

Section 1

Overview

Various element libraries can be used for geometric modeling on FEA NX. These elements can be used for linear and nonlinear stress analysis, seepage analysis, consolidation analysis and various other coupled analyses. It is important to understand the usable elements and their respective properties to perform accurate finite element analysis. This chapter introduces the various theories behind techniques and backgrounds used in finite element modeling in FEA NX.

The usable element types in FEA NX and their shapes and properties can be classified as follows:

Scalar Element

A single node, whose motion has a displacement or kinetic energy defined by the relative value with respect to a ground point. The element can be defined using 2 nodes, but the shape information such as the distance between the nodes is not taken into account. Point spring, matrix spring, mass and pile tip elements are classified as scalar elements.

1 Dimensional shape element

A line shape with two nodes, whose shape information (such as the distance between nodes) is used. 1D shape elements include truss, embedded truss, beam, embedded beam, geogrid 1D, pile and elastic link elements.

2 Dimensional shape element

A triangular or rectangular shape that can have 3/4/6/8 nodes. The 2D element can have a curvature in space. 2D shape elements include plane strain, plane stress, shell, axisymmetric solid, geogrid 2D and gauging shell elements. Plain strain and axisymmetric solid elements are 2D shape elements, but they are used to express 3D stress states.

3 Dimensional shape element

A tetrahedral, pentahedral or hexahedral shape that can have 4/5/6/8/10/13/15/20 nodes. A pentahedral element has either a wedge shape or a pyramid shape. Solid elements are classified as 3D shape elements.

**Interface element**

An element used to express the relative behavior between faces or line-to-line. Interface elements include face-to-face Interface elements with 6/8/12/16 nodes, or line-to-line Interface elements with 4/6 nodes.

Rigid link/interpolation element

An element used to define rigid body motion between nodes or interpolate relative motion. It has characteristics similar to multi-point constraints. Rigid link and interpolation elements are classified as such.

Infinite element

Infinite Element consists of line/surface type element. Line type infinite element consists of 4/8 nodes and surface type infinite element consists of 6/8/15/20 nodes.

Section 2

Finite Element Formulation

Applying the stress-strain and strain-stress relationships from the principle of virtual work due to stress in variational form and using it as a constraint condition results in the Hu-Washizu principle^{1,2}, which can be expressed as follows.

$$\delta G_{ext} = \int_{\Omega} (\nabla \delta \mathbf{u})^T \boldsymbol{\sigma} + \delta \boldsymbol{\varepsilon}^T (\boldsymbol{\sigma}(\boldsymbol{\varepsilon}) - \boldsymbol{\sigma}) + \delta \boldsymbol{\sigma}^T (\nabla \mathbf{u} - \boldsymbol{\varepsilon}) d\Omega \quad (3.2.1)$$

δG_{ext} : Virtual work due to external force

\mathbf{u} : Displacement

$\boldsymbol{\sigma}$: Stress

$\boldsymbol{\varepsilon}$: Strain

$\boldsymbol{\sigma}(\boldsymbol{\varepsilon})$: Stress calculated from strain

∇ : Strain-stress relationship operator

The equation above is the most general form that includes the equilibrium equation, constitutive equation and compatibility condition. If the strain and stress relationship is assumed to always be satisfied by the constitutive equation, it results in the Hellinger-Reissner principle^{3,4} as follows:

$$\delta G_{ext} = \int_{\Omega} (\nabla \delta \mathbf{u})^T \boldsymbol{\sigma} + \delta \boldsymbol{\sigma}^T (\nabla \mathbf{u} - \boldsymbol{\varepsilon}(\boldsymbol{\sigma})) d\Omega \quad (3.2.2)$$

$\boldsymbol{\varepsilon}(\boldsymbol{\sigma})$: Strain calculated from stress

Assuming that the relationship between $\boldsymbol{\varepsilon}$ and $\nabla \mathbf{u}$ is satisfied by additional suitable conditions, the equation becomes the general principle of virtual work:

$$\delta G_{ext} = \int_{\Omega} (\nabla \delta \mathbf{u})^T \boldsymbol{\sigma}(\mathbf{u}) d\Omega \quad (3.2.3)$$

¹ Hu, H.C., "On some variational principles in the theory of elasticity and the theory of plasticity," *Scintia Sinica*, Vol. 4, 1955.

² Washizu, K., *On the Variational Principles of Elasticity*, Aeroelastic and Structural Research Laboratory, MIT, Technical Report, 1955.

³ Hellinger, E., "Der allgemeine Ansatz der Mechanik der Continua," *Encyclopadie der Mathematischen Wissenschaften*, Vol. 4, 1914.

⁴ Reissner, E., "On a variational theorem in elasticity," *Journal of Mathematical Physics*, Vol. 29, 1950.

Let us limit the integration region as a single element to apply the principle of virtual work on the finite element method. Interpolating the displacement \mathbf{u} as a shape function within a single element gives the following equation:

$$\mathbf{u}^h = \mathbf{N}\mathbf{d}^e \quad (3.2.4)$$

\mathbf{N} : Shape function

\mathbf{d}^e : Element node DOF

Using the strain-displacement relationship $\boldsymbol{\varepsilon}^h = \nabla \mathbf{u}^h = \mathbf{B}\mathbf{d}^e$, the principle of virtual work for the entire element can be expressed as follows:

$$\delta G_{ext} = \delta \mathbf{d}^T \mathbf{F} = \delta \mathbf{d}^T \left[\sum \int_{\Omega_e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \right] \mathbf{d} = \delta \mathbf{d}^T \mathbf{K} \mathbf{d} \quad (3.2.5)$$

\mathbf{D} : Stress-strain relationship matrix

In linear analysis, the total stiffness matrix \mathbf{K} is independent of the total node DOF \mathbf{d} , and the stiffness of individual elements \mathbf{K}^e can be expressed as follows:

$$\mathbf{K}^e = \int_{\Omega_e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega \quad (3.2.6)$$

This equation is applicable for the analysis of elastic structures with small displacements, but it can also be applied to nonlinear analysis using the same principle. The finite element formulation process used in seepage analysis and consolidation analysis are explained in 3.10 and 3.11.

Section 3

Shape Function

Element definition starts from assuming a displacement field caused by a shape function. The basis function of a displacement field is equally applied to the pore pressure field in a seepage or consolidation element, unless specified otherwise. The examples in this section do not follow the summation convention. 1,2,3 dimensional shape functions are expressed in the natural coordinate system (ξ, η, ζ) .

3.1

1 Dimensional shape

2 node shape function

$$N_i = \frac{1 + \xi_i \xi}{2}, \quad -1 \leq \xi \leq 1$$

$$\xi_1 = -1, \quad \xi_2 = 1$$

2 node Hermite shape function

$$N_1 = 1 - 3\xi^2 + 2\xi^3, \quad N_2 = l\xi - 2l\xi^2 + l\xi^3, \quad N_3 = 3\xi^2 - 2\xi^3, \quad N_4 = -l\xi^2 + l\xi^3, \quad 0 \leq \xi \leq 1$$

l : Element length

3 node shape function

$$N_1 = \frac{1}{2}\xi(\xi-1), \quad N_2 = \frac{1}{2}\xi(\xi+1), \quad N_3 = \frac{1}{2}(1-\xi^2), \quad -1 \leq \xi \leq 1$$

3.2

2 Dimensional shape

3 node triangle

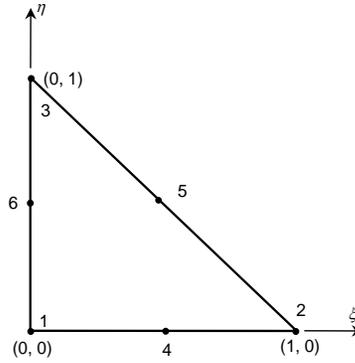
$$N_1 = 1 - \xi - \eta, \quad N_2 = \xi, \quad N_3 = \eta$$

6 node triangle

$$N_1 = (1 - \xi - \eta)(1 - 2\xi - 2\eta), \quad N_2 = \xi(2\xi - 1), \quad N_3 = \eta(2\eta - 1)$$

$$N_4 = 4\xi(1 - \xi - \eta), \quad N_5 = 4\xi\eta, \quad N_6 = 4\eta(1 - \xi - \eta)$$

Figure 3.3.1 Node position and natural coordinate system for triangular elements



4 node rectangle

$$N_i = \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta)$$

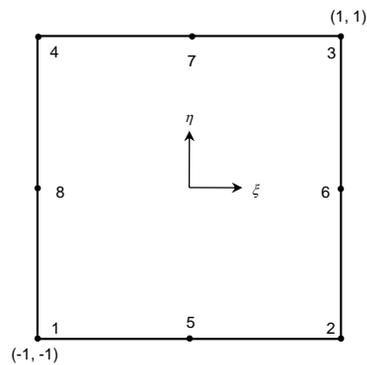
8 node rectangle

$$N_i = \frac{1}{4}(1 + \xi_i \xi)(1 + \eta_i \eta)(\xi_i \xi + \eta_i \eta - 1), \quad i = 1, 2, 3, 4$$

$$N_i = \frac{1}{2}(1 - \xi^2)(1 + \eta_i \eta), \quad i = 5, 7$$

$$N_i = \frac{1}{2}(1 - \eta^2)(1 + \xi_i \xi), \quad i = 6, 8$$

Figure 3.3.2 Node position and natural coordinate system for rectangular elements



3.3

3 Dimensional shape

4 node tetrahedron

$$N_1 = 1 - \xi - \eta - \zeta, \quad N_2 = \xi, \quad N_3 = \eta, \quad N_4 = \zeta$$

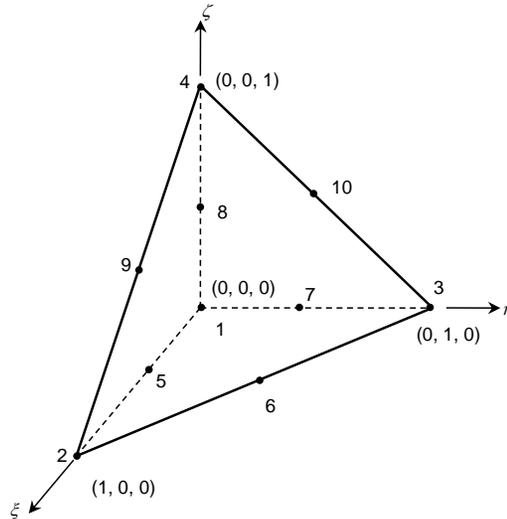
10 node tetrahedron

$$N_1 = 2(1 - \xi - \eta - \zeta)\left(\frac{1}{2} - \xi - \eta - \zeta\right), \quad N_2 = 2\xi\left(\xi - \frac{1}{2}\right), \quad N_3 = 2\eta\left(\eta - \frac{1}{2}\right),$$

$$N_4 = 2\zeta\left(\zeta - \frac{1}{2}\right), \quad N_5 = 4\xi(1 - \xi - \eta - \zeta), \quad N_6 = 4\xi\eta, \quad N_7 = 4\eta(1 - \xi - \eta - \zeta),$$

$$N_8 = 4\xi(1 - \xi - \eta - \zeta), \quad N_9 = 4\xi\zeta, \quad N_{10} = 4\eta\zeta$$

Figure 3.3.3 Node position and natural coordinate system for tetrahedral elements



6 node pentahedron

$$N_i = \frac{1}{2}(1 - \xi - \eta)(1 + \zeta_i \zeta), \quad i = 1, 4$$

$$N_i = \frac{1}{2}\xi(1 + \zeta_i \zeta), \quad i = 2, 5$$

$$N_i = \frac{1}{2}\eta(1 + \zeta_i \zeta), \quad i = 3, 6$$

15 node pentahedron

$$N_i = \frac{1}{2}(1 - \xi - \eta)(1 + \zeta_i \zeta)(\zeta_i \zeta - 2\xi - 2\eta), \quad i = 1, 4$$

$$N_i = \frac{1}{2}\xi(1 + \zeta_i \zeta)(\zeta_i \zeta + 2\xi - 2\eta), \quad i = 2, 5$$

$$N_i = \frac{1}{2}\eta(1 + \zeta_i \zeta)(\zeta_i \zeta + 2\eta - 2), \quad i = 3, 6$$

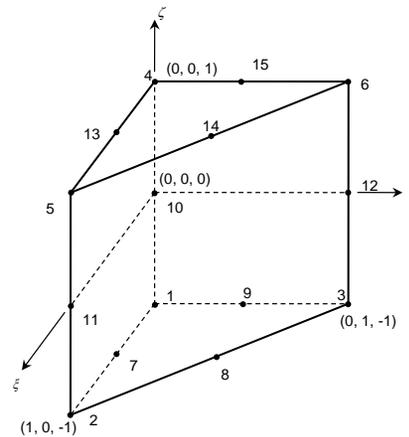
$$N_i = 2\xi(1 - \xi - \eta)(1 + \zeta_i \zeta), \quad i = 7, 13$$

$$N_i = 2\xi\eta(1 + \zeta_i \zeta), \quad i = 8, 14$$

$$N_i = 2\eta(1 - \xi - \eta)(1 + \zeta_i \zeta), \quad i = 9, 15$$

$$N_{10} = (1 - \xi - \eta)(1 - \zeta^2), \quad N_{11} = \xi(1 - \zeta^2), \quad N_{12} = \eta(1 - \zeta^2)$$

Figure 3.3.4 Node position and natural coordinate system for pentahedral (wedge) elements



5 node or 13 node pentahedra are pyramid-shaped and the degenerated shape function due to node coupling is widely used. However, because this shape function is known to have problems with numerical integration⁵, FEA NX uses the following form.

5 node pentahedron

$$N_i = \frac{1}{4} \left\{ (1 + \zeta_i \zeta)(1 + \eta, \eta) - \zeta + \xi, \eta, \frac{\xi \eta \zeta}{1 - \zeta} \right\}, \quad i = 1, 2, 3, 4$$

$$N_5 = \zeta$$

⁵ Bedrosian, G., "Shape functions and integration formulas for three-dimensional finite element analysis", International Journal for Numerical Methods in Engineering, Vol. 35, 1992

13 node pentahedron

$$N_i = \frac{1}{4} (\xi_i \xi + \eta_i \eta - 1) \{ (1 + \xi_i \xi)(1 + \eta_i \eta) - \zeta + \xi_i \eta_i \frac{\xi \eta \zeta}{1 - \zeta} \}, \quad i = 1, 2, 3, 4$$

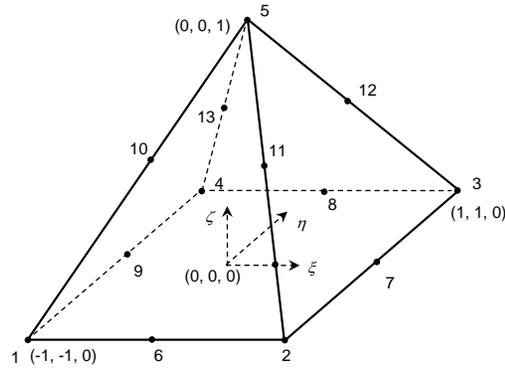
$$N_i = \frac{(1 + \xi - \zeta)(1 - \xi - \zeta)(1 + \eta_i \eta - \zeta)}{2(1 - \zeta)}, \quad i = 6, 8$$

$$N_i = \frac{(1 + \eta - \zeta)(1 - \eta - \zeta)(1 + \xi_i \xi - \zeta)}{2(1 - \zeta)}, \quad i = 7, 9$$

$$N_i = \frac{\zeta(1 + 2\xi_i \xi - \zeta)(1 + 2\eta_i \eta - \zeta)}{(1 - \zeta)}, \quad i = 10, 11, 12, 13$$

$$N_5 = \zeta(2\zeta - 1)$$

Figure 3.3.5 Node position and natural coordinate system for pentahedral (pyramid) elements



8 node hexahedron

$$N_i = \frac{1}{8} (1 + \xi_i \xi)(1 + \eta_i \eta)(1 + \zeta_i \zeta), \quad i = 1, 2, 3, \dots, 8$$

20 node hexahedron

$$N_i = \frac{1}{8} (1 + \xi_i \xi)(1 + \eta_i \eta)(1 + \zeta_i \zeta) (\xi_i \xi + \eta_i \eta + \zeta_i \zeta - 2), \quad i = 1, 2, 3, \dots, 8$$

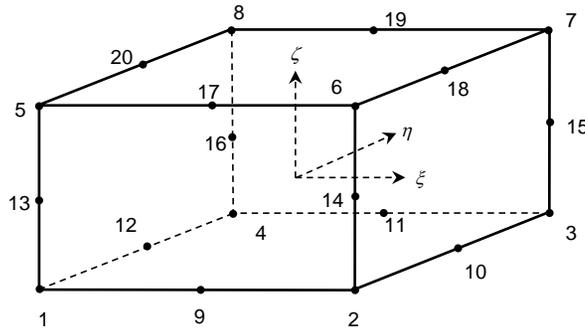
$$N_i = \frac{1}{4} (1 - \xi^2)(1 + \eta_i \eta)(1 + \zeta_i \zeta), \quad i = 9, 11, 17, 19$$

$$N_i = \frac{1}{4} (1 - \eta^2)(1 + \xi_i \xi)(1 + \zeta_i \zeta), \quad i = 10, 12, 18, 20$$

$$N_i = \frac{1}{4}(1 - \zeta^2)(1 + \xi_i \xi)(1 + \eta_i \eta)$$

$i = 13, 14, 15, 16$

Figure 3.3.6 Node position and natural coordinate system for hexahedral elements



Numerical integration is needed to apply the shape functions above to the formulation process outlined in section 3.1. Numerical analysis is needed to calculate the stiffness matrix, mass matrix, load vector, element internal force, etc. FEA NX provides two numerical integration methods: the Gaussian Quadrature and the Lobatto Quadrature.

Table 3.3.1 Numerical integration method types and applicable elements

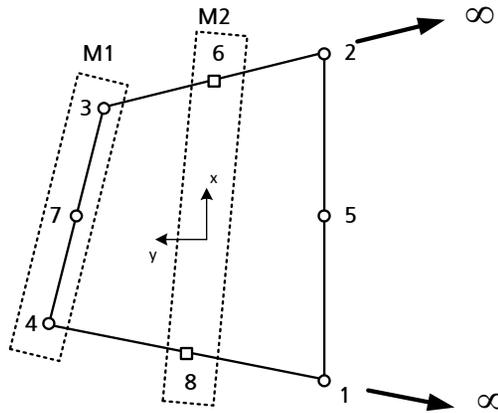
Numerical integration method	Matrix type	Applicable elements	
Gaussian quadrature	Stiffness Matrix	Structural element All elements that use numerical integration	
		Seepage element All elements that use numerical integration	
	Mass Matrix	Consistent mass	All elements
		Lumped mass	All elements using diagonal scaling ⁶
Lobatto quadrature	Stiffness Matrix	Structural element -	
		Seepage element -	
		Consistent mass -	
	Mass Matrix	Lumped mass 3 node triangle, 4 node rectangle 4 node tetrahedron, 6 node pentahedron 8 node hexahedron	

⁶ Hinton, E., Tock, T. and Zienkiewicz, O.C., "A Note on mass lumping and related processes in the finite element method," Earthquake Engineering and Structural Dynamics, Vol. 4, 1976

3.4

Mapping Function for Infinite Element

Same function is applied for displacement shape function and geometrical shape function for general element. However infinite element uses individual functions for displacement and geometrical shape. Infinite element uses following mapping function in order to express geometrically infinite shape.



Infinite direction

$$M_1 = \frac{2\eta}{1+\eta}, \quad M_2 = \frac{1-\eta}{1+\eta}$$

Section 4 Porous Medium Consideration

The consideration of porous media including pore water is a fundamental aspect of geotechnical analysis. Porous media are considered as multiphase materials on FEA NX and are based on an extension of Bishop's stress relationship⁷ which considers partially saturated states for Terzaghi's effective stress principle⁸.

$$\boldsymbol{\sigma}' = \boldsymbol{\sigma} - \mathbf{m}[(\chi - 1)p_a - \chi p] \quad (3.4.1)$$

- $\boldsymbol{\sigma}$: Total stress
- $\boldsymbol{\sigma}'$: Effective stress
- \mathbf{m} : Second rank unit tensor
- p : Pore pressure
- p_a : Atmospheric pressure
- χ : Effective stress parameter

Here, atmospheric pressure p_a has an even distribution within models for practical problems; and because its size is relatively small, its effects are not considered.

The effective stress parameter χ represents the changing properties according to the degree of saturation of the porous medium. It has a value of '1' at the saturated state and a value between '0' and '1' at the unsaturated state. FEA NX assumes that the effective stress parameter and effective saturation are the same ($\chi = S_e$). The effective saturation is proportional to the change in actual degree of saturation and has a value between '0' and '1' for the minimum and maximum degree of saturation.

$$0 \leq S_e = \frac{S - S_{\min}}{S_{\max} - S_{\min}} \leq 1 \quad (3.4.2)$$

- S, S_e : Saturation and effective degree of saturation
- S_{\max}, S_{\min} : Maximum and minimum value of the saturation function

⁷ Bishop, A.W., The principle of effective stress, Teknisk Ukeblad, 39 (1959) 859–863.

⁸ Terzaghi, K. v., "Die Berechnung der Durchlässigkeitsziffer des tones aus dem verlauf der hydrodynamischen spannungserscheinungen", Technical report II a, 132 N 3/4, 125, 138, Akademie der Wissenschaften in Wien. Sitzungsberichte Mathnaturwiss Klasse Abt, 1923.

Using the assumptions made above and in equation (3.4.1), the total stress and effective stress relationship can be ultimately expressed as follows.

$$\begin{aligned}\boldsymbol{\sigma} &= \boldsymbol{\sigma}' + \boldsymbol{\sigma}_p \\ \boldsymbol{\sigma}_p &= -S_e \mathbf{m} p\end{aligned}\quad (3.4.3)$$

$\boldsymbol{\sigma}_p$: Pore stress

Terzaghi's effective stress can be used as a foundation for analysis when the partially saturated state is not considered in the stress relationship. In this case, the pore stress can be expressed in the following form (Later explanations will be based on the use of Bishop's effective stress relationship for a more general concept):

$$\boldsymbol{\sigma}_p = -\mathbf{m} p \quad (3.4.4)$$

The effective stress $\boldsymbol{\sigma}'$ is obtained through the constitutive equation, and represents the stress held by the soil skeleton.

In FEA NX, the pore pressure p is set using the following methods:

- ▶ Hydrostatic pressure due to water level
- ▶ Nodal pore pressure of seepage analysis in seepage-stress coupled analysis
- ▶ Nodal pore pressure DOF in consolidation analysis, stress-seepage coupled analysis

The saturation function $S = S(p)$ needs to be defined to perform analysis that considers partial saturation. The degree of saturation is a function of pore pressure and is defined by various function forms or a user input table form. The degree of saturation is fundamentally calculated using the pore pressure of the current state in undrained analysis (where the pore pressure changes during analysis) or consolidation analysis. However, to maintain the linearity of linear undrained analysis, the degree of saturation is calculated using the initial pore pressure.

FEA NX provides various stress analysis for porous media with pore water, and the effects of the existing pore water is reflected in the calculation of the internal force and self weight. The change in density due to the existence of pore water is reflected in the calculation of the mass matrix for dynamic analysis and eigenvalue analysis.

$$\begin{aligned}\mathbf{f}_l^e &= \int_{\Omega_e} \mathbf{B}^T (\boldsymbol{\sigma}' - S_e \mathbf{m} p) d\Omega \\ \mathbf{f}_g^e &= \int_{\Omega_e} \mathbf{N}^T \rho \mathbf{g} d\Omega \\ \mathbf{M}^e &= \int_{\Omega_e} \mathbf{N}^T \mathbf{N} \rho d\Omega\end{aligned}\quad (3.4.5)$$

- \mathbf{f}_I^e : Internal force of element e
 \mathbf{f}_g^e : Body force of element e due to gravity
 \mathbf{M}^e : Mass matrix of element e
 ρ : Homogenized density
 \mathbf{g} : Gravitational acceleration vector

The homogenized density needed to calculate the self weight and inertial force is also assumed to be a function of the degree of saturation to accurately represent the partially saturated state, and can be expressed using the following equation:

$$\rho = (1 - S_e) \rho_{unsat} + S_e \rho_{sat} \quad (3.4.6)$$

Here, ρ_{sat} and ρ_{unsat} are the soil densities at the saturated and unsaturated state, respectively. In other words, when considering partial saturation, the density has a value between these two densities.

On the other hand, if partial saturation is not considered, the unsaturated state density is used for self weight and inertial force calculations. This occurs when the pore pressure is 0 or a negative value. If the pore pressure is positive, the saturated state density is used.

$$\rho = \begin{cases} \rho_{unsat} & (p \leq 0) \\ \rho_{sat} & (p > 0) \end{cases} \quad (3.4.7)$$

Table 3.4.1 lists the element types for porous medium analysis, supported analysis types and considered calculations for porous medium simulation.

Table 3.4.1 Porous medium consideration: element and analysis type

Element type	Applicable analysis type	Porous medium consideration
Linear elements	Linear static analysis	Self weight calculation Undrained stiffness calculation Internal force calculation
	Eigenvalue analysis	Mass matrix calculation Undrained stiffness calculation
	Linear time history analysis	Self weight calculation Mass matrix calculation Undrained stiffness calculation Internal force calculation
Nonlinear elements	(Nonlinear static analysis Slope stability analysis)	Self weight calculation Undrained stiffness calculation Internal force calculation
	Nonlinear time history analysis	Self weight calculation Mass matrix calculation Undrained stiffness calculation Internal force calculation
Nonlinear consolidation elements	Consolidation analysis Stress/Seepage coupled analysis	Self weight calculation Undrained stiffness calculation Internal force calculation

Section 5

Supplement for Locking Phenomena

The solution for finite elements obtained from the assumed displacement method is generally known to have very low accuracy. This is due to the locking phenomena. Resolving this problem to increase the accuracy of the solution is vital to the utilization of finite element programs. FEA NX uses the following methods to increase the precision of each element. Each method is not used independently, and 2 or more methods can be used simultaneously depending on the element.

5.1

Mixed-Hybrid Formulation

The mixed-hybrid formulation method can be classified depending on the assumed component from mixing variational theories or displacements. FEA NX uses the assumed stress method and mixed u-p method.

The variational equation that has displacement and stress as variables according to the Hellinger-Reissner principle is as follows:

$$\delta G_{ext} = \int_{\Omega} (\nabla \delta \mathbf{u})^T \boldsymbol{\sigma} + \delta \boldsymbol{\sigma}^T (\nabla \mathbf{u} - \mathbf{D}^{-1} \boldsymbol{\sigma}) d\Omega \quad (3.5.1)$$

Assuming the displacement and stress of an arbitrary element due to the shape function as $\mathbf{u}^h = \mathbf{N}\mathbf{d}^e$ and $\boldsymbol{\sigma}^h = \mathbf{D}\mathbf{d}^e$ respectively, substituting these values gives the right hand side of the equation above as follows:

$$\delta \mathbf{d}^{eT} \bar{\mathbf{Q}}^T \boldsymbol{\beta}^e + \delta \boldsymbol{\beta}^{eT} (\bar{\mathbf{Q}}\mathbf{d}^e - \bar{\mathbf{P}}\boldsymbol{\beta}^e) \quad (3.5.2)$$

Here:

$$\bar{\mathbf{Q}} = \int_{\Omega_e} \mathbf{P}^T \mathbf{B} d\Omega_e \quad (3.5.3)$$

$$\bar{\mathbf{P}} = \int_{\Omega_e} \mathbf{P}^T \mathbf{D}^{-1} \mathbf{P} d\Omega_e \quad (3.5.4)$$

$\boldsymbol{\beta}^e$ is assumed not to be continuous between elements and can be eliminated within the element as shown below:

$$\boldsymbol{\beta}^e = \bar{\mathbf{P}}^{-1} \bar{\mathbf{Q}}\mathbf{d}^e \quad (3.5.5)$$

Substituting this into equation(3.5.2), the element stiffness can be expressed as follows:

$$\mathbf{K}^e = \bar{\mathbf{Q}}^T \bar{\mathbf{P}}^{-1} \bar{\mathbf{Q}} \quad (3.5.6)$$

Appropriately selecting the function \mathbf{P} for stress assumption is an essential aspect of determining the performance of the element. For example, the stress of a plane stress element or the in-plane direction stress⁹ of a shell element is assumed as follows:

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix} = \mathbf{P}\boldsymbol{\beta} = \mathbf{T}\hat{\mathbf{P}}\boldsymbol{\beta} = \mathbf{T} \begin{bmatrix} 1 & 0 & 0 & \eta & 0 \\ 0 & 1 & 0 & 0 & \xi \\ 0 & 0 & 1 & 0 & 0 \end{bmatrix} \boldsymbol{\beta} \quad (3.5.7)$$

Here, \mathbf{T} is the coordinate transformation matrix of the contravariant stress component as shown below:

$$\boldsymbol{\sigma} = \mathbf{T}\hat{\boldsymbol{\sigma}} = \begin{bmatrix} j_{11}^2 & j_{21}^2 & 2j_{11}j_{21} \\ j_{12}^2 & j_{22}^2 & 2j_{12}j_{22} \\ j_{11}j_{12} & j_{21}j_{22} & j_{11}j_{22} + j_{12}j_{21} \end{bmatrix} \begin{Bmatrix} \sigma_{\xi\xi} \\ \sigma_{\eta\eta} \\ \tau_{\xi\eta} \end{Bmatrix} \quad (3.5.8)$$

The transformation matrix is calculated from the Jacobian of each element, and the value at the element center is often used.

$$\mathbf{J} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} = \begin{bmatrix} j_{11} & j_{12} \\ j_{21} & j_{22} \end{bmatrix} \quad (3.5.9)$$

The mixed u-p method assumes the stress $\boldsymbol{\sigma}$ of all components, but only assumes the hydrostatic stress or pressure p to be used as a solution¹⁰ to the locking phenomena in incompressible materials. The stress tensor is disassembled into the deviatoric stress and pressure as shown below, and the Hu-Washizu variational principle is used:

$$\begin{aligned} \boldsymbol{\sigma} &= \boldsymbol{\sigma}_{dev} - p\mathbf{I} \\ p &= Ktr(\boldsymbol{\epsilon}) \end{aligned} \quad (3.5.10)$$

$\boldsymbol{\sigma}_{dev}$: Deviatoric stress

K : Bulk modulus

$tr(\boldsymbol{\epsilon})$: Trace of strain

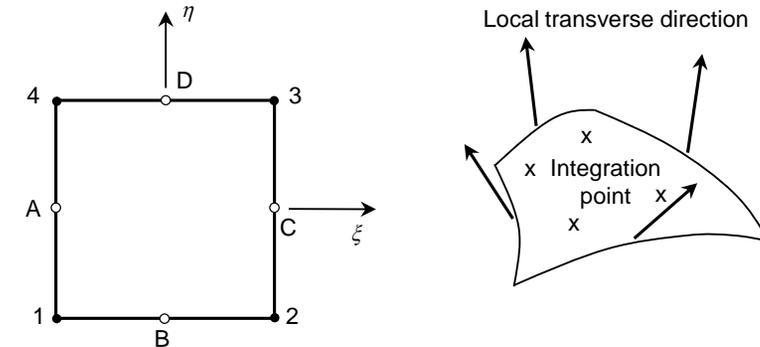
⁹ Pian, T.H.H. and Sumihara, K., "Rational approach for assumed stress finite elements," International Journal for Numerical Methods in Engineering, Vol. 20, 1984

¹⁰ Zienkiewicz, O.C., Rojek, J., Taylor, R.L. and Pastor, M., "Triangles and tetrahedral in explicit dynamics codes for solids," Computer Methods in Applied Mechanics and Engineering, Vol. 43, 1998

5.2

ANS: Assumed Natural Strain

Figure 3.5.1 Assumption for lateral shear strain



For example, let us apply the ANS method on the lateral shear strain of a 4 node shell element as shown above. The components $\gamma_{\xi z}$ and $\gamma_{\eta z}$ of the natural coordinate system are accurate at positions B,D and A,C respectively. The strain at the integral point is interpolated using these values as shown below.

$$\gamma_{\xi z} = \frac{1}{2}(1-\eta)\gamma_{\xi z}^B + \frac{1}{2}(1+\eta)\gamma_{\xi z}^D \quad (3.5.11)$$

$$\gamma_{\eta z} = \frac{1}{2}(1-\xi)\gamma_{\eta z}^A + \frac{1}{2}(1+\xi)\gamma_{\eta z}^C \quad (3.5.12)$$

The strain in the natural coordinate system can be changed to the spatial coordinate system using the following transformation formula.

$$\boldsymbol{\gamma} = \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \mathbf{T}^{-T} \begin{Bmatrix} \gamma_{\xi z} \\ \gamma_{\eta z} \end{Bmatrix} \quad (3.5.13)$$

Here, T is the coordinate transformation matrix of the covariant component as shown below:

¹¹ MacNeal, R.H., "Derivation of element stiffness matrices by assumed strain distribution," Nuclear Engineering and Design, Vol. 70, 1982

¹² Hughes, T.J.R. and Tezduyar, T.E., "Finite elements based upon Mindlin plate theory with particular reference to the four-node bilinear isoparametric element," Journal of Applied Mechanics, Vol. 48, 1981

¹³ Bathe, K.J. and Dvorkin, E.N., "A formulation of general shell elements-The use of mixed interpolation of tensorial components," International Journal for Numerical Methods in Engineering, Vol. 22, 1986

¹⁴ Hughe, T. J. R., The Finite Element Method, Prentice Hall Inc., Englewood Cliffs, NJ, 1987

$$\mathbf{T} = \begin{bmatrix} j_{11} & j_{21} \\ j_{12} & j_{22} \end{bmatrix} \quad (3.5.14)$$

The transformation matrix is calculated from the Jacobian of each element. Because the ANS method only changes the strain using equation (3.5.11-13), the matrix \mathbf{B} of the classical displacement assumption method is modified as follows:

$$\boldsymbol{\varepsilon} = \nabla \mathbf{u} = \bar{\mathbf{B}} \mathbf{d}^e \quad (3.5.15)$$

5.3 EAS: Enhanced Assumed Strain

The enhanced assumed strain method is very similar to the method using the incompatible mode¹⁵ and their results are identical. However, the two differ in that the EAS is theoretically based on the Hu-Washizu principle and starts from the strain assumption, not displacement¹⁶. The following Hu-Washizu variational equation assumes 3 variables (displacement, strain, stress).

$$\delta G_{ext} = \int_{\Omega} (\nabla^T \delta \mathbf{u}) \boldsymbol{\sigma} + \delta \boldsymbol{\varepsilon}^T (\mathbf{D} \boldsymbol{\varepsilon} - \boldsymbol{\sigma}) + \delta \boldsymbol{\sigma}^T (\nabla \mathbf{u} - \boldsymbol{\varepsilon}) d\Omega \quad (3.5.16)$$

The strain $\boldsymbol{\varepsilon}$ is assumed to be a sum of the compatible and incompatible (improved strain assumption) elements calculated from the displacement.

$$\boldsymbol{\varepsilon} = \nabla \mathbf{u} + \bar{\boldsymbol{\varepsilon}} \quad (3.5.17)$$

Substituting this into equation (3.5.16) and simplifying gives the following equation:

$$\delta G_{ext} = \int_{\Omega} (\nabla^T \delta \mathbf{u}) \mathbf{D} (\nabla \mathbf{u} + \bar{\boldsymbol{\varepsilon}}) + \delta \bar{\boldsymbol{\varepsilon}}^T (\mathbf{D} \nabla \mathbf{u} + \mathbf{D} \bar{\boldsymbol{\varepsilon}} - \boldsymbol{\sigma}) - \delta \boldsymbol{\sigma}^T \bar{\boldsymbol{\varepsilon}} d\Omega \quad (3.5.18)$$

If the stress distribution and incompatible strain is assumed to be perpendicular within the element, the equation consisting of only displacement and improved strain are as follows:

$$\delta G_{ext} = \int_{\Omega} \nabla^T \delta \mathbf{u} \mathbf{D} (\nabla \mathbf{u} + \bar{\boldsymbol{\varepsilon}}) + \delta \bar{\boldsymbol{\varepsilon}}^T (\mathbf{D} \nabla \mathbf{u} + \mathbf{D} \bar{\boldsymbol{\varepsilon}}) d\Omega \quad (3.5.19)$$

Substituting the displacement and improved strain due to the shape function as $\mathbf{u}^h = \mathbf{N} \mathbf{d}^e$ and $\bar{\boldsymbol{\varepsilon}} = \mathbf{G} \boldsymbol{\alpha}^e$ respectively for an arbitrary element, the right hand side of the equation above is as follows:

$$\delta \mathbf{d}^{eT} \mathbf{K}_{dd}^e \mathbf{d}^e + \delta \mathbf{d}^{eT} \mathbf{K}_{d\alpha}^e \boldsymbol{\alpha}^e + \delta \boldsymbol{\alpha}^{eT} \mathbf{K}_{\alpha d}^e \mathbf{d}^e + \delta \boldsymbol{\alpha}^{eT} \mathbf{K}_{\alpha\alpha}^e \boldsymbol{\alpha}^e \quad (3.5.20)$$

¹⁵ Taylor, R.L., Beresford, P.J. and Wilson, E.L., "A non-conforming element for stress analysis," International Journal for Numerical Methods in Engineering, Vol. 10, 1976

¹⁶ Simo, J.C. and Rifai, M.S., "A class of mixed assumed strain methods and the method of incompatible modes," International Journal for Numerical Methods in Engineering, Vol. 29, 1990

Here, \mathbf{K}_{dd}^e is the classical element stiffness from the assumed displacement and \mathbf{K}_{da}^e and \mathbf{K}_{aa}^e are as follows:

$$\mathbf{K}_{da}^e = \int_{\Omega_e} \mathbf{B}^T \mathbf{D} \mathbf{G} d\Omega_e \quad (3.5.21)$$

$$\mathbf{K}_{aa}^e = \int_{\Omega_e} \mathbf{G}^T \mathbf{D} \mathbf{G} d\Omega_e \quad (3.5.22)$$

$\boldsymbol{\alpha}^e$ is assumed to be the discontinuity between elements and since there is no external work for $\delta \boldsymbol{\alpha}^e$, it can be eliminated within the element as shown below:

$$\boldsymbol{\alpha}^e = -\mathbf{K}_{aa}^{e-1} \mathbf{K}_{ad}^e \mathbf{d}^e \quad (3.5.23)$$

Substituting this into equation (3.5.20) gives the element stiffness as follows:

$$\mathbf{K}^e = \mathbf{K}_{dd}^e - \mathbf{K}_{da}^e \mathbf{K}_{aa}^{e-1} \mathbf{K}_{ad}^e \quad (3.5.24)$$

Selecting the function \mathbf{G} for improved strain assumption is a vital part in determining the performance of the element.

5.4 Reduced Integration

The strain at the integral point of a low order has a more accurate value than at different positions^{17,18}. Also, because the element locking phenomena is generally caused by the unnecessary powers of strain, these higher order strain shapes can be removed by numerical integration. However, reduced integration can worsen the numerical properties of the stiffness matrix and can cause the spurious zero energy mode or hourglass mode.

5.5 Reduced Integration Stabilization

Generally, the strain of 3D lower order elements can be simplified using the following equation:

$$\boldsymbol{\varepsilon} = \nabla \mathbf{u} \approx (\mathbf{B}_0 + \mathbf{B}_1(\xi, \eta, \zeta)) \mathbf{d}^e \quad (3.5.25)$$

Removing the element locking phenomena by applying a lower integration order is the same as using just \mathbf{B}_0 , and the same as evaluating the strain at the center of the element. When using the strain at the element center, the patch test where uniform deformation is applied may not be satisfied. This problem is generally solved by changing \mathbf{B}_0 to $\bar{\mathbf{B}}_0$ ¹⁹. The average strain \mathbf{B}_0 satisfies the following equation.

¹⁷ Barlow, J., "A stiffness matrix for a curved membrane shell," Conf. Recent Advances in Stress Analysis, Royal Aeron. Soc., 1968

¹⁸ Barlow, J., "Optimal stress locations in finite element models," International Journal for Numerical Methods in Engineering, Vol. 10, 1976

¹⁹ Flanagan, D.P. and Belytschko, T., "A uniform strain hexahedron and quadrilateral with orthogonal hourglass control," International Journal for Numerical Methods in Engineering, Vol. 17, 1981

$$\bar{\mathbf{B}}_0 = \frac{1}{V_e} \int_{\Omega_e} \mathbf{B} d\Omega_e \quad (3.5.26)$$

Here, V_e is the element volume. When using only the corresponding average strain $\bar{\mathbf{B}}_0$, the deformation energy for the spurious zero energy mode is not considered. Since this phenomenon is particularly severe for lower order elements, a stabilization method is needed. Various stabilization methods called "Hourglass Control" exist, and FEA NX uses the physical stabilization method²⁰ suggested by Puso. For example, the $\tilde{\mathbf{B}}_1$ necessary to calculate the stabilized strain for an 8 node hexagonal element in the natural coordinate system is as follows:

$$\tilde{\mathbf{B}}_1 = \xi \tilde{\mathbf{B}}_{\xi} + \eta \tilde{\mathbf{B}}_{\eta} + \zeta \tilde{\mathbf{B}}_{\zeta} + \xi \eta \tilde{\mathbf{B}}_{\xi\eta} + \eta \zeta \tilde{\mathbf{B}}_{\eta\zeta} + \zeta \xi \tilde{\mathbf{B}}_{\zeta\xi} \quad (3.5.27)$$

Using all of the strain from the equation above erases the effects of the reduced integration and hence, some of the shear strain is removed.

Using the average strain and applying the stabilization method has the same effect as selective reduced integration. It also has a fast calculation speed because the numerical integration process is substituted by the following equation:

$$\int_{\Omega_e} \square d\Omega_e = \frac{V_e}{8} \iiint \square d\xi d\eta d\zeta \quad (3.5.28)$$

5.6 Non-conforming Element

For non-conforming elements, the method of strain disassembly is used to make the compatibility condition between elements into integral form. The EAS method above can also be classified as a type of non-conforming element. Generally, the integral form of the compatibility condition between elements is expressed as follows:

$$\int_{\Omega_e} u_{i,j}^* d\Omega = \int_{\partial\Omega_e} \bar{u}_i n_j dS \quad (3.5.29)$$

- u^* : Assumed displacement within the element
- \bar{u} : Assumed displacement on the outer surface of the element
- n_j : Vector perpendicular to the outer surface of the element (direction cosine)
- j : j direction differentiation

The displacement assumed in the element is generally composed of the shape function part and other additional parts.

²⁰ Puso, M.A., "A highly efficient enhanced assumed strain physically stabilized hexahedral element," International Journal for Numerical Methods in Engineering, Vol. 49, 2000

$$\mathbf{u}^* = \mathbf{N}\mathbf{d}^e + \mathbf{P}\lambda \quad (3.5.30)$$

The displacement assumed on the outer surface of the element is also composed of the shape function part and other additional parts. However, the assumed displacement is expressed as an interpolated form of the node displacement:

$$\bar{\mathbf{u}} = \mathbf{N}\mathbf{d}^e + \alpha\mathbf{M}\mathbf{d}^e \quad (3.5.31)$$

\mathbf{M} : Added shape function ($\neq \mathbf{N}$)

The α in the equation above is an arbitrary coefficient and an appropriate value is used, based on the element convergence or the accuracy of the solution. Substituting equations (3.5.30) and (3.5.31) into equation (3.5.29) can calculate λ using \mathbf{d}^e . Using the calculated λ , the element strain can be expressed as follows:

$$\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{d}^e + (\nabla\mathbf{P})\lambda \quad (3.5.32)$$

The \mathbf{B} matrix in the assumed displacement method is modified for non-conforming elements as shown below:

$$\boldsymbol{\varepsilon} = \nabla\mathbf{u} = \bar{\mathbf{B}}\mathbf{d}^e \quad (3.5.33)$$

Section 6

Continuum Elements

Continuum elements are used for volume modeling, such as soils in ground analysis, solid elements in 3D analysis, plane strain elements in 2D analysis and axisymmetric solid elements in axisymmetric analysis are all examples of continuum elements.

6.1

Solid Elements

Solid elements are often used for modeling structures with volume, such as soils, thick walls, etc. The usable solid elements in FEA NX are tetrahedral, pentahedral and hexahedral shapes with 4/5/6/8/10/13/15/20 nodes. Pentahedral elements include wedge shapes and pyramid shapes.

Coordinate system

The entity coordinate system (ECS) of a tetrahedral element is the same as applying the ECS definition rule on a plane stress element on a triangular shape composed of nodes 1,2,3.

The ECS of a pentahedral wedge element is the same as applying the ECS definition rule on a plane stress element on a triangular shape composed of the intermediate points between nodes 1 and 4, nodes 2 and 5, and nodes 3 and 6.

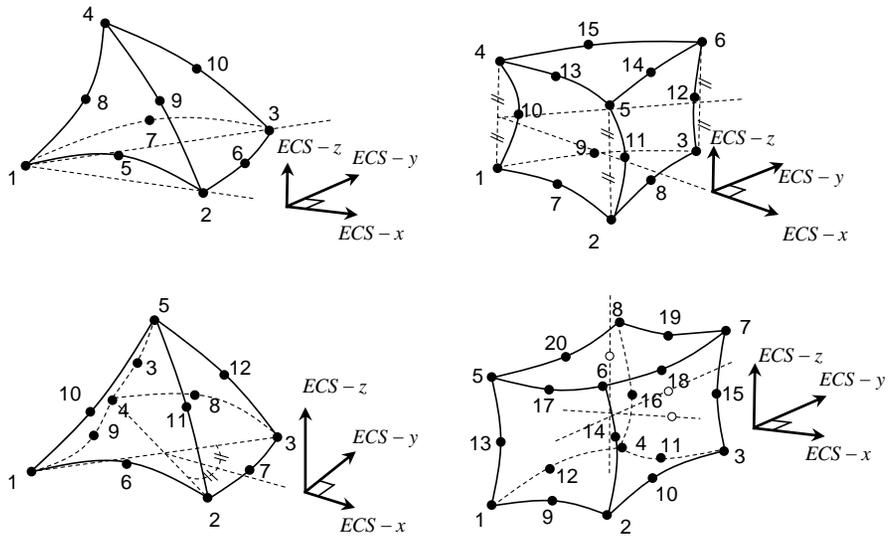
The ECS of a pentahedral pyramid element is the same as applying the ECS definition rule on a plane stress element on a rectangular shape composed of nodes 1,2,3,4.

For a hexahedral element, a vector adjacent to the ECS is defined first as shown below:

- ▶ \mathbf{r} : Vector at the intermediate point of nodes 1,5,8,4 in the direction of the intermediate point of nodes 2,6,7,3
- ▶ \mathbf{s} : Vector at the intermediate point of nodes 1,2,6,5 in the direction of the intermediate point of nodes 4,3,7,8
- ▶ \mathbf{t} : Vector at the intermediate point of nodes 1,2,3,4 in the direction of the intermediate point of nodes 5,6,7,8

The rectangular coordinate system that is closest to the 3 vectors above becomes the ECS of the hexagonal element.

Figure 3.6.1 Coordinate system of Solid elements



DOF

The global coordinate system (GCS) of a Solid element has a displacement DOF in the x, y, z axis directions.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T \tag{3.6.1}$$

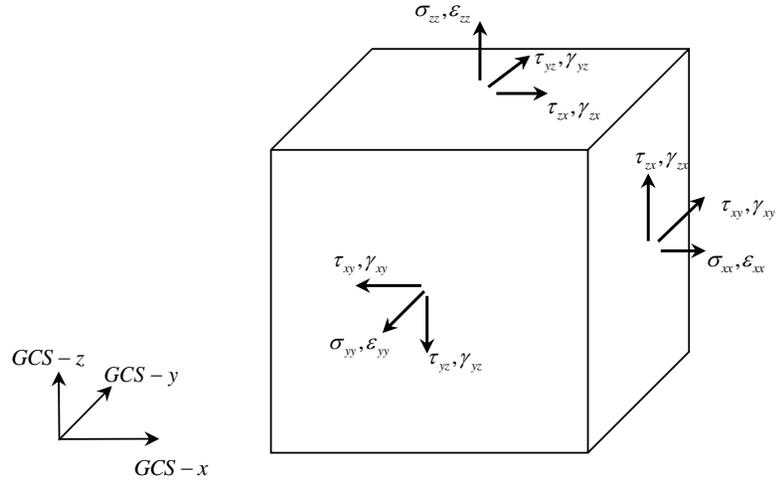
Stress and strain

A solid element considers the stress and strain defined by the GCS. Its components are as follows:

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}, \quad \boldsymbol{\epsilon} = \begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \tag{3.6.2}$$

(3 dimensional stress and strain)

Figure 3.6.2 Stress/strain of a Solid element



Load

The loads applied to a Solid element are as follows:

Table 3.6.1 Loads applied on Solid elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Pressure load	Distributed load applied on an element face
Water pressure load	Distributed load applied on an element face Size determined by pore pressure/water level
Element temperature load	Element temperature that causes volumetric deformation
Prestress/Initial equilibrium force	Initial stress of an element and the corresponding equilibrium force

Element result

On FEA NX, the results of a solid element are printed in the user defined reference coordinate system. The selectable element result coordinate systems (ERCS) are the ECS, MCS and arbitrary coordinate system.



Table 3.6.2 Result article of Solid elements

	Result article	Explanation
Stress	Stress component	Position : vertex/element center $\sigma_{xx} , \sigma_{yy} , \sigma_{zz} , \tau_{xy} , \tau_{yz} , \tau_{zx}$
	Principal stress	Position : vertex/element center P_1 , P_2 , P_3 , Principal stress direction
	Von-Mises stress	Position : vertex/element center σ_v
	Max shear stress	Position : vertex/element center τ_{max}
	Octahedral stress	Position : vertex/element center τ_o
	Mean pressure	Position : vertex/element center p_0
Strain	Strain component	Position : vertex/element center $\epsilon_{xx} , \epsilon_{yy} , \epsilon_{zz} , \gamma_{xy} , \gamma_{yz} , \gamma_{zx}$
	Principal strain	Position : vertex/element center E_1 , E_2 , E_3 , Principal strain direction
	Von-Mises strain	Position : vertex/element center ϵ_v
	Max shear strain	Position : vertex/element center γ_{max}
	Octahedral strain	Position : vertex/element center γ_o
	Mean compression	Position : vertex/element center c_0

The results of a Solid element including pore water are printed in the effective stress and total stress components, and the total pore stress and excess pore stress are additionally printed.

Table 3.6.3 Results of a Solid element including pore water

	Result article	Explanation
Stress	Stress component (Effective and total)	Position : vertex/element center $\sigma'_{xx} , \sigma'_{yy} , \sigma'_{zz} , \tau'_{xy} , \tau'_{yz} , \tau'_{zx}$ $\sigma_{xx} , \sigma_{yy} , \sigma_{zz}$
	Pore stress (Total and excessive)	Position : vertex/element center $\sigma_{p,total} , \sigma_{p,excessive}$
	Mean pressure (Effective and total)	Position : vertex/element center

P'_0, P_0

Element method selection
 The usable Solid elements of FEA NX depend on the various element performance improvement methods. The following table lists the name, related finite element method and integration method. The standard value is shaded.

Table 3.6.4 Performance improvement methods used on Solid elements

Shape	Number of nodes	Name	Element method	Numerical integration of stiffness matrix	Calculation method for concentrated mass
Tetrahedron	4	Full integration	Assumed displacement	1 Point	Lobatto
		Enhanced	EAS, u-p mixing	4 Points	Lobatto
		Full integration	Assumed displacement	3X2 Points	Lobatto
Wedge	6	Reduced integration (stabilized)	Reduced Integration (stabilized)	1X1 Point	Lobatto
		Hybrid	Mixed method	3X2 Points	Lobatto
Pyramid	5	Full integration	Assumed displacement	4X2 Points	Diagonal scaling term
		Reduced integration	Reduced Integration	1X1 Point	Diagonal scaling term
		Hybrid	Mixed method	4X2 Points	Diagonal scaling term
Hexahedron	8	Full integration	Assumed displacement	2X2X2 Points	Lobatto
		Reduced integration (stabilized)	Reduced Integration (stabilized)	1X1X1 Point	Lobatto
		Hybrid	Mixed method	2X2X2 Points	Lobatto
Tetrahedron	10	Full integration	Assumed displacement	4 Points	Diagonal scaling term
		Enhanced	Nonconforming element	4 Points	Diagonal scaling term
Wedge	15	Full integration	Assumed displacement	3X3 Points	Diagonal scaling term
		Reduced integration	Reduced Integration	3X2 Points	Diagonal scaling term
		Hybrid	Mixed method	3x3 Points	Diagonal scaling term

Pyramid	13	Full integration	Assumed displacement	9X3 Points	Diagonal term scaling
		Full integration	Assumed displacement	3X3X3 Points	Diagonal term scaling
Hexahedron	20	Reduced integration	Reduced Integration	2X2X2 Points	Diagonal term scaling
		Hybrid	Mixed method	3X3X3 Points	Diagonal term scaling

The characteristics and precautions for each element method are as follows:

- ▶ 4 node element : The displacement results are similar regardless of the method, but elements using the EAS and u-p mixed methods display more accurate stress results.
- ▶ 6 node element : The performance is vastly superior for elements using the mixed method for thin structures.
- ▶ 8 node element : The performance is vastly superior for elements using the mixed method or reduced integration method for structures experiencing bending.
- ▶ 10 node element : The results are similar regardless of the method, but non-conforming elements for thin structures display relatively flexible results.
- ▶ 20 node element : All methods display accurate results. Elements using the mixed method for thin structures display superior performance.

Nonlinear analysis

Geometric nonlinearity can be considered for solid elements and material nonlinearity can be considered for elastic and nonlinear elastic materials. The additional result articles for nonlinear material usage is listed in the table below, and the material states are expressed using symbols.

Table 3.6.5 Nonlinear analysis result article of Solid elements

Result article		Explanation
Stress	Equivalent stress	Position : Integral point Calculated according to plastic model, σ_{eq}
	Material status	Position : Integral point Elastic: - Plastic/Nonlinear elastic: ○ Unloading/reloading: ⊕ Tension failure: ⊕ Cap failure: ⊖
Strain	Equivalent strain	Position : Integral point Calculated according to plastic model, ε_{eq}

Effective plastic strain	Position : Integral point e_p
--------------------------	------------------------------------

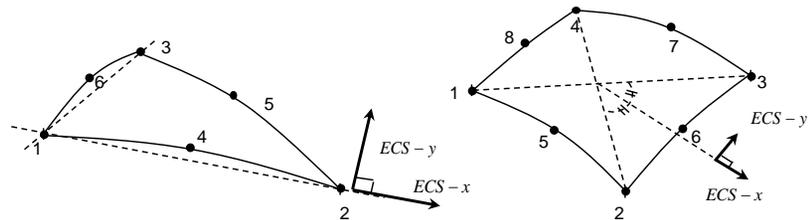
6.2 Plane strain Element

A plane strain element is a triangular or rectangular element consisting of 3/4/6/8 nodes in a plane. It is mainly used for the ground and structural analysis of dams or tunnels, which maintains a consistent section and has a long length in the direction normal to the section. Because stress exists in the element thickness direction, it is technically not a 2D stress state. A plane strain element can be modeled in 3D space but generally it is modeled in the particular coordinate plane (x-y, x-z, y-z) on the GCS for convenience. 2D modeling on FEA NX is done in the x-y plane.

Coordinate system

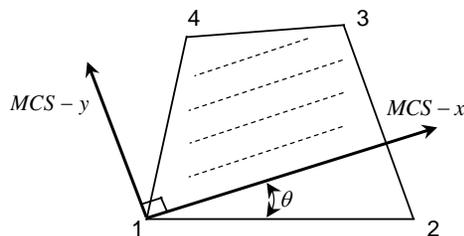
The ECS of a triangular plane strain element has the z axis in the direction normal to the element plane and the x axis in the direction from node 1 to 2. Similarly, for a rectangular element, the z axis is set in the direction normal to the element plane and the x axis is set in the direction that bisects the intersection angle between diagonal lines made by nodes 1 and 3, and nodes 4 and 2. The finite element formulation of a plane strain element is done about the ECS.

Figure 3.6.3 Coordinate system of Plane strain elements



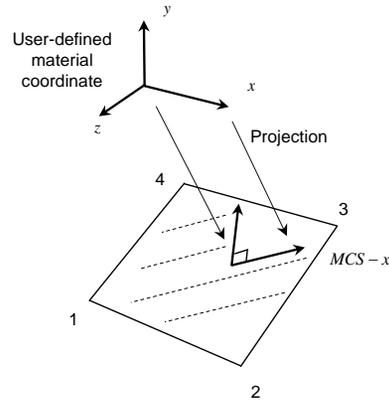
To use a transversely isotropic material on a plane strain element, the major material axis needs to be in an appropriate direction. In this case, the MCS is used, and FEA NX uses 2 methods to decide the material direction of a plane strain element. The first method uses the rotation angle between the line made by nodes 1 and 2, as shown in figure 3.6.3.

Figure 3.6.4 Definition of material axis for Plane strain elements using angles



The second method uses an arbitrary coordinate system where the x axis is projected onto the element plane and that direction is assumed to be the major material axis, as shown in figure 3.6.4. The dotted lines represent the major material axes specified within the element. This projection method is equally applied when setting the ERCS for element result verification.

Figure 3.6.5 Definition of material axis for Plane strain elements using a coordinate system



DOF

Plane strain elements have directional DOF in the ECS x and y axis directions.

$$\mathbf{u}_i = \{u_i \quad v_i\}^T \quad (3.6.3)$$

Stress and strain

Plane strain elements are expressed using the 2D stress state and lateral normal stress. However, only 2D stresses are used for finite element formulation and thus the resultant force and in plane directional strain defined by the ECS are taken into account.

$$\mathbf{N} = \begin{Bmatrix} N_{xx} \\ N_{yy} \\ N_{xy} \end{Bmatrix}, \quad \boldsymbol{\epsilon} = \begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \quad (3.6.4)$$

(In plane directional resultant force and strain)

Load

The loads applied to a plane strain element are as follows:

Table 3.6.6 Loads applied on Plane strain elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Pressure load	Distributed load applied on an element side
Water pressure load	Distributed load applied on an element side Size determined by pore pressure/water level
Element temperature load	Element temperature that causes in-plane directional deformation
Prestress/Initial equilibrium force	Initial stress of an element and the corresponding equilibrium force

When a temperature load is applied to a plane strain element, thermal expansion in the thickness direction is constrained by the planar deformation state and thus the in plane directional deformation due to the Poisson effect increases.

Element result

On FEA NX, the results of a plane strain element are printed in the user defined reference coordinate system. The selectable element result coordinate systems (ERCS) are the ECS, MCS and arbitrary coordinate system.

Table 3.6.7 Result article of Plane strain elements

Result Article	Explanation
Stress	Stress component Position : vertex/element center $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \tau_{xy}$
	Principal stress Position : vertex/element center P_1, P_2 , Principal stress direction
	Von-Mises stress Position : vertex/element center σ_v
	Max shear stress Position : vertex/element center τ_{max}
	Mean pressure Position : vertex/element center P_0
	Strain
Principal strain Position : vertex/element center E_1, E_2 , Principal strain direction	
Von-Mises strain Position : vertex/element center ϵ_v	

	Max shear strain	Position : vertex/element center γ_{\max}
	Mean compression	Position : vertex/element center c_0
Force/ Moment	In-plane force	Position : vertex N_{xx}, N_{yy}, N_{xy}

The results of a solid element including pore water are printed in the effective stress and total stress components, and the total pore stress and excess pore stress are additionally printed.

Table 3.6.8 Result article of Plane strain elements including pore water

	Result article	Explanation
Stress	Stress component (Effective and total)	Position : vertex/element center $\sigma'_{xx}, \sigma'_{yy}, \sigma'_{zz}, \tau'_{xy}$ $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}$
	Pore stress (Total and excessive)	Position : vertex/element center $\sigma_{p,\text{total}}, \sigma_{p,\text{excessive}}$
	Mean pressure (Effective and total)	Position : vertex/element center P'_0, P_0

Element thickness

The thickness of a plane strain element needs to be consistent to satisfy the planar deformation condition. The Plane strain element thickness can be directly input in FEA NX, or assumed to be 1 if no value is input.

Element method selection

The usable Plane strain elements of FEA NX depend on the various element performance improvement methods. The following table lists the name, related finite element method and integration method. The standard value is shaded.

Table 3.6.9 Performance improvement methods used on Plane strain elements

Shape	Number of nodes	DOF	Name	Element method	Numerical integration of stiffness matrix	Calculation method for concentrated mass
Triangle	3			Assumed displacement	1 Point	Lobatto
Rectangle	4		Full integration	Assumed displacement	2X2 Points	Lobatto

		Reduced integration (stabilized)	Reduced Integration (stabilized)	1X1 Point	Lobatto
		Hybrid	Mixed method	2X2 Points	Lobatto
Triangle	6		Assumed displacement	3 Points	Diagonal term scaling
		Full integration	Assumed displacement	3X3 Points	Diagonal term scaling
Rectangle	8	Reduced integration	Reduced Integration	2X2 Points	Diagonal term scaling
		Hybrid	Mixed method	3X3 Points	Diagonal term scaling

The characteristics and precautions for each element method are as follows:

- ▶ 3 node element: Be aware that the element flexibility is greatly decreased when not enough elements are used, resulting in an inaccurate solution.
- ▶ 4 node element: The accuracy is generally high except for isoparametric elements which only use the assumed displacement method.
- ▶ 6 node element: The element performance can be greatly diminished when the node on an element side is not at its center.
- ▶ 8 node element: The results are generally accurate for all methods. Elements using reduced integration show similar performance for elements using the mixed method and show high calculation efficiency. However, the spurious zero energy mode can appear.

Nonlinear analysis

Geometric nonlinearity can be considered for Plane strain elements and material nonlinearity can be considered for elastic and nonlinear elastic materials. The additional result articles for nonlinear material usage are listed in the table below, and the material states are expressed using symbols.

Table 3.6.10 Nonlinear analysis result article of Plane strain elements

	Result article	Explanation
Stress	Equivalent stress	Position : Integral point Calculated according to plastic model, σ_{eq}
	Material status	Position : Integral point Elastic: - Plastic/Nonlinear elastic: ○ Unloading/reloading: ⊕ Tension failure: ⊕ Cap failure: ⊖



Strain	Equivalent strain	Position : Integral point Calculated according to plastic model, ϵ_{eq}
	Effective plastic strain	Position : Integral point e_p

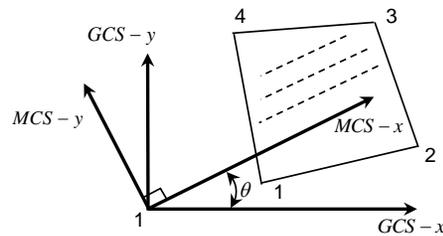
6.3 Axisymmetric Solid Elements

Axisymmetric solid elements use models, whose shape, material, load conditions etc. satisfy the axisymmetric condition about an arbitrary axis for analysis. Axisymmetric solid elements cannot be mixed with other element types and can be modeled using 3/4/6/8 node triangular and rectangular shapes.

Coordinate system

To use a transversely isotropic material on an axisymmetric solid, the major material axis needs to be in an appropriate direction. In this case, the MCS is used. FEA NX uses two methods to decide the material direction of a plane strain element. The first method uses the rotation angle between the GCS x axis, as shown in figure 3.6.5.

Figure 3.6.6 Definition of material axis for Axisymmetric solid elements using angles



The second method uses an arbitrary coordinate system, where the x axis is projected onto the element plane and that direction is assumed to be the major material axis. The dotted lines represent the major material axes specified within the element. This projection method is equally applied when setting the ERCS for element result verification.

DOF

Axisymmetric solid elements have displacement DOF in the GCS x (radial direction) and y directions.

$$\mathbf{u}_i = \{u_i \quad v_i\}^T \quad (3.6.5)$$

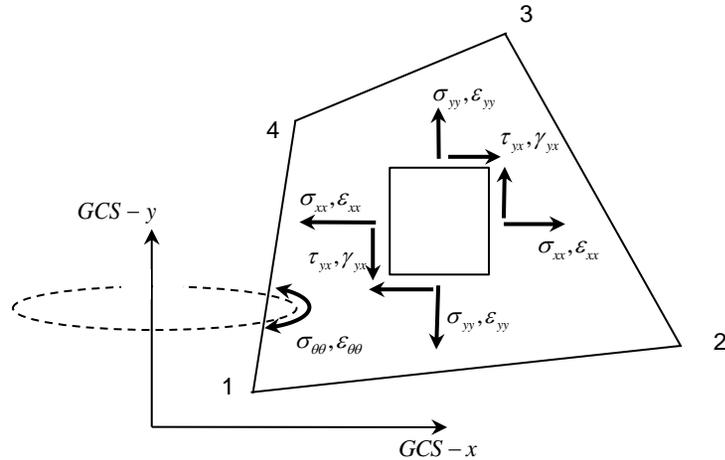
Stress and strain

Axisymmetric solid elements consider strain and stress defined on the GCS, and the components are as follows:

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{\theta\theta} \\ \sigma_{yy} \\ \tau_{yx} \end{Bmatrix}, \quad \boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{yy} \\ \gamma_{yx} \end{Bmatrix} \quad (3.6.6)$$

(In-plane/circumferential direction stress and strain)

Figure 3.6.7 Stress/strain of an Axisymmetric solid element



Load

The loads applied to axisymmetric solid elements are as follows:

Table 3.6.11 Loads applied on Axisymmetric solid elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density Distributed load applied on an element side
Pressure load	(Load acting on the outer surface of a structure when considering axisymmetry.) Distributed load applied on an element side
Water pressure load	Size determined by pore pressure/water level (Load acting on the outer surface of a structure when considering axisymmetry.)
Element temperature load	Element temperature that causes deformation of the axisymmetric section
Prestress/Initial equilibrium force	Initial stress of an element and the corresponding equilibrium force

Element result

On FEA NX, the results of an axisymmetric solid element are printed in the user defined reference coordinate system. The selectable systems are the ECS, MCS and arbitrary coordinate system. The element result component direction has the x, y direction and a circumferential direction of θ .



Table 3.6.12 Result article of Axisymmetric solid elements

	Result Article	Explanation
Stress	Stress component	Position : vertex/element center $\sigma_{xx} \mid \sigma_{yy} \mid \sigma_{\theta\theta} \mid \tau_{xy}$
	Principal stress	Position : vertex/element center $P_1 \mid P_2$
	Von-Mises stress	Position : vertex/element center σ_v
	Max shear stress	Position : vertex/element center τ_{max}
Strain	Strain component	Position : vertex/element center $\epsilon_{xx} \mid \epsilon_{yy} \mid \epsilon_{\theta\theta} \mid \gamma_{xy}$
	Principal strain	Position : vertex/element center $E_1 \mid E_2$
	Von-Mises strain	Position : vertex/element center ϵ_v
	Max shear strain	Position : vertex/element center γ_{max}

The results of an axisymmetric solid element including pore water are printed in the effective stress and total stress components, and the total pore stress and excess pore stress are additionally printed.

Table 3.6.13 Results of an Axisymmetric solid element including pore water

	Result article	Explanation
Stress	Stress component (Effective and total)	Position : vertex/element center $\sigma'_{xx} \mid \sigma'_{yy} \mid \sigma'_{\theta\theta} \mid \tau'_{xy}$ $\sigma_{xx} \mid \sigma_{yy} \mid \sigma_{\theta\theta}$
	Pore stress (Total and excessive)	Position : vertex/element center $\sigma_{p,excessive} \mid \sigma_{p,total}$
	Mean pressure (Effective and total)	Position : vertex/element center $P'_0 \mid P_0$

Element method selection

The usable axisymmetric solid elements of FEA NX depend on the various element performance improvement methods. The following table lists the name, related finite element method and integration method. The standard value is shaded.

Table 3.6.14 Performance improvement methods used on Axisymmetric solid elements

Shape	Number of nodes	Name	Element method	Numerical integration of stiffness matrix	Calculation method for concentrated mass
Triangle	3		Assumed displacement	3 Points	Lobatto
Rectangle	4	Full integration	Assumed displacement	2X2 Points	Lobatto
		Hybrid	Mixed method	2X2 Points	Lobatto
Triangle	6		Assumed displacement	3 Points	Diagonal term scaling
Rectangle	8		Assumed displacement	3X3 Points	Diagonal term scaling

Nonlinear analysis

Geometric nonlinearity can be considered for axisymmetric solid elements and material nonlinearity can be considered for elastic and nonlinear elastic materials. The additional result articles for nonlinear material usage are listed in the table below. The material states are expressed using symbols.

Table 3.6.15 Nonlinear analysis result article of Axisymmetric solid elements

	Result article	Explanation
Stress	Equivalent stress	Position : Integral point Calculated according to plastic model, σ_{eq}
	Material status	Position : Integral point Elastic: - Plastic/Nonlinear elastic: ○ Unloading/reloading: ⊕ Tension failure: ⊕ Cap failure: ⊖
Strain	Equivalent strain	Position : Integral point Calculated according to plastic model, ϵ_{eq}
	Effective plastic strain	Position : Integral point e_p

Section 7 Structural Elements

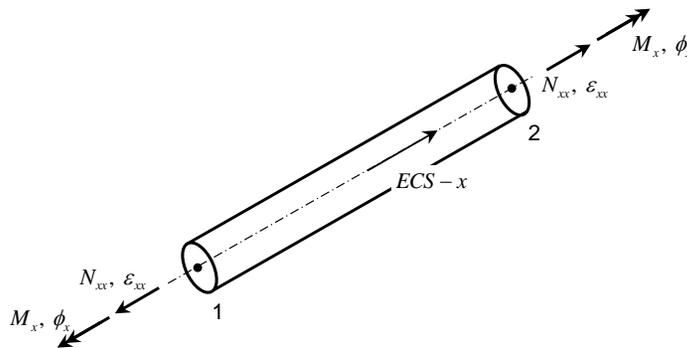
Structural elements are used to effectively express particular load states and Truss, Embedded Truss, Beam, Embedded Beam, Plane Stress, and Shell are classified as such. During modeling, the load state of the structural element and its deformation condition need to be checked to ensure that they are appropriate for the assumptions made.

7.1

Truss Element

Truss elements are a 1D line element defined by 2 nodes. They are generally used to model space Trusses or diagonal braces that have a relatively longer length than the cross section. It is generally used for modeling structural elements such as anchors, nails and rockbolts, which ignore flexural behavior.

Figure 3.7.1 Stress/strain and coordinate system of Truss elements



Coordinate system

The ECS x axis direction of a truss element is the direction from node 1 to node 2. Finite element formulation is done with reference to the ECS.

DOF

Truss elements have a displacement and rotation DOF in the ECS x axis direction.

$$\mathbf{u}_i = \{u_i\}_t, \quad \boldsymbol{\theta}_i = \{\theta_{xi}\} \quad (3.7.1)$$

Stress and strain

Truss elements express the ECS defined axial deformation and torsion, as shown in figure 3.7.1.

$$\mathbf{N} = \{N_{xx}\}_t, \quad \boldsymbol{\epsilon} = \{\epsilon_{xx}\} \quad (3.7.2)$$

(Axial force and strain)

$$\mathbf{T} = \{M_x\}, \phi = \{\phi_x\}$$

(3.7.3)

(Torsional moment and torsion)

Load

The loads applied to Truss elements are as follows:

Table 3.7.1 Loads applied on Truss elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Element temperature load	Element temperature that causes length deformation
Prestress/Initial equilibrium force	Initial element axial force and corresponding equilibrium force

Element result

The truss element result articles are as follows. The reference coordinate system is always the ECS.

Table 3.7.2 Result articles of Truss elements

	Result article	Explanation
Stress	Axial stress	Position : element center, σ_{xx}
	Torsional stress	Position : element center Calculated from torsional stress coefficient c ($\tau = Tc / J$)
Strain	Axial strain	Position : element center, ε_{xx}
	Torsional strain	Position : element center
Force /Moment	Axial force	Position : element center, N_{xx}
	Torque	Position : element center, M_x

Nonlinear analysis

Geometric nonlinearity can be considered for truss elements; and von Mises elastic materials can be applied. Also, truss elements can have gaps/hooks with a set initial spacing and compressive/tensile and nonlinear elastic material properties with allowable loading.

Table 3.7.3 Nonlinear analysis result articles of Truss elements

	Result article	Explanation
Stress	Equivalent stress	Position : element center Calculated according to plastic model, σ_{eq}
	Plastic status	Position : element center Elastic/Plastic, 0 / 1
Strain	Equivalent strain	Position : element center

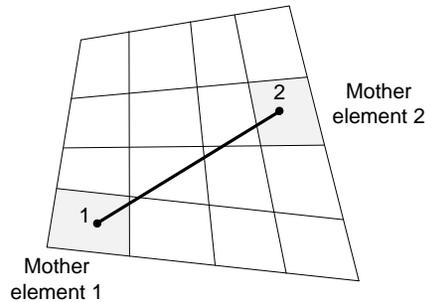
	Calculated according to plastic model, ϵ_{eq}
Effective plastic strain	Position : element center e_p

7.2 Embedded truss Element

The input shape, coordinate system, material property, etc of embedded truss elements are identical to that of truss elements. Embedded truss elements are generally used to model structural elements such as anchors, nails and rockbolts, which ignore flexural behavior. Also, gaps/hooks with a set initial spacing and compressive/tensile only and nonlinear elastic material properties with allowable loading can be applied. This information is introduced in chapter 4.

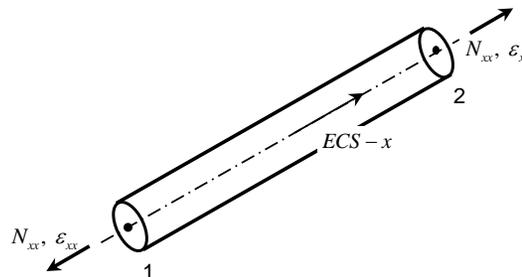
When using truss elements with different elements, the nodes must be shared. However, embedded truss elements do not require node sharing and are hence more convenient for modeling and analysis. Embedded truss elements are used in an embedded form inside a mother element, and the mother element can be a plane strain element or a solid element.

Figure 3.7.2 Embedded truss element within a mother element



The mother element is determined as the element that includes each embedded truss element node within itself, and the multi-point constraint is used to automatically constrain the nodal displacement of the embedded truss element to be the same as the internal displacement of the mother element.

Figure 3.7.3 Coordinate system and strain of Embedded truss elements



Coordinate system

The ECS x axis direction of an embedded truss element is the direction from node 1 to node 2. Finite element formulation is done with reference to the ECS.

DOF

Embedded truss elements have a displacement DOF in the ECS x axis direction.

$$\mathbf{u}_i = \{u_i\} \quad (3.7.4)$$

Stress and strain

Truss elements express the ECS defined axial deformation, as shown in figure 3.7.2.

$$\mathbf{N} = \{N_{xx}\}, \quad \boldsymbol{\varepsilon} = \{\varepsilon_{xx}\} \quad (3.7.5)$$

(Axial force and strain)

Load

The loads applied to Embedded truss elements are as follows:

Table 3.7.4 Embedded truss - Load type

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Element temperature load	Element temperature that causes length deformation
Prestress/Initial equilibrium force	Initial element axial force and corresponding equilibrium force

Element result

The embedded truss element result articles are as follows. The reference coordinate system is always the ECS.

Table 3.7.5 Result articles of Embedded truss elements

Result article	Explanation
Stress Axial stress	Position : element center σ_{xx}
Strain Axial strain	Position : element center ε_{xx}
Force Axial force	Position : element center N_{xx}

Nonlinear analysis

Geometric nonlinearity can be considered for embedded truss elements and von Mises elastic materials can be applied. Also, embedded truss elements can have gaps/hooks with a set initial spacing and compressive/tensile

only and nonlinear elastic material properties with allowable loading, depending on the axial direction. The nonlinear analysis result articles are as follows:

Table 3.7.6 Nonlinear result articles of Embedded truss elements

	Result article	Explanation
Stress	Equivalent stress	Position : element center Calculated according to plastic model, σ_{eq}
	Plastic status	Position : element center Elastic/Plastic, 0 / 1
Strain	Equivalent strain	Position : element center Calculated according to plastic model, ε_{eq}
	Effective plastic strain	Position : element center e_p

7.3

Beam Element

Beam elements are 1D line elements defined by 2/3 nodes and are often used when long elements, whose length is longer than its cross sectional size, experiences bending deformation. When the ratio of length to section width or height is larger than 1/5, the effects of shear deformation becomes very large and the use of shell or solid elements is recommended.

Coordinate system

The ECS x axis direction of a beam element is the direction from node 1 to node 2. The ECS y and z axis directions are determined by a reference node or reference vector. Figure 3.7.4 displays the determination of the x-y plane using a reference node. Here, reference node should not be on the ECS x axis. Figure 3.7.4 displays the determination of the x-y plane using a reference vector. The reference vector cannot be defined parallel to the ECS x axis. Finite element formulation is done with reference to the ECS.

Figure 3.7.4 Coordinate element setting using a reference node for a Beam element

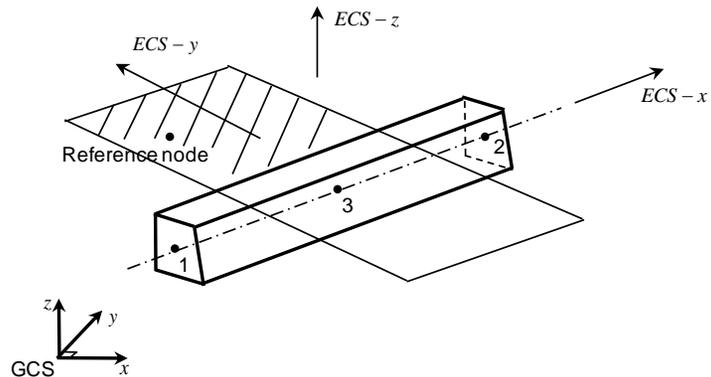
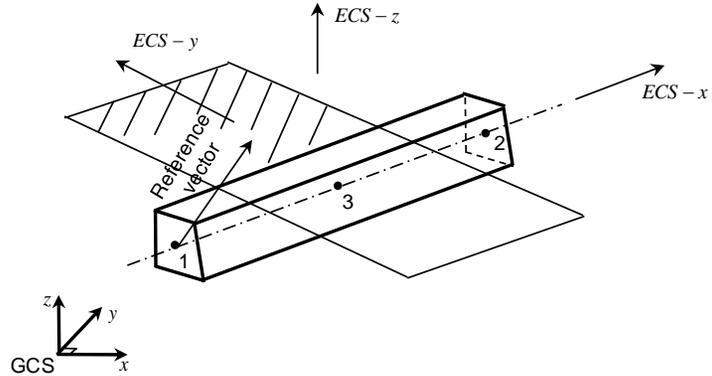


Figure 3.7.5 Coordinate element setting using a reference vector for a Beam element



DOF

Beam elements have a displacement and rotation DOF in all ECS axis directions.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T, \quad \boldsymbol{\theta}_i = \{\theta_{xi} \quad \theta_{yi} \quad \theta_{zi}\}^T \quad (3.7.6)$$

Stress and strain

Beam elements can consider the ECS defined axial deformation, bending, torsion, shear deformation etc., as shown in figure 3.7.6. When Euler's theory (which does not consider shear deformation) is applied, the shear area factor is input as 0.

$$\mathbf{N} = \{N_{xx}\}, \quad \boldsymbol{\varepsilon} = \{\varepsilon_{xx}\} \quad (3.7.7)$$

(Axial force and strain)

$$\mathbf{M} = \begin{Bmatrix} M_y \\ M_z \end{Bmatrix}, \quad \boldsymbol{\kappa} = \begin{Bmatrix} \kappa_y \\ \kappa_z \end{Bmatrix} \quad (3.7.8)$$

(Bending moment and curvature)

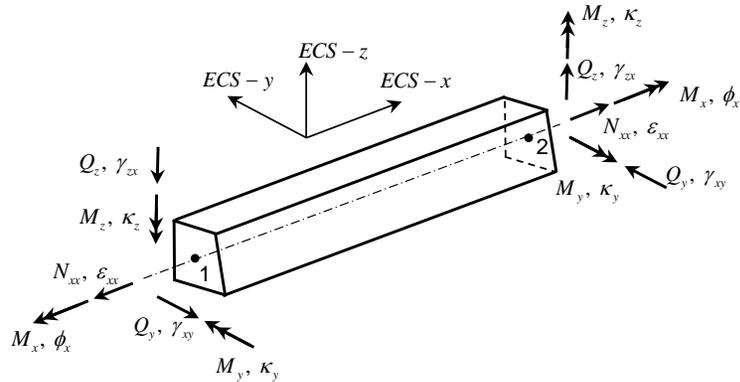
$$\mathbf{T} = \{M_x\}, \quad \boldsymbol{\phi} = \{\phi_x\} \quad (3.7.9)$$

(Torsional moment and torsion)

$$\mathbf{Q} = \begin{Bmatrix} Q_y \\ Q_z \end{Bmatrix}, \quad \boldsymbol{\gamma} = \begin{Bmatrix} \gamma_{xy} \\ \gamma_{zx} \end{Bmatrix} \quad (3.7.10)$$

(Shear force and shear strain)

Figure 3.7.6 Coordinate system and stress/strain of Beam elements



Load

The loads applied to beam elements are as follows:

Table 3.7.7 Loads applied on Beam elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density and nonstructural mass
Beam element load	Distributed load applied on an arbitrary section between element nodes, or concentrated load applied to an arbitrary position between element nodes
Beam element temperature load	Average section temperature that causes axial deformation Temperature gradient that causes bending
Prestress/Initial equilibrium force	Initial resultant force/moment of an element and corresponding equilibrium force/moment

Beam element loads can be applied as a distributed or concentrated load form as shown in figure 3.7.7 and can be set in the ECS or GCS direction. When defining a distributed load using the GCS, the effective distributed load that considers the angle between GCS and ECS is used, such that the load is applied to the length of the line normal to the load direction (figure 3.7.8).

Figure 3.7.7 Example of applied Beam element load

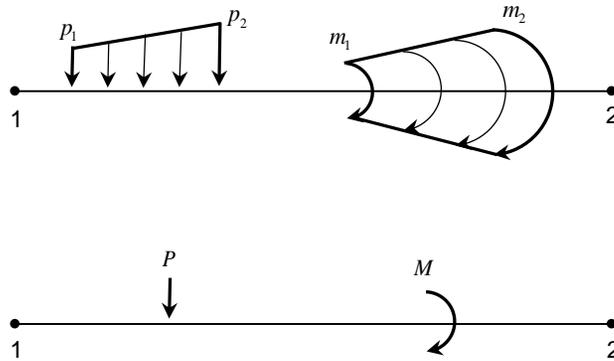
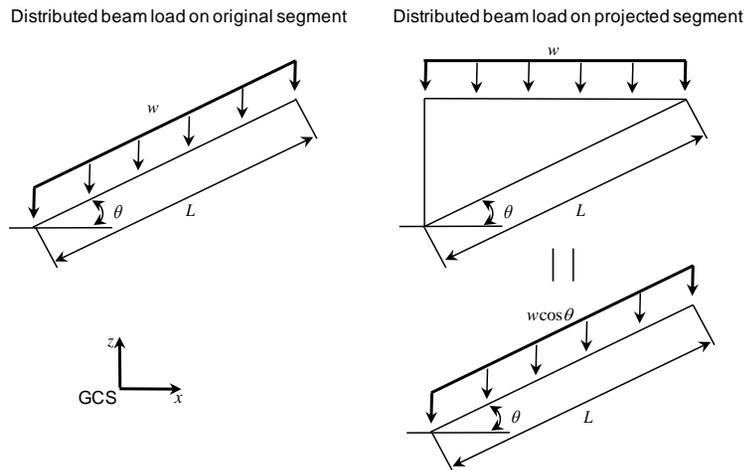


Figure 3.7.8 Beam element load adjustment according to load direction



Element result

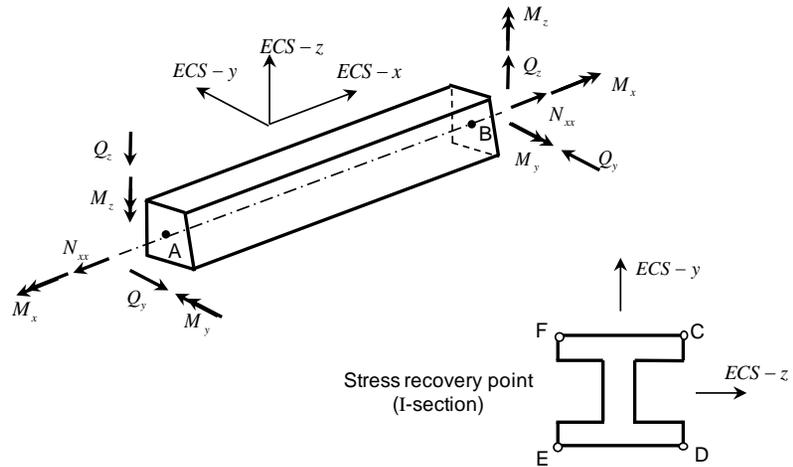
To effectively check the end results as well as internal results of a beam element, each element can be defined using multiple output segments. Beam elements results can be checked at each end (A-B) of the output segment and the result articles are as follows. The reference coordinate system is always the ECS.

Table 3.7.8 Result articles of Beam elements

	Result article	Explanation
Stress	Axial stress	Position : A-B of each segment σ_{xx}
	Torsional stress	Position : A-B of each segment Calculated from torsional stress coefficient c ($\tau = Tc / J$)
	Shear stress	Position : A-B of each segment Calculated from shear modulus of the section where max/min shear stress occur $\tau_{xy} = Q_y G_y$, $\tau_{xz} = Q_z G_z$
	Point stress	Position : A-B of each segment Stress due to bending at user defined position (C,D,E,F) σ_{xx}
	Max/Min stress	Position : A-B of each segment Max/min of axial and point stress sum at positions C-F
	Von-mises stress	Position : A-B of each segment $\sigma_v = \sqrt{\sigma_{xx}^2 + 3(\tau_{xy}^2 + \tau_{xz}^2)}$
Strain	Axial strain	Position : A-B of each segment ϵ_{xx}
	Torsional strain	Position : A-B of each segment
	Shear strain	Position : A-B of each segment
	Point strain	Position : A-B of each segment Strain due to bending at user defined position (C,D,E,F) ϵ_{xx}
	Max/Min strain	Position : A-B of each segment Max/min of axial and point strain sum at positions C-F
	Von-mises strain	Position : A-B of each segment Calculated using 3 stress components (ϵ_v)
Force /Moment	Axial force	Position : A-B of each segment N_{xx}
	Bending moment	Position : A-B of each segment M_y , M_z
	Torque	Position : A-B of each segment M_x
	Shear force	Position : A-B of each segment



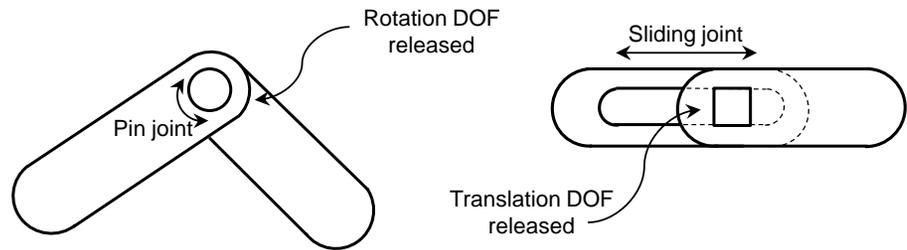
Figure 3.7.9 Output position and direction of Beam elements



Release condition

The end release condition is used when mutual constraint for motion in a particular direction does not occur, such as for pin joints at each end of a member. Because this condition is applied to the ECS, the coordinate relationship must be accurately understood and applied when using connection release in the GCS. Also, an unconstrained DOF is added to the applied node, resulting in instability of the entire structure. Hence, comprehensive examination is needed for structural safety when applying the end release condition.

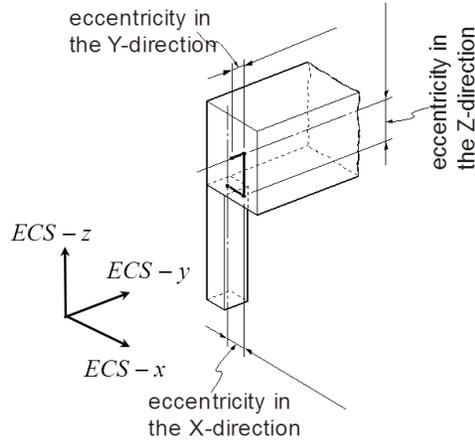
Figure 3.7.10 Example of applied end release condition



Offset

Offset can be used on beam elements when the neutral axis is isolated from the nodes, or when the neutral axis between connecting elements are not the same. Offset is applied to the set NCS on a beam element node, and change in element length is assumed when the offset is set in the axial direction of the element.

Figure 3.7.11 Example of applied offset



Tapered section

The change in section of a beam element is defined about the ECS axis. Using this, the property change (depending on the sectional change for each axis) is assumed as follows:

Table 3.7.9 Order depending on the sectional changes in a tapered section of a Beam element

y-axis	z-axis	A	I_{xx}	I_{yy}	I_{zz}
constant	constant	1	1	1	1
constant	linear	1	1	3	1
linear	constant	1	1	1	3
linear	linear	2	2	4	4

$$A(x) = \left\{ \sqrt[m]{A_1} + \left(\sqrt[m]{A_2} - \sqrt[m]{A_1} \right) \left(\frac{x}{L} \right) \right\}^m, \quad m = 1, 2 \tag{3.7.11}$$

$$I(x) = \left\{ \sqrt[n]{I_1} + \left(\sqrt[n]{I_2} - \sqrt[n]{I_1} \right) \left(\frac{x}{L} \right) \right\}^n, \quad n = 1, 3, 4 \tag{3.7.12}$$

Linear stiffness considers tapered sections for nonlinear analysis, but sections with geometric stiffness substitute an equivalent section for application.

$$A_{off} = \begin{cases} \frac{A_1 + A_2}{2}, & m = 1 \\ \frac{A_1 + \sqrt{A_1 A_2} + A_2}{3}, & m = 2 \end{cases} \quad (3.7.13)$$

$$I = \frac{I_1 + 4\sqrt{I_1^3 I_2} + \sqrt{I_1 I_2} + 4\sqrt{I_1 I_2^3} + I_2}{5} \quad (3.7.14)$$

Nonlinear analysis

Beam elements can only consider geometric nonlinearity, and nonlinear or non-elastic materials cannot be used. Hence, there are no additional result articles when performing nonlinear analysis.

7.4 Embedded beam Element

The input shape, coordinate system, DOF and result of embedded beam elements are identical to that of beam elements. The end release condition and offset can be applied in embedded beam elements. Nonlinear material model cannot be used for embedded beam elements, but these elements can be used in geometric nonlinear analysis. (For more details, please refer to the 7.3 Beam Element.)

When using beam elements with different elements, the nodes must be shared. However, embedded beam elements do not require node sharing and are hence more convenient for modeling similar to the embedded truss elements. Embedded beam elements are used in an embedded form inside a mother element as shown in Figure 3.7.2, and the mother element can be a plane strain element or a solid element. The mother element is determined as the element that includes each embedded beam element node within itself, and the multi-point constraint is used to automatically constrain the nodal displacement of the embedded beam element to be the same as the internal displacement of the mother element. (For more details about multi-point constraint, please refer to the 8.9 Rigid Link/Interpolation Element.)

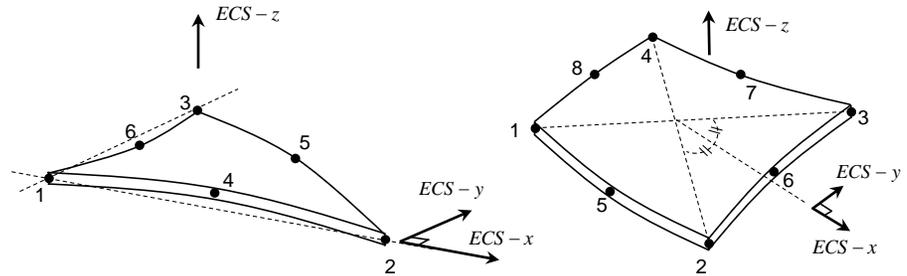
7.5 Plane stress Element

Plane stress elements are triangular or rectangular elements consisting of 3/4/6/8 nodes in space. It is mainly used to model sheets with uniform thickness and has 2D stress states.

Coordinate system

The ECS of a triangular plane stress element has the z axis in the normal direction to the element plane and the x axis in the direction from node 1 to node 2. For rectangular plane stress elements, the z axis is in the direction normal to the element plane and the x axis in the direction that bisects the intersection angle between diagonal lines made by nodes 1 and 3, and nodes 4 and 2. The finite element formulation of a plane stress element is done about the ECS.

Figure 3.7.12 Coordinate system of Plane stress elements



FEA NX uses 2 methods to decide the material direction of a plane stress element. The methods are the same for plane strain elements.

DOF

Plane stress elements have a displacement DOF in the ECS x and y axis direction.

$$\mathbf{u}_i = \{u_i \quad v_i\}^T \quad (3.7.15)$$

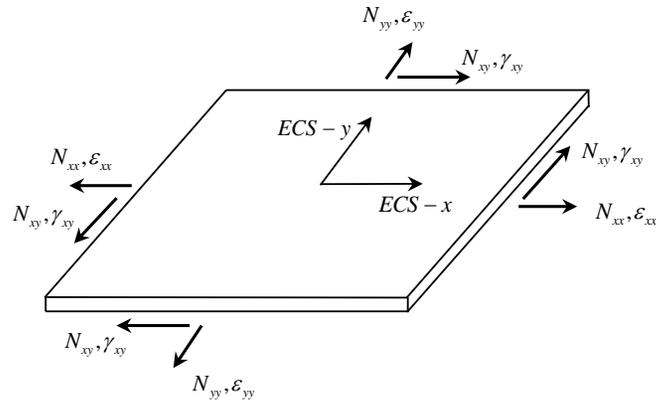
Stress and strain

Because the basic assumption of plane stress elements is the 2D stress state, the in-plane direction resultant force and strain defined on the ECS can be considered.

$$\mathbf{N} = \begin{Bmatrix} N_{xx} \\ N_{yy} \\ N_{xy} \end{Bmatrix}, \quad \boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \quad (3.7.16)$$

(In-plane direction resultant force and strain)

Figure 3.7.13 Stress/strain of Plane stress elements

**Load**

The loads applied to plane stress elements are as follows:

Table 3.7.10 Loads applied on Plane stress elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Pressure load	Distributed load applied on an element plane
Water pressure load	Distributed load applied on an element plane Size determined by pore pressure/water level
Element temperature load	Element temperature that causes in-plane deformation
Prestress/Initial equilibrium force	Initial stress of an element and corresponding equilibrium force

Plane stress elements do not have stiffness in the lateral direction, but this lateral component is considered for loads applied on the element and the mass.

Element result

On FEA NX, the results of a plane stress element are printed in the user defined reference coordinate system. The selectable element result coordinate systems (ERCS) are the ECS, MCS and arbitrary coordinate system.

Table 3.7.11 Result articles of Plane stress elements

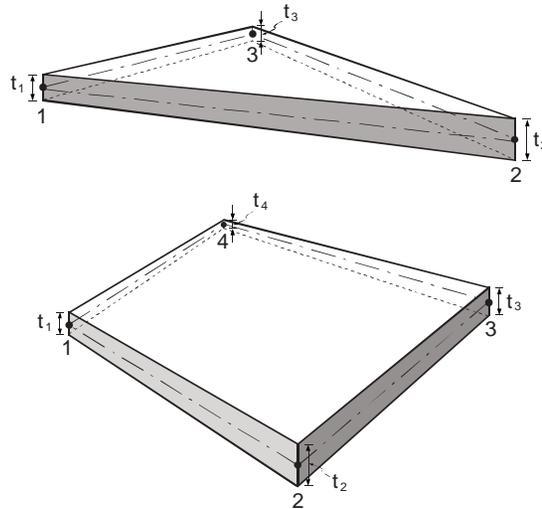
Result article	Explanation
In-plane stress	Position : vertex/element center $\sigma_{xx} , \sigma_{yy} , \tau_{xy}$
Stress	
Principal stress	Position : vertex/element center P_1 , P_2 , Principal stress direction
Von-Mises stress	Position : vertex/element center σ_v

Strain	Max shear stress	Position : vertex/element center τ_{\max}
	In-Plane strain	Position : vertex/element center $\epsilon_{xx}, \epsilon_{yy}, \epsilon_{zz}, \gamma_{xy}$
	Principal strain	Position : vertex/element center E_1, E_2 , Principal strain direction
	Von-Mises strain	Position : vertex/element center ϵ_v
	Max shear strain	Position : vertex/element center γ_{\max}
Force/ Moment	In-plane force	Position : vertex N_{xx}, N_{yy}, N_{xy}

Element thickness

On FEA NX, the thickness of Plane stress elements can be set as shown in figure 3.7.14. Keep in mind that the thickness of high order elements (6/8 nodes) can only be defined at vertices.

Figure 3.7.14 Thickness specification of Plane stress elements

**Element method selection**

The usable plane stress elements on FEA NX depend on the various element performance improvement methods. Each finite element method and integration method is equal to that of plane strain elements.

Nonlinear analysis

Geometric nonlinearity can be considered for plane stress elements and von Mises elastic materials can be applied. The nonlinear analysis result articles are identical to that of plane strain elements.

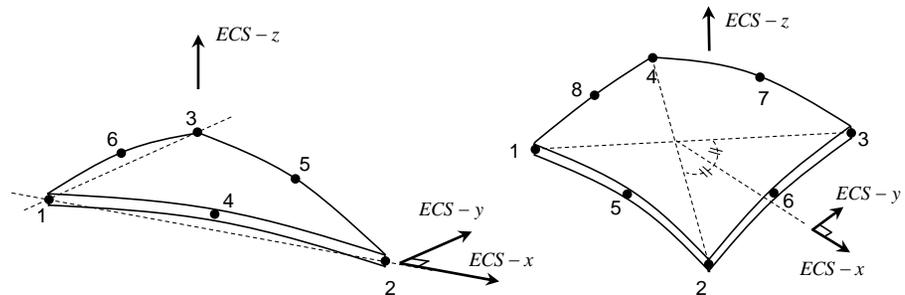
7.6 Shell Element

Shell elements are triangular or rectangular elements consisting of 3/4/6/8 nodes on a curve surface. It is mainly used on thin structures that experience bending deformation and 2D stress states. Bending and shear deformation can be considered.

Coordinate system

Shell elements exist on curved surfaces and the nodes may not exist on the same plane. This needs to be considered when defining the ECS. The ECS of a triangular plane stress element has the x axis in the direction from node 1 to node 2, and the z axis direction is the outer product between this vector and the vector from node 1 to node 3. For rectangular shell elements, the x axis is in the direction that bisects the intersection angle between diagonal lines made by nodes 1 and 3, and the z axis direction is the outer product direction of the two vectors. The finite element formulation of a shell element is done about the ECS.

Figure 3.7.15 Coordinate system of shell elements



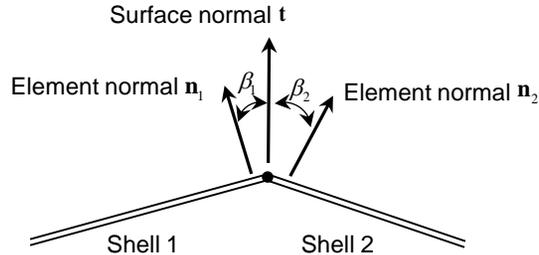
The method for defining the MCS is the same as for plane stress elements: using the angle from the line connecting nodes 1 and 2 or using an arbitrary coordinate system.

Curved surface modeling

When using a shell element on FEA NX, the deformation is expressed using the motion of a 'director', the inherent normal vector that is assumed to exist at each node²¹. Because the rotational DOF direction at an element node is defined with reference to the director, moment in the director direction does not exist as an internal force of the element. This vector can be normal to the element plane, but it is not so for curved surfaces modeled as a shell element.

²¹ Simo, J.C. and Fox, D.D., "On a stress resultant geometrically exact shell model. Part I: Formulation and optimal parametrization," Computer Methods in Applied Mechanics and Engineering, Vol. 72, 1989

Figure 3.7.16 Angle between shell elements used to model a curved surface



For example, if a small corrugation angle exists between adjacent elements as shown in figure 3.7.16, the vector normal to the curved surface can be calculated as follows:

$$\mathbf{t} = \frac{\sum \mathbf{n}_i}{\|\sum \mathbf{n}_i\|} \quad (3.7.17)$$

\mathbf{t} : curved surface normal vector

\mathbf{n}_i : vector normal to element plane

Here, if the angle β between \mathbf{t} and \mathbf{n}_i is above the tolerance, it is considered as a bent structure rather than a segment of a curved surface. Hence, the curved surface normal vector is not defined. A normal vector at a node where the curved surface normal vector is not defined is classified as a director.

Accurately expressing the geometric shape by creating curved surface normal vectors contributes greatly to the accuracy of the results, but precautions need to be made when modeling half or 1/4 of a cylindrical shape using symmetric conditions. The side where the symmetric condition is applied does not have Shell 2 in figure 3.7.16 and thus a geometrically correct curved surface normal vector cannot be obtained. In this case, it is better not to generate a curved surface normal vector.

DOF

Shell elements have a displacement DOF in the ECS x, y, z axis directions.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T \quad (3.7.18)$$

The rotational DOF is defined by the two directions normal to the director.

$$\boldsymbol{\theta}_i = \{\theta_{xi} \quad \theta_{yi}\} \quad (3.7.19)$$

The director is the curved surface normal vector or element normal vector, as explained above.

Stress and strain

Truss elements consider the ECS defined 2D stress state and bending, shear deformation, as shown in figure 3.7.17. FEA NX always considers shear deformation for shell elements.

$$\mathbf{N} = \begin{Bmatrix} N_{xx} \\ N_{yy} \\ N_{xy} \end{Bmatrix}, \boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \tag{3.7.20}$$

(In-plane resultant force and strain)

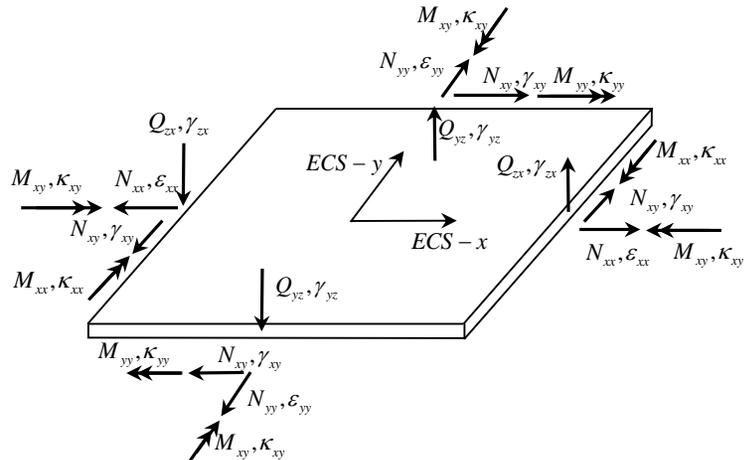
$$\mathbf{M} = \begin{Bmatrix} M_{xx} \\ M_{yy} \\ M_{xy} \end{Bmatrix}, \boldsymbol{\kappa} = \begin{Bmatrix} \kappa_{xx} \\ \kappa_{yy} \\ \kappa_{xy} \end{Bmatrix} \tag{3.7.21}$$

(Bending moment and curvature)

$$\mathbf{Q} = \begin{Bmatrix} Q_{zx} \\ Q_{yz} \end{Bmatrix}, \boldsymbol{\gamma} = \begin{Bmatrix} \gamma_{zx} \\ \gamma_{yz} \end{Bmatrix} \tag{3.7.22}$$

(Shear strength and shear strain)

Figure 3.7.17 Stress/strain of shell elements



Load

The loads applied to shell elements are as follows:

Table 3.7.12 Loads applied on shell elements

Load type	Explanation
Self-weight due to gravity	Applied to the material density
Pressure load	Distributed load applied on an element plane or

	distributed load applied on an element side
Water pressure load	Distributed load applied on an element plane Size determined by pore pressure/water level
Element temperature load	Element temperature that causes in-plane direction deformation Temperature gradient that causes bending deformation
Prestress/Initial equilibrium force	Initial resultant force/moment of an element and corresponding equilibrium force/moment

Element result

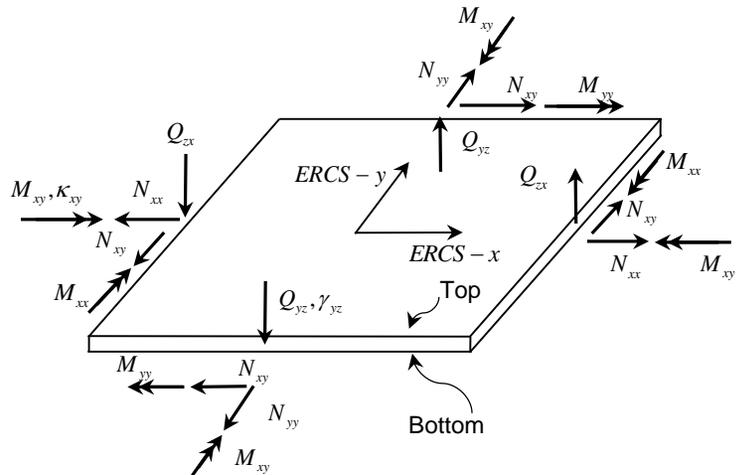
The shell element results on FEA NX are provided at 2 points (top/bottom) or 3 points (top/middle/bottom) in the element thickness direction. When a shell element is used, the result articles are as follows and the reference coordinate system can be specified by the user. The selectable coordinate systems are the ECS, MCS and arbitrary coordinate system.

Table 3.7.13 Result articles of shell elements

	Result article	Explanation
Stress	In-plane stress	Position : top/(middle)/bottom, vertex/element center $\sigma_{xx}, \sigma_{yy}, \tau_{xy}$
	Normal stress	Position : top/(middle)/bottom, vertex/element center σ_{zz}
	Principal stress	Position : top/(middle)/bottom, vertex/element center P_1, P_2 , Principal stress direction
	Von-Mises stress	Position : top/(middle)/bottom, vertex/element center σ_v
	Max shear stress	Position : top/(middle)/bottom, vertex/element center τ_{max}
	Fiber distance	Position : top/(middle)/bottom, vertex/element center Stress calculation position in thickness direction
	Maximum values	Position : Maximum value among vertex/element center and top/bottom, ($P_1, P_2, \sigma_v, \tau_{max}$)
Strain	In-plane strain	Position : top/(middle)/bottom, vertex/element center $\epsilon_{xx}, \epsilon_{yy}, \gamma_{xy}$

Normal strain	Position : top/(middle)/bottom, vertex/element center ϵ_{zz}
Principal strain	Position : top/(middle)/bottom, vertex/element center E_1, E_2 , Principal strain direction
Von-Mises strain	Position : top/(middle)/bottom, vertex/element center ϵ_v
Max shear strain	Position : top/(middle)/bottom, vertex/element center γ_{max}
Maximum values	Position : Maximum value among vertex/element center and top/bottom, ($E_1, E_2, \epsilon_v, \gamma_{max}$)
Force/ Moment	In-plane force N_{xx}, N_{yy}, N_{xy}
	Bending moment M_{xx}, M_{yy}, M_{xy}
	Shear force Q_{zx}, Q_{zy}

Figure 3.7.18 Result output direction of shell elements



Element thickness and material

The thickness of a shell element can be defined at the vertices in the same way as plane stress elements. The material and effective thickness for bending and shear deformation can also be assigned individually. For example, assuming the thickness for in-plane direction behavior (membrane thickness) to be t , the following values can be set:

- ▶ $12I / t^3$: Bending stiffness ratio, calculated using the actual bending stiffness I and t
- ▶ t_s / t : Ratio between actual shear deformation t_s and t

The effective thickness above is only used to calculate the stiffness and internal forces and is not used to calculate the mass matrix. Also, when gravity, rotational inertial force or any other mass effects are considered, the membrane material is used.

Offset

Offset can be used on shell elements when the neutral axis is isolated from the nodes, or when the neutral axis between connecting elements are not the same. The offset of shell elements can have a constant value within the element in the director direction.

Release condition

The end release condition is used when mutual constraint for motion in a particular direction does not occur, such as for pin joints. Unlike beam elements, the NCS is applied to the end release conditions of a shell element. When entering the connection release about the GCS, the coordinate relationship needs to be accurately understood. Also, an unconstrained DOF is added to the applied node, resulting in instability of the entire structure. Hence, comprehensive examination is needed for structural safety when applying the end release condition.

Element method selection

The usable shell elements of FEA NX depend on the various element performance improvement methods. Particularly for shell elements, many different methods are applied depending on the direction of element deformation (such as in-plane direction or lateral direction). The following table lists the name, related finite element method and integration method. The standard value is shaded.

Table 3.7.14 Performance improvement methods used on shell elements

Shape	Number of nodes	Node DOF	Name	Element method (In-plane/lateral)	Numerical integration of stiffness matrix	Calculation method for concentrated mass
Triangle	3	5	Full integration	Assumed displacement /ANS	1 Point	Lobatto

			Hybrid	Mixed method /ANS+ Mixed method	3 Points	Lobatto
			Full integration	Assumed displacement /ANS	2X2 Points	Lobatto
Rectangle	4	5	Reduced integration (stabilized)	Reduced Integration /ANS (stabilized)	1x1 Point	Lobatto
			Hybrid	Mixed method /ANS+ Mixed method	2X2 Points	Lobatto
Triangle	6	5		Assumed displacement /ANS	3 Points	Diagonal term scaling
			Full integration	Assumed displacement /ANS	3X3 Points	Diagonal term scaling
Rectangle	8	5	Reduced integration	Reduced Integration	2X2 Points	Diagonal term scaling
			Hybrid	Mixed method /ANS+ Mixed method	3X3 Points	Diagonal term scaling

The characteristics and precautions for each element method are as follows:

- 6 node element: The lateral displacement is generally large compared to other elements. The element performance can be greatly diminished when the node on an element side is not at its center.
- 8 node element: The results are generally accurate for all methods. Elements using reduced integration show similar performance to elements using the mixed method and show high calculation efficiency. However, the spurious zero energy mode can appear.

Nonlinear analysis

Geometric nonlinearity can be considered for shell elements and von Mises elastic materials can be applied. When an elastic material is applied, different material behavior is shown depending on the position in the thickness direction. In this case, the numerical calculation of in-plane direction resultant force and bending moment or the stiffness is needed, and FEA NX uses the Simpson integration method. Particularly, when



applying the elastic constitutive equation that calculates stress from strain, the planar stress state excluding shear deformation is fundamentally assumed. The nonlinear analysis result articles are as follows:

Table 3.7.15 Nonlinear analysis result articles of shell elements

	Result article	Explanation
Stress	Equivalent stress	Position : top/bottom, integral point Calculated according to plastic model, σ_{eq}
	Plastic status	Position : top/bottom, integral point Elastic/Plastic, 0 / 1
Strain	Equivalent strain	Position : top/bottom, integral point Calculated according to plastic model, ε_{eq}
	Effective plastic strain	Position : top/bottom, integral point e_p

7.7 Layered Shell Element

Layered shell elements are used to effectively analyze thin structures of laminated layers comprising different materials or different principal axes in the thickness direction such as composite materials and sandwiches. The basic coordinate system, curved surface model and the degrees of freedom are identical to those of the general shell elements. A layered shell element is a triangular or a quadrilateral element consisted of 3, 4, 6 or 8 nodes. That is, a layered shell element, which is based on the primary shear deformation theory, has the identical finite element formulation as the shell element.

Shear Stiffness Calculation & Shear Stress Restoration in the Transverse Direction

Unlike an isotropic material, the stiffness calculation in the transverse direction, which is based on the shear correction factor, cannot be generalized for a laminated composite material. Therefore, it is desirable to calculate the shear stiffness in the transverse direction using a number of assumed deformation shapes and stress equilibrium equations.²² The use of the series of the above procedures also renders an advantage of the possibility of restoring shear stress in the transverse direction.

The in-plane stresses based on the primary shear deformation theory and the transverse shear stresses based on the 3-dimensional stress equilibrium equations are expressed as follows:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \mathbf{C}^{(k)} (\boldsymbol{\varepsilon}_o + z\boldsymbol{\kappa}) \quad (3.7.23)$$

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \int_{\zeta=0}^{\zeta=z} \left(\begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix} \mathbf{C}^{(k)} (\boldsymbol{\varepsilon}_{o,x} + z\boldsymbol{\kappa}_x) + \begin{bmatrix} 0 & 0 & 1 \\ 0 & 1 & 0 \end{bmatrix} \mathbf{C}^{(k)} (\boldsymbol{\varepsilon}_{o,y} + z\boldsymbol{\kappa}_y) \right) d\zeta \quad (3.7.24)$$

If the effects of the in-plane forces are ignored in the constitutive equation of a layered shell ($\mathbf{N} = \mathbf{0}$), the strain and curvature at the neutral plane are expressed as follows:

$$\begin{aligned} \boldsymbol{\varepsilon}_o &= -\mathbf{A}^{-1}\mathbf{B}\boldsymbol{\kappa} \\ \boldsymbol{\kappa} &= \mathbf{D}^{*-1}\mathbf{M} \end{aligned} \quad (3.7.25)$$

where, $\mathbf{D}^* = \mathbf{D} - \mathbf{B}^T \mathbf{A}^{-1} \mathbf{B}$. Using Equation (3.7.25), the transverse shear stresses can be expressed again as follows:

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix} \mathbf{F}(z) \mathbf{M}_{,x} + \begin{bmatrix} 0 & 0 & 1 \\ 0 & 1 & 0 \end{bmatrix} \mathbf{F}(z) \mathbf{M}_{,y} \quad (3.7.26)$$

$$\mathbf{F}(z) = \left(\int_{\zeta=0}^{\zeta=z} \mathbf{C}^{(k)} d\zeta \mathbf{A}^{-1} \mathbf{B} - \int_{\zeta=0}^{\zeta=z} \zeta \mathbf{C}^{(k)} d\zeta \right) \mathbf{D}^{*-1} \quad (3.7.27)$$

When cylindrical bending behaviors about the x- and y-axes are assumed, the in-plane differential values and the transverse shear forces can be expressed in a simple relationship.

$$\begin{Bmatrix} Q_{xz} \\ Q_{yz} \end{Bmatrix} = - \begin{Bmatrix} M_{x,x} \\ M_{y,y} \end{Bmatrix} \quad (3.7.28)$$

Using the above, the transverse shear stresses are expressed as,

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} F_{11} & F_{32} \\ F_{31} & F_{22} \end{bmatrix} \begin{Bmatrix} Q_{xz} \\ Q_{yz} \end{Bmatrix} \quad (3.7.29)$$

That is, the transverse shear stresses are determined by the transverse shear forces and the shape function, $\mathbf{F}(z)$, which is based on the properties, thicknesses and the lamination angles of the materials constituting the laminated plate. Using this, the transverse shear stiffness, \mathbf{G} , which is calculated by integration based on the lamination theory, can be expressed as follows:

$$\tilde{\mathbf{H}} = \left[\int \begin{bmatrix} F_{11} & F_{32} \\ F_{31} & F_{22} \end{bmatrix}^T \mathbf{G}^{-1} \begin{bmatrix} F_{11} & F_{32} \\ F_{31} & F_{22} \end{bmatrix} \right]^{-1} \quad (3.7.30)$$

Element Results

The layered shell elements in FEA NX generate results at the center or top and bottom of each ply in the thickness direction. Also, the results include the maximum and minimum values of the entire laminated plate. Stress and strain results are produced in the principal axis directions of the material in each ply.

Table 3.7.16 Result Items of a layered shell element

	Result Items	Descriptions
Stress	In-plane stress	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center $\sigma_{11}, \sigma_{22}, \tau_{12}$
	Transverse shear stress	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center



		τ_{1z}, τ_{2z}
	Principal stress	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center P_1, P_2
	Von-mises stress	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center σ_v
	Max Shear Stress	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center τ_{max}
	In-plane strain	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center $\epsilon_{11}, \epsilon_{22}, \gamma_{12}$
Strain	Principal strain	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center E_1, E_2 Principal strain directions
	Von-Mises strain	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center ϵ_v
	Max Shear Strain	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center γ_{max}
Force/ Moment	In-plane force	Location: Vertices/element center N_{xx}, N_{yy}, N_{xy}
	Bending moment	Location: Vertices/element center M_{xx}, M_{yy}, M_{xy}
	Shear force	Location: Vertices/element center Q_{xz}, Q_{yz}
Misc	Failure index/strength ratio	Location: Center/top-bottom of each ply, maximum/minimum of laminated plate, vertices/element center



	Tsai-Wu, Tsai-Hill, Hoffman, max-strain, max-stress or LaRC02
--	---

Nonlinear Analysis

When using elasto-plastic materials, the number of Simpson integration points can be defined at each ply. The result items from nonlinear analysis are as follows:

Table 3.7.16 Result items of a layered shell element in nonlinear analysis

	Result Items	Descriptions
Stress	Equivalent stress	Location: Center/top-bottom of each ply, maximum/minimum integration points of laminated plate Calculated on the basis of plastic models, σ_{eq}
	Plastic status	Location: Center/top-bottom of each ply, maximum/minimum integration points of laminated plate Elasticity/plasticity, 0/1
Strain	Equivalent strain	Location: Center/top-bottom of each ply, maximum/minimum integration points of laminated plate Calculated on the basis of plastic models, ε_{eq}
	Effective plastic strain	Location: Center/top-bottom of each ply, maximum/minimum integration points of laminated plate e_p

Section 8

Special Purpose Elements

8.1

Interface Element

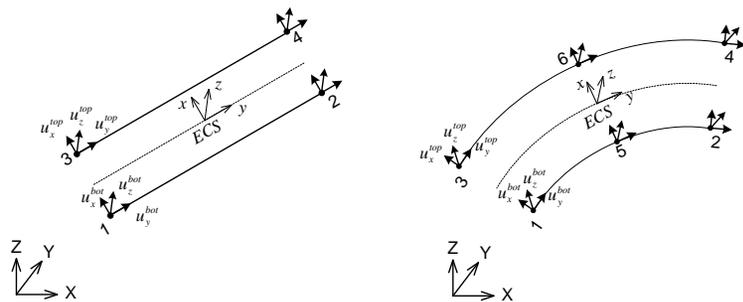
Interface elements account for normal and shear directional relative displacement and interface traction that simulate face/face and line/line behavior. FEA NX divides Interface elements into line Interface elements and plane Interface elements. Line Interface elements are often used to simulate behavior between planar elements (Plane stress, Plane strain) or between planar elements and Truss/Beam elements. Plane Interface elements are often used to simulate behavior between solid elements or between planar elements and solid elements.

Coordinate system

In FEA NX, the element coordinate system and node order of line and plane Interface elements are as shown figures 3.8.1 and 3.8.2.

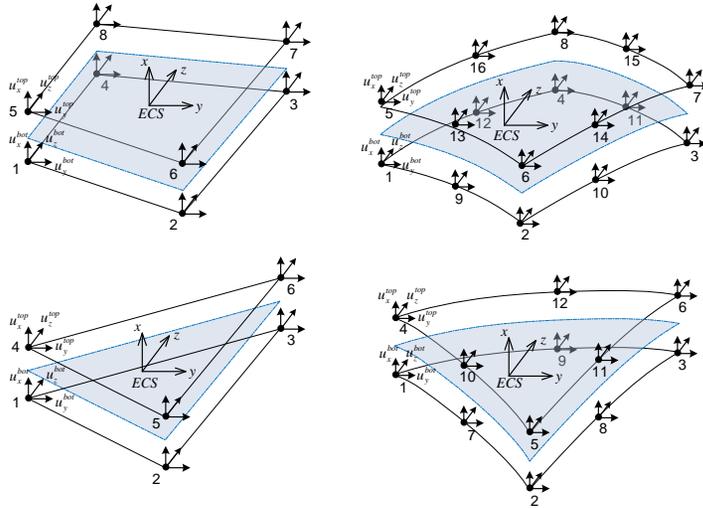
Line Interface elements can use 4 node and 6 node elements.

Figure 3.8.1 Line Interface elements



Plane Interface elements can use triangular shapes (6 nodes, 12 nodes) or rectangular shapes (8 nodes, 16 nodes).

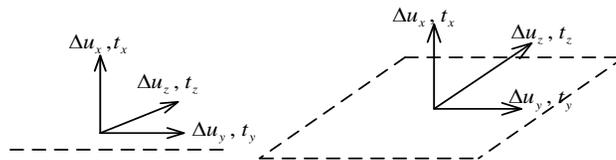
Figure 3.8.2 Plane Interface elements



DOF

Interface elements have 3 displacement DOF in the 3 axes of the ECS. Finite element formulation is done with reference to the ECS.

Figure 3.8.3 DOF of Interface elements



$$\mathbf{t} = \begin{Bmatrix} t_x \\ t_y \\ t_z \end{Bmatrix}, \Delta \mathbf{u} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \\ \Delta u_z \end{Bmatrix} = \begin{Bmatrix} u_x^{top} - u_x^{bot} \\ u_y^{top} - u_y^{bot} \\ u_z^{top} - u_z^{bot} \end{Bmatrix} \quad (3.8.1)$$

$t_x / \Delta u_x$: Normal interface traction / Normal relative displacement

$t_y, t_z / \Delta u_y, \Delta u_z$: Tangential interface traction / Tangential relative displacement

The relationship between relative displacement $\Delta \mathbf{u}$ and interface traction \mathbf{t} can be defined using the following equation:

$$\mathbf{t} = \mathbf{D} \cdot \Delta \mathbf{u} \quad (3.8.2)$$

$$\mathbf{D} = \begin{bmatrix} k_x & 0 & 0 \\ 0 & k_y & 0 \\ 0 & 0 & k_z \end{bmatrix} \quad (3.8.3)$$

\mathbf{D} : interface linear constitutive equation

Here, the displacement of the top/bottom faces is interpolated using the shape function.

The \mathbf{D} from equation (3.8.3) is the linear elastic constitutive equation. Here, it can be verified that the relative displacement and interface traction do not have a correlation in each direction. Detailed information is introduced in Chapter 4: Material Models.

Numerical integration uses the Newton-Cotes integration method, and the number of integral points depending on the element shape is as follows. When the Gaussian integral is used for Interface elements, the position of integral points exists within the element and nodal behavior of the interface cannot be accurately simulated.

Table 3.8.1 Number of integral points of Interface elements

Interface element	Number of integral points	
Line interface	4 Nodes	2
	6 Nodes	4
Triangle plane interface	6 Nodes	3
	12 Nodes	6
Quadrilateral plane interface	8 Nodes	4
	16 Nodes	16

Element result

The Interface element results are output with reference to a user specified coordinate system. The selectable coordinate systems are ECS and GCS.

Table 3.8.2 Result articles of Interface elements

Interface element	Result article
Stress	Position : element node/center point $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}$
	Position : element node/center point Pore stress (Total and excessive) $\sigma_{p,total}, \sigma_{p,excessive}$
Strain	Position : element node/center point $\Delta u_x, \Delta u_y, \Delta u_z$

Nonlinear analysis

Geometric nonlinearity can be considered for Interface elements and Coulomb friction materials can be applied. The nonlinear analysis result articles are as follows:

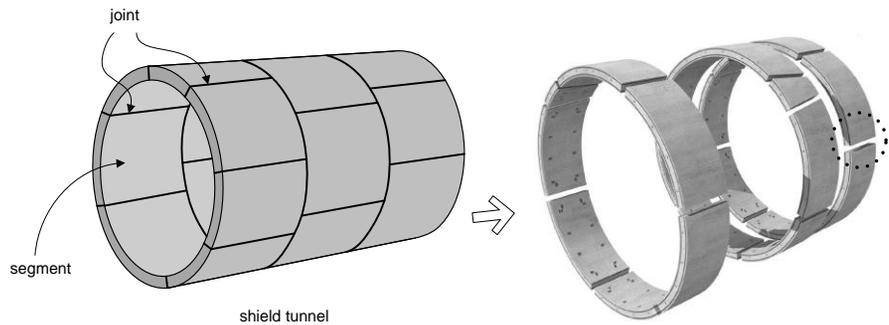
Table 3.4.3 Nonlinear analysis result articles of Interface elements

Result article	Explanation
Strain	Position : element node/center point $\Delta u_x^p, \Delta u_y^p, \Delta u_z^p$

8.2 Shell interface Element

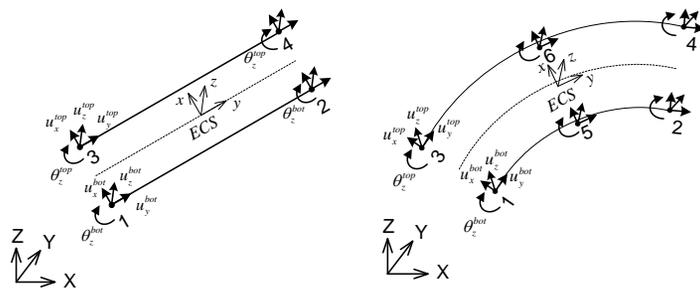
Shell interface elements are used to simulate the behavior between shell elements. Shell interface elements can be used to express the nonlinearity at joints between lining segments in a tunnel, as shown below. Shell interface elements are especially needed for TBM tunnels which are constructed through segmented lining because while the other segment elements display rigid behavior, the rotational and sliding behavior is dominant at the joints. The basic concept is similar to that of line Interface elements. An axial rotation DOF is added.

Figure 3.8.4 Segmented lining



On FEA NX, the 4 node and 6 node shell interface elements can be selected. both elements can only be used in 3D analysis. The ECS and node order are shown in figure 3.8.5.

Figure 3.8.5 Coordinate system and DOF of Shell interface elements



Coordinate system

The ECS y axis of a shell interface element is the direction from node 1 to node 2, and the z axis is determined by the average normal direction of adjacent shell elements. The x axis is the tangential direction of the contact plane.

DOF

Shell interface elements have a y axis direction rotational DOF and 3 displacement DOF in the ECS axis directions.

The relative displacement, interface traction and ECS of a shell interface element are shown in equation 3.8.4 and figure 3.8.6. Finite element formulation is done with reference to the ECS.

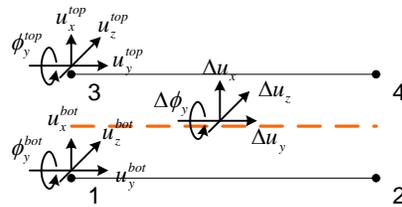
$$\mathbf{t} = \begin{Bmatrix} n_x \\ t_y \\ t_z \\ m_y \end{Bmatrix}, \quad \Delta \mathbf{u} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \\ \Delta u_z \\ \Delta \phi_y \end{Bmatrix} = \begin{Bmatrix} u_x^{top} - u_x^{bot} \\ u_y^{top} - u_y^{bot} \\ u_z^{top} - u_z^{bot} \\ \phi_y^{top} - \phi_y^{bot} \end{Bmatrix} \quad (3.8.4)$$

$t_x / \Delta u_x$: Normal interface traction / Normal relative displacement

$t_y, t_z / \Delta u_y, \Delta u_z$: Tangential interface traction / Tangential relative displacement

$m_y, \Delta \phi_y$: Axial moment / Rotation angle

Figure 3.8.6 Relative displacement and interface traction of Shell interface elements



Here, the displacement of the top/bottom faces is interpolated using the shape function. The constitutive equation for finding the interface traction of a shell interface element is introduced in Chapter 4: Material Models.

Like interface models, the Newton-Cotes integration method is used and the number of integral points according to the element shape is as follows:

Table 3.8.4 Number of integral points of Shell interface elements

Shell interface element	Number of integral points	
Line shell interface	4 Nodes	3
	6 Nodes	4



Element result

The shell interface element results are output with reference to a user specified coordinate system. The selectable coordinate systems are ECS and GCS.

Table 3.8.5 Result articles of Shell interface elements

Shell interface element	Result article
Force	Position : element node/center point t_x, t_y, t_z, m_y
Strain	Position : element node/center point $\Delta u_x, \Delta u_y, \Delta u_z, \Delta \phi_y$

Nonlinear analysis

Geometric nonlinearity can be considered for Shell interface elements, and Coulomb friction and Janssen materials can be applied. The nonlinear analysis result articles are as follows:

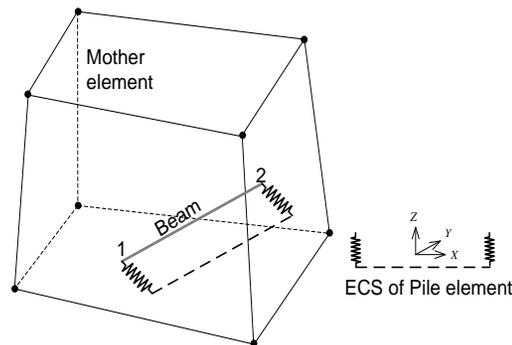
Table 3.8.6 Nonlinear analysis result articles of Shell interface elements

Shell interface element	Result article
Strain	Position : element node/center point $\Delta u_x^p, \Delta u_y^p, \Delta u_z^p, \Delta \phi_y^p$

8.3 Pile/Pile Tip Elements

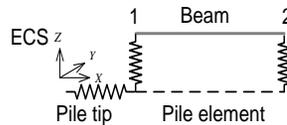
Pile elements are the linear Interface elements between the buried beam elements and the ground, as shown in figure 3.8.7. To simulate piles in the ground without pile elements, a beam element needs to be modeled by node sharing with the ground element, and a line Interface element is used to separate the beam element nodes and ground element nodes. FEA NX overcomes this tedious modeling process by inserting a pile element between the virtual beam element nodes and ground element. This is done when a beam element is buried within the ground element in order to simulate the behavior between ground and beam elements. However, in FEA NX, pile elements can be applied to solid and plane strain elements as shown in figure 3.8.7, and only 4 node pile elements can be used.

Figure 3.8.7 Pile element and ECS



Pile tip elements can be generated in FEA NX at the end of pile elements that are buried in a solid element, as shown in figure 3.8.7. This pile tip element acts as a solid-point interface. This is the same as adding spring stiffness in the element axial direction between the pile element and the pile tip node.

Figure 3.8.8 ECS and block diagram of a Pile tip element



Coordinate system

The ECS x axis direction of a pile element is the direction from node 1 to node 2, and the ECS x direction of a pile tip element is the same as the x axis direction of the pile element. Finite element formulation for both elements is done with reference to the ECS.

DOF, relative displacement and force

Pile elements have 3 DOF, like interface elements.

$$\mathbf{t} = \begin{Bmatrix} t_x \\ t_y \\ t_z \end{Bmatrix}, \Delta \mathbf{u} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \\ \Delta u_z \end{Bmatrix} = \begin{Bmatrix} u_x^{top} - u_x^{bot} \\ u_y^{top} - u_y^{bot} \\ u_z^{top} - u_z^{bot} \end{Bmatrix} \quad (3.8.5)$$

On the other hand, pile tip elements have 1 DOF in the axial direction.

$$\mathbf{t} = \{t_x\}, \Delta \mathbf{u} = \{\Delta u_x\} = \{u_x^{top} - u_x^{bot}\} \quad (3.8.6)$$

Element result

The pile element results are output with reference to the ECS and GCS. The pile tip element results are output with reference to the ECS.

Table 3.8.7 Result articles of Pile elements

Pile element	Result article
Force	Position : element node/center point t_x, t_y, t_z
Strain	Position : element node/center point $\Delta u_x, \Delta u_y, \Delta u_z$



Table 3.8.8 Result articles of Pile tip elements

Pile tip element	Result article
Force	Position : element node f_x
Strain	Position : element node Δu_x

Nonlinear analysis

A Pile element can simulate a perfectly plastic model in the axial direction (ECS x axis) and only linear behavior is possible in the normal (ECS y,z axis) direction. Pile tip elements can be a perfectly plastic model in the axial direction (ECS x axis). Both elements do not have an additional result article.

8.4 Geogrid Element

Geogrid elements are thin elements that do not have flexural rigidity, but have axial stiffness. Also, geogrid elements are textile products used for reinforcement and segregation in construction work, and only have stiffness during tension.

Element

Geogrid elements can be classified into geogrid 1D elements with 2 nodes or triangular/rectangular geogrid 2D elements with 3/4/6/8 nodes. The behavior of geogrid 1D elements is the same as that of truss elements. The behavior of geogrid 2D elements is the same as that of plane stress elements.

Coordinate system/DOF

The ECS and DOF of geogrid 1D/2D elements are defined in the exact same way as that of truss and plane stress elements.

Load

The loads applied on truss and plane stress elements can be equally applied on geogrid 1D and 2D elements respectively. However, generally only the very small self weight, such as the simulation of target element geogrids, exists. Additional load is not applied during construction to prevent geogrid damage.

Element result

The element results of geogrid 1D/2D elements are the same as that of truss and plane stress elements respectively.

Nonlinear analysis

Material nonlinearity where the stiffness only exists for tension needs to be considered for geogrid elements. Even when a linear elastic material is selected, the Inverse Rankine material is used internally and material nonlinear analysis is necessary. Hence, a nonlinear result (equivalent stress, plastic status, equivalent strain, effective plastic strain) is output.

Element texture

The texture of geogrid elements do not need additional specification; selecting an elastic texture analyzes it as an Inverse Rankine material with an allowable compressive strength of 0 (zero). Here, if it is a 1D element, it is the same as tension only behavior.

Figure 3.8.9 Tension only behavior of Geogrid 1D elements

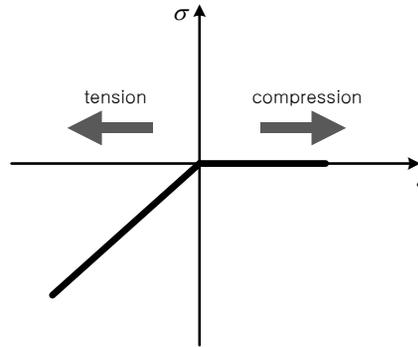
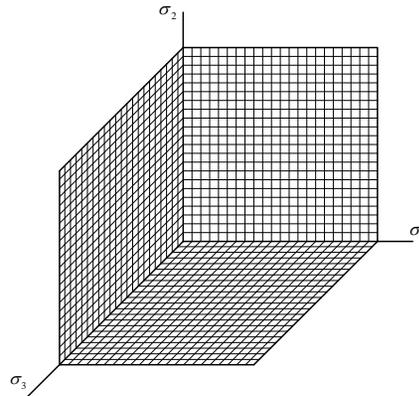


Figure 3.8.10 Inverse Rankine of Geogrid 2D elements



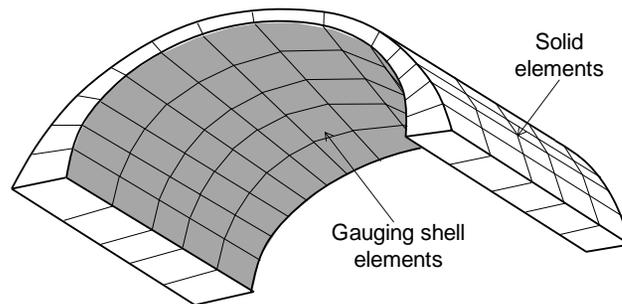
Element thickness and area

Geogrid 1D elements use the same method as truss elements to input the area. Geogrid 2D elements use same method as plane stress elements to input the thickness.

8.5 Gauging Shell Element

Gauging shell elements are triangular or rectangular elements consisting of 3/4/6/8 nodes on a plane. Gauging shell elements are special elements that are used to measure the force or moment at the neutral axis of a thin plate modeled as a solid element, as shown in figure 3.8.11. This is similar to a shell element, with the same thickness as the target solid element, positioned at the neutral axis of the target solid element. However, unlike general shell elements, it does not have a mass and has a very small stiffness. Hence, gauging shell elements only provide the element results, such as force and moment etc., and have almost no effect on the behavior of the entire analysis model.

Figure 3.8.11 Solid elements and Gauging shell elements



Coordinate system/DOF

The ECS and DOF of gauging shell elements are defined in the exact same way as that of shell elements.

Load

Loading can be applied to gauging shell elements but because they are elements that measure the results of the target element, the load should not be directly applied such that it does not affect the entire structural model. However, thermal expansion is considered to correctly calculate the stress and strain. Also, there are no mass effects such as self weight because the mass density is not used.

Element result

The gauging shell element results are output with reference to a user specified coordinate system. The selectable coordinate systems are ECS, MCS and arbitrary coordinate system.

Table 3.8.9 Result articles of Gauging shell elements

	Result article	Explanation
Force/ Moment	In-plane force	Position : vertex/element center N_{xx} , N_{yy} , N_{xy}
	Bending moment	Position : vertex/element center M_{xx} , M_{yy} , M_{xy}
	Shear force	Position : vertex/element center Q_{xz} , Q_{yz}

Nonlinear analysis

Geometric nonlinearity can be considered for gauging shell elements, but only linear elastic materials can be applied. Hence, there are no additional result articles for nonlinear analysis.

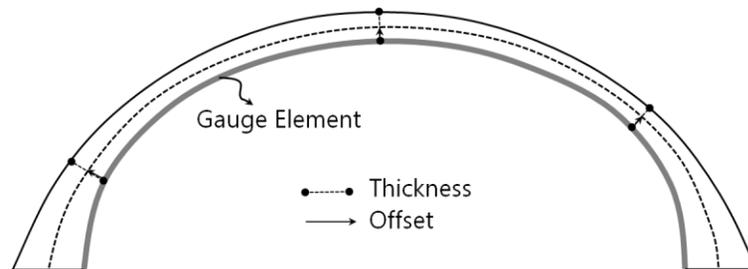
Element texture

The same texture as the target solid element is applied, but the mass density is not used. For modulus of elasticity, the input stiffness control coefficient is multiplied so as not to affect the total deformation result for accurate measurement. The modulus of elasticity is only used for printing results.

Element thickness and offset

Because the gauging shell element is generated on the surface of the target solid element, an offset of half the thickness around the target element is automatically considered for result measurement at the neutral axis. Like the shell element offset, a constant value is used for each element in the director direction, but the thickness is considered differently for each node to fit the target solid element.

Figure 3.8.12 Thickness and offset of Gauging shell elements

**Precaution – rotational DOF, thick target element**

Because the target solid element does not have a rotational DOF, if a separate element with a rotational DOF is not adjoined, the rotation amount of the gauging shell element is the result of rigid rotation about the displacement DOF. Hence for result measurement, the rotational DOF needs to be fixed when the end rotation is not fixed. If this is not possible, the gauging shell element needs to be attached slightly longer than the actual end measurement position such that the amount of rotation is appropriately measured.

Because gauging shell elements use the out-plane direction offset of shell elements to output results, the error becomes larger because the amount of rotation at the top and bottom parts of an element differ for thick elements. In this case, adding gauging shell elements to the top and bottom and using the average value is recommended.

8.6

Point Spring/Damper Element

Point spring/damper elements are used to prevent singular errors that can occur when elements with insufficient DOF (truss, plane stress elements etc.) are connected together, or when considering the elastic stiffness of adjacent structures or ground boundary conditions etc. at the model boundary. It can also be used to model the cohesive boundary conditions of the ground using the damping coefficient input. Nodal damping is not applied to general static analysis and it is only applied in dynamic analysis due to its properties.

Coordinate system

Point spring/damper elements do not specify a separate coordinate system and references the GCS.

DOF

Point spring/damper elements have a displacement and rotational DOF in all axial directions.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T, \quad \boldsymbol{\theta}_i = \{\theta_{xi} \quad \theta_{yi} \quad \theta_{zi}\}^T \quad (3.8.7)$$

Element force

Point spring/damper elements can consider force and moment in all axial directions.

$$\mathbf{N} = \begin{Bmatrix} N_{xx} \\ N_{yy} \\ N_{zz} \end{Bmatrix}, \quad \mathbf{M} = \begin{Bmatrix} M_{xx} \\ M_{yy} \\ M_{zz} \end{Bmatrix} \quad (3.8.8)$$

(Axial direction force and moment)

Load

Because point spring/damper elements only have a simple stiffness and no material properties, no loads except the concentrated load and moment can be applied.

Element result

On FEA NX, the Point spring/damper element result articles are output as a value about the GCS.

Table 3.8.10 Result articles of Point spring/damper elements

	Result article	Explanation
Force/ Moment	Force	Position : element center N_{xx} , N_{yy} , N_{zz}
	Bending moment	Position : element center M_{xx} , M_{yy} , M_{zz}

8.7

Matrix Spring Element

Matrix spring elements are used to supplement the point spring element, which only inputs the stiffness in one DOF direction without any behavioral correlations with other DOF. When modeling the elastic boundary condition in detail, not only the stiffness of each DOF but also the coupled stiffness between DOF needs to be considered. Hence, coupled stiffness needs to be used when considering rotational displacement that occurs simultaneously with displacement. Matrix spring elements are suitable for modeling these elastic boundary conditions. For example, when matrix spring elements are used to model piles used in the foundation of a structure, the coupled stiffness can be input as well as the stiffness of each direction for more detailed analysis.

Coordinate system/DOF

Matrix spring elements reference the GCS and have a displacement and rotational DOF in each axial direction, like point spring elements.

Element force

Force and moment in each axial direction can be considered, just like point spring elements.

Load

No loads other than the concentrated load and moment can be applied, just like point spring elements.

Element result

Matrix spring elements output the same result articles in the GCS as point spring elements.

Precaution - positive definite, symmetry

The user can input an arbitrary value for the stiffness of a matrix spring element, but the entire matrix needs to be positive. If not, an appropriate solution cannot be found. Also, FEA NX only allows the input of the upper triangular matrix such that the stiffness matrix of the matrix spring element is symmetric.

8.8

Elastic Link Element

Elastic link elements connect two nodes by the user input stiffness and do not have any other structural property, just like point spring elements or matrix spring elements. Generally, elastic link elements are composed of displacement and rotational stiffness in each axial direction. Other than this, the tension-only and compression-only properties can be assigned and in this case, only the stiffness in the ECS x direction can be input.

Elastic link elements can be appropriately used for elastomeric bearings (elastic links) that connect the top and bottom piers of bridge structures and for ground boundary conditions that have compressive-only properties.

Coordinate system/DOF

The coordinate system and DOF of elastic link elements are defined in the same way as beam elements.

Element force

Force and moment in each axial direction can be considered, just like point spring elements.

Load

No loads other than the concentrated load and moment can be applied, just like point spring elements.

Element result

The result articles of elastic link elements are the same as that of point spring elements, but the reference coordinate system is always the ECS.

8.9

Rigid

Link/Interpolation

Element

Rigid link and interpolation elements mutually constrain the relative motion between nodes. Here, the principal constraint node is the independent node and the principal constraint DOF is the independent DOF.

The constrained node is the dependent node and the constrained DOF is the dependent DOF.

Rigid link elements constrain the geometrically relative behavior of different nodes due to one node. Hence, it has a form of one independent node with multiple connected dependent nodes. The coupled relationship equation between the independent and dependent nodes is as follows:

$$\begin{aligned}\mathbf{u}^D &= \mathbf{u}^I + \mathbf{r} \times \boldsymbol{\theta}^I = \mathbf{u}^I + (\Delta \mathbf{x}) \times \boldsymbol{\theta}^I \\ \boldsymbol{\theta}^D &= \boldsymbol{\theta}^I\end{aligned}\quad (3.8.9)$$

$\mathbf{u}^D, \boldsymbol{\theta}^D$: Displacement and rotation of dependent nodes

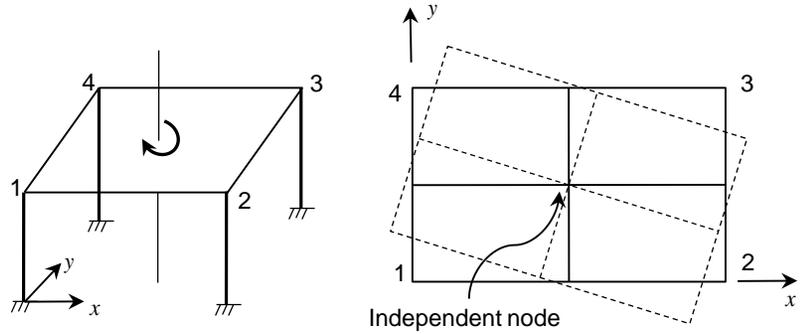
$\mathbf{u}^I, \boldsymbol{\theta}^I$: Displacement and rotation of independent nodes

$\Delta \mathbf{x}$: Vector from a dependent node to an independent node ($\mathbf{x}^I - \mathbf{x}^D$)

The DOFs that are constrained by the independent nodes can be selected out of the 6 DOF of dependent nodes, and this can be used to generate directional selective rigid link elements. The following example shows the constrained rigid behavior in the x-y plane:

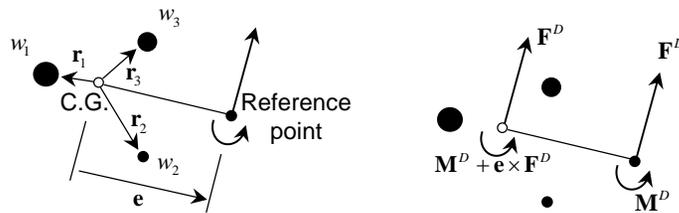
$$u^D = u^I - \theta_z^I \Delta y, \quad v^D = v^I + \theta_z^I \Delta x, \quad \theta_z^D = \theta_z^I \quad (3.8.10)$$

Figure 3.8.13 Example of rigid behavior in a plane



Interpolation elements display the relative behavior of one node depending on the motion of the other nodes. Hence, it has a form of one dependent node with multiple connected independent nodes. Interpolation nodes are used when distributing the force or mass to multiple nodes. The binding force is smaller than the rigid link element because the number of constrained nodes is smaller. The following example displays the force distribution process to show the displacement relationship between the independent and dependent nodes on a 2D x-y plane:

Figure 3.8.14 Relationship between center of mass of an independent node and force acting on a dependent node



When the dependent nodes are located at a distance e from the center of mass of independent nodes that are distributed with a weighed value w_i , then the force F^D and moment M^D acting on the dependent node acts with reference to the center of mass as a moment of $M^D + e \times F^D$, as shown in figure 3.8.14. The force and moment acting on the center of mass due to the dependent node can be distributed as a weighted average force for each independent node, as shown below:

$$\mathbf{F}_i \stackrel{\text{def}}{=} \hat{w}_i \left(\mathbf{F}^D + \mathbf{T}^{-1} \left(\mathbf{M}^D + \mathbf{e} \times \mathbf{F}^D \right) \times \mathbf{r}_i \right) \quad (3.8.11)$$

Here, \hat{w}_i is the weight normalized by the sum of weights, and \mathbf{T} is the average inertia tensor at the center of mass of dependent nodes:

$$\hat{w}_i = \frac{w_i}{\sum w_i} \quad (3.8.12)$$

$$\mathbf{T} = \sum_i \hat{w}_i \left[(\mathbf{r}_i^T \mathbf{r}_i) \mathbf{I} - \mathbf{r}_i \mathbf{r}_i^T \right] \quad (3.8.13)$$

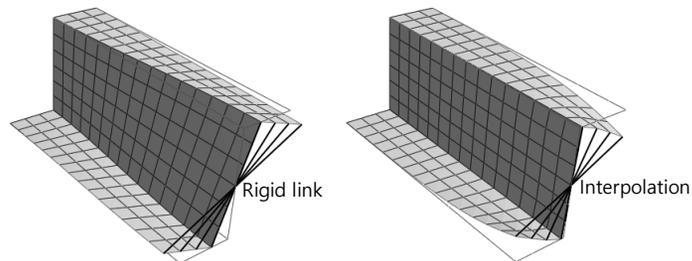
This force relationship can be converted to the following displacement and rotation relationship:

$$\mathbf{u}^D = \sum_i \hat{w}_i \mathbf{u}^I + \left(\mathbf{T}^{-1} \sum_i \hat{w}_i (\mathbf{r}_i \times \mathbf{u}^I) \right) \times \mathbf{e} \quad (3.8.14)$$

$$\theta^D = \mathbf{T}^{-1} \sum_i \hat{w}_i (\mathbf{r}_i \times \mathbf{u}^I) \quad (3.8.15)$$

Eventually, the average behavior of independent nodes determines the motion of dependent nodes and because of this property, a smaller number of DOF constraints occur.

Figure 3.8.15 Behavioral comparison between rigid link/Interpolation elements



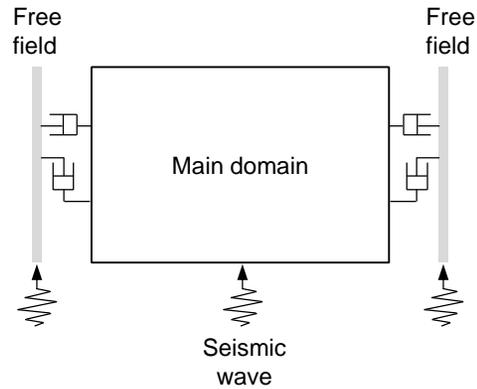
8.10 Free Field Element

In case of seismic analysis, one would generally model some part of ground surrounding main domain since it is impossible to model infinite ground as finite element model. In order to get similar results with infinite ground, one should create a kind of load which is traction from free field analysis and use absorbent boundary condition to remove the reflected wave generated in the model boundary. These two features are implemented with free field element in FEA NX²².

A traction generated from free field analysis is internal force and delivered to main domain and the reflected wave is removed using absorbent boundary condition. An example including free field element is shown in the below:

Figure 3.8.16 Model for seismic analysis including free field element

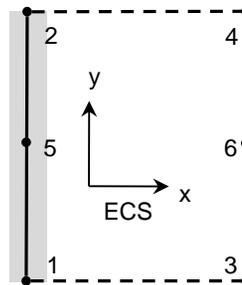
²² Nielsen, A. H., "Towards a Complete Framework for Seismic Analysis in Abaqus", Engineering and Computational Mechanics, Vol 167, EM1, 2013, pp. 3-12



Coordinate system

The element coordinate system and order of node in 2D column element for 2D analysis are as below. Here, shaded area is free field domain and non-shaded area is connection part to main domain. The dotted lines are damper connected between free field and main domain.

Figure 3.8.17 2D column free field element



The element coordinate system and order of node in 3D column element and plane element for 3D analysis are as below.

Figure 3.8.18 3D column free field element

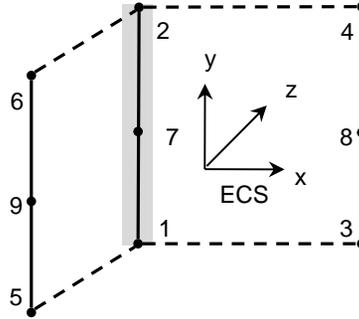
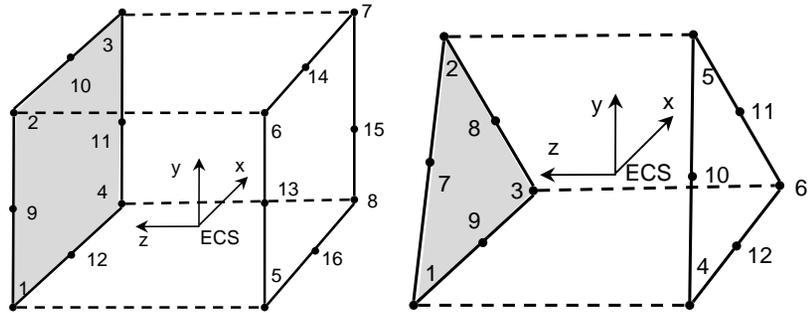


Figure 3.8.19 6/8/12/16 nodes free field element



DOF

Free field elements have displacement DOF for all directions in the x, y, z axes of the ECS.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T \quad (3.8.16)$$

Stiffness matrix

The stiffness matrix of free field elements consists of a part for applying internal force from stiffness matrix of free field domain (\mathbf{K}_{ff}) and traction by free field behavior (\mathbf{K}_{mf}) to main domain.

$$\begin{bmatrix} \mathbf{K}_{ff} & 0 \\ \mathbf{K}_{mf} & 0 \end{bmatrix} \quad (3.8.17)$$

The stiffness matrix is a non-symmetric matrix since free field domain is not influenced from the model of main domain. In static analysis, it solves equations with assumption that the stiffness matrix is a symmetric matrix. In this case, it makes the stiffness matrix to a symmetric matrix to symmetry \mathbf{K}_{mf} matrix.

**Damping matrix**

Free field elements can consider Rayleigh damping of material. The matrix by Rayleigh damping and absorbent boundary condition is included in damping matrix of free field elements.

Nonlinear analysis

It can use nonlinear material which can be used for plain strain and solid element, but it isn't able to consider geometry nonlinearity.

Element thickness and width

Element thickness of 2D model is same with element thickness of plain strain model. Element width is free field width. In case of using non-symmetric stiffness matrix, it doesn't affect to the solution. In case of using symmetric stiffness matrix, element width should be increased to minimize that free field analysis is affected by the model of main domain such as using penalty factor. In this case, element width should be larger than minimum 10^4 times of model width of main domain.

Absorbent boundary condition

In order to absorb the shock wave from model boundary, the method proposed by Lysmer and Kuhlemeyer²³ is used. This method gives a damping to the boundary with material density and wave velocity. A traction added to elements by absorbent boundary condition is as follows.

$$\begin{aligned} t_n &= -\rho C_p (v_n^m - v_n^{ff}) \\ t_s &= -\rho C_s (v_s^m - v_s^{ff}) \end{aligned} \quad (3.8.18)$$

C_p, C_s : Wave velocity of p, s direction

v^m : Velocity of model boundary

v^{ff} : Velocity of free field domain

8.11

Inelastic Element

When crack or yield occurs due to irregular cyclic load such as seismic load, very complex behavior appears since displacement history to the current affects to the restoring force and displacement relationship. It is called that hysteresis model which regulates this relationship and is considered to inelastic hinge at inelastic element.

Inelastic beam element is a beam element, which is assigned inelastic hinge properties. The inelastic beam element is limited to having a prismatic section whose hinge properties are identical for the single beam element. The stiffness of the inelastic beam element is formalized by the flexibility method. The shape function on which the existing stiffness method is based may differ from the true deformed shape in inelastic analysis. Whereas, the element section force distribution on which the flexibility method is based coincides

²³ Lysmer, J. and Kuhlemeyer, R. L. "Finite Dynamic model for infinite media", Journal of the Engineering Mechanics Division, Proc. ASCE, Vol. 95, No. EM4, 1969, pp. 859-876



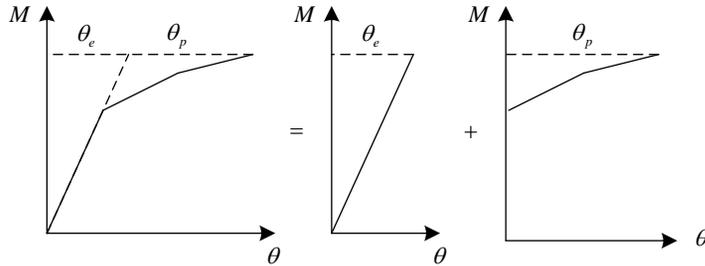
with the true distribution, which results in much higher accuracy. It has been known that the use of the flexibility method allows us to accurately model with a much less number of elements and as a result, the analysis speed can be much faster.

Inelastic beam elements are classified into lumped type and distributed type.

Lumped type inelastic element

The formulation is represented by inserting inelastic translational and rotational springs of non-dimensional lengths, which can deform plastically, into the beam element. The remaining parts other than the lumped type inelastic hinges are modeled as an elastic beam. The locations for inserting the inelastic springs for axial and flexural deformation components are assigned to the middle and both ends of the beam element respectively. The lumped type hinge is defined by a force-displacement relationship for the axial component and a moment-rotational angle relationship at the ends for the flexural components. The stiffness matrix of the beam element, which has been assigned lumped type hinges, is calculated by the inverse matrix of the flexibility matrix. The flexibility matrix of the total element is formulated by adding the flexibility matrices of the inelastic springs and the elastic beam.

Figure 3.8.20 Flexibility of lumped type inelastic element



The relationship of moment-rotational angle of a flexural deformation hinge is influenced by the end moments as well as by the flexural moment distribution within the member. In order to determine the relationship, the distribution of flexural moment needs to be assumed. The initial stiffness based on assumed moment distribution is shown below.

Figure 3.8.21 Initial stiffness relative to flexural deformations

Deflection Shape	Moment Distribution	Initial Stiffness
		$\frac{6EI}{L}$
		$\frac{3EI}{L}$
		$\frac{2EI}{L}$

Distributed type inelastic element

The flexibility of a section at an integration point in the longitudinal direction is obtained by state determination by the hysteresis models of uni-axial or multi-axial hinges. The distributed type hinge is defined by a force-deformation relationship for the axial component and a moment-curvature rate relationship for the flexural components at the section. The flexibility matrix of a beam element, which has been assigned distributed type inelastic element, is defined by the following equations and calculated through the Gauss-Lobatto integration.

$$F = \int_0^L b^T(x) f(x) b(x) dx = K^{-1} \quad (3.8.19)$$

- F : Element flexibility matrix
 K : Element stiffness matrix
 L : Length of member
 x : Location of section
 $f(x)$: Flexibility matrix of the section at the location x
 $b(x)$: Matrix of the section force distribution function for the location x

The locations of the integration points are determined by the number of integration points. The distances between the integration points are closer as the points near both ends. A maximum of 8 integration points can be specified.

8.12 Infinite Element

Stiffness matrix and Jacobian matrix

Same function is applied for displacement shape function and geometrical shape function for general element. However infinite element uses individual functions for displacement and geometrical shape. Function defines infinite area of global coordinate system as finite area of natural coordinate system. It is then used for calculation of Jacobian matrix.

$$[K] = \int_{\Omega} [B]^T [C][B] J d\Omega \quad (3.8.20)$$

- K : Stiffness matrix
 B : Deformation matrix
 C : Constitutive equations matrix
 J : Jacobian matrix

Jacobian matrix is defined as relationship of mapping function unlike general finite element. In case of 2D, it is defined as follows:

$$[J] = \begin{bmatrix} \sum_{node} \frac{\partial M_i}{\partial \xi} x_i & \sum_{node} \frac{\partial M_i}{\partial \xi} y_i \\ \sum_{node} \frac{\partial M_i}{\partial \eta} x_i & \sum_{node} \frac{\partial M_i}{\partial \eta} y_i \end{bmatrix} \quad (3.8.21)$$

- M : Mapping function
 x, y : Nodal coordinates

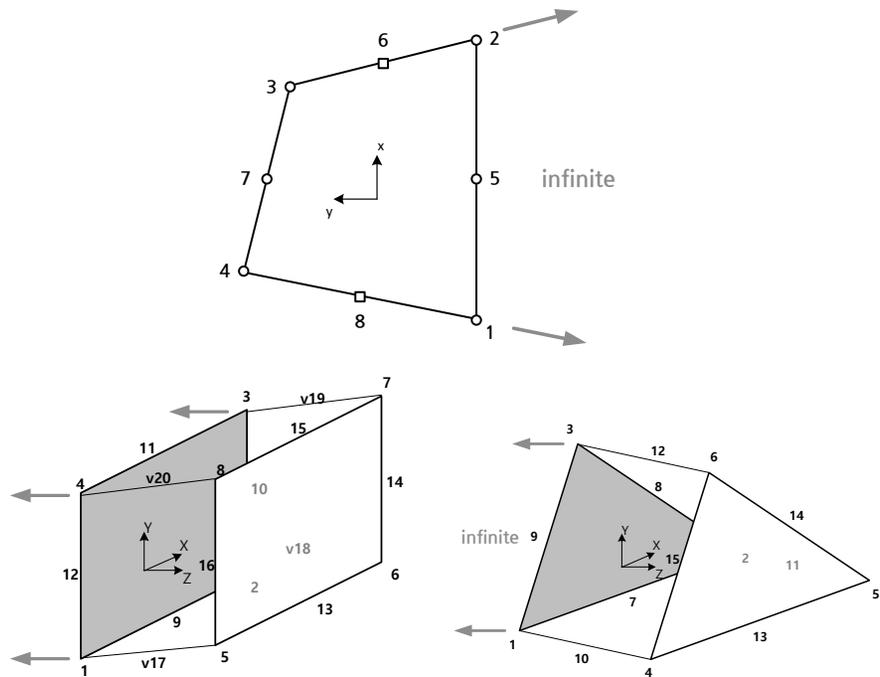
In case of axisymmetric stiffness matrix, additional radius (r) should be considered thus mapping function is defined as follows:

$$r = \sum_{node} M_i r_i \tag{3.8.22}$$

Coordinate System

The coordinate system can be expressed as below because selected area (area adjacent to finite element) and infinite area should be differentiated when defining infinite element. Polar coordinate is used to create imaginary mid-node because mapping function only uses high order element.

Figure 3.8.22 2D/3D Infinite element coordinate system



Degree of Freedom

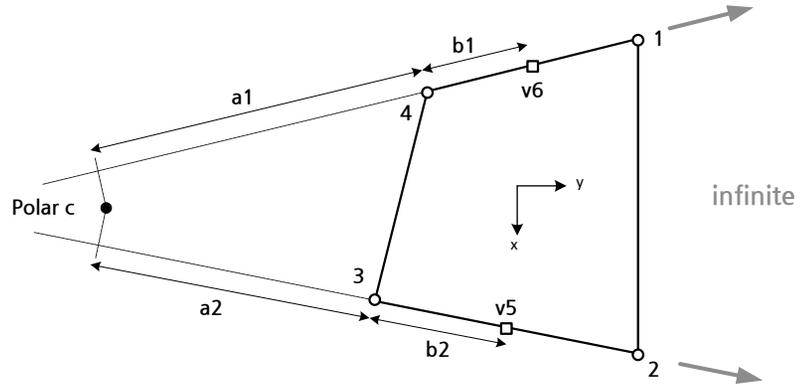
Free field element takes degree of freedom for displacement in all x, y and z direction of ECS.

$$\mathbf{u}_i = \{u_i \quad v_i \quad w_i\}^T \tag{3.8.23}$$

Polar Coordinate

Generally, polar coordinate of infinite element is set to be located at the center of the model. This is very important as results of infinite element change with the polar coordinate. Imaginary mid-node is created at the point $(a_1=b_1, a_2=b_2)$ and mapping function is considered.

Figure 3.8.23 Polar coordinate system in 2D infinite element



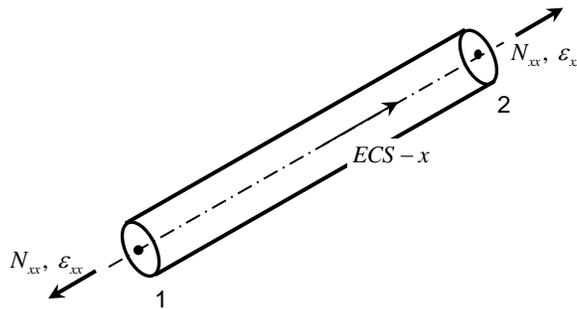
8.13

Tendon Element

As concrete structures are highly susceptible to tensile stress, they can be offset by applying a certain size of compressive stress to offset tensile stress. As such, concrete under compression stress is called prestressed concrete and tensile force is applied to high-strength steel through prestressing to act on the concrete.

Tendon elements are used to represent prestressed effects, and basic element actions do not require node-sharing, such as Embedded Truss, for convenient modeling and interpretation. It is also used as a buried form in mother elements and can be buried in plate elements and solid elements. For each of the nodes, such as the Embedded Truss element, the element that contains the node inside is determined as the mother element, and the element's nodal displacement is automatically constrained through multi-point constraints to match the internal displacement of the mother element. The difference from Embedded Truss is that for the Tendon element, it is assumed that stiffness is not introduced if the prestress is not applied.

Figure 3.8.23 Tendon Element Coordinate System and Strain



Coordinate System

In the ECS of the Tendon element, the direction of the x-axis is from node 1 to node 2. The formulation of finite elements is based on the ECS.

Degree of Freedom

The tendon element has a degree of freedom of the displacement in the x-axis direction of the ECS.

$$\mathbf{u}_i = \{u_i\} \quad (3.8.24)$$

Stress and Strain

The Tendon element represents the axial deformation defined in ECS as shown in Figure 3.8.23.

$$\mathbf{N} = \{N_{xx}\}, \quad \boldsymbol{\varepsilon} = \{\varepsilon_{xx}\} \quad (3.8.25)$$

(Axial force and strain)

Load

The loads applied to the tendon element are as follows.

Table 3.8.11 Loads applied to the tendon element

Load Types	Description
Self Weight	Applied based on material density
Element Temperature	Element temperature causing length deformation
Prestress	Initial axial force of element and corresponding equilibrium force

Element Result

When a tendon element is used, the element results always follows ECS.

Table 3.8.12 Tendon Element Result Item

	Result Item	Description
Stress	Axial stress	Position: Element Center σ_{xx}
Strain	Axial strain	Position: Element Center ε_{xx}
Force	Axial force	Position: Element Center N_{xx}

Section 9

Geometric Stiffness

Geometric stiffness or stress stiffness is the stiffness induced by the internal force change when geometric shape change occurs in a structure with internal forces. Geometric stiffness is used for nonlinear analysis, and the element types on FEA NX that consider geometric stiffness is as follows.

Table 3.9.1 Element types that consider geometric stiffness

Element type	Internal force component	DOF component
Truss	Axial force N_{xx}	Axial force N_{xx}
Beam	Axial force N_{xx}	Axial force N_{xx}
Plane stress, Plane strain	In-plane resultant force N_{xx}, N_{yy}, N_{xy}	In-plane resultant force N_{xx}, N_{yy}, N_{xy}
Shell	In-plane resultant force N_{xx}, N_{yy}, N_{xy} Bending moment M_{xx}, M_{yy}, M_{xy} Shear strength Q_{zx}, Q_{yz}	Axial force N_{xx}
Axisymmetric solid	In-plane stress $\sigma_{xx}, \sigma_{yy}, \tau_{xy}$ Circumferential stress $\sigma_{\theta\theta}$	u, w
Solid	Stress component $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \tau_{xy}, \tau_{yz}, \tau_{zx}$	u, v, w

Apart from this, the geometric stiffness is considered for geometric nonlinear analysis on a rigid link element.

9.1

Geometric Stiffness

Calculation for

General Elements

The calculation of geometric stiffness in FEA NX is based on the updated Lagrangian formulation that assumes the Jaumann stress rate as an objective stress rate. For example, the internal forces of a solid element are calculated from the stress and virtual deformation as follows.

$$\delta u_i f_i = \int \sigma_{ij} \delta D_{ij} dV \quad (3.9.1)$$

$$\delta D_{ij} : \text{virtual deformation} \quad \frac{1}{2} \left(\frac{\partial \delta u_i}{\partial x_j} + \frac{\partial \delta u_j}{\partial x_i} \right)$$

Because the tangent of the internal force corresponds to the stiffness, applying the variation method again leads to the following integrand.

$$d\sigma_{ij}\delta D_{ij} + \sigma_{ij}d\delta D_{ij} \quad (3.9.2)$$

The equation above ignores integrands within the integral area. The EFCS of solid elements is the GCS and is fixed, regardless of the structural deformation. Hence, $d\delta D_{ij} = 0$ and the stress increment due to the objective stress rate from the first term is as follows.

$$d\sigma_{ij} = dw_{ik}\sigma_{kj} + \sigma_{ij}dw_{jk} + C_{ijkl}dD_{kl} \quad (3.9.3)$$

$$\delta w_{ij} : \text{spin increment } \frac{1}{2}\left(\frac{\partial\delta u_i}{\partial x_j} - \frac{\partial\delta u_j}{\partial x_i}\right)$$

By substituting equation (3.9.2) and (3.9.3) into equation (3.9.1) and simplifying, the following tangential stiffness can be obtained.

$$\delta u_i K_{ij} du_j = \int \delta D_{ij} C_{ijkl} dD_{kl} + \sigma_{ij}(\delta L_{ki} dL_{kj} - 2\delta D_{ik} dD_{kj}) dV \quad (3.9.4)$$

$$\delta L_{ij} : \text{displacement gradient increment, } \delta D_{ij} + \delta w_{ij}$$

The first term of the integrand value is the material stiffness and the second term is the geometric stiffness.

9.2

Geometric Stiffness

Calculation for Rigid

Link Elements

The geometric stiffness of rigid link elements is generated by the force acting on the dependent node. Setting the force and moment acting on the dependent node as \mathbf{f}^s , \mathbf{m}^s , the virtual work due to these values is as follows.

$$\delta W = \mathbf{f}^s \cdot \delta \mathbf{u}^s + \mathbf{m}^s \cdot \delta \boldsymbol{\theta}^s \quad (3.9.5)$$

$$\delta \mathbf{u}^s, \delta \boldsymbol{\theta}^s : \text{displacement and rotation of the dependent node}$$

Applying the variation method again on the virtual work generates the following basic equation for calculating the stiffness.

$$d\delta W = \mathbf{f}^s \cdot d\delta \mathbf{u}^s + \mathbf{m}^s \cdot d\delta \boldsymbol{\theta}^s = \mathbf{f}^s \cdot d\delta \mathbf{u}^s \quad (3.9.6)$$

The following equation is used to convert the dependent node displacement to the displacement and rotation of the independent node.

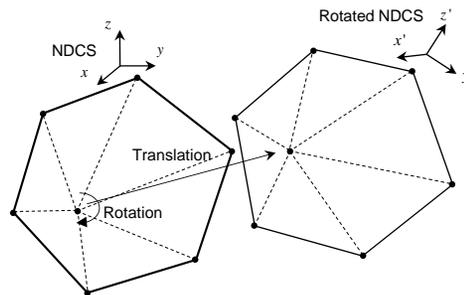
$$\delta \mathbf{u}^s = \delta \mathbf{u}^m + \delta \boldsymbol{\theta}^m \times (\mathbf{x}^s - \mathbf{x}^m) \quad (3.9.7)$$

Applying the variation method again to the equation above and substituting it into equation (3.9.6) can calculate the stiffness.

$$d\delta W = \mathbf{f}^s \cdot d\delta \mathbf{u}^s = \mathbf{f}^s \cdot (\delta \mathbf{\theta}^m \times (d\mathbf{\theta}^m \times (\mathbf{x}^s - \mathbf{x}^m))) \quad (3.9.8)$$

It can be shown that the geometric stiffness of rigid link elements are composed of the forces and relative distance acting on dependent nodes, and has a relationship with the rotational DOF of independent nodes. When the constraints are lifted for some DOFs of the dependent nodes, the now unconstrained DOF direction rotates in the motion of the rigid link element. Figure 3.9.1 shows the NDCCS of the motion of the dependent node due to the rotation of the independent node, and the removal of constraints also occurs in the changing coordinate system. Or when partial DOF are constrained using the rigid link element, the constraint direction is not constantly maintained and continuously changes.

Figure 3.9.1 Changing dependent node DOF direction due to independent node rotation



Section 10 Seepage Element

Various seepage elements exist in FEA NX to analyze the pore water seepage within saturated and unsaturated ground. This chapter focuses on the introduction of the governing equation based on the continuity equation, the finite element formulation process, and element results etc. of seepage elements. The constitutive equation and hydraulic model for unsaturated ground are explained in chapter 4.

10.1 Governing Equation

The mass concentration of water in the ground is $\rho_w n S$. When considering the continuity equation of mass for micro-volumes, the amount of water escaping from the micro-volume is the same as the change in mass concentration.

$$\nabla^T (\rho_w \mathbf{q}) = \frac{\partial}{\partial t} (\rho_w n S) \quad (3.10.1)$$

n : porosity

S : degree of saturation

The right term of the equation above can be expressed using the changes in water density, saturation and porosity with time.

$$\frac{\partial}{\partial t} (\rho_w n S) = n S \frac{\partial \rho_w}{\partial t} + \rho_w n \frac{\partial S}{\partial t} + \rho_w S \frac{\partial n}{\partial t} \quad (3.10.2)$$

Because the change in porosity with time is not considered in seepage analysis, this term can be ignored and Darcy's law can be applied. However, the porosity change is considered for the formulation process of consolidation elements introduced later.

FEA NX uses the pore pressure (p) as a variable in seepage analysis, and Darcy's law expressed as pore pressure can be applied to obtain the governing equation for seepage analysis.

$$\frac{1}{\gamma_w} \nabla^T (\mathbf{k} \nabla p) - \nabla^T (\mathbf{k} \mathbf{n}_g) = \left(\frac{n S}{\rho_w} \frac{\partial \rho_w}{\partial p} + n \frac{\partial S}{\partial p} \right) \frac{\partial p}{\partial t} \quad (3.10.3)$$

\mathbf{k} : coefficient of permeability matrix

\mathbf{n}_g : unit vector in gravitational direction

In steady state seepage analysis, the pore pressure does not change with time and the differential term for time in the equation above is '0'.

10.2

Finite Element Equation

Applying the variational method on the governing equation gives the following integral.

$$\int_{\Omega} \frac{1}{\gamma_w} \nabla^T (\mathbf{k} \nabla p) d\Omega - \int_{\Omega} \nabla^T (\mathbf{k} \mathbf{n}_g) d\Omega + \int_{\partial\Omega} q_{ext} dS = \int_{\Omega} \left(\frac{nS}{\rho_w} \frac{\partial \rho_w}{\partial p} + n \frac{\partial S}{\partial p} \right) \frac{\partial p}{\partial t} d\Omega \quad (3.10.4)$$

Here, q_{ext} is the velocity of flow at the model surface. Interpolating the pore pressure using a shape function in the form $p = N_i(\mathbf{x})P_i$ gives the following nonlinear simultaneous equation for time that includes the differential term for pore pressure.

$$\mathbf{C}(P_i)\dot{\mathbf{P}} + \mathbf{K}(P_i)\mathbf{P} = \mathbf{R}(q_{ext}, \mathbf{n}_g) \quad (3.10.5)$$

The integral term for position head in equation (3.10.4) is included as an internal force.

10.3

Time Integration

The time integration for the existing time term in equation (3.10.4) consists of the finite difference approach, and FEA NX applies the backward difference method. The backward difference method is an implicit time integration method that can analyze a long time period, and solution vibrations do not occur. The equation with applied backward difference is as follows.

$$\frac{\mathbf{C}(P_i^{n+1}, P_i^n)}{\Delta t} (\mathbf{P}^{n+1} - \mathbf{P}^n) + \mathbf{K}(P_i^{n+1})\mathbf{P}^{n+1} - \mathbf{R}(q_{ext}, z) = 0 \quad (3.10.6)$$

Here, the superscript $n, n+1$ represents the values at time $t^{n+1}, t^{n+1} = t^n + \Delta t$. To calculate the solution of the next step for time integration with applied backward difference, the solution of the previous step is needed. Hence, the initial conditions are needed for transient analysis, and FEA NX uses the steady-state analysis results at the initial time step load, or specifies the water level as the initial conditions for transient analysis.

10.4

Element Matrix

\mathbf{K} and \mathbf{C} from equation (3.10.5) are used to calculate the element tangent matrix and are expressed depending on the element type, as shown below.

1D element (Cross sectional area : A)

$$\mathbf{K}_{ij}^e = \int k \frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} AdL, \quad \mathbf{C}_{ij}^e = \int \frac{\partial \theta}{\partial p_w} N_i N_j AdL$$

2D element (Thickness : t)

$$\mathbf{K}_{ij}^e = \int k_{kl} \frac{\partial N_i}{\partial x_k} \frac{\partial N_j}{\partial x_l} t dA, \quad \mathbf{C}_{ij}^e = \int \frac{\partial \theta}{\partial p_w} N_i N_j t dA, \quad k, l = 1, 2$$

3D element

$$K_{ij}^e = \int k_{kl} \frac{\partial N_i}{\partial x_k} \frac{\partial N_j}{\partial x_l} dV, \quad C_{ij}^e = \int \frac{\partial \theta}{\partial p_w} N_i N_j dV, \quad k, l = 1, 2, 3$$

Relationship with structural element

Unlike structural analysis models, the seepage load and seepage boundary conditions need to be defined for seepage analysis models. However, because there is no difference in the modeling process between structural analysis and seepage analysis when the seepage load and boundary conditions are disregarded, the beam, shell, solid etc. elements used in structural analysis can be used to perform seepage analysis. The following table lists the relationship between structural elements and seepage elements.

Table 3.10.1 Relationship between seepage element and structural element

Seepage element type	Structural element
1D element	Truss, Beam
2D element	Plane strain, Plane stress, Shell, Axisymmetric solid
3D element	Solid

Apart from the the rigid link, the other elements in the table above can simulate the connected behavior with the pore pressure DOF. Elastic links act as a spring for the pore pressure difference.

10.5

Element Analysis

Results

Table 3.10.2 Result articles of Seepage elements

On FEA NX, the seepage element results are output with reference to a user specified coordinate system. The applicability or way of use for each element is the same as for structural elements.

Result article	Explanation
Hydraulic gradient component	Position : element center g_x, g_y, g_z
Hydraulic gradient resultant	Position : element center $\ \mathbf{g}\ = \sqrt{g_x^2 + g_y^2 + g_z^2}$
Seepage flow velocity component	Position : element center q_x, q_y, q_z
Seepage flow velocity resultant	Position : element center $\ \mathbf{q}\ = \sqrt{q_x^2 + q_y^2 + q_z^2}$
Principal permeability component	Position : element center k_a, k_b, k_c
Volumetric water content	Position : element center θ
Degree of saturation	Position : element center S

Section 11

Consolidation Element

Consolidation continuum elements are specifically used to simulate stress-seepage coupled phenomena in FEA NX. Consolidation elements can simultaneously use general structural/special elements or seepage elements for seepage analysis to perform consolidation analysis.

The starting points of basic finite element formulation for consolidation analysis are the stress equilibrium equation, for porous media with pore water, and the continuity equation of pore water. The discrete vector equation can be obtained by considering the coupled stress/seepage analysis, based on the finite element formulation process for stress and seepage analysis introduced above.

11.1

Pore Pressure DOF

Consolidation elements use both the displacement and pore pressure as nodal DOF simultaneously. FEA NX uses 2 types of consolidation elements, depending on the purpose.

- Total pore pressure based consolidation element, using the total pore pressure as a DOF

The simulation of transient state seepage phenomena-stress coupled analysis is suitable when the steady-state water pressure cannot be defined, due to the change with respect to the seepage boundary conditions and flow time. It is appropriate for fully-coupled consolidation analysis, where the seepage and structural load/boundary change simultaneously.

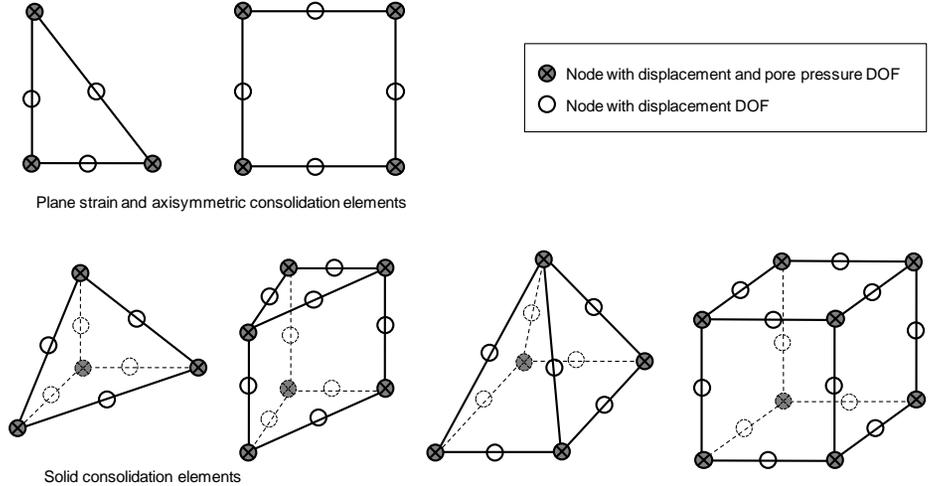
- Excessive pore pressure based consolidation element, using the excessive pore pressure as a DOF

The pore pressure can be classified into the steady state and transient state. The user defined initial water level or water pressure distribution is considered as the steady state pore pressure, and the excessive pore pressure during consolidation is considered as the transient state pore pressure. Simulation of the dissipation process for excessive pore pressure is appropriate for consolidation analysis in the general sense.

$$\begin{aligned} \mathbf{u}^e &= \mathbf{N}\mathbf{d}^e \\ p^e &= \mathbf{N}_p\mathbf{p}^e \quad \text{or} \quad p^e = p_{steady}^e + \mathbf{N}_p(\mathbf{p}_{unsteady}^e) \end{aligned} \quad (3.11.1)$$

Here, \mathbf{N} is the shape function for the displacement DOF of a consolidation element, \mathbf{N}_p is FEA NX uses low order shape functions for the displacement and pore pressure of low order consolidation elements. On the other hand, high order consolidation elements have a high order shape function for displacement and a low order shape function for pore pressure (Figure 3.11.1).

Figure 3.11.1 DOF of high order continuum consolidation element



The later part of the formulation process for consolidation elements explains the case where the total pore pressure is used as a DOF. The formulation process for consolidation elements where the excess pore pressure is used as a DOF is similar, and hence omitted from this manual.

11.2 Time Integration

The transient state is the fundamental state of consolidation analysis. The backward difference method is applied to the differentiation with time. In other words,

$$\begin{aligned}\frac{\partial \mathbf{d}^{e,n+1}}{\partial t} &= \frac{1}{\Delta t} (\mathbf{d}^{e,n+1} - \mathbf{d}^{e,n}) \\ \frac{\partial \mathbf{p}^{e,n+1}}{\partial t} &= \frac{1}{\Delta t} (\mathbf{p}^{e,n+1} - \mathbf{p}^{e,n})\end{aligned}\quad (3.11.2)$$

Here, the superscript n , $n+1$ represents the values at time t^{n+1} , $t^{n+1} = t^n + \Delta t$. The equilibrium equation of porous media, with applied backward difference for time, and the continuity equation of pore water can be expressed using the following vector equation.

$$\begin{aligned}\mathbf{g}_u^{n+1} &= \mathbf{r}_{ext}^{n+1} - \sum_e \mathbf{f}_I^{e,n+1} = \mathbf{0} \\ \mathbf{g}_p^{n+1} &= \mathbf{r}_{ext}^{n+1} - \sum_e \left[\frac{1}{\Delta t} \mathbf{C}^{e,n+1} (\mathbf{p}^{e,n+1} - \mathbf{p}^{e,n}) + \mathbf{K}^e \mathbf{p}^{e,n+1} \right] = \mathbf{0}\end{aligned}\quad (3.11.3)$$

\mathbf{g}_u^{n+1} and \mathbf{g}_p^{n+1} each represent the unbalanced force and unbalanced discharge, and are introduced for the Newton-Raphson based nonlinear analysis. Also, the internal force of consolidation element e includes the effects of effective stress and pore pressure, and can be expressed as follows.

$$\mathbf{f}_I^e = \int_{\Omega_e} \mathbf{B}^T (\boldsymbol{\sigma}' - S_e \mathbf{m} \mathbf{N}_p \mathbf{P}^e) d\Omega \quad (3.11.4)$$

The solution at time t^n can be used to calculate the solution that satisfies the nonlinear vector equation from equation (3.11.3) at time $t^{n+1} = t^n + \Delta t$, and this series of procedures is used to calculate the solution changing with time.

11.3 Nonlinear Increment Solution

Consolidation analysis is fundamentally performed as a nonlinear analysis in FEA NX. In other words, material and geometric nonlinearity can be included, and changing material properties due to change in pore pressure can also be considered in the seepage equation. The nonlinear solution uses the Newton-Raphson method and detailed explanations are provided in chapter 6.

The increment DOF of displacement and pore pressure are defined for nonlinear analysis using the Newton-Raphson method, as shown below.

$$\begin{aligned} \mathbf{u}_{I+1}^{n+1} &= \mathbf{u}_I^{n+1} + \delta \mathbf{u} \\ \mathbf{p}_{I+1}^{n+1} &= \mathbf{p}_I^{n+1} + \delta \mathbf{p} \end{aligned} \quad (3.11.5)$$

The subscripts $I, I+1$ represent the I th or $I+1$ th solution of the recursive calculation. Assuming that $\mathbf{u}_{I+1}^{n+1}, \mathbf{p}_{I+1}^{n+1}$ are the converged solution, the unbalanced force of the $I+1$ th recursive calculation is 0, and the Taylor expansion that considers the 1st differential term of the unbalanced force for the converged solution can be expressed as follows.

$$\mathbf{0} = \mathbf{g}_u(\mathbf{u}_{I+1}^{n+1}, \mathbf{p}_{I+1}^{n+1}) \cong \mathbf{g}_u(\mathbf{u}_I^{n+1}, \mathbf{p}_I^{n+1}) + \left[\frac{\partial \mathbf{g}_u}{\partial \mathbf{u}} \right] \delta \mathbf{u} + \left[\frac{\partial \mathbf{g}_u}{\partial \mathbf{p}} \right] \delta \mathbf{p} \quad (3.11.6)$$

When considering the geometric and material nonlinearity due to the use of nonlinear materials such as large scale deformation or plastic materials, the differential matrix for displacement DOF of unbalanced forces are expressed as a sum of the material tangent matrix and the geometric tangent matrix, and the differential matrix for the pore pressure DOF of unbalanced forces are expressed as follows.

$$-\left[\frac{\partial \mathbf{g}_u}{\partial \mathbf{u}} \right] = \mathbf{K}_{mat} + \mathbf{K}_{geo} \quad (3.11.7)$$

$$-\left[\frac{\partial \mathbf{g}_u}{\partial \mathbf{p}} \right] = \mathbf{K}_C = \sum_e \int_{\Omega^e} S_e \mathbf{B}^T \mathbf{m} \mathbf{N}_p d\Omega \quad (3.11.8)$$

The equation for unbalanced discharge is also expressed in a similar form.

$$\mathbf{0} = \mathbf{g}_p(\mathbf{u}^{n+1}, \mathbf{p}^{n+1}) \cong \mathbf{g}_p(\mathbf{u}^n, \mathbf{p}^n) + \left[\frac{\partial \mathbf{g}_p}{\partial \mathbf{u}} \right] \delta \mathbf{u} + \left[\frac{\partial \mathbf{g}_p}{\partial \mathbf{p}} \right] \delta \mathbf{p} \quad (3.11.9)$$

The differential matrix for displacement DOF and pore pressure DOF of the unbalanced discharge is expressed as follows.

$$-\left[\frac{\partial \mathbf{g}_p}{\partial \mathbf{u}} \right] = -\frac{1}{\Delta t} \mathbf{K}_C^T = -\frac{1}{\Delta t} \sum_e \int_{\Omega^e} S_e \mathbf{N}_p \mathbf{m}^T \mathbf{B} d\Omega \quad (3.11.10)$$

$$-\left[\frac{\partial \mathbf{g}_p}{\partial \mathbf{p}} \right] = \mathbf{K}_p = \sum_e \left[\frac{1}{\Delta t} \mathbf{C}^{e,n+1} + \mathbf{K}^e \right] \quad (3.11.11)$$

Arranging equations (3.11.7), (3.11.8), (3.11.10), and (3.11.11) gives the following coupled simultaneous equation for increment displacement and increment pore pressure DOFs.

$$\begin{bmatrix} \mathbf{K}_{mat} + \mathbf{K}_{geo} & \mathbf{K}_C \\ \mathbf{K}_C^T & -\Delta t \mathbf{K}_p \end{bmatrix} \begin{Bmatrix} \delta \mathbf{u} \\ \delta \mathbf{p} \end{Bmatrix} = \begin{Bmatrix} \mathbf{g}_u \\ -\Delta t \mathbf{g}_p \end{Bmatrix} \quad (3.11.12)$$

The total displacement and pore pressure of a node can be calculated by substituting the calculated increment solution into equation (3.11.5). This process is repeated until the unbalanced component or the increment DOF value is within a certain tolerance.

11.4 Analysis Result of Element

On FEA NX, the consolidation element analysis results are output with reference to a user specified coordinate system. The applicability and way of use for each element is the same as for structural elements and seepage elements. The calculated result articles are composed of the structural element result articles for porous medium analysis and seepage element analysis articles.

Section 12 Heat Conduction Element

The heat transfer analysis of FEA NX is based on the thermal equilibrium equation for transient and steady-state. In this section, the spatial discretization and the time integration method by the finite element method are explained. In FEA NX, only linearly conductive heat transfer elements are supported. Also, since the method of shape function or numerical integration is not much different from the structural element, and the degree of freedom is expressed by one point (temperature) per node, the analysis can be performed with a relatively small calculation cost as compared with the structural analysis.

12.1 Finite Element Formulation

The transient state energy equilibrium equation considering the increase and decrease of internal heat energy due to heat flux, internal heat, and specific heat transmitted to the boundary is as follows:

$$\int_{\partial\Omega} q dS + \int_{\Omega} r d\Omega = \int_{\Omega} c \rho \dot{T} d\Omega \quad (3.12.1)$$

q : Heat flux

r : Quantity of heat generated per unit volume

c : Specific heat

ρ : Mass density

The relationship between heat flux and temperature founded on the Fourier's Law is expressed as follows:

$$f_i = -k_{ij}(T) \frac{\partial T}{\partial x_j} = -k_{ij}(T) g_j \quad (3.12.2)$$

$k_{ij}(T)$: Conductivity

g_j : Temperature gradient

If the Fourier's Law is substituted into the energy equilibrium equation and a variation is taken, the following equation is attained:

$$\int_{\Omega} c \rho \dot{T} \delta T d\Omega + \int_{\Omega} \frac{\partial \delta T}{\partial x_i} k_{ij}(T) \frac{\partial T}{\partial x_j} d\Omega = \int_{\partial\Omega_2} q_{ext} \delta T dS + \int_{\Omega} r \delta T d\Omega \quad (3.12.3)$$

q_{ext} : Heat flux flowing in from outside

If the temperature is interpolated as a shape function of the form of $T = N_i(\mathbf{x})T_i$, the above equation becomes the following nonlinear simultaneous equation consisted of nodal temperature, which includes a temperature differential term with respect to time.

$$\mathbf{C}(T_i)\dot{\mathbf{T}} + \mathbf{K}(T_i)\mathbf{T} = \mathbf{R}(q_{ext}, r) \quad (3.12.4)$$

In order to calculate the time history of temperature distribution based on the above equation, the backward difference method is applied. This method being a type of implicit time integration can perform analysis of long time intervals and does not generate any oscillation of solutions. The equation using the backward difference method is as follows:

$$\left[\frac{\mathbf{C}(T_i(t + \Delta t))}{\Delta t} + \mathbf{K}(T_i(t + \Delta t)) \right] \mathbf{T}(t + \Delta t) - \frac{\mathbf{C}(T_i(t + \Delta t))}{\Delta t} \mathbf{T}(t) - \mathbf{R}(q_{ext}, r) = 0 \quad (3.12.5)$$

The Newton-Raphson method is used to calculate the time history of temperature distribution by repeatedly applying the above nonlinear equation to calculate the temperature at the next time step.

The conductivity and capacitance matrices are calculated respectively as follows

► 1-dimensional element (cross-sectional area: A)

$$K_{ij}^e = \int k \frac{\partial N_i}{\partial x} \frac{\partial N_j}{\partial x} AdL, \quad C_{ij}^e = \int \rho c N_i N_j AdL$$

► 2-dimensional element (thickness: t)

$$K_{ij}^e = \int k_{kl} \frac{\partial N_i}{\partial x_k} \frac{\partial N_j}{\partial x_l} t dA, \quad C_{ij}^e = \int \rho c N_i N_j t dA, \quad k, l = 1, 2$$

► 3-dimensional element

$$K_{ij}^e = \int k_{kl} \frac{\partial N_i}{\partial x_k} \frac{\partial N_j}{\partial x_l} dV, \quad C_{ij}^e = \int \rho c N_i N_j dV, \quad k, l = 1, 2, 3$$

12.2

Relationship with Structural Elements

Unlike a structural analysis model, thermal loads and thermal boundary conditions must be defined in a heat transfer analysis model. Except for the thermal loads and boundary conditions, its modeling does not differ from that of a structural analysis. Heat transfer analysis thus can be performed using the same structural elements such as bar, shell and solid elements. The following table summarizes the relationship between the structural elements and the heat conduction elements.

Table 3.12.1 Relationship between heat conduction elements and structural elements

Heat Conduction Elements	Structural Elements
1-dimensional Element	Truss, Beam
2-dimensional Element	Plane stress, Shell, Plane strain, Axisymmetric
3-dimensional Element	Solid

In addition to the elements in the above table, the total behavior of the temperature degrees of freedom can be simulated with the rigid body or rigid bar elements.

12.3

Analysis Result of Heat Transfer Element

The analysis results of heat conduction elements in FEA NX are produced in a user-defined reference coordinate system. The use of coordinate systems and the methods of using the elements are identical to those of structural elements.

Table 3.12.2 Analysis result items of heat conduction elements

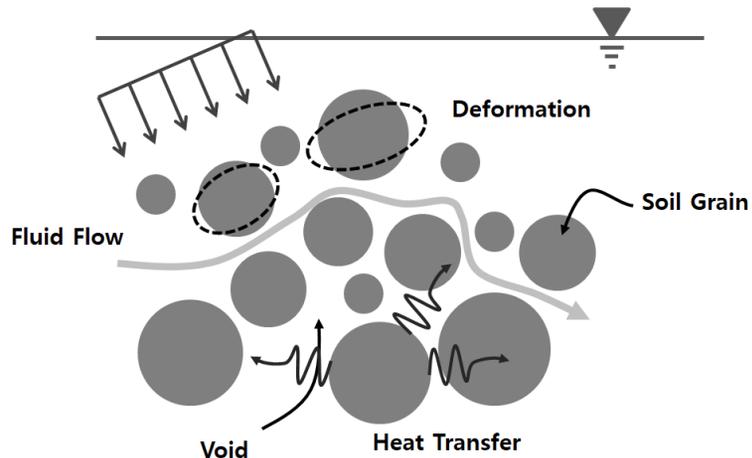
	Result Items	Descriptions
Element thermal results	Flux component	Location: Element center f_x, f_y, f_z
	Flux resultant	Location: Element center $\ \mathbf{f}\ = \sqrt{f_x^2 + f_y^2 + f_z^2}$
	Thermal gradient component	Location: Element center g_x, g_y, g_z
	Thermal gradient resultant	Location: Element center $\ \mathbf{g}\ = \sqrt{g_x^2 + g_y^2 + g_z^2}$

Section 13 Fully-Coupled Element(Thermal-Hydro-Mechanical)

In FEA NX, the dedicated fully-coupled element is used to simulate the linked phenomenon of thermal-hydro-mechanical effects. The fully-coupled element is used to perform the fully-coupled analysis by considering the degree of freedom of structural element, seepage element and heat transfer element at the same time and the relationship between the degree of freedom.

The correlation of seepage and heat has not been much researched since it is relatively insignificant compared to the correlation of structure and seepage. However, the interest of this research has been increased because heat has a great influence on seepage and deformation in case of high temperature heat source in the ground such as the nuclear waste storage.

Figure 3.13.1 Conceptual diagram of fully-coupled element



13.1

Governing Equation

In short, the fully-coupled element is an extension of the heat transfer element to the consolidation element in Section 11. In the fully-coupled element, deformation, pore water pressure and heat are used simultaneously for the degree of freedom of node. Therefore, the following governing equation considers the moment equilibrium equation, fluid mass conservation equation and energy conservation equation. Here, the limitation is the air pressure is assumed to be constant at the atmospheric pressure throughout the analysis, and the flow of air is not considered. In terms of the sign convention, compression is negative (-).

Also, regarding the degree of freedom, please refer to those of pore water pressure in the 'Section 11 Consolidation Element'. Here, in case of quadratic element, the degree of freedom of heat transfer uses the lower order shape function same as those of pore water pressure.

Moment Equilibrium Equation

The linear moment equilibrium equation for small volume of the porous medium is as follows:

$$\nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{g} = 0 \quad (3.13.1)$$

ρ : Density of the porous medium

\mathbf{g} : Gravity acceleration

And, the total stress of the porous medium which is defined based on the stress relationship is as follows:

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}' - \mathbf{m} [\chi p_w + (1 - \chi) p_a] \quad (3.13.2)$$

$\boldsymbol{\sigma}$: Total stress

$\boldsymbol{\sigma}'$: Effective stress

\mathbf{m} : Second rank unit tensor

p : Pore water pressure

p_a : Atmospheric pressure

χ : Effective stress parameter

For more details, please refer to the 'Section 4 Porous Medium Consideration'.

Here, if the air pressure is assumed to be constant at the atmospheric pressure, it can be expressed as follows:

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}' - \mathbf{m} \chi p_w \quad (3.13.3)$$

The constitutive equation of effective stress considering thermal strain is as follows:

$$d\boldsymbol{\sigma}' = \mathbf{C}_{ep} (d\boldsymbol{\varepsilon} - d\boldsymbol{\varepsilon}_T) \quad (3.13.4)$$

\mathbf{C}_{ep} : Constitutive equation of stress-strain

$\boldsymbol{\varepsilon}$: Total strain

$\boldsymbol{\varepsilon}_T$: Thermal strain

And, the thermal strain can be expressed as follows:

$$d\boldsymbol{\varepsilon}_T = \boldsymbol{\alpha}_T \cdot \mathbf{m} \cdot dT \quad (3.13.5)$$

$\boldsymbol{\alpha}_T$: Thermal expansion coefficient of porous medium

Therefore, substituting the equation(3.13.5) into the equation(3.13.4), it is as follows:

$$d\boldsymbol{\sigma}' = \mathbf{C}_{ep} (d\boldsymbol{\varepsilon} - \boldsymbol{\alpha}_T \cdot \mathbf{m} \cdot dT) \quad (3.13.6)$$

The governing equation of deformation can be expressed as follows:

$$\nabla \cdot [\mathbf{C}_{ep} (d\boldsymbol{\varepsilon} - \boldsymbol{\alpha}_T \cdot \mathbf{m} \cdot dT) - \mathbf{m} \chi p_w] + \rho \mathbf{g} = 0 \quad (3.13.7)$$

Fluid Mass Conservation Equation

The seepage element mentioned in Section 10 is as follows:

$$\frac{\partial}{\partial t} (\rho n S) = n S \frac{\partial \rho}{\partial t} + \rho n \frac{\partial S}{\partial t} + \rho S \frac{\partial n}{\partial t} \quad (3.13.8)$$

Although the change of porosity with time is not considered in seepage analysis, because there is no need to take into account the structural deformation, the equation can be expanded by considering the change of porosity as follows (Rutqvist et al., 2001):

$$n \frac{\partial}{\partial t} (S_\kappa \rho_\kappa) + (S_\kappa \rho_\kappa) \left(\frac{\partial \varepsilon_v}{\partial t} + \frac{(1-n)}{\rho_s} \frac{\partial \rho_s}{\partial t} \right) = -\nabla \cdot (\mathbf{q}_\kappa) \quad (3.13.9)$$

Here, κ is soil component, and it is composed of soil particle(s), groundwater(w) and vapor(v). Assuming that water and vapor are only existed in void of soil, it can be expressed as follows:

$$n \frac{\partial}{\partial t} (S_w \rho_w + S_v \rho_v) + (S_w \rho_w + S_v \rho_v) \left(\frac{\partial \varepsilon_v}{\partial t} + \frac{(1-n)}{\rho_s} \frac{\partial \rho_s}{\partial t} \right) = -\nabla \cdot (\mathbf{q}_w + \mathbf{q}_v) \quad (3.13.10)$$

ρ : Density

n : Porosity

S : Degree of saturation

ε_v : Volumetric strain

\mathbf{q} : Discharge of water and vapor

In the right side of the above equation, assuming that liquid water in the voids between the soil particle advectons by the Darcy's law, and gas state water, vapor does not advection, but only diffusions by the Fick's law, it can be expressed as follows:

$$\mathbf{q}_w = \rho_w \left(\frac{\mathbf{k}}{\gamma_w} (-\nabla \mathbf{p} - \rho_w \mathbf{g}) \right) \quad (3.13.11)$$

$$\mathbf{q}_v = -D_v \mathbf{m} \rho_v \quad (3.13.12)$$

\mathbf{k} : Coefficient of permeability

γ_w : Unit weight of water

D_v : Diffusion coefficient of vapor

Here, ρ_g is assumed to be constant.

And the density of vapor(ρ_v) can be defined as follows:

$$\rho_v = R_H \rho_{vs} = \left(\frac{P}{\rho_w R_v T} \right) \rho_{vs} \quad (3.13.13)$$

R_H : Relative humidity

ρ_{vs} : Saturated unit density of vapor

R_v : Specific gas constant of vapor (461.5J / kgK)

Using the equation(3.13.12) and (3.13.12), it is as follows:

$$\mathbf{q}_v = -\rho_w (D_{pv} \nabla p_w + D_{Tv} \nabla T) \quad (3.13.14)$$

Here, the coefficient of diffusion is as follows:

$$D_{pv} = \frac{D_v \rho_v}{\rho_w^2 R_v T} \quad (3.13.15)$$

$$D_{T_v} = f_{T_v} D_v \left(\frac{R_H}{\rho_w} \frac{\partial \rho_{vs}}{\partial T} - \frac{\rho_v \rho_w}{\rho_w^2 R T^2} \right) \quad (3.13.16)$$

f_{T_v} : Diffusion coefficient of temperature

Therefore, the above formulas are summarized as follows:

$$\begin{aligned} & [S \rho_w + (1-S) \rho_v] \frac{\partial \varepsilon_v}{\partial t} \\ & + \left[n(\rho_w - \rho_v) \frac{\partial S}{\partial p} + nS \rho_w \beta_{wp0} - n(1-S) \frac{\rho_v}{\rho_w R T} \right] \frac{\partial p_w}{\partial t} \\ & + \left[n(\rho_w - \rho_v) \frac{\partial S}{\partial T} - nS \rho_w \beta_{wT0} - n(1-S) \left(R_H \frac{\partial \rho_{vs}}{\partial T} - \frac{\rho_v \rho_w}{\rho_w^2 R T^2} \right) - (S \rho_w + (1-S) \rho_v) (1-n) \beta_{sT} \right] \frac{\partial T}{\partial t} \\ & + \nabla \cdot (\mathbf{q}_w + \mathbf{q}_v) = 0 \end{aligned} \quad (3.13.17)$$

β_{wp0} : Reference compressibility of water

β_{wT0} : Volumetric thermal expansion of water ($2.1 \times 10^{-4} K^{-1}$ at 293.15K)

β_{sT} : Volumetric thermal expansion of soil particle

Energy Conservation Equation

The heat conduction is follows the Fourier's law.

$$\mathbf{q}_T = -\lambda \nabla T \quad (3.13.18)$$

Here, the heat conduction in porous media is defined as follows:

$$\lambda = (1-n) \lambda_s + nS \lambda_w + n(1-S) \lambda_v \quad (3.13.19)$$

The energy conservation equation of porous media is summarized as follows: (Rutqvist et al., 2001)

$$\frac{\partial}{\partial t} (C \rho) - Q = -\nabla \cdot (\mathbf{q}_T + C_w (T - T_0) \mathbf{q}_w) \quad (3.13.20)$$

\mathbf{q}_T : Heat flux

$C_w (T - T_0) \mathbf{q}_w$: Heat flux by water advection

Here, the heat capacity in porous media is as follows:

$$C\rho = (1-n)\rho_s C_s + nS_w\rho_w C_w + nS_v\rho_v C_v \quad (3.13.21)$$

The above formula is summarized as follows:

$$\rho C \frac{\partial T}{\partial t} - \nabla \cdot (\lambda \nabla T) + \rho_w C_w \left(\frac{\mathbf{k}}{\gamma_w} (-\nabla \mathbf{p} - \rho_w \mathbf{g}) \right) \cdot \nabla T + \rho_w C_w (T - T_0) \nabla \cdot \left(\frac{\mathbf{k}}{\gamma_w} (-\nabla \mathbf{p} - \rho_w \mathbf{g}) \right) - Q = 0 \quad (3.13.22)$$

13.2

Finite Element

Equation

By transform and discretization the governing equation, the matrix formula can be defined as below. It can be represented in an expanded form including degree of freedom related to temperature from the equation (3.11.12) of the consolidation element. At this time, the diffusion of vapor is considered additionally.

$$\begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{uP} & \mathbf{K}_{uT} \\ \mathbf{K}_{Pu} & \mathbf{K}_{PP} & \mathbf{K}_{PT} \\ 0 & -\Delta t \mathbf{K}_{TP} & \mathbf{K}_{TT} \end{bmatrix} \begin{bmatrix} \delta \mathbf{u} \\ \delta p_w \\ \delta T \end{bmatrix} = \begin{bmatrix} \mathbf{g}_u \\ \Delta t \mathbf{g}_P \\ \Delta t \mathbf{g}_T \end{bmatrix} \quad (3.13.23)$$

Here,

$$\mathbf{K}_{uu} = \sum_e \int_{\Omega^e} \mathbf{B}^T \mathbf{C}_{ep} \mathbf{B} d\Omega$$

$$\mathbf{K}_{uP} = -\sum_e \int_{\Omega^e} S \mathbf{B}^T \mathbf{m} \mathbf{N}_p d\Omega$$

$$\mathbf{K}_{uT} = -\sum_e \int_{\Omega^e} \mathbf{B}^T \mathbf{C}_{ep} \alpha_T \mathbf{m} \mathbf{N}_p d\Omega$$

$$\mathbf{K}_{Pu} = \sum_e \int_{\Omega^e} \mathbf{N}_p^T (S \rho_w + (1-S) \rho_v) \mathbf{m} \mathbf{B} d\Omega$$

$$\begin{aligned} \mathbf{K}_{PP} = & \sum_e \int_{\Omega^e} \mathbf{N}_p^T \left(n(\rho_w - \rho_v) \frac{\partial S}{\partial p} + S \rho_w \frac{S_s}{\gamma_w} + n(1-S) \frac{\rho_v}{\rho_w R_v T} \right) \mathbf{N}_p d\Omega \\ & - \Delta t \sum_e \int_{\Omega^e} \nabla \mathbf{N}_p^T \left(\rho_w \frac{\mathbf{k}}{\gamma_w} + \frac{D_v \rho_v}{\rho_w R T} \right) \nabla \mathbf{N}_p d\Omega \end{aligned}$$

$$\begin{aligned}
\mathbf{K}_{pT} &= \sum_e \int_{\Omega^e} \mathbf{N}_p^T \left(n(\rho_w - \rho_v) \frac{\partial S}{\partial T} - nS\rho_w\beta_{wT0} + n(1-S) \left(R_H \frac{\partial \rho_{vs}}{\partial T} + \frac{\rho_v p}{\rho_w R_v T^2} \right) \right) \mathbf{N}_p d\Omega \\
&\quad - \Delta t \sum_e \int_{\Omega^e} \nabla \mathbf{N}_p^T f_{Tv} D_v \left(H_R \frac{\partial \rho_{vs}}{\partial T} + \frac{\rho_v \mathbf{p}}{\rho_w R \mathbf{T}^2} \right) \nabla \mathbf{N}_p d\Omega \\
\mathbf{K}_{TT} &= \sum_e \int_{\Omega^e} \mathbf{N}_p^T C \rho \mathbf{N}_p d\Omega \\
&\quad + \Delta t \sum_e \int_{\Omega^e} \nabla \mathbf{N}_p^T \lambda \nabla \mathbf{N}_p d\Omega + \Delta t \sum_e \int_{\Omega^e} \mathbf{N}_p^T \left(\rho_w C_w \frac{k_{rel}}{\gamma_w} \mathbf{k}^{sat} (-\nabla \mathbf{p} - \rho_w \mathbf{g}) \right) \nabla \mathbf{N}_p d\Omega \\
\mathbf{K}_{Tp} &= \sum_e \int_{\Omega^e} \nabla \mathbf{N}_p^T \left(\rho_w C_w \mathbf{T} \frac{\mathbf{k}}{\gamma_w} \right) \nabla \mathbf{N}_p d\Omega
\end{aligned}$$

13.3

Analysis Result of Element

The analysis result of fully-coupled element is printed according to the coordinate system specified by the user. The calculated result item is consist of structural element result for the analysis of porous medium, seepage element result and heat transfer element result.