, and setting the direction for constraint condition application appropriately is important. For example in the figure below, the node where the two truss elements meet only has an axial DOF and thus DOFs in the other directions should not be reflected in the analysis. These DOFs need to be appropriately constrained and the singularity needs to be removed from the stiffness matrix, but an appropriate constraint direction cannot be specified when the coordinate axis is defined in the x-y coordinate system, as shown in the figure. Hence, a new nodal displacement coordinate system needs to be defined, including the element axial direction and the vector $\mathbf{n}$ normal to it, and the $\mathbf{n}$ direction needs to be constrained to achieve an appropriate constraint condition required for analysis.

Figure 6.2.1 Example of nodal displacement coordinate system use



2.2 Multiple Node Constraint

Multiple node constraints use the linear relationship between multiple nodal DOFs to apply the constraint condition, and the general linear relationship form is as follows:

$$R_j u_j = 0 \tag{6.2.1}$$

R_j : Linear relationship coefficient

u_j : DOF related to constraint condition

When there are multiple node constraints, it can be expressed in the following matrix form:

$$\mathbf{R}_M \mathbf{u}_M = \mathbf{0} \tag{6.2.2}$$

To apply the multiple node constraint to simultaneous equations, $\mathbf{u}_M$ is classified into the independent and dependent DOFs, and the dependent DOFs are eliminated from the simultaneous equation. First, the DOFs that are involved in the multiple node constraint are classified into the independent DOF $\mathbf{u}_I$ and dependent DOF $\mathbf{u}_D$.

$$\mathbf{u}_M = \begin{Bmatrix} \mathbf{u}_I \\ \mathbf{u}_D \end{Bmatrix}, \ \mathbf{R}_M = \begin{bmatrix} \mathbf{R}_I & \mathbf{R}_D \end{bmatrix} \tag{6.2.3}$$

Using the equation above, equation (6.2.2) can be expressed as follows:

$$\mathbf{R}_I \mathbf{u}_I + \mathbf{R}_D \mathbf{u}_D = \mathbf{0} \tag{6.2.4}$$

If the $\mathbf{R}_D$ inverse matrix exists in the given equation, the relationship between the independent and dependent DOFs can be reorganized as follows:

$$\mathbf{u}_D = -\mathbf{R}_D^{-1} \mathbf{R}_I \mathbf{u}_I = \mathbf{G} \mathbf{u}_I \tag{6.2.5}$$

This equation can be used to eliminate the dependent DOF $\mathbf{u}_D$ from the simultaneous equation that is composed of the entire model.

The applicable range of multiple node constraints is very large and it can be used for the following cases:

- ▶ When simulating relative motion between two nodes
- ▶ When simulating hinges or sliding joints
- ▶ When combining the adjacent sections of two elements with different number of nodal DOFs
- ▶ When applying the load after dispersing it first

▶ When applying the constraint condition in a direction not equal to the displacement coordinate system assigned to the node

The constraint between DOF that occurs in rigid/interpolated elements are a type of multiple node constraint. For behavior expressed using rigid/interpolated elements, it is more convenient to use these elements than to use the multiple node constraint.

2.3 Automatic Single-point Constraint

GTS NX provides the automatic single-point constraint function, which automatically finds stiffness matrix singularities and applies the constraint conditions in the node units. When this function is used, the 3x3 stiffness matrix consisting of the nodal displacement or rotation is analyzed and the constraint conditions are generated in the direction where the stiffness closes to '0'.

When the automatic constraint is applied to models consisting of truss elements as shown in figure 6.2.1, constraint conditions are automatically generated in the $\mathbf{n}$ direction, where there is no stiffness component, without the need for defining the single node constraint mentioned above. In this case, it is important to define the nodal displacement coordinate system that includes the element axial direction and the vector $\mathbf{n}$ normal to it.

2.4 Constraint Force Calculation

Constraint forces act on DOFs where constraint conditions are applied.When the solution is found, the constraint forces from the single and multiple node constraints can be calculated. The single node constraint force $\mathbf{f}_S$ and multiple node constraint force $\mathbf{f}_M$ need to satisfy the following equilibrium equation:

$$\mathbf{f}_{\text{int}} = \mathbf{f}_{ext} + \mathbf{f}_S + \mathbf{f}_M \tag{6.2.6}$$

$\mathbf{f}_{ext}$: External load vector

$\mathbf{f}_{\text{int}}$: Internal load vector ($\int_\Omega \mathbf{B}^T \boldsymbol{\sigma} d\Omega$)

Dividing the equation above into the independent DOF (I) and dependent DOF (D) components, and using the condition that the single node constraint and multiple node constraint cannot be applied to the same DOF, gives the following equation:

$$\begin{Bmatrix} \mathbf{f}_{\text{int},I} \\ \mathbf{f}_{\text{int},D} \end{Bmatrix} = \begin{Bmatrix} \mathbf{f}_{ext,I} \\ \mathbf{f}_{ext,D} \end{Bmatrix} + \begin{Bmatrix} \mathbf{f}_{S,I} \\ \mathbf{0} \end{Bmatrix} + \begin{Bmatrix} \mathbf{f}_{M,I} \\ \mathbf{f}_{M,D} \end{Bmatrix} \tag{6.2.7}$$

Hence, the constraint force for the dependent DOF $\mathbf{f}_{M,D}$ can be calculated as follows:

$$\mathbf{f}_{M,D} = \mathbf{f}_{\text{int},D} - \mathbf{f}_{ext,D} \tag{6.2.8}$$

Also, the constraint force for the independent DOF $\mathbf{f}_{M,I}$ satisfies the following relationship when equation (6.2.5) is used:

$$\mathbf{f}_{M,I} = -\mathbf{G}^T \mathbf{f}_{M,D} \tag{6.2.9}$$

After the multiple node constraint force is calculated, the single node constraint force $\mathbf{f}_{S,I}$ can be calculated from equation (6.2.7).

2.5 Singularity Error

When the singularity error for the stiffness matrix occurs, a single solution does not exist and this implies that there is an error in the finite element mode. Singularity errors can be divided into the single node singularity (which can be evaluated at one node) and the mechanism singularity (which evaluates the total stiffness and has a mechanism form).

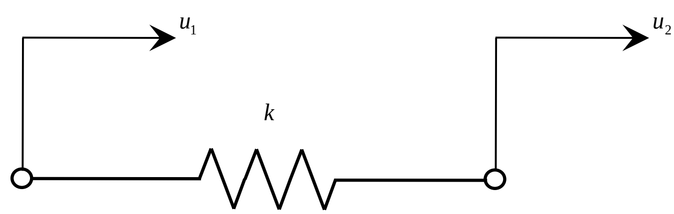
Single node singular error

Single node singular errors occur when elements are used without proper understanding of their properties. For example, when beam or shell elements (which do not have stiffness in a particular direction) are used or when the stiffness direction of a spring element is set in a particular direction only. In this case, the single node constraint needs to be used appropriately to remove the singular errors and find the normal solution. Because single node singular errors can be evaluated without needing to decompose the stiffness matrix, the single node constraint function can be used to remove the error.

Mechanism singular error

Mechanism form singular errors are expressed by two or more nodes related to each other. In particular, this error often occurs when the constraint conditions are inappropriately set.

Figure 6.2.2 Elastic connection without constraint conditions



For example, the equilibrium equation for the system above can be expressed as follows:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} \tag{6.2.10}$$

The inverse of this stiffness matrix does not exist and when the load is $P_1 = -P_2$, it has many solutions. Here, the eigenvalue of the stiffness matrix includes '0' and the corresponding eigenvector is the structural shape with displacement when there is no deformation energy. For general structures with no constraint conditions, the stiffness matrix has 6 '0' eigenvalues corresponding to the rigid-body motion and the single solution cannot be found because of the singularity error.

The existence of a mechanism singular error can be found from the decomposition process of the stiffness matrix, and a singular error is assumed when a value close to '0' is found on the diagonal term during stiffness decomposition. When the error is found, the program execution is stopped, or a small stiffness can be added to the diagonal term for continued calculation. Adding stiffness to the diagonal term gives the same result as adding a spring element on a finite element model.

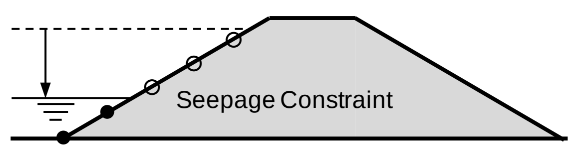
2.6 Seepage Boundary Condition

The single node and multiple node constraints are applicable to the pore pressure DOF in seepage analysis. Apart from this, there are particular boundary conditions applied to seepage analysis such as the change in boundary condition with time or review boundary.

Change in boundary condition with time

In seepage analysis, a boundary condition for the changing water level with time may be needed. When this condition is assigned to the boundary of the analysis region, the boundary condition may also be assigned to nodes which are higher than the water level. GTS NX provides a function which automatically removes the nodes in a higher region than the water level from the boundary constraint.

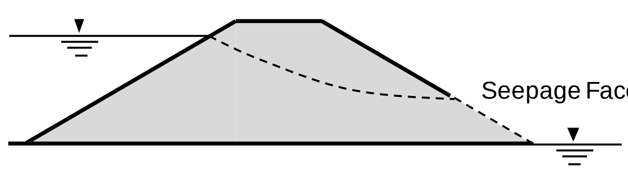
Figure 6.2.3 Problem of decreasing water level with time



Boundary review

In seepage analysis, the boundary conditions may change according to the flow process. When performing seepage analysis in dams such as figure 6.2.4, the seepage face at the downstream region needs to be considered and assigned a water level boundary condition. However, this seepage face cannot be predicted during modeling. In this case, if the boundary review condition is assigned to the area where the seepage face is expected to form, the seepage face is found through nonlinear solutions. GTS NX uses the penalty method to perform boundary review:

Figure 6.2.4 Problem that requires review boundary



Drained condition

The drained condition is a boundary condition used for excessive pore pressure based consolidation analysis. The excessive pore pressure is kept as '0' in the region where the drained condition is applied, and this implies that the water can escape due to the loads acting on the ground. As water dissipates out according to the drained condition, the internal force part shared by water is gradually converted and resisted by the ground.

2.7 Transfer Boundary Condition

It is difficult to model the nearly infinite ground accurately using the 2D model used for ground-structure analysis. Hence, the model boundary needs to be set at an engineering appropriate position and the set boundaries need to be processed to simulate actual site conditions.

The boundary conditions in ground modeling can be divided into the boundary load, viscous boundary and transmitting boundary conditions. The boundary load condition is divided into the free end (where the force of the earthquake response load at the boundary point for the free field is input) as well as the fixed end (which inputs the displacement). The boundary load condition can sufficiently consider the effects of free field and earthquake waves, but it cannot consider the effects of waves reflected from the foundation slab when a structure is present. Also, the effects increase as the boundary position approaches the foundation slab.

The viscous boundary condition was developed to by Lysmer and Kuhlemeyer[2], Ang and Newmark[3] to solve the weaknesses of the load boundary condition. This condition absorbs the matter waves that have a certain angle at the boundary. However, because the viscous boundary condition also cannot perfectly handle the effects of complex surface waves, the boundary position also needs to be set at a certain distance from the foundation slab.

The transmitting boundary condition supplements the problems of the viscous boundary condition and can consider the effects of nearly all types of matter waves and surface waves. This condition is expressed using a spring and damper, which has a function of frequency in the horizontal direction soil layer. Because the transmitting boundary condition assumes the horizontal direction properties of each soil layer to be homegeneous, satisfactory results can be obtained even when the boundary condition is attached to the structure itself. However, maintaining a certain distance with the foundation slab is effective to accurately consider the property changes according to the horizontal strain intensity.

---

[2] Lysmer, J., and Kuhlemeyer, R.L., "Finite dynamic model for infinite media", Jour. Engng., Mech. Div., ASCE, Vol 95, No EM4, Aug 1969, pp 859-877

[3] Ang, A.H., and Newmark, N.M., "Development of a transmitting boundary for a numerical wave motion calculation", Report to the Defense Atomic Support Agency, Wash. D.C., contract DASA-01-0040, 1971

Section 3

Pore Pressure/Initial Condition

3.1 Pore Pressure Definition

Pore pressure definition is essential to the analysis of porous media including pore water. The defined pore pressure is used as an initial condition, or participates in the analysis as a pore-pressure field form. The former is generally used for seepage analysis and the latter is used for stress analysis of the porous ground.

Pore pressure can be largely defined using two methods: either specifying a water table or phreatic line/surface, or using the nodal pore pressure results obtained from seepage analysis.

Definition using water table

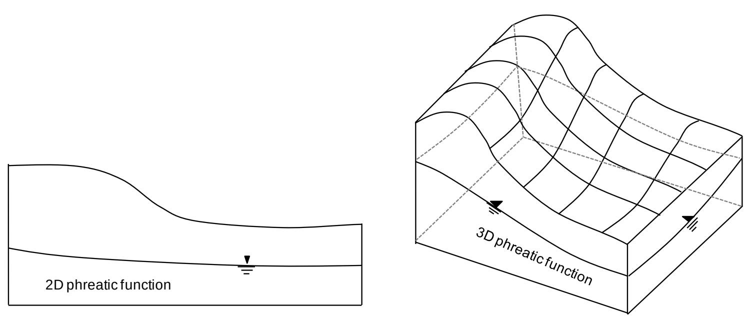
The water table is the boundary line or surface in saturated or unsaturated state ground, and it generally represents the height information at the corresponding position. GTS NX uses the following methods to define the water table (assuming that the z axis and self weight axis are identical):

▶ Specify the height using a constant $z_{phreatic} - c = 0$

▶ Function on 2D plane: specify water table height using water table: $z_{phreatic} - T_I(x) = 0$

▶ Function in 3D space: specify water table height using a set of position vectors in 3D space: $z_{phreatic} - T_{II}(x, y) = 0$

Figure 6.3.1 Definition of 2/3D water table function



The method of using 2/3D functions generally uses the geometric shape of the model as a reference. In other words, the discrete sampling points are extracted based on the selected geometric shape, and the extracted points are used as a reference to generate the 2/3D functions. Especially for the water table condition of the 3D function, the Quadratic Shepard[4] method is used to find the water level at an

[4] Robert J. Renka, "Multivariate Interpolation of Large Sets of Scattered Data", ACM TRANS. on Mathematical Software, Vol. 14, 1988

arbitrary height. This method uses the least square method to find the local interpolation function, and uses the weight according to distance to find the interpolated values in an arbitrary region. In this case, the values outside the sampling region cannot be obtained and a large water table region that is sufficient for analysis is required.

The water table can be confined for a partial section of the finite element model. Hence, various modeling including the impermeable layer can be effectively performed. Assuming a hydrostatic pore pressure distribution for the defined water table, the pore pressure at an arbitrary position can be calculated as follows:

$$p = \rho_f \, g \, \mathbf{n}_g \, (\mathbf{X} - \mathbf{X}_{phreatic}) \tag{6.3.1}$$

ρ_f	: Water density	$\mathbf{X}$	: Position vector
g	: Gravitational acceleration	$\mathbf{X}_{phreatic}$	: Position vector of phreatic line/surface
$\mathbf{n}_g$	: Unit vector in gravitational direction		

Definition using seepage analysis results

Seepage analysis has the pore pressure nodal DOF fundamentally. Hence, performing the steady state or transient state seepage analysis gives the nodal pore pressure distribution that considers actual seepage phenomena. This can be used in the following analysis as an initial condition or pore pressure field. The pore pressure is calculated using the inner product of the nodal pore pressure p_i and the shape function N_i of the corresponding position:

$$p = N_i p_i \tag{6.3.2}$$

Especially for transient state seepage analysis and fully coupled stress-seepage analysis, the nodal pore pressure is required as an initial condition and the defined pore pressure distribution from the two methods above can be used. Information on the initial condition will be explained later.

The pore pressure calculated above can be used for ground analysis including pore water to calculate the effective stress, total stress, degree of saturation etc. Detailed information is available in Chapter 3.

3.2 Initial Condition

Unit analysis, which includes each stage in construction stage analysis, aims to simulate the changes that occur from external environment changes such as load, boundary condition, etc. as it moves from the initial state to the final state. Here, information at the initial state needs to be defined.

The initial information is called the initial condition, and can be largely divided into the initial condition for nodes and initial conditions for the element integral point. The usable conditions for each type are listed in table 6.3.1:

Table 6.3.1 Usable initial conditions in GTS NX

Initial condition type	Content
Node condition	nodal displacement/velocity, nodal pore pressure
Element integral point condition	stress, strain, void ratio, OCR/pre-consolidation pressure

Nodal displacement/velocity

Assuming the basic initial displacement in GTS NX to be '0', the initial values of the nodal velocity can be input for linear/nonlinear dynamic analysis. However, if self weight analysis is conducted to calculate the in-situ state, its endpoint is assumed to be the initial state and the initial deformation can be set as '0'.

If the initial displacement/velocity is not defined for the analysis stage, the last value from the stage before is used. If the previous analysis stage does not exist, the initial value is assumed to be '0'.

Nodal pore pressure

The nodal pore pressure can be specified as an initial condition for seepage analysis and fully coupled stress-seepage analysis. The pore pressure can be defined using two methods. Firstly, specifying a water table or phreatic line/surface and secondly, using the nodal pore pressure results obtained from seepage analysis.

Initial strain condition

If the in-situ state is calculated using self weight analysis, the initial state can be assumed to be the state where the analysis ends. Here, the initial strain can be specified as '0'.

Initial stress condition

The initial stress components that need to be defined are different for each element type, and the stress/resultant force component for each element type is listed in table 6.3.2.

For 2/3D elements, the initial stress component can specify the defined coordinate system. If the coordinate system of the initial stress component is given for a 2D element, the stress component is assumed to be defined in the coordinate system that projects the given input axes in the 2D plane.

Table 6.3.2 Initial stress component for different element types

Type	Component
Truss/Embedded Truss	N_{xx}
Beam	N_{xx} , M_x , M_y , M_z
Plane Strain/Plane Stress	σ_{xx} , σ_{yy} , τ_{xy} , σ_{zz}
Shell	N_{xx} , N_{yy} , N_{xy} , M_{xx} , M_{yy} , M_{xy} , Q_{xz} , Q_{yz}
Axisymmetric	σ_{xx} , σ_{yy} , τ_{xy} , $\sigma_{\theta\theta}$
Solid	σ_{xx} , σ_{yy} , σ_{zz} , τ_{xy} , τ_{yz} , τ_{zx}

The initial stress can also be calculated from self weight analysis, and the state at which the analysis ends is assumed as the initial state.

Initial void ratio

The void ratio can have an effect on the permeability coefficient in seepage and consolidation analysis and has a great effect on the behavior of ground which uses the modified Cam-clay material model. The initial void ratio is updated using the changes in volumetric strain.

OCR/ pre-consolidation pressure

To use the Modified Cam-clay model, the in-situ stresses and pre-consolidation pressure p_c' are also needed in addition to the initial void ratio. GTS NX uses a directly input initial value for the pre-consolidation pressure p_c' , or automatically calculates it from the in-situ stresses and over-consolidation ratio (OCR). Details on the Modified Cam-clay material model are explained in Chapter 4.

3.3 Ground Stress Initialization

General ground analysis uses the in-situ stress of the ground state as the initial values. GTS NX considers the calculation of in-situ stresses from self weight analysis to be the base. Here, self weight analysis is the static analysis that only considers the self weight as the external force when appropriate ground boundary conditions are applied. For consolidation analysis, the drained condition is added such that the excessive pore pressure does not occur.

K_0 method

A function that modifies the stress calculated from self weight analysis to satisfy the K_0 condition is provided. The K_0 condition for the stress components can be expressed using vertical stress σ_v and horizontal stress σ_h as follows:

$$K_0 = \frac{\sigma_h}{\sigma_v} \tag{6.3.3}$$

The K_0 method uses the vertical stress calculated from self weight analysis to calculate the horizontal stress. The modified stress field generally does not maintain the equilibrium state with the self weight. If the stress is adjusted when the equilibrium state is not maintained, the stress in the continued stress analysis continuously changes to be in equilibrium with the external force, even when there is no change in external force, hence generating a deformation. Therefore, the K_0 method is applicable when this additional stress change is relatively small.

The general usable conditions for stress modification using the K_0 method are as follows:

- ▶ When the ground shape change in the horizontal direction is insignificant
- ▶ When the pore pressure distribution does not change in the horizontal direction

► When horizontal stress is generated by the horizontal free edge/surface boundary condition

►When the material axis is perpendicular or identical to the horizontal axis when a transversely isotropic material is used