Part II

Ground

Nonlinear Finite Element Analysis

Chapter 1 Static Analysis

Chapter 2 Construction Stage Analysis

Chapter 3 Stress-Seepage Coupled Analysis

Chapter 4 Finite Elements

Ground

Nonlinear Finite Element Analysis / 001

Chapter 1 **Static Analysis / 002**

1.1 Introduction / 002
1.2 Linear Static Analysis / 004
1.3 Nonlinear Static Analysis / 014
1.3.1 Introduction / 014
1.3.2 Nonlinear Elastic Analysis / 014
1.3.3 Elasto-plastic Analysis / 015

Chapter 2 **Construction Stage Analysis / 026**

2.1 Introduction / 026
2.2 Activation and Deactivation of Elements / 029
2.3 Activation and Deactivation of Loads / 032
2.4 Activation and Deactivation of Boundary Conditions / 034
2.5 Load Distribution Factor / 035
2.6 Initialization of Displacements / 036
2.7 Change in Material Properties / 037
2.8 In-situ Stress / 038
2.8.1 K_0 Method / 38
2.8.2 Self Weight Analysis Method / 38
2.9 Undrained Analysis / 039

Chapter 3 **Stress-Seepage Coupled Analysis / 040**

3.1 Introduction / 040
3.2 Effective Stress / 041
3.3 Governing Equation / 042

Chapter 4 **Finite Elements / 043**
4.1 Plane Strain Element / 044
4.1.1 Overview / 044
4.1.2 Element Shape, Nodal Connectivity and Degrees of Freedom / 045
4.1.3 Element Material Properties / 048
4.1.4 Finite Element Formulation / 049
4.1.5 Generation of Element Results / 051
4.2 Truss element / 054
4.2.1 Overview / 054
4.2.2 Element Shape, ECS, Nodal Connectivity & Degrees of Freedom / 054
4.2.3 Element Material Properties / 055
4.2.4 Finite Element Formulation / 056
4.2.5 Internal Element Results / 058
4.3 Embedded Truss Element / 060
4.3.1 Overview / 060
4.3.2 Finite Element Formulation / 060
4.3.3 Internal Element Results / 063
4.4 Beam Element / 065
4.4.1 Overview / 065
4.4.2 Element Shape, ECS & Degrees of Freedom / 066
4.4.3 Element Material Properties / 067
4.4.4 Finite Element Formulation / 067
4.4.5 Internal Element Results / 072
4.5 Interface Element / 075

4.5.1 Overview / 075
4.5.2 ECS, Element Shape & Degrees of Freedom / 077
4.5.3 Element Properties / 077
4.5.4 Finite Element Formulation / 077
4.5.5 Internal Element Results / 080
4.6 Pile/Anchor/Nail/Rockbolt Elements / 081
4.6.1 Overview / 081
4.6.2 Element Shape, ECS & Degrees of Freedom / 082
4.6.3 Element Properties / 083
4.6.4 Finite Element Formulation / 083
4.6.5 Internal Element Results / 087
4.7 Pile Tip Bearing Element / 088
4.7.1 Overview / 088
4.7.2 Elements Shape, ECS & Degrees of Freedom / 088
4.7.3 Element Properties / 090
4.7.4 Finite Element Formulation / 090
4.7.5 Internal Element Results / 092
4.8 Geogrid Element / 093
4.9 Elastic Link / 094
4.10 Rigid Link / 095
4.11 Point Spring/Damping / 096

Nonlinear Finite Element Analysis

In general, geotechnical analysis is often compared with structural analysis for its fundamental concept. In structural analysis, the focus is placed on the uncertainty of applied loads such that various load cases are analyzed for a given structure, and the load cases are combined to determine the maximum force effects on each member for its design. On the other hand, geotechnical analysis is focused on construction sequence and uncertainty of ground materials. Geotechnical engineers primarily place emphasis on analyzing the physical state and behavior of soils for numerical analysis. Therefore, the analysis model must reflect the actual ground composition and construction conditions as much as possible using solid elements. In addition, the analysis must reflect nonlinear and anisotropic characteristics of soils and the initial state of stresses of the ground in order to reflect the in-situ conditions as closely as possible.

A geotechnical analysis software package must be able to simulate the actual site conditions in numerical modeling in order to determine the suitability of design and construction conditions. As a total solution package for geotechnical engineering, SoilWorks is designed to provide all the necessary functions required for numerical analysis for tunnel engineering and soil/rock mechanics. The analysis features included in are outlined below.

A. Static Analysis

1) Linear static analysis
2) Nonlinear static analysis (Nonlinear elastic & elasto-plastic analysis)

B. Construction Stage Analysis

1) Nonlinear Analysis
2) Steady Flow Analysis
3) Transient Flow Analysis
4) Consolidation Analysis

C. **Seepage-Stress Coupled Analysis**

Chapter 1

Static Analysis

1.1 Introduction

Static analysis is theoretically used for analyzing structures, which are not subjected to any kind of vibration. However, in practice, it is often assumed that static analysis can be performed on a structure if the externally applied load has a frequency that is equal to or less than one third of the natural frequency of the structure. Static analysis problems can be divided into linear and nonlinear behaviors according to the behavior type. Linear behavior can be recognized as a special case of nonlinear behavior. The primary sources of nonlinearity in static analysis are as follows:

- ✓ Material nonlinear behavior
- ✓ Geometric nonlinear behavior due to finite strain
- ✓ Nonlinear behavior due to slip and separation at boundaries

In geotechnical analysis, material nonlinear behavior is mainly used to analyze the soil behavior. Material nonlinear behavior is often imposed even to a structure to simulate its detail behavior. The types of material models that can be used to consider the material nonlinearity are as follows:

- ✓ Linear elastic model
- ✓ Nonlinear elastic model
- ✓ Elasto-plastic model
- ✓ Visco-elastic model
- ✓ Visco-elasto-plastic model

Among the above models, SoilWorks provides linear elastic, nonlinear elastic and elasto-plastic models. A more detailed list of the models is shown in the Material manual.

Strains caused by the action of an object may be divided into infinitesimal strain and finite strain. Most geotechnical problems relate to infinitesimal strain, and the mechanical relationship is defined based on the assumption that the strain in the ground is very small. The geometric nonlinear behavior exhibiting finite strain becomes apparent on significantly deformed structures consisted of ductile materials such as mild steel and rubber. Even in

the case of a structure consisted of such ductile materials, the geometric nonlinear effects may be insignificant if the structure exhibits small deformation. Therefore, the geometric nonlinear effects are not commonly considered in geotechnical problems provided that the deformation of the object in question is not so large.

Nonlinear effects at boundary surfaces reflect the nonlinear behaviors taking place at the interfaces between different materials. A variety of behaviors may occur at the interfaces between different materials such as shear slip, (crack) opening and contact depending on the types of external loadings. In order to take into account such behaviors, it is common to define the behaviors of interfaces with the special types of elements. The special types of elements and behaviors are summarized below.

- ✓ Interface, Contact Element: slip, cracking or opening, contact, bending
- ✓ Elastic Link: tension-only, compression-only, hook, gap
- ✓ Spring[1]: tension-only, compression-only, hook, gap

The static analysis types provided by SoilWorks can be classified into the following based on the linearity of the behavior:

- ✓ Linear static analysis
- ✓ Nonlinear static analysis

Even though linear behaviors are viewed as special forms of nonlinear behaviors, linear analysis is separated into an independent analysis type in SoilWorks because it is more convenient and intuitive to use. In linear static analysis, the material model is limited to linear elastic models, which include truss element and elastic link element of tension-only, compression-only, hook or gap behavior. Therefore, linear static analysis can be useful to observe the approximate behavior of the ground, to compose the initial condition of construction stage analysis or to perform a tunnel lining analysis.

When linear static analysis is performed on the ground or a structure, which has been assigned with nonlinear material models, SoilWorks internally converts the material models into linear types and performs a linear static analysis.

---

[1] Currently the spring element in SoilWorks does not consider the nonlinear behavior.

1.2 Linear Static Analysis

Linear static analysis is used to determine the overall behavior of soil and rock structures by assuming that all ground elements are composed of linear elastic materials. Linear elastic behaviors of ground materials can be observed at the initial stage of loading application, which generates very small strain. Linear static analysis neglects the failure mode and idealizes the stress-strain relationship as a straight line and is thus used for simple analysis such as determining the initial stress distribution or stress concentration in the original ground. Theoretically, the solutions to nonlinear finite element equations are found by incremental equations. The increments used in the incremental equations are formulated on the basis of linear elastic equations founded on Hooke's Law. Therefore, nonlinear elastic analysis and elasto-plastic analysis can be ultimately executed in the form of the linear elastic equations.

The inception of nonlinear elastic or elasto-plastic analysis by finite element method became common in the practice of geotechnical engineering in early 1990s. Initially, there were technical difficulties in performing nonlinear analysis in practice. Relative to linear elastic analysis, long computation time was required for the convergence of nonlinear elastic or elasto-plastic analysis. With the advent of the rapid development of computer analysis speed and the advancement in nonlinear elastic and elasto-plastic analysis algorithms, nonlinear analysis has been widely used. Nevertheless, linear analysis is still widely used in practice for efficiency and intuitive results in conducting preliminary analysis of nonlinear materials and simple analysis involving materials of relatively small inherent nonlinearity.

In most civil engineering analysis problems, questions can be condensed to "whether or not a structure is safe under a given set of loads" and "what the magnitude of deformation is prior to structural failure". In order to obtain geotechnical deformation, a stress-strain relationship is required. However, the stress-strain relationships of geo-materials are very complex. A stress-strain relationship varies widely depending upon the material composition, porosity, stress history and loading method.

The simplest way to obtain the stress-strain relationship is assuming the linear elastic behavior of the ground. Since stress-strain relationships exhibit very complex nonlinear material characteristics, it is not practical to mathematically represent them in numerical models. In practice, the material models are idealized into a simpler form to model stress-

strain relationships. In the process of idealizing the material models, the modulus of elasticity and Poisson's ratio alone cannot adequately address the ground behavior realistically. But approximations can be made to analyze the behaviors at some specific stress states. In such cases, a clear definition of the modulus of elasticity is required to idealize the material model.

The key material parameter representing the material properties is the modulus of elasticity in which the concepts of Tangent modulus and Secant modulus are most widely used. For a perfectly linear elastic material model, the Tangent and Secant moduli are identical. The Secant modulus is often referred to as the deformation modulus of a nonlinear material in the stress range of interest.

The equilibrium equation among the basic equations used for linear elastic analysis is expressed in Eq. (1.2.1).

$$\mathbf{Ku} = \mathbf{p} \tag{1.2.1}$$

where,

$\mathbf{K}$: Global stiffness matrix of ground and structures

$\mathbf{u}$: Displacement vector

$\mathbf{p}$: Load vector or unbalanced force vector

From Eq. (1.2.1), the displacement vector can be determined. This method is known as the displacement method, which obtains the displacement vector from the first found analysis results. The strain is calculated from the displacement vector through the compatibility equation, and then the stress is determined from the strain through the constitutive equation.

When a material undergoes deformation, as shown in Fig. 1.2.1, an internal point (x, y, z) of the material moves to new coordinates, (x+u, y+v, z+w). Unless the element is a rigid body, the displacement vector *U* (*u, v, w*) continuously changes, which can be expressed as a function of x, y and z coordinates. The infinitesimal length (dx, dy, dz) inside the material space having 3 directions generates new displacement vectors after deformation has taken place.

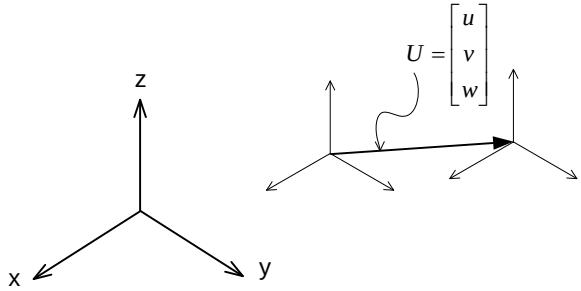


Figure 1.2.1 Definition of displacement (u, v, w)

$$
\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} \\
\varepsilon_y &= \frac{\partial v}{\partial y} \\
\varepsilon_z &= \frac{\partial w}{\partial z} \\
\gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \\
\gamma_{yz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \\
\gamma_{zx} &= \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}
\end{aligned}
\tag{1.2.2}
$$

Applying uniaxial stress in an elastic material results in strain in the stress direction and also strain in the transverse direction due to the Poisson's effect.

$$
\begin{aligned}
\varepsilon_z &= \frac{\sigma_z}{E} \\
\varepsilon_x &= \varepsilon_y = -\nu\varepsilon_z
\end{aligned}
\tag{1.2.3}
$$

where,

$\varepsilon_x, \varepsilon_y, \varepsilon_z$: Strain components in the x, y and z directions

E : Modulus of elasticity

ν : Poisson's ratio

If shear stress, τ_{zx} , acts on an element, the shear strain can be expressed as Eq. (1.2.4)

$$\gamma_{zx} = \frac{\tau_{zx}}{G} \tag{1.2.4}$$

where,

G : Shear modulus

Shear modulus can be derived from the modulus of elasticity and Poisson's ratio.

$$G = \frac{E}{2(1+\nu)} \tag{1.2.5}$$

The volumetric strain of a soil material is expressed as follows:

$$\frac{\Delta V}{V} = \varepsilon_x + \varepsilon_y + \varepsilon_z = \frac{(\sigma_x + \sigma_y + \sigma_z)}{E}(1-2\nu) \tag{1.2.6}$$

where,

$$\begin{aligned} \varepsilon_x &= \frac{1}{E}[\sigma_x - \nu(\sigma_y + \sigma_z)] \\ \varepsilon_y &= \frac{1}{E}[\sigma_y - \nu(\sigma_z + \sigma_x)] \\ \varepsilon_z &= \frac{1}{E}[\sigma_z - \nu(\sigma_x + \sigma_y)] \end{aligned} \tag{1.2.7}$$

Thus, the bulk modulus, K, can be defined as follows:

$$K = \frac{[(\sigma_x + \sigma_y + \sigma_z)/3]}{\Delta V / V} = \frac{E}{3(1-2\nu)} \tag{1.2.8}$$

Due to the characteristics of the ground continuum, the use of the bulk modulus (K) and shear modulus (G) may be a little controversial. But they are more conveniently used as they can be expressed more concisely and clearly compared to E and ν. Figure 1.2.2 illustrates the physical significance of K and G.

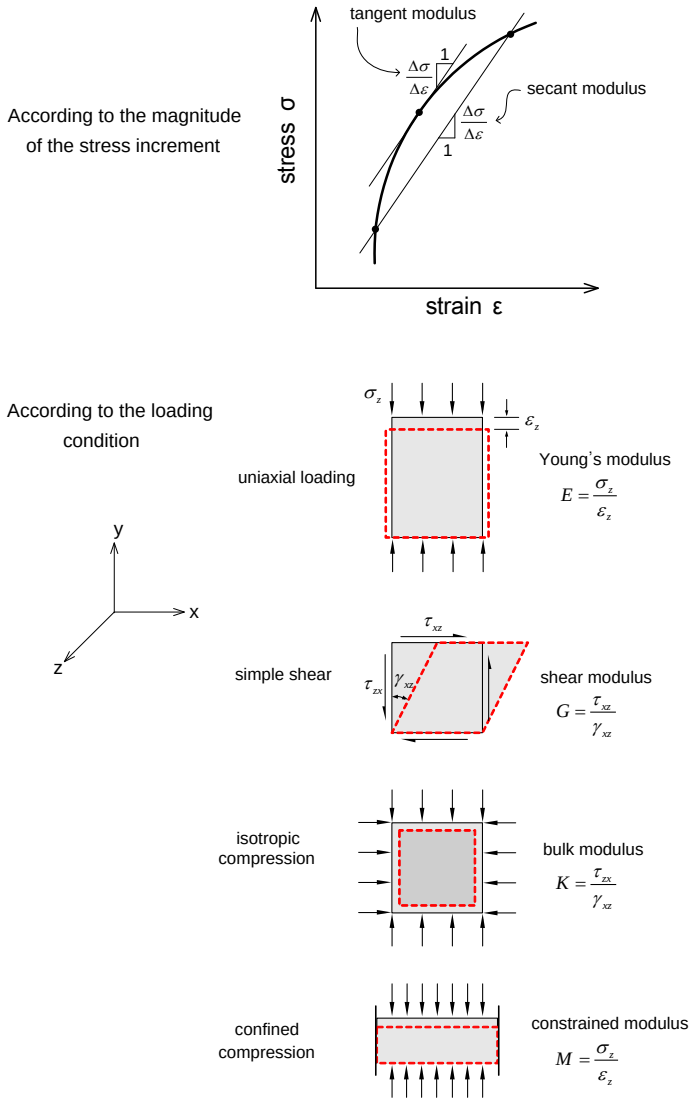


Figure 1.2.2 Various types of moduli

When boundaries are constrained horizontally and deformation occurs in one dimension, the constrained modulus, M, can be determined. Eq. (1.2.9) and (1.2.10) give the equations for horizontal stresses and the constrained modulus for $\varepsilon_x = \varepsilon_y = 0$ respectively.

$$\sigma_x = \sigma_y = \frac{\nu}{1-\nu}\sigma_z \tag{1.2.9}$$

$$M = \frac{(1-\nu)}{(1+\nu)(1-2\nu)}E \tag{1.2.10}$$

The modulus of elasticity obtained from field testing is one of the moduli above. This can be used in practice after appropriately converting the modulus.

The boundary condition for one dimensional consolidation behavior is identical to that for the constrained modulus, which is closely related to the properties of one dimensional consolidation behavior in soft ground. Table (1.2.1) summarizes the relationships between the one dimensional consolidation property parameters and the constrained modulus.

Table 1.2.1 Relationships between Consolidation Parameters and Constrained Modulus

Consolidation Parameter	Relationship with M
Coefficient of volumetric change, m_v	$m_v = \frac{1}{M}$
Coefficient of compressibility, a_v	$a_v = \frac{1+e_0}{M}$
Compression index, c_c	$c_c = \frac{(1+e_0)\sigma_{va}}{0.435M}$

Table 1.2.2 Moduli of Elasticity and Poisson's Ratios of rocks & other materials

Material	Modulus of Elasticity (tonf/m^2)	Poisson's Ratio
Amphibolites	9.4~12.1 ×106	0.28~0.30
Anhydrite	6.8 ×106	0.30
Siabase	8.7~11.7 ×106	0.27~0.30
Siorite	7.5~10.8 ×106	0.26~0.29
Solomite	11.0~12.1 ×106	0.30
Sunite	14.9~18.3 ×106	0.26~0.28
deldspathic gneiss	8.3~11.9 ×106	0.15~0.20
Gabbro	8.9~11.7 ×106	0.27~0.31
Granite	7.3~8.6 ×106	0.23~0.27
Ice	7.1 ×106	0.36
Limestone	8.7~10.8 ×106	0.27~0.30
Marble	8.7~10.8 ×106	0.27~0.30
mica Schist	7.9~10.1 ×106	0.15~0.20
Obsidian	6.5~8.0 ×106	0.12~0.18
Oligoclasite	8.0~8.5 ×106	0.29
Quartzite	8.2~9.7 ×106	0.12~0.15
rock salt	3.5 ×106	0.25
Slate	7.9~11.2 ×106	0.15~0.20
Aluminum	5.5~7.6 ×106	0.34~0.36
Steel	20.0 ×106	0.28~0.29

The values shown in Table (1.2.2) are based on testing results obtained from uncracked small specimens of various materials. Accordingly, reduced moduli of elasticity of rock materials should be used to consider the discontinuity of materials in a large scale. Figure (1.2.3) shows a graph indicating the testing data of elastic modulus reduction ratio relative to Rock Quality Designation (RQD). RQD is defined as the cumulative length of core pieces longer than 100 mm within 1m of a core sample divided by the core length of 1m expressed in a percentage. The higher the RQD value, the better is the rock quality. An

RQD of 100% does not necessarily mean that the rock is perfectly continuous without defects. Weathered rocks exhibit small RQD values.

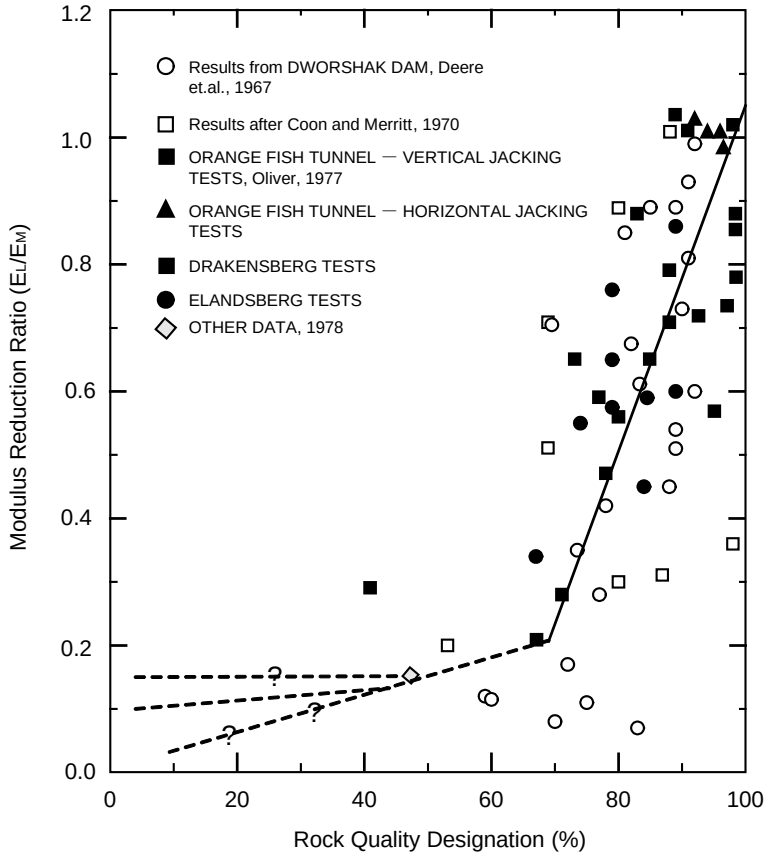


Figure 1.2.3 Reduction in Ratio of Modulus of Elasticity (E_M / E_L) relative to RQD

As shown in Figure 1.2.3, the Modulus of Elasticity reduces from 100% to 20% even at the RQD of 70%.

The 3D stress-strain relationship of a material is expressed as follows:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & 1/E & -\nu/E & 0 & 0 & 0 \\ -\nu/E & -\nu/E & 1/E & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} \tag{1.2.11}$$

The above equation can be rewritten in an inverse matrix form as,

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = A \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-\nu \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} \tag{1.2.12}$$

where,

$$A = \frac{E}{(1-2\nu)(1+\nu)}$$

That is,

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon} \tag{1.2.13}$$

$$(\sigma_x + \sigma_y + \sigma_z)/3 = K(\varepsilon_x + \varepsilon_y + \varepsilon_z) \tag{1.2.14}$$

where,

$$K = \frac{E}{3(1-2\nu)}$$

The matrix $\mathbf{D}$ related to the compatibility matrix is as follows:

$$\begin{bmatrix} D_1 & D_2 & D_2 & 0 & 0 & 0 \\ D_2 & D_1 & D_2 & 0 & 0 & 0 \\ D_2 & D_2 & D_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & D_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & D_3 & 0 \\ 0 & 0 & 0 & 0 & 0 & D_3 \end{bmatrix} \tag{1.2.15}$$

where,

$$D_1 = K + (4/3)G$$

$$D_2 = K - (2/3)G$$

$$D_3 = G$$

1.3 Nonlinear Static Analysis

1.3.1 Introduction

In geotechnical analysis, nonlinear behaviors are induced by the material nonlinearity of the ground. The material nonlinear behavior is classified into nonlinear elastic, elasto-plastic and visco-elasto-plastic models depending on the behavioral assumption. For a small strain problem, a nonlinear elastic model can be usefully applied. For a large strain problem, which is related to the speed of applied load or settlement, visco-elasto-plastic analysis is appropriate. In the majority of other cases, elasto-plastic analysis is used. SoilWorks supports only nonlinear elastic and elasto-plastic analyses.

1.3.2 Nonlinear Elastic Analysis

The elastic characteristic of a nonlinear elastic material changes with the analysis results. A hyperbolic model such as the Duncan-Chang model is a typical example of a nonlinear material model in which the stress and strain relationship can be expressed as a hyperbolic curve. The ground parameters are a function of confinement stress and shear stress. The model is quite useful because it can be easily defined by ground material properties obtained from triaxial tests or published literatures. But its shortcomings include inability to express a reduction in stiffness after failure has taken place.

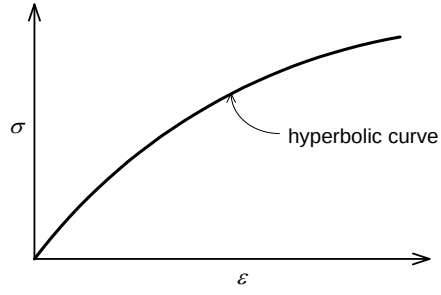


Figure 1.3.1 Stress-Strain Curve of Duncan-Chang model

1.3.3 Elasto-plastic Analysis

1.3.3.1 Introduction

Unlike the elastic behavior, the plastic behavior presents permanent deformation on the structure even after removing the applied load. In order to represent such a behavioral characteristic, the total strain can be divided into the elastic and plastic components as follows:

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^{e} + \boldsymbol{\varepsilon}^{p} \tag{1.3.1}$$

where,

$\boldsymbol{\varepsilon}$: Total Strain

$\boldsymbol{\varepsilon}^{e}$: Elastic Strain

$\boldsymbol{\varepsilon}^{p}$: Plastic Strain

Hook's Law defines the stress-strain relationship in the elastic range, which is applied to Eq. (1.3.1) to define the stress as,

$$\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon}^{e} = \mathbf{D}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{p}) \tag{1.3.2}$$

where,

$\boldsymbol{\sigma}$: Stress Vector

$\mathbf{D}$: Material Stiffness Matrix

The stresses at a specific point in the structure subjected to loading become the measure of the state of elasto-plasticity. Such a measure of evaluating the elasticity and plasticity is defined with respect to the characteristics of materials such as steel and concrete, which is referred to as the yield criterion. The yield criterion of a material is defined on the basis of experimental data. The stresses at the point of inducing the plastic flow can be expressed as a function in the stress space as below.

$$f(\boldsymbol{\sigma}, \kappa) = 0 \tag{1.3.3}$$

where,

f : Yield function

κ : Hardening parameter

No plastic flow takes place if the value of the yield function, f is less than zero. Plastic flow occurs if f is greater than zero.

1.3.3.2 Plastic Flow Rule

The yielding of a material initiates the plastic flow, which induces stress redistribution to maintain the equilibrium of the material. The plastic flow is calculated in a nonlinear form and is generally formulated using an incremental form. The representative variables for computing the plastic flow in the elasto-plastic analysis of a material are the incremental stress direction and the plastic strain direction. Between these, the incremental stress direction can be written as below.

$$\mathbf{n}_i = \frac{\partial f_i}{\partial \boldsymbol{\sigma}} \tag{1.3.4}$$

$\mathbf{n}$ is the slope vector representing the direction of the incremental stress perpendicular to the yield surface. $\mathbf{i}$ is the total number of yield functions in case of more than one.

The incremental plastic strain can be separated into the magnitude and direction components following the Koiter's law as follows:

$$\dot{\boldsymbol{\varepsilon}}_p = \sum_{i=1}^{n} \dot{\lambda}_1 \frac{\partial g_i}{\partial \boldsymbol{\sigma}} = \sum_{i=1}^{n} \dot{\lambda}_1 \mathbf{m}_i \tag{1.3.5}$$

g_i , being the plastic potential function, is a function ($g_i(\boldsymbol{\sigma}, \kappa)$) of the stress and the hardening parameter, κ which is obtained from material testing. The plastic multiplier $\dot{\lambda}_i$ must satisfy the Kuhn-Tucker conditions as follows:

$$f \le 0 \quad ; \quad \dot{\lambda}_i \ge 0 \quad ; \quad \dot{\lambda}_i f = 0 \tag{1.3.6}$$

These conditions show that if the yield function f is less than zero, then $\dot{\lambda}_i$ is always

zero and no plastic flow will take place.

In Eq. (1.3.5), $\mathbf{m}$ is defined as the directional vector of incremental plastic strain. When $\partial f / \partial \boldsymbol{\sigma}$ is used to define the direction of the incremental plastic strain, using the yield function f rather than using the plastic potential function g, this method is referred to as the associated flow rule. When $\partial g / \partial \boldsymbol{\sigma}$ is used to define the direction of the incremental plastic strain, using the plastic potential function, the method is referred to as the non-associated flow rule. In general, the associated flow rule is applied to material models, which exhibit constant deviatoric stress distribution along the hydrostatic axis in the stress space such as the von Mises and Tresca models.

The non-associated flow rule is used for material models in which the deviatoric stress changes with the hydrostatic axis in the stress space such as the Mohr-Coulomb and Drucker-Prager models. When the non-associated flow rule is used for such models with varying deviatoric stress along the hydrostatic axis, it has an effect of restraining excessive volume expansion (dilatancy effects) induced by the inconsistency of the directions of the stress and strain. However, it is important to note that slower convergence is expected because the resulting non-symmetric stiffness matrix needs to be solved by a non-symmetric solver.

1.3.3.3 Linearized Consistency Condition

The state of internal change in an elasto-plastic material is defined by the incremental constitutive relationship that is expressed in the form of infinitesimal increments. The plastic flow begins when the stress point reaches the yield criterion, which is controlled by the parameter of plastic state such as κ.

The constitutive equation for the state of infinitesimal strain in a non-associated plastic flow is founded on Eq. (1.3.2) as below.

$$\dot{\boldsymbol{\sigma}} = \mathbf{D}\,(\dot{\boldsymbol{\varepsilon}} - \dot{\boldsymbol{\varepsilon}}^{\mathrm{p}}) = \mathbf{D}\,(\dot{\boldsymbol{\varepsilon}} - \dot{\lambda}\,\mathbf{m}) \tag{1.3.7}$$

When the stress point lies on the yield surface, the linearized consistency condition is defined through the first order expansion of Taylor series.

$$\dot{f}(\boldsymbol{\sigma}, \kappa) = \left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^{\mathrm{T}} \dot{\boldsymbol{\sigma}} + \frac{\partial f}{\partial \kappa}\frac{\partial \kappa}{\partial \lambda}\dot{\lambda} = \mathbf{n}^{\mathrm{T}}\dot{\boldsymbol{\sigma}} - \mathrm{h}\dot{\lambda} = 0 \tag{1.3.8}$$

where, $\mathbf{n} = \dfrac{\partial f}{\partial \boldsymbol{\sigma}}$, and the hardening modulus $h = -\dfrac{\partial f}{\partial \kappa}\dfrac{\partial \kappa}{\partial \lambda}$.

$\partial f / \partial \kappa$ is calculated from the $f - \kappa$ relationship based on experiments. $\partial \kappa / \partial \lambda$ depends on the material's yield model, and it is calculated using the $\kappa - \lambda$ relationships defined for the individual yield models outlined in Section 1.2.

Substituting Eq. (1.3.7) into Eq. (1.3.8) and rearranging it for the plastic multiplier $\dot{\lambda}$ give the following:

$$\dot{\lambda} = \frac{\mathbf{n}^{\mathrm{T}}\mathbf{D}}{h + \mathbf{n}^{\mathrm{T}}\mathbf{D}\mathbf{m}}\dot{\boldsymbol{\varepsilon}} \tag{1.3.9}$$

Substituting Eq. (1.3.9) into Eq. (1.3.7) gives the incremental form of a stress-strain relationship:

$$\dot{\boldsymbol{\sigma}} = \left(\mathbf{D} - \frac{\mathbf{D}\mathbf{m}\mathbf{n}^{\mathrm{T}}\mathbf{D}}{\mathrm{h} + \mathbf{n}^{\mathrm{T}}\mathbf{D}\mathbf{m}}\right)\dot{\boldsymbol{\varepsilon}} = \mathbf{D}^{\mathrm{ep}}\dot{\boldsymbol{\varepsilon}} \tag{1.3.10}$$

where, $\mathbf{D}^{\mathrm{ep}}$ is the elastic-plastic tangent operator that represents the tangent stiffness matrix of a material. Introducing the non-associated flow rule at this point, because $\mathbf{m} \neq \mathbf{n}$, $\mathbf{D}^{\mathrm{ep}}$ becomes a non-symmetric matrix.

1.3.3.4 Integration of incremental equations

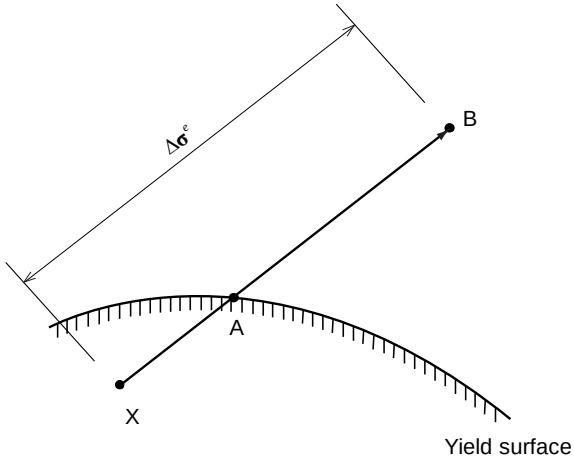
The integration methods for incremental equations can be largely divided into the explicit forward Euler algorithm with sub-increment and the implicit backward Euler algorithm.

In the explicit forward Euler algorithm, the direction of plastic flow is obtained at the point A (refer to Figure 1.3.2 & 1.3.3), where the incremental stress intersects with the yield surface. In the implicit backward Euler algorithm, it is calculated at the point B (refer to Figure 1.3.4) of the final stress point.

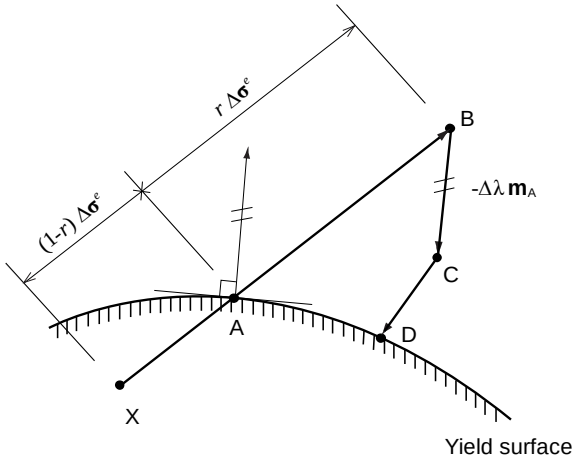
The explicit forward Euler algorithm is relatively simple and does not require iterative calculations at the integration points, but the following shortcomings are present:

- ✓ Conditionally stable
- ✓ Sub-increments are required to attain permissible accuracy (Figure 1.3.3)
- ✓ Additional calibration is required for returning to the yield surface (Figure1.3.2)

In addition, the consistent tangent stiffness matrix cannot be derived in the explicit forward Euler algorithm. The implicit backward Euler algorithm produces sufficiently accurate results and is unconditionally stable without sub-increments or artificial returns, but it requires iterative calculations at the integration points. However, this method is more efficient than the explicit forward Euler algorithm since a consistent tangent stiffness matrix can be constituted.



(a) Location of the intersection point A



(b) Moving from A to C in the tangential direction and calibrating to point D

Figure 1.3.2 Explicit forward Euler algorithm

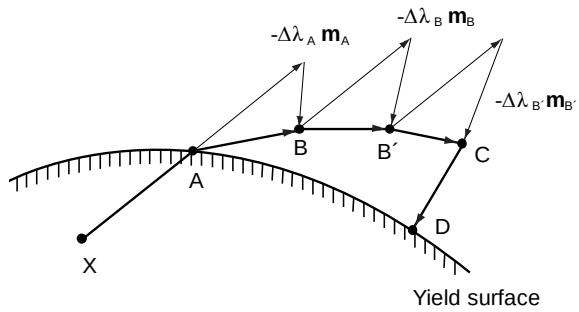


Figure 1.3.3 Sub-increments in the explicit forward Euler algorithm

1.3.3.5 Explicit forward Euler method

In the explicit forward Euler method, the elastic stress increment is first calculated based on the assumption of elastic strain. Refer to the point B in Figure 1.3.2(a).

$$\begin{aligned} \Delta\boldsymbol{\sigma}^{e} &= \mathbf{D}\,\Delta\boldsymbol{\varepsilon} \\ \boldsymbol{\sigma}_{\mathrm{B}} &= \boldsymbol{\sigma}_{\mathrm{X}} + \Delta\boldsymbol{\sigma}^{\mathrm{e}} \end{aligned} \tag{1.3.11}$$

Next, the incremental stress defining the elastic limit is calculated. The initial elastic stress increment is decomposed into the allowable stress increment $(1-r)\Delta\boldsymbol{\sigma}^{\mathrm{e}}$ within the elastic region and the unallowable stress increment $r\Delta\boldsymbol{\sigma}^{\mathrm{e}}$ outside the yield function. The stress increment defining the elastic limit is calculated from the following equation. Refer to the point A in Figure 1.3.2(a).

$$\begin{aligned} & f(\boldsymbol{\sigma}_{\mathrm{X}} + (1-r)\Delta\boldsymbol{\sigma}^{\mathrm{e}}, \kappa) = 0 \\ & r = \frac{f_{\mathrm{B}}}{f_{\mathrm{B}} - f_{\mathrm{X}}} \end{aligned} \tag{1.3.12}$$

For the subscripts in Eq. (1.3.10) and Eq. (1.3.11), refer to Figure (1.3.2).

In order to apply the sub-increment method, the stress increment $r\Delta\boldsymbol{\sigma}^{\mathrm{e}}$ outside the yield function is subdivided into a number (k) of small sub-increments for approximation (Figure 1.3.3). The number of sub-increments is directly related to the margin of error, which is calculated from Eq. (1.3.13).

$$k = \mathrm{INT}\left[8\left(\sigma_{\mathrm{effB}} - \sigma_{\mathrm{effA}}\right)/\sigma_{\mathrm{effA}}\right] + 1 \tag{1.3.13}$$

where, σ_{effA} and σ_{effB} are the effective stresses at the points A and B in Figure 1.3.2(a) respectively.

When the final stress point is not on the yield surface, the stress point is moved to the yield surface through iteration using a returning algorithm. Refer to point D in Figure 1.3.3.

$$\begin{aligned} \Delta\lambda_{\mathrm{C}} &= \frac{f_{\mathrm{C}}}{h + \mathbf{n}_{\mathrm{C}}^{\mathrm{T}}\mathbf{D}\mathbf{m}_{\mathrm{C}}} \\ \boldsymbol{\sigma}_{\mathrm{D}} &= \boldsymbol{\sigma}_{\mathrm{C}} - \Delta\lambda_{\mathrm{C}}\mathbf{D}\mathbf{m}_{\mathrm{C}} \end{aligned} \tag{1.3.14}$$

1.3.3.6 Implicit backward Euler method

The explicit backward Euler method always requires the calculation of the intersection point since the next stress point is estimated using the direction vector perpendicular to the yield surface at the intersection point A. However, the calculation of the intersection point is not required if the direction vector perpendicular to the yield surface is estimated at the point B as shown in Figure 1.3.4.

$$\boldsymbol{\sigma}_{\mathrm{C}} = \boldsymbol{\sigma}_{\mathrm{B}} - \Delta\lambda\mathbf{D}\mathbf{m}_{\mathrm{C}} \tag{1.3.15}$$

Iterative analysis is required to find the solution since both the left and right hand side terms have values at the unknown point C as shown in Eq. (1.3.15). For such iterative analysis, the residual vector, $\mathbf{r}$ is introduced, which represents the difference between the current stress and the backward-Euler stress.

$$\boldsymbol{\sigma}_{\mathrm{C}} = \boldsymbol{\sigma}_{\mathrm{B}} - \Delta\lambda\mathbf{D}\mathbf{m}_{\mathrm{C}} \tag{1.3.16}$$

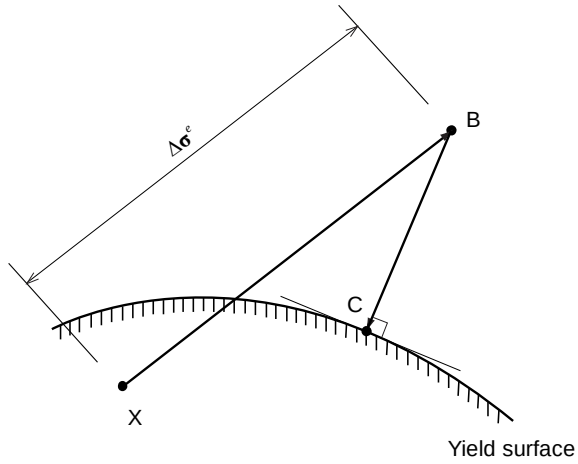


Figure 1.3.4 Implicit backward Euler algorithm

The residual vector $\mathbf{r}$ converges to zero when the final stress point is on the yield surface. During the iterative analysis, the first order expansion of Taylor series is used to define the new residual vector $\mathbf{r}_{\text{new}}$ as follows:

$$\mathbf{r}_{\text{new}} = \mathbf{r}_{\text{old}} + \dot{\boldsymbol{\sigma}} + \dot{\lambda}\mathbf{Dm} + \Delta\lambda\,\mathbf{D}\,\frac{\partial\mathbf{m}}{\partial\boldsymbol{\sigma}}\,\dot{\boldsymbol{\sigma}} \tag{1.3.17}$$

The residual vector $\mathbf{r}_{\text{new}}$ becomes zero due to the converged final stress. Substituting $\mathbf{r}_{\text{new}} = 0$ into Eq. (1.3.17) and rearranging it for $\dot{\boldsymbol{\sigma}}$ gives Eq. (1.3.18).

$$\dot{\boldsymbol{\sigma}} = -\left(\mathbf{I} + \Delta\lambda\,\mathbf{D}\,\frac{\partial\mathbf{m}}{\partial\boldsymbol{\sigma}}\right)^{-1}\left(\mathbf{r}_{\text{old}} + \dot{\lambda}\mathbf{Dm}\right) = -\mathbf{R}^{-1}\left(\mathbf{r}_{\text{old}} + \dot{\lambda}\,\mathbf{Dm}\right) \tag{1.3.18}$$

Also using the first order expansion of Taylor series, the yield function can be written as,

$$f_{new} = f_{old} + \frac{\partial f}{\partial\boldsymbol{\sigma}}\,\dot{\boldsymbol{\sigma}} + \frac{\partial f}{\partial\kappa}\dot{\kappa} = f_{old} + \boldsymbol{n}^T\dot{\boldsymbol{\sigma}} + h\dot{\lambda} = 0 \tag{1.3.19}$$

Now substituting Eq. (1.3.19) into Eq. (1.3.18) yields the following equation for $\dot{\lambda}$:

$$\dot{\lambda} = \frac{f_{\text{old}} - \mathbf{n}^T \mathbf{R}^{-1} \mathbf{r}_{\text{old}}}{h + \mathbf{n}\mathbf{R}^{-1}\mathbf{D}\mathbf{m}} \tag{1.3.20}$$

1.3.3.7 Constitutive matrix

Constructing the plastic constitutive equation is explained below. The infinitesimal stress increment is determined by the elastic part of the infinitesimal strain increment vector. That is,

$$\dot{\boldsymbol{\sigma}} = \mathbf{D}\,(\,\dot{\boldsymbol{\varepsilon}} - \dot{\boldsymbol{\varepsilon}}^{\mathrm{p}}) = \mathbf{D}\,\dot{\boldsymbol{\varepsilon}} - \dot{\lambda}\mathbf{D}\mathbf{m} \tag{1.3.21}$$

Since the current stress point must be always on the yield surface, the consistency condition must be satisfied. Substituting Eq. (1.3.9) into Eq. (1.3.21) and arranging it for the infinitesimal strain increment, the infinitesimal stress increment can be calculated as Eq. (1.3.22).

$$\dot{\boldsymbol{\sigma}} = \left(\mathbf{D} - \frac{\mathbf{D}\mathbf{m}\mathbf{n}^{\mathrm{T}}\mathbf{D}}{\mathrm{h} + \mathbf{n}^{\mathrm{T}}\mathbf{D}\mathbf{m}}\right)\dot{\boldsymbol{\varepsilon}} = \mathbf{D}^{\mathrm{ep}}\dot{\boldsymbol{\varepsilon}} \tag{1.3.22}$$

The converged solution can be obtained faster due to the quadratic convergence property of the Newton-Raphson method when the consistent tangent stiffness matrix is used in the iterative process of the Newton-Raphson method in the explicit forward Euler method. In order to consider this quadratic property, Eq. (1.3.15) is differentiated.

$$\dot{\boldsymbol{\sigma}} = \mathbf{D}\dot{\boldsymbol{\varepsilon}} - \dot{\lambda}\mathbf{D}\mathbf{m} - \Delta\lambda\mathbf{D}\frac{\partial\mathbf{m}}{\partial\boldsymbol{\sigma}}\dot{\boldsymbol{\sigma}} - \Delta\lambda\mathbf{D}\frac{\partial\mathbf{m}}{\partial\kappa}\frac{\partial\kappa}{\partial\lambda}\dot{\lambda} \tag{1.3.23}$$

where, $\dot{\lambda}$ is the rate of change in $\Delta\lambda$.

Eq. (1.3.23) is rewritten as,

$$\mathbf{A}\dot{\boldsymbol{\sigma}} = \mathbf{D}\dot{\boldsymbol{\varepsilon}} - \dot{\lambda}\mathbf{D}\bar{\mathbf{m}} \tag{1.3.24}$$

where, $\mathbf{A} = \mathbf{I} + \Delta\lambda\mathbf{D}\dfrac{\partial\mathbf{m}}{\partial\boldsymbol{\sigma}}$, $\bar{\mathbf{m}} = \mathbf{m} + \Delta\lambda\dfrac{\partial\mathbf{m}}{\partial\kappa}\dfrac{\partial\kappa}{\partial\lambda}$

Setting $\mathbf{H} = \mathbf{A}^{-1}\mathbf{D}$, Eq. (1.3.24) can be rewritten as,

$$\dot{\boldsymbol{\sigma}} = \mathbf{H}\,(\,\dot{\boldsymbol{\varepsilon}} - \dot{\lambda}\bar{\mathbf{m}}) \tag{1.3.25}$$

Rewriting Eq. (1.3.25) with respect to the total strain term based on the linearized consistency condition gives the following:

$$\dot{\boldsymbol{\sigma}} = \left(\mathbf{H} - \frac{\mathbf{H}\bar{\mathbf{m}}\mathbf{n}^{\mathrm{T}}\mathbf{H}}{\mathrm{h} + \mathbf{n}^{\mathrm{T}}\mathbf{H}\bar{\mathbf{m}}}\right)\dot{\boldsymbol{\varepsilon}} = \mathbf{C}^{\mathrm{ep}}\dot{\boldsymbol{\varepsilon}} \tag{1.3.26}$$

$\mathbf{D}^{\mathrm{ep}}$ In Eq. (1.3.22) is referred to as the tangent stiffness matrix, and $\mathbf{C}^{\mathrm{ep}}$ in Eq. (1.3.26) is referred to as the consistent tangent stiffness matrix.

Chapter 2

Construction Stage Analysis

2.1 Introduction

Numerical analyses in typical geotechnical applications can be considered to take place in construction stages. Geotechnical analysis usually involves material nonlinear analysis in which the material nonlinear properties can be obtained from the initial condition of the ground. The initial condition, which is the condition prior to any construction events, refers to the in-situ condition or initial ground stress. In SoilWorks, all the material models are assumed to be linear elastic for the analysis of the initial ground condition.

Once the initial ground stress is obtained, excavation loads can be obtained. In addition, the shear strength of a nonlinear material can be calculated by applying material properties such as Mohr-Coulomb. Therefore, the construction stage analysis entails the entire construction process from the initial ground condition. Since the actual sequence of construction on site is very complex and variable, the analysis is simplified to focus on the major construction events in sequence.

For example, construction stages for a tunnel are shown below.

1st Stage : Initial ground stress
2nd Stage: First face excavation
3rd Stage : Install reinforcement to first face + second face excavation
4th Stage : Install reinforcement to second face + third face excavation
5th Stage : Install reinforcement to third face + fourth face excavation
…… (Continue) ……

SoilWorks considers the following items in construction stage analysis:

1. Expression of Construction Stages

Activate/Deactivate ground elements
Activate/Deactivate loads
Modify boundary condition
Reflect change in material properties

Load distribution ratio
Groundwater level by construction stages
Drained/undrained analysis
Displacement initialization
In-situ analysis

2. Seepage Analysis

Steady state flow analysis by construction stages
Transient state flow analysis by construction stages

3. Stress-Seepage Coupled Analysis

Effective stress analysis using pore water pressure results obtained from ground water seepage analysis.

4. Consolidation Analysis

Consolidation analysis due to embankment, etc.

Activation and deactivation of elements, activation and deactivation of loads and changes in boundary conditions take place at the first step of every construction stage. New construction stages are created whenever any structural changes occur in the process. The number of construction stages thus increases with frequent changes in the structural system.

When new elements are activated in a particular construction stage, the elements are not affected by the load history or internal stresses from the previous construction stages. This means that the newly activated elements in the corresponding construction stage will have zero stresses irrespective of the loadings acting on the structure.

In case certain elements having a 100% load distribution factor are deactivated, all the internal stresses in the elements being deactivating are redistributed to the remaining elements of the structure causing stress changes in the remaining elements. Conversely, if certain elements having a 0% load distribution factor are deactivated, no redistribution of the stresses in the elements being deactivated takes place, resulting in no changes in the stresses in the remaining elements in the structure.

The load distribution factor controls the magnitude of stress transfer from the deactivated elements to the remaining elements. This function becomes very useful when the user has

to consider partial stress relaxation even after the elements have been removed. In the case of tunnel analysis, the stresses in the excavated elements are not transferred at once. Instead, the stresses are gradually redistributed to the rock bolts and shotcrete over a number of construction stages. In such a case, the load distribution factor is used to reflect partial relaxation of stresses over the ensuing stages.

SoilWorks does not perform analyses of independent analytical models for individual construction stages. Instead, only the changes in the structural system, loads and boundaries are defined at each stage, and the analysis results from the previous stages are reflected and analyzed with the changes at the current stage in the concept of an accumulated model. Therefore, any changes in the structure or loadings in the preceding stages influence the results in the current and subsequent construction stages. For example, once a load is applied to the structure in a preceding construction stage, the load remains active for the subsequent construction stages. Even the elements activated in the preceding stages remain active for the current and subsequent stages. Only the new elements or elements to be removed at each stage need to be defined.

2.2 Activation and Deactivation of Elements

The elements that are activated in a construction stage are assigned with zero initial stresses at which point their self weights are applied to the existing structure. These activated elements will become part of the total structure in the subsequent stages.

All ground elements are subject to stresses until they are deactivated. If any loads are acting around the deactivated elements, the remaining elements must not impose stresses on the new free faces through appropriate stress relaxation.

In Figure 2.2.1, the object A is assumed to be removed from the object B.

The stresses in the object A and the object B are σ_{AO} and σ_{BO} respectively prior to removal of the object A. All the external forces have been reflected in these stresses. Since these two objects are in equilibrium, the load, F_{AB} must be applied to the object B to maintain the stress, σ_{BO} .due to the object A. Similarly, the load, F_{BA} must be applied to the object A. Therefore, any excavation load acting on a boundary depends on the stress state of the elements being excavated and the self weights of the elements. This is expressed in Eq. 2.2.1.

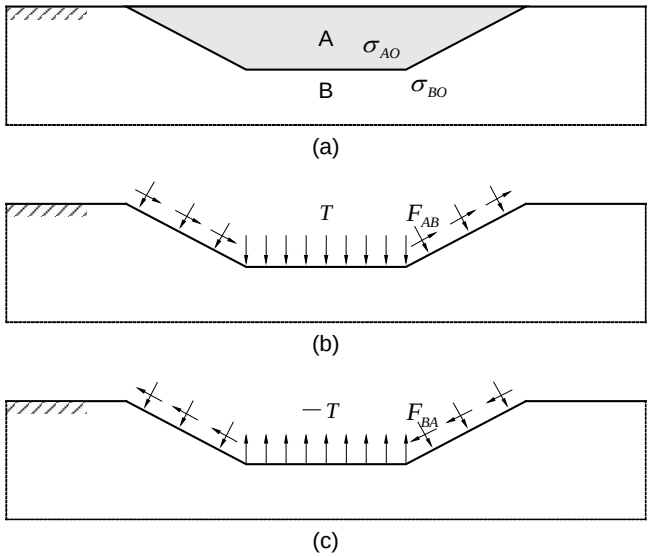


Figure 2.2.1 Concept of Element Deactivation & Equilibrium Forces

$$F_{BA} = -\int_{V_A} \mathbf{B}^T \sigma_{AO} dV_A + \int_{V_A} \mathbf{N}^T \gamma dV_A \tag{2.2.1}$$

where,

$\mathbf{B}$	: Strain-Displacement matrix
V_A	: Volume of excavation
$\mathbf{N}$	: Element shape function
γ	: Unit weight of ground

The global stiffness for the current construction stage is calculated by assembling the stiffness of each activated element as shown in Eq. (2.2.2). All the elements used in all the construction stages are stored in the database, and the element data pertaining to the current stage is added through searching all the elements activated from the initial stage to the current stage. And the data for the deactivated elements is removed.

$$\mathbf{K} = \sum_{i=1}^{n} \mathbf{L}_i^T \mathbf{k}_i \mathbf{L}_i \tag{2.2.2}$$

where,

$\mathbf{K}$	: Global stiffness of the current construction stage
$\mathbf{L}_i$	: Matrix defining the location of element stiffness within the structural stiffness
$\mathbf{k}_i$	: Element stiffness activated at the current construction stage
n	: Total number of elements activated at the current construction stage

2.3 Activation and Deactivation of Loads

The construction stage analysis consists of phased analysis in an incremental form for which the incremental load of the current construction stage is required. The incremental load $\Delta\mathbf{F}^{\text{current}}$ at the current construction stage is the difference between the total load $\mathbf{F}^{\text{current}}$ at the current stage and the total load $\mathbf{F}^{\text{previous}}$ at the immediately preceding stage.

$$\Delta\mathbf{F}^{\text{current}} = \mathbf{F}^{\text{current}} - \mathbf{F}^{\text{previous}} \tag{2.3.1}$$

All the loads used in all the construction stages are stored in the database, and the total load $\mathbf{F}^{\text{current}}$ pertaining to the current stage is added through searching all the loads activated from the initial stage to the current stage. And the data for the deactivated loads is removed.

The total loads at the current construction stage can be classified into surface forces and body forces.

$$\mathbf{F}^{\text{current}} = \mathbf{F}_{\text{surface}}^{\text{current}} + \mathbf{F}_{\text{body}}^{\text{current}} \tag{2.3.2}$$

The surface forces include the loads related to nodes such as nodal or distributed loads, and the body forces include the loads related to elements such as self-weights. While the surface forces are assembled by using the information of the degrees of freedom of the nodes to which the loads are imposed, the body forces are assembled by using the matrix $\mathbf{L}_i$ in Eq. (2.2.2) since the forces are related to adding and subtracting elements.

The sub-incremental load step at a given construction stage is calculated by the incremental load $\Delta\mathbf{F}^{\text{current}}$ in Eq. (2.3.1) multiplied by a load factor α already defined within the current construction stage. It can be expressed for a specific load step i as,

$$\mathbf{F}_i^{\text{current}} = \mathbf{F}^{\text{previous}} + \alpha_i^{\text{current}} \cdot \Delta\mathbf{F}^{\text{current}} \tag{2.3.3}$$

The linear algebraic equation of static analysis to calculate the response of a structure for a specific load step i is given as,

$$\mathbf{F}_i^{\text{ubf}} = \mathbf{K}_i \mathbf{d}_i \tag{2.3.4}$$

$\mathbf{K}_i$ above represents the global stiffness at the ith load step, and $\mathbf{d}_i$ represents the nodal displacement at the ith load step. $\mathbf{F}_i^{\text{ubf}}$ is the unbalance force at the ith step, which is defined by subtracting the total internal force at the previous step from the total current load.

$$\mathbf{F}_i^{\text{ubf}} = \mathbf{F}_i^{\text{current}} - \mathbf{F}_{i-1}^{\text{internal}} \tag{2.3.5}$$

In Eq. (2.3.5), $\mathbf{F}_i^{\text{current}}$ has been already defined in Eq. (2.3.3), and $\mathbf{F}_{i-1}^{\text{internal}}$ is the total internal force of the structure at the previous load step, which is calculated using the element stresses at the previous load step.

$$\mathbf{F}_{i-1}^{\text{internal}} = \sum_{j=1}^{n} \mathbf{L}_j^T \mathbf{f}_j = \sum_{j=1}^{n} \mathbf{L}_j^{\text{T}} \left(\int_V \mathbf{B}_j^{\text{T}} \boldsymbol{\sigma}_j \, dV \right) \tag{2.3.6}$$

where,

$\mathbf{f}_j$	: Element member force
$\mathbf{L}_j$	: Matrix defining the location of element loading within the total load
$\mathbf{B}_j$	: Matrix of slope of displacement-strain relationship
$\boldsymbol{\sigma}_j$	: Element stress
V	: Volume
n	: Number of elements activated at the current construction stage

For $i = 1$, $\mathbf{F}_0^{\text{internal}}$ is the total internal force of the final load step of the previous construction stage. The element stress is updated after calculating for each load step within the construction stage.

2.4 Activation and Deactivation of Boundary Conditions

Similar to the elements and loads, all the boundary conditions used in all the construction stages are stored in the database, and the activated boundary conditions are added through searching all the boundary conditions activated from the initial stage to the current stage. And the data for the deactivated boundary conditions is removed. At this point, free nodes may exist, which are not part of any elements due to boundary conditions added and elements removed at the current stage. Such free nodes are removed using the nodal data associated with the active elements, thereby improving the analysis speed through eliminating the redundant nodal degrees of freedom.

2.5 Load Distribution Factor

In order to simplify the construction stage analysis, the concept of load distribution factors is introduced. This is a numerical technique, which enables the user to sequentially reflect the effects of change in elements to simulate a number of construction stages. Distribution factors are applied when construction stages are condensed in a 3 dimensional analysis, or when a 3 dimensional model is simplified to a 2 dimensional model.

For example, suppose that stress relaxation takes place sequentially over 3 stages by 40%, 30% and 30% from an excavation stage. In SoilWorks, the user can define the excavation stage and activate the load distribution factor option. The user can then simply enter 0.4, 0.3 and 0.3 for 'Subsequent Stage 0, 1 and 2' respectively.

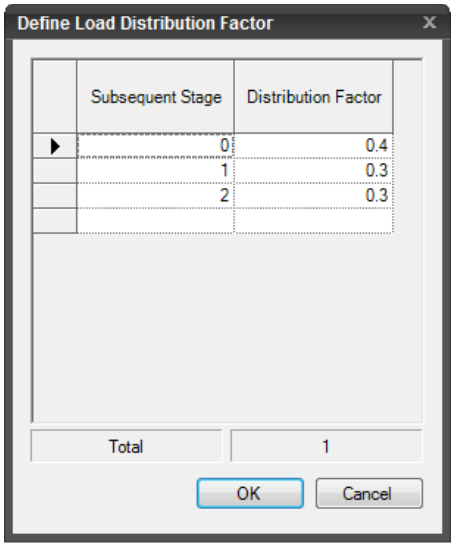


Figure 2.5.1 Load Distribution Factor Dialog Box

2.6 Initialization of Displacements

During a construction stage analysis, the condition for initializing displacements may become necessary. For instance, the displacements must be initialized after performing construction stage analysis including the initial condition. Since the displacements can be initialized at any construction stage in SoilWorks, a number of intermediate stage analyses can be considered to obtain the initial stage condition.

2.7 Change in Material Properties

Changes in ground material properties may be required in construction stage analysis such as to reflect any ground improvements or disturbance and material hardening over time. Structural properties may also change due to curing of lining concrete or a change in lining thickness. SoilWorks allows changes in material properties an unlimited number of times. The analysis of changed properties is performed in continuation from the results of the preceding stage (displacements, stresses, strains, etc.).

In case of linear elastic materials, the material properties can be freely changed, but care should be used when dealing with nonlinear material models.

For instance, if a nonlinear material model such as soil is excavated and backfilled with a different material in a construction stage analysis, the user may attempt to change the material properties only without actually removing and adding the elements. Such a practice is physically incorrect because the stress state from the previous stage will be reflected in the next stage. Therefore, changing the material properties of a nonlinear material model requires removal of the elements first and addition of new elements with the change in material properties. In case new elements are added, SoilWorks initializes the stresses, strains and internal state variables of the new elements to zero.

2.8 In-situ Stress

SoilWorks provides the following two methods for computing in-situ stresses.

2.8.1 K_0 Method

The K_0 method defines the initial stress by calculating the horizontal stress from the vertical stress using a non-dimensional coefficient, K_0, where $K_0 = \sigma_h / \sigma_v$.

The vertical stress σ_v is first obtained by performing a self-weight analysis, and then the horizontal stress is obtained using the relationship, $\sigma_h = K_0 \sigma_v$. In this case, the shear stresses on the vertical and horizontal planes of the soil element remain unchanged from the results produced from the analysis.

If the ground surface is horizontal, this method presents no problem. If the surface, however, is not horizontal, the stress state determined by this method under the self weight will not be in the state of equilibrium. As such, the un-equilibrated internal forces from the selfweight and the stress state calculated above must be used for analysis to reach an equilibrium state. This stage can be performed by introducing a null stage at which no conditions change. After the analysis for the null stage, the total system becomes in equilibrium, which is used as the state of initial stress.

2.8.2 Self Weight Analysis Method

The initial stress state is determined by the stress state obtained by self weight analysis. If the ground surface is horizontal, this method becomes identical to that of the K_0 method, $K_0 = \nu / (1 - \nu)$. Otherwise, the results will differ from those of the K_0 method, and shear stress will exist because horizontal strains exist.

Generally, this method is suitable for a sloped ground surface. This method, however, does not allow the user to specify a K_0 value greater than 1. If K_0 is required to be defined with a value greater than 1, then the K_0 method is first used and a null stage is used to satisfy an equilibrium condition.

2.9 Undrained Analysis

In SoilWorks, undrained analysis can be applied only to the selected elements and associated construction stages. There are two mandatory steps to perform undrained analysis in SoilWorks. First, the material properties for the undrained analysis are defined in the material model's drainage parameters. The second step involves definition of undrained analysis when construction stages are defined. If the drainage parameter option "undrained" is selected, and undrained analysis is not performed in the construction stage, the material will exhibit drained behavior. Conversely, if the drainage parameter option "drained" has been selected, and undrained analysis is performed in the construction stage, the material will also exhibit drained behavior. In order to perform undrained analysis, both the undrained properties in the material properties and undrained analysis in the construction stage must be defined.

It is also possible to simulate drained and undrained analyses alternately by construction stages. When undrained analysis was conducted in the previous stage, and drained analysis is conducted in the current stage, it can be observed that the pore water pressure decreases due to the dissipation of excessive pore water pressure. When drained analysis was conducted in the previous stage, and undrained analysis is conducted with applied loading in the current stage, it can be also observed that the pore pressure changes due to an increase in excessive pore water pressure.

Chapter 3

Stress-Seepage Coupled Analysis

3.1 Introduction

Groundwater seepage phenomenon is caused by the difference in water heads at the boundaries in the seepage zone or the flux boundary condition. As the groundwater flows, seepage force is developed due to the friction between the water and soil particles. This process induces displacements and stresses. Seepage force is expressed as the integration of the pore water pressure.

SoilWorks attains the seepage force effect using the pore water pressure obtained by seepage analysis.

Pore water pressure is the product of the weight density of water and the pressure head (total head - elevation head). In general, the seepage force is concentrated in the zone adjacent to the outflow boundaries where the total head suddenly reduces. Due to the low constraint pressure, shear and tensile strengths are relatively low in this zone. As a result, when the effective stress is analyzed considering the seepage pressure in soils, the ground can easily fail unlike rock conditions. Therefore, stress-seepage coupled analysis is a very important aspect in the stability analysis of soils.

3.2 Effective Stress

Pore water pressure in ground influences the total stress. According to the Terzaghi's principle, the total pressure can be divided into effective stress (σ') and pore water pressure (p_w).

Since water cannot resist any shear stress, the effective shear stress is equal to the total shear stress. Thus, the total stress is expressed as,

$$\begin{aligned}
\sigma_{xx} &= \sigma'_{xx} + p_w \\
\sigma_{yy} &= \sigma'_{yy} + p_w \\
\sigma_{zz} &= \sigma'_{zz} + p_w \\
\tau_{xy} &= \tau'_{xy} \\
\tau_{yz} &= \tau'_{yz} \\
\tau_{zx} &= \tau'_{zx}
\end{aligned} \tag{3.2.1}$$

where, the pore water pressure (p_w) in turn can be divided into steady state pore water pressure $\left(p_{steady}\right)$ and excess pore water pressure (p_{excess}).

$$p_w = p_{steady} + p_{excess} \tag{3.2.2}$$

The steady state pore water pressure, being the input data obtained from the groundwater analysis, is created by the height of the groundwater table. The excess pore water pressure is created during the stress calculation of a material, which exhibits an undrained behavior, or the consolidation analysis. Calculation of excess pore water pressure is described in more detail in the Chapter 2.9 Undrained Analysis or Consolidation Manual.

3.3 Governing Equation

The inverse elastic Hooke's law is considered to derive the Hooke's law expressed in infinitesimal strains.

$$\begin{bmatrix} \dot{\varepsilon}^e_{xx} \\ \dot{\varepsilon}^e_{yy} \\ \dot{\varepsilon}^e_{zz} \\ \dot{\gamma}^e_{xy} \\ \dot{\gamma}^e_{yz} \\ \dot{\gamma}^e_{zx} \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2+2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 2+2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 2+2\nu \end{bmatrix} \begin{bmatrix} \dot{\sigma}'_{xx} \\ \dot{\sigma}'_{yy} \\ \dot{\sigma}'_{zz} \\ \dot{\tau}'_{xy} \\ \dot{\tau}'_{yz} \\ \dot{\tau}'_{zx} \end{bmatrix} \tag{3.3.1}$$

where,

E, ν : Effective material properties – modulus of elasticity and Poisson's ratio respectively.

However, since the derivative of the steady state component is zero,

$$\dot{p}_w = \dot{p}_{excess} \tag{3.3.2}$$

By applying the terms in Eq. 3.2.1, the above Eq. (3.3.1) can be rewritten as follows:

$$\begin{bmatrix} \dot{\varepsilon}^e_{xx} \\ \dot{\varepsilon}^e_{yy} \\ \dot{\varepsilon}^e_{zz} \\ \dot{\gamma}^e_{xy} \\ \dot{\gamma}^e_{yz} \\ \dot{\gamma}^e_{zx} \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2+2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 2+2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 2+2\nu \end{bmatrix} \begin{bmatrix} \dot{\sigma}_{xx} - \dot{p}_{excess} \\ \dot{\sigma}_{yy} - \dot{p}_{excess} \\ \dot{\sigma}_{zz} - \dot{p}_{excess} \\ \dot{\tau}'_{xy} \\ \dot{\tau}'_{yz} \\ \dot{\tau}'_{zx} \end{bmatrix} \tag{3.3.3}$$

Chapter 4

Finite Elements

The element library in SoilWorks is classified as follows depending on the purpose of use:

1) Geotechnical element

- ✓ Plane strain element

2) Structural element

- ✓ Truss element
- ✓ Embedded truss element
- ✓ Beam element

3) Applied element

- ✓ Interface element
- ✓ Pile, rock bolt, anchor & nail elements
- ✓ Pile tip bearing element
- ✓ Geogrid element
- ✓ Elastic link
- ✓ Rigid link
- ✓ Nodal spring

Finite elements are created by the connecting nodes, which define the locations, shapes and sizes. The element types, material properties and stiffness data are also defined.

4.1 Plane Strain Element

4.1.1 Overview

Plane strain elements in SoilWorks can be utilized to simplify a long structure having a uniform cross section throughout its entire length, such as dams and tunnels. Figure 4.1.1 depicts the simplification of a 3-D shape into a 2-D shape section represented by the grey unit thickness.

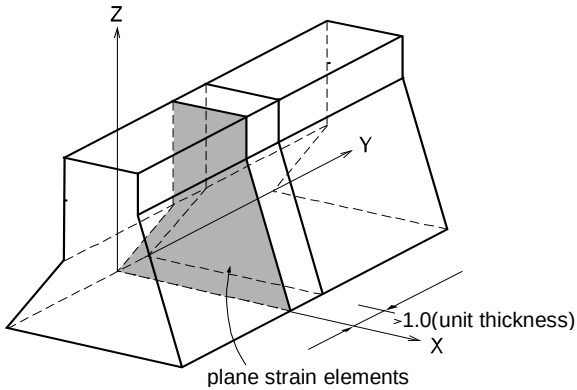


Figure 4.1.1 Thickness of two-dimensional plane strain elements

Because a plane strain element is formulated on the basis of its plane strain properties, no strain is assumed to exist in the thickness direction and the stress component in the thickness direction can be determined through the Poisson's effect.

The plain strain elements can be used together with all other structural and application elements such as truss, beam, interface and elastic link elements and can be used in finite element analysis involving dynamic, tunnel, slope stability and soft ground analyses.

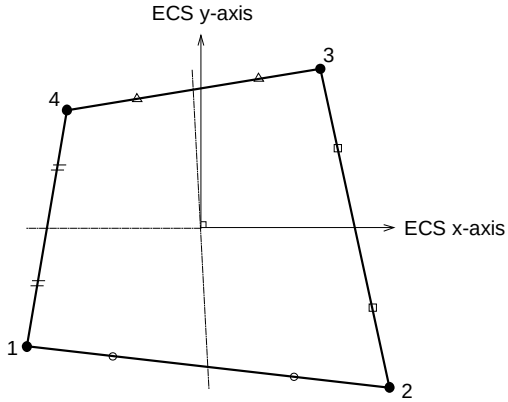
Triangular or quadrilateral shapes are available for plane strain elements. Plane strain elements retain tension, compression and shear stiffness in the in-plane direction, and tension and compression stiffness in the thickness direction. It is recommended that 4-node quadrilateral elements or higher order 6-node or 8-node elements be used rather than using 3-node triangular elements as shown in Figure 4.1.2. It is also recommended that

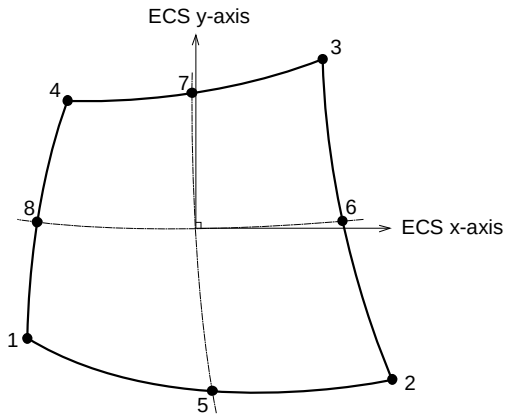
the aspect ratio of an element shape be close to 1.0, as is in the case of an equilateral triangle or a square.

4.1.2 Element Shape, Nodal Connectivity and Degrees of Freedom

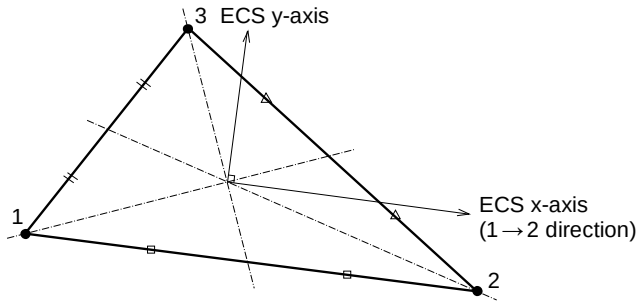
The element shapes, element local coordinate systems, degrees of freedom and node numbering convention supported by SoilWorks are defined in Figure 4.1.2. The analysis results for plane strain elements are produced in the GCS since the GCS is more effective in representing the characteristics of ground elements. Therefore, the element coordinate systems shown in Figure 4.1.2 are not so meaningful, and the nodal forces are produced in reference to the GCS.

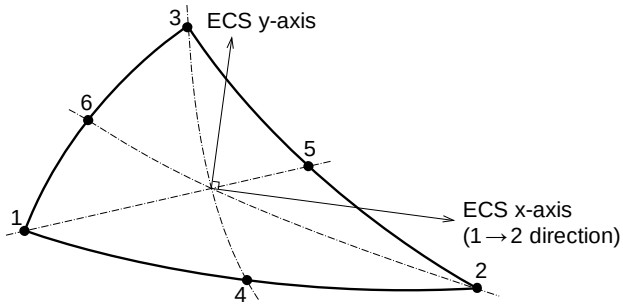
The plane strain elements retain only two translational degrees of freedom in the horizontal and vertical directions at each node on the selected plane in the GCS.





(a) Quadrilateral Elements





(b) Triangular elements

Figure 4.1.2 Arrangement of plane strain elements, element coordinate systems and nodal forces

The node numbering sequence follows the counter-clockwise direction. For higher order elements, the mid-node numbers are assigned after the corner nodes have been numbered.

The plane strain elements in SoilWorks are used in a 2-dimensional model. If the plane of the elements is set to the X-Z plane, the gravity direction is defined in the negative (-) Z direction.

4.1.3 Element Material Properties

When ground elements are modeled as plane strain elements in a linear elastic model, the following material parameters are basically required:

Modulus of Elasticity	:	E
Poisson's Ratio	:	ν
Weight Density	:	γ_t
Saturated Weight Density	:	γ_{sat}
Initial Stress Parameter	:	K_0
Thermal Coefficient	:	α

In addition, SoilWorks supports all the nonlinear ground material models of 2D ground elements using plane strain elements. Refer to the manual of Material Nonlinear Models for additional material properties.

Plane strain elements can be utilized to simplify the 3-dimensional behavior of a long structure. Since the normal strain, shear strain and shear stress in the thickness direction are zero, the elements are assumed to behave in the 2-dimension. The constitutive equation of a plain strain element is $\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\varepsilon}$. In the case of an isotropic linear elastic material, the equation is expressed in a matrix form as follows:

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \dfrac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} = 0 \\ \gamma_{xy} \end{Bmatrix} \qquad (4.2.1)$$

The thickness being the structural property of plane strain elements, a unit thickness (1.0) is automatically considered in the direction normal to X-Z plane as shown in Figure 4.1.1.

4.1.4 Finite Element Formulation

In SoilWorks, plane strain elements are formulated on the basis of the isoparametric plane strain, and the numerical equation implemented is shown below.

$$t\int_A \mathbf{B}^T\mathbf{D}\mathbf{B}dA\mathbf{u} = bt\int_A \mathbf{N}^T dA + pt\int_L \mathbf{N}^T dL + \mathbf{p}_n \qquad (4.1.2)$$

where,

L : Length of element boundary

t : 1.0 (unit thickness)

From Eq. (4.1.2), the shape function, N, can be of a maximum of a quadratic function. The 3-node triangular and 4-node quadrilateral elements are linear elements, and the 6-node triangular and 8-node quadrilateral elements are quadratic elements as shown in Figure 4.1.2.

The shape functions are defined as follows depending on the element types:

- ✓ 3-Node element

$$\begin{aligned} N_1 &= 1-\xi-\eta \\ N_2 &= \xi \\ N_3 &= \eta \end{aligned} \qquad (4.1.3)$$

- ✓ 4-Node element

$$\begin{aligned} N_1 &= \frac{1}{4}(1-\xi)(1-\eta),\ \ N_2 = \frac{1}{4}(1+\xi)(1-\eta) \\ N_3 &= \frac{1}{4}(1+\xi)(1+\eta),\ \ N_4 = \frac{1}{4}(1-\xi)(1+\eta) \end{aligned} \qquad (4.1.4)$$

- ✓ 6-Node element

$$\begin{aligned} N_1 &= (1-\xi-\eta)(1-2\xi-2\eta),\ \ N_2 = \xi(2\xi-1) \\ N_3 &= \eta(2\eta-1),\ \ N_4 = 4\xi(1-\xi-\eta) \\ N_5 &= 4\xi\eta,\ \ N_6 = 4\eta(1-\xi-\eta) \end{aligned} \qquad (4.1.5)$$

- ✓ 8-Node element

$$
\begin{aligned}
N_1 &= \frac{1}{4}(1-\xi)(1-\eta) - \frac{1}{2}N_5 - \frac{1}{2}N_8 \\
N_2 &= \frac{1}{4}(1+\xi)(1-\eta) - \frac{1}{2}N_5 - \frac{1}{2}N_6 \\
N_3 &= \frac{1}{4}(1+\xi)(1+\eta) - \frac{1}{2}N_6 - \frac{1}{2}N_7 \\
N_4 &= \frac{1}{4}(1-\xi)(1+\eta) - \frac{1}{2}N_7 - \frac{1}{2}N_8 \\
N_5 &= \frac{1}{2}(1-\xi^2)(1-\eta), \quad N_6 = \frac{1}{2}(1+\xi)(1-\eta^2) \\
N_7 &= \frac{1}{2}(1-\xi^2)(1+\eta), \quad N_8 = \frac{1}{2}(1-\xi)(1-\eta^2)
\end{aligned}
\tag{4.1.6}
$$

The strain of a plane strain element is expressed as the following engineering strain:

$$
\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \partial u/\partial x \\ \partial v/\partial y \\ 0 \\ \partial u/\partial y + \partial v/\partial x \end{Bmatrix}
\tag{4.1.7}
$$

Accordingly, the strain-displacement matrix is defined as,

$$
\mathbf{B} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \cdots & \frac{\partial N_{np}}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & \cdots & 0 & \frac{\partial N_{np}}{\partial y} \\ 0 & 0 & \cdots & 0 & 0 \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \cdots & \frac{\partial N_{np}}{\partial y} & \frac{\partial N_{np}}{\partial x} \end{bmatrix}
\tag{4.1.8}
$$

The Jacobian matrix is expressed in Eq. (4.1.9):

$$
\mathbf{J} = \begin{bmatrix} \partial x/\partial \xi & \partial y/\partial \xi \\ \partial x/\partial \eta & \partial y/\partial \eta \end{bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \cdots & \frac{\partial N_{np}}{\partial \xi} \\ \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \cdots & \frac{\partial N_{np}}{\partial \eta} \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ \vdots & \vdots \\ x_{np} & y_{np} \end{bmatrix}
\tag{4.1.9}
$$

The numerical integration for the plane strain elements is shown as,

$$t\int_A \mathbf{B}^T\mathbf{D}\mathbf{B}dA \rightarrow t\sum_{j=1}^{n}\mathbf{B}_j^T\mathbf{D}_j\mathbf{B}_j\left|\mathbf{J}_j\right|W_{1j}W_{2j} \tag{4.1.10}$$

where,

W_{1j}, W_{2j} : Weight factors for the directions of ξ, η of integration points

4.1.5 Generation of Element Results

The results of 2-D plane strain elements include internal forces, stresses and strains. There are no differences in the result items between triangular and quadrilateral elements or low and higher order elements. The results are produced in the GCS due to the characteristics of ground elements.

4.1.5.1 Element Force Results

The internal forces of the elements are produced at each node. The sign convention for the internal forces is shown in Figure 4.1.3. The directions of the arrows represent the positive (+) directions.

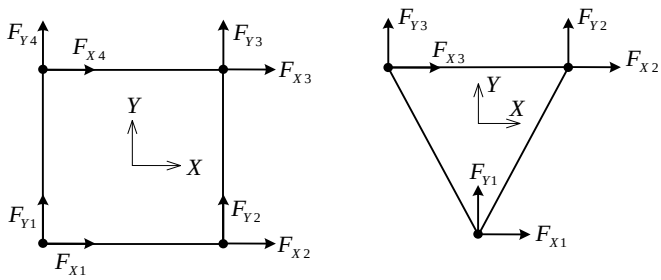


Figure 4.1.3 Sign convention and nodal forces of plane strain elements

4.1.5.2 Element Stress and Strain Results

The stresses and strains of an element are converted into the nodal results by extrapolating the stresses calculated at the integration points (Gauss Points).

The results at the center point of an element are also produced by using the average of the values at the integration points.

Figure 4.1.4 shows the sign convention for element stresses and strains. The arrows represent the positive (+) directions.

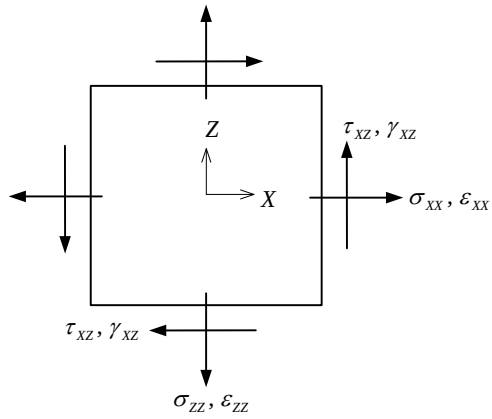


Figure 4.1.4 Sign convention for stresses and strains of a plane strain element

The stress results for the plain strain elements are generated as follows:

σ'_{XX} : Effective normal stress in the GCS X direction

σ'_{YY} : Effective normal stress in the GCS Y direction

σ'_{ZZ} : Effective normal stress in the GCS Z direction

σ_{XX} : Total normal stress in the GCS X direction

σ_{YY} : Total normal stress in the GCS Y direction

σ_{ZZ} : Total normal stress in the GCS Z direction

$\tau_{XZ} = \tau_{ZX}$: Shear stress in the GCS X-Z plane

$\sigma_1,\ \sigma_2,\ \sigma_3$: Effective principal stresses in the directions of principal axes, 1, 2 and 3

τ_{max} : Tresca (maximum shear stress) = $\max\left[\frac{\sigma_3-\sigma_2}{2}, \frac{\sigma_2-\sigma_1}{2}, \frac{\sigma_3-\sigma_1}{2}\right]$

σ_{vm} : von Mises stress = $\sqrt{\frac{1}{2}\left[(\sigma_1-\sigma_2)^2-(\sigma_2-\sigma_3)^2-(\sigma_3-\sigma_1)^2\right]}$

σ'_{mean} : Mean effective stress = $\frac{\sigma'_{XX}+\sigma'_{YY}+\sigma'_{ZZ}}{3}$

σ_{mean} : Mean total stress = $\frac{\sigma_{XX}+\sigma_{YY}+\sigma_{ZZ}}{3}$

The strain results for the plain strain elements are generated as follows:

ε_{XX} : Normal strain in the GCS X direction

ε_{YY} : Normal strain in the GCS Y direction

ε_{ZZ} : Normal strain in the GCS Z direction

$\gamma_{XZ} = \gamma_{ZX}$: Shear strain in the GCS X-Z plain

$\varepsilon_1,\ \varepsilon_2,\ \varepsilon_3$: Principal strains in the directions of principal axes, 1, 2 and 3

γ_{max} : Tresca (maximum shear strain) = $\max\left[\frac{\varepsilon_3-\varepsilon_2}{2},\ \frac{\varepsilon_2-\varepsilon_1}{2},\ \frac{\varepsilon_3-\varepsilon_1}{2}\right]$

ε_{dev} : Deviatoric strain = $\sqrt{\frac{2}{9}\left\{(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2\right\}}$

ε_{vol} : Volumetric strain = $\varepsilon_{XX}+\varepsilon_{YY}+\varepsilon_{ZZ}$

ε^p_{XX} : Plastic normal strain in the GCS X direction

ε^p_{YY} : Plastic normal strain in the GCS Y direction

ε^p_{ZZ} : Plastic normal strain in the GCS Z direction

$\gamma^P_{XZ} = \gamma^P_{ZX}$: Plastic shear strain in the GCS X-Z plain

$\varepsilon^p_1,\ \varepsilon^p_2,\ \varepsilon^p_3$: Principal plastic strains in the directions of principal axes, 1, 2 and 3

ε^p : Equivalent plastic strain = $\sqrt{\frac{2}{3}\left(\varepsilon_1^{p2}+\varepsilon_2^{p2}+\varepsilon_3^{p2}\right)}$

4.2 Truss element

4.2.1 Overview

The truss element is a structural element, which is used to simulate anchors, nails, rock bolts and piles with neglected bending behavior. The truss element is generally used for modeling space trusses and/or diagonal braces and can be used for both static (linear & nonlinear) and dynamic analyses.

SoilWorks supports only a low order 2-node truss element.

Since a truss element retains no rotational stiffness or rotational degrees of freedom at the end nodes, singularity may occur in analysis when other truss elements or the elements having no rotational degrees of freedom are connected to a truss element. SoilWorks automatically restrains the rotational degrees of freedom at such a node to prevent singular errors.

Singular errors do not occur when truss elements are connected to beam elements having rotational degrees of freedom.

4.2.2 Element Shape, ECS, Nodal Connectivity & Degrees of Freedom

The element shape and element coordinate system are given in Figure 4.2.1. The truss element is a structural element, and the element's local x-axis (axial direction) is the only direction of interest in analysis and results, unlike ground elements. The element local x-axis is assumed to orient in the direction from the node 1 to node 2.

A truss element retains only one translational degree of freedom in the direction of the ECS x-axis. When a truss element is inclined on a 2-dimensional plane, the boundary condition should be defined in the direction of the inclined ECS x-axis to obtain accurate results. The boundary condition in the element's x-axis direction can be defined by aligning the node local coordinate system with the element's ECS x-axis.

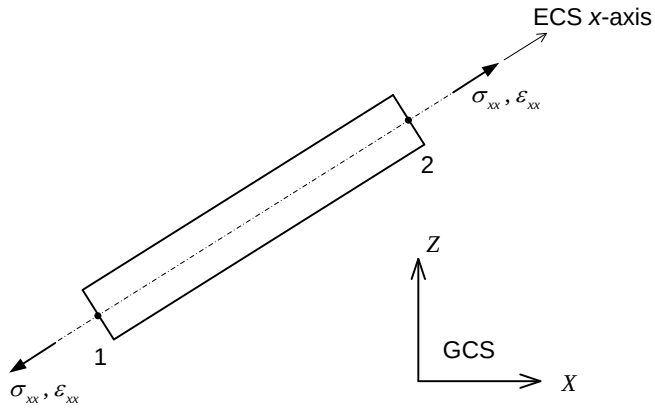


Figure 4.2.1 Element Coordinate System of a truss element and the sign convention for Stress/Strain

4.2.3 Element Material Properties

A truss element being a structural element, its linear elastic material properties are defined as follows:

Modulus of Elasticity	:	E
Poisson's Ratio	:	ν
Weight Density	:	γ_t
Thermal Coefficient	:	α

Truss elements only transmit axial forces. Truss elements can be assigned tension-only (with a gap distance), compression-only (with a hook distance) and nonlinear elastic properties.

The constitutive equation for a truss element is identical to Eq. (4.2.1), and an isotropic linear elastic material can be expressed in a matrix form as,

$$\sigma_{xx} = E\varepsilon_{xx} \tag{4.2.1}$$

The structural properties of a truss element are a cross sectional area and spacing.

SoilWorks' Section Library supports various shapes. Simply entering the section dimensions will automatically calculate the cross sectional area. The prismatic section of a constant cross sectional area is assumed along the length. SoilWorks supports the following section shapes:

Solid Round :
Solid Rectangle :
Pipe :
Box :
H-Section :
T-Section :
Rebar : Cross sectional area defined by the user
Strand : Cross sectional area defined by the user
User-defined section : Cross sectional area defined by the user

The spacing feature enables the user to reflect the effect of three-dimensional spatial reinforcement. For example, when two truss elements are placed per meter in the thickness direction in a two-dimensional model, the spacing becomes 0.5 and the stiffness of the truss reinforcement in a unit thickness becomes doubled. In analysis, the stiffness of a unit thickness is used. The truss forces in the results are however produced for each truss reinforcement element.

4.2.4 Finite Element Formulation

A truss element retains only the translational displacement, u, in the direction of ECS x-axis.

$$\mathbf{u}_i = \{u_i\} \tag{4.2.2}$$

A specific coordinate *x* and the translational displacement *u* are expressed in Eq. (4.2.3) and (4.2.4), using the shape functions.

$$x = \sum_{i=1}^{2} N_i x_i \ , \quad u = \sum_{i=1}^{2} N_i u_i \tag{4.2.3}$$

$$N_1 = \frac{1-\xi}{2}, \quad N_2 = \frac{1+\xi}{2} \quad \left(-1 \le \xi \le 1\right) \tag{4.2.4}$$

The nodal displacement can be related to strain by $\mathbf{B}_i$, as shown in Eq. (4.2.5).

$$\boldsymbol{\varepsilon} = \sum_{i=1}^{2} \mathbf{B}_i \mathbf{u}_i \tag{4.2.5}$$

The $\mathbf{B}_i$ matrix is the derivative of the shape functions, expressed in Eq. (4.2.6) as,

$$\mathbf{B}_i = \left\{ \frac{\partial N_i}{\partial x} \right\} \tag{4.2.6}$$

The element stiffness matrix related to the axial deformation can be expressed using the $\mathbf{B}_i$ matrix as follows:

$$\mathbf{K}_{ij} = \int_{L_e} \mathbf{B}_i^T \mathbf{D} \mathbf{B}_j \, dL \ , \quad \mathbf{D} = A\left\{E\right\} \tag{4.2.7}$$

where,

A : Cross sectional area

L_e : Element length

Rearranging Eq. (4.2.7), the stiffness matrix of a truss element can be calculated as,

$$\mathbf{K} = \frac{EA}{L_e} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{4.2.8}$$

4.2.5 Internal Element Results

The analysis results of truss elements are element internal forces, stresses and strains at Nodes I and J in the element coordinate system.

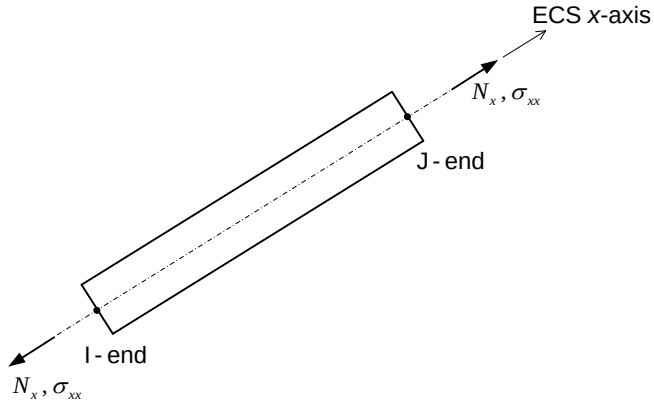


Figure 4.2.2 Output locations and results for truss elements

4.2.5.1 Element Member Force Results

The element force result is the axial force, $N_x(=A\sigma_{xx})$. The sign convention is shown in Figure 4.2.2 where a positive (+) stress is in tension. The element internal forces at both ends are generally equal unless selfweight is assigned.

Fx(N_x) : Axial force in the ECS x direction

4.2.5.2 Element Stress and Strain Results

The element stress and strain are constant along the length since the truss element is a 2-node element and the shape functions are assumed to be linear. Therefore, the stress and strain at a specific point can represent the stress and strain values of the entire

element. The axial stress, σ_{xx}, total axial strain, ε_{xx}, and depending on the material type plastic axial strain, ε_{xx}^{p} can be generated.

Sx (σ_{xx}) : Axial stress in the ECS x -direction

Ex (ε_{xx}) : Total axial strain in the ECS x -direction

PE (ε_{xx}^{p}) : Total axial plastic strain in the ECS x -direction

4.3 Embedded Truss Element

4.3.1 Overview

An embedded truss element is used for the elements with the bending behavior neglected, such as an anchor, nail and rockbolt. The input data (element shape, element coordinate system, material properties, etc.) are identical to those for the truss element. In addition, tension only and compression only behaviors having an allowable load with a hook distance or a gap distance and nonlinear elastic properties can be assigned to an embedded truss element.

The nodes of a truss element always must be coincident with the nodes of adjoining elements such as 2D plain strain elements. The embedded truss element, on the other hand, need not have coincident nodes with the nodes of adjoining elements as long as the element is in contact with ground elements such as plain strain elements or tunnel elements such as beam elements.

4.3.2 Finite Element Formulation

Mother elements representing the ground, which can be used with embedded truss elements are plain strain elements as follows:

Plane strain element

- ✓ 3-node/6-node triangular element
- ✓ 4-node/8-node quadrilateral element

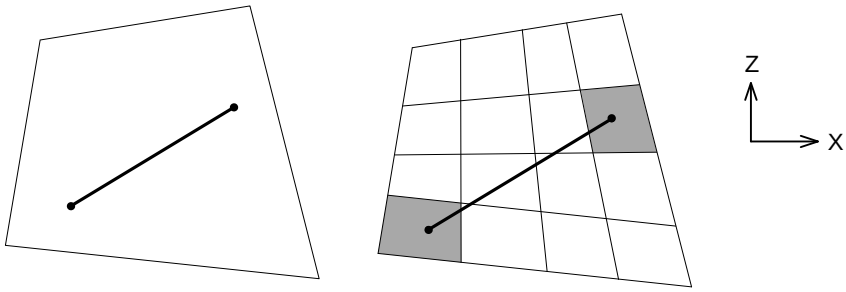


Figure 4.3.1. Embedded truss element in mother elements

An embedded truss element is always placed within the mother elements as shown in Figure 4.3.1. The equations for the embedded truss element use a subscript “tru”, and the mother elements use a subscript “mot.”

In the element coordinate system of the mother elements, u and v are the translational displacements in the local x and y directions respectively.

$$\mathbf{u}^i_{mot} = \left\{ u^i_{mot},\ v^i_{mot} \right\} \tag{4.3.1}$$

The matrix of axis transformation between the mother and truss elements is obtained from the directional vector, $\mathbf{d}$.

$$\mathbf{T} = \begin{bmatrix} d_1 & d_2 & 0 & 0 \\ 0 & 0 & d_1 & d_2 \end{bmatrix} \tag{4.3.2}$$

Since a truss is a one-dimensional element, the transformation matrix can comprise only of the directional vector in the element local x-axis.

$$\mathbf{d} = \left\{ d_1,\ d_2 \right\} \tag{4.3.3}$$

The nodal coordinates of a truss element can be defined in the isoparametric coordinate system of the mother elements as follows:

$$\mathbf{a}_{tru}^{i} = \left\{ \xi^{i},\ \eta^{i} \right\} \tag{4.3.4}$$

Accordingly, the translational displacements of the truss element are given in the mother element coordinate system as follows:

$$\begin{aligned} u_{tru} &= \sum_{i=1}^{N_{mot}} N_{mot}^{i}\left(\xi^{i},\ \eta^{i}\right) u_{mot}^{i} \\ v_{tru} &= \sum_{i=1}^{N_{mot}} N_{mot}^{i}\left(\xi^{i},\ \eta^{i}\right) v_{mot}^{i} \end{aligned} \tag{4.3.5}$$

The translational displacements of the truss element in the truss element coordinate system can be obtained from the transformation matrix.

$$\hat{\mathbf{u}}_{tru} = \mathbf{T} \cdot \mathbf{u}_{tru} \tag{4.3.6}$$

The relationship between the nodal displacements, $\hat{\mathbf{u}}_{tru}$ and strains, $\boldsymbol{\varepsilon}_{tru}$ can be expressed by the strain-displacement relationship matrix, $\mathbf{B}_{tru}^{i}$ as follows:

$$\boldsymbol{\varepsilon}_{tru} = \sum_{i=1}^{N_{tru}} \hat{\mathbf{B}}_{tru}^{i}\, \mathbf{u}_{tru}^{i} \tag{4.3.7}$$

The $\mathbf{B}_{tru}^{i}$ matrix is the derivative of the shape functions, which is expressed as,

$$\mathbf{B}_{tru}^{i} = \left[\frac{\partial N_{tru}}{\partial x_{tru}} \right] \tag{4.3.8}$$

The stiffness matrix of the embedded truss element is given as,

$$\mathbf{K}_{ij} = \int_{L_{tru}} \left(\mathbf{B}_{tru}\mathbf{T}\mathbf{N}_{mot}\right)^{T} \mathbf{D}\left(\mathbf{B}_{tru}\mathbf{T}\mathbf{N}_{mot}\right) dL_{tru}\ , \qquad \mathbf{D} = A\left[E\right] \tag{4.3.9}$$

where, A is the cross sectional area, and L_{tru} is the length of the truss element.

4.3.3 Internal Element Results

The analysis results of truss elements are element internal forces, stresses and strains at Nodes I and J in the element coordinate system.

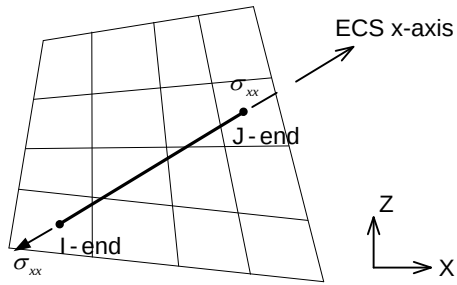


Figure 4.3.2 Output locations and results for embedded truss elements

4.3.3.1 Element Member Force Results

The element force result is the axial force, $N_x(=A\sigma_{xx})$. The sign convention is shown in Figure 4.3.2 where a positive (+) stress is in tension. The element internal forces at both ends are generally equal unless selfweight is assigned.

Fx$\left(N_x\right)$: Axial force in the ECS x -direction

4.3.3.2 Element Stress and Strain Results

The element stress and strain are constant along the length since the truss element is a 2-node element and the shape functions are assumed to be linear. Therefore, the stress and strain at a specific point can represent the stress and strain values of the entire element. The axial stress, σ_{xx}, total axial strain, ε_{xx}, and depending on the material type plastic axial strain, ε_{xx}^{p} can be generated.

Sx (σ_{xx}) : Axial stress in the ECS x -direction

Ex (ε_{xx}) : Total axial strain in the ECS x -direction

PEx (ε_{xx}^{p}) : Total axial plastic strain in the ECS x -direction

4.4 Beam Element

4.4.1 Overview

A beam element is a structural element, which is mainly used to model a long member relative to the section dimensions and is subjected to bending. Beam elements are mostly used for modeling structural members. In geotechnical engineering, beam elements can be used to model underground structures such as retaining walls, piles, tunnel lining, culverts etc. Since a beam element retains 6 degrees of freedom at a node, the element can be used as a load-transfer element connecting other elements having different degrees of freedom.

The beam element is formulated on the basis of the Timoshenko beam theory (the cross sectional plane initially normal to the neutral axis of the beam remains plane after deformation but not necessarily normal to the neutral axis in the deformed state), and shear deformations are taken into account. When zero is entered for shear areas, the corresponding shear deformations of the beam element are ignored.

If the aspect ratio of the section depth to the length is greater than 1/5, shear deformations becomes significant. In such a case, it is recommended that fine mesh consisted of plane strain elements of elastic material properties be used.

The torsional resistance of a beam element differs from the sectional polar moment of inertia (they are the same for circular and cylindrical sections). Caution should be exercised when the effects of torsional deformation are large since the torsional resistance is generally determined by experimental methods.

When elements with different DOF adjoin one another, the use of rigid beam elements is effective. In order to avoid numerical errors, the stiffness of the rigid beam elements is generally set to 105~108 times the stiffness of the adjoining elements.

4.4.2 Element Shape, ECS & Degrees of Freedom

The beam element retains three translational and three rotational degrees of freedom per each node in the element coordinate system. The local x-axis is defined in the direction from Node i to Node j. The local z-axis is defined in the direction of the global Z-axis. The local y-axis is defined using the right hand rule. If the element x-axis is parallel with the global Z-axis, the local y-axis is defined in the direction parallel with the global Y-axis, and the local z-axis is defined using the right hand rule.

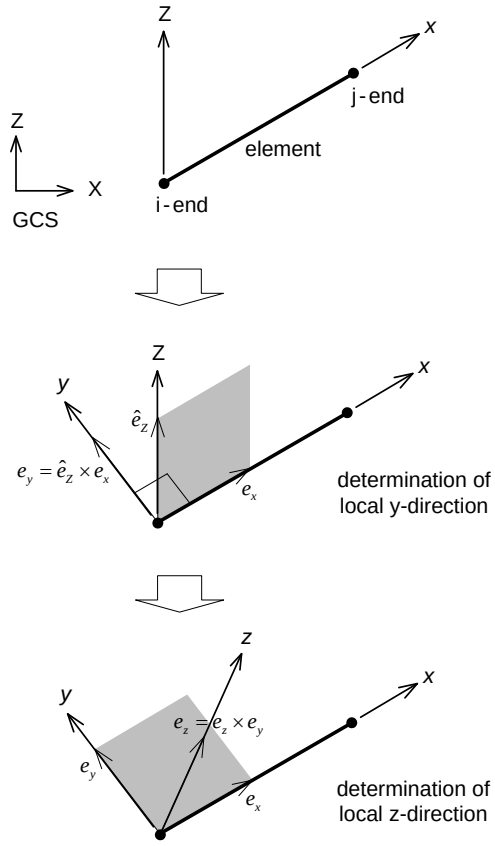


Figure 4.4.1 Definition of Beam Element Coordinate System

4.4.3 Element Material Properties

A beam element being a structural element, its linear elastic material properties are defined as follows:

Modulus of Elasticity	:	E
Poisson's Ratio	:	ν
Weight Density	:	γ_t
Thermal Coefficient	:	α

The structural properties of a beam element are a cross sectional area and spacing. SoilWorks' Section Library supports various shapes. Simply entering the section dimensions will automatically calculate the section properties listed below.

Cross section area	:	A
Torsional constant	:	I_x
Moment of inertia about local y-axis	:	I_y
Moment of inertia about local z-axis	:	I_z
Effective shear area for local y-axis	:	A_y
Effective shear area for local z-axis	:	A_z

The spacing feature is identical to that of the truss element.

4.4.4 Finite Element Formulation

The beam element in SoilWorks is formulated on the basis of the Timoshenko beam theory, which renders the stiffness for tension, compression, shear, bending and torsion.

For the sake of convenience, the finite element formulation of the beam element is explained for the 2-dimensional behavior on the X-Z plane.

The deformation of a beam element is independently assumed as follows:

$$u = z\beta_y(x) \quad \text{and} \quad w = w(x) \tag{4.4.1}$$

where,

u	: Displacement in the x direction
z	: Coordinate in the z direction
β_y	: Angle of rotation about y-axis
w	: Displacement in the z direction

The strain and curvature are expressed as,

$$\begin{aligned} \varepsilon_b &= \varepsilon_x = \frac{\partial u}{\partial x} = z\frac{\partial \beta_y}{\partial x} = z\kappa_y \\ \gamma_s &= \gamma_{xz} = \frac{\partial w}{\partial x} + \beta_y \end{aligned} \tag{4.4.2}$$

where,

ε_b	: Axial strain due to bending
γ_s	: Shear strain
κ_y	: Curvature about y-axis

The moment - curvature relationship and the shear force - shear strain relationship are as follows:

$$\begin{aligned} M_y &= D_b\kappa_y = EI\frac{\partial \beta_y}{\partial x} \\ V_y &= D_s\overline{\gamma}_{xz} = GA_s\overline{\gamma}_{xz} \end{aligned} \tag{4.4.3}$$

where,

D_b	: Bending stiffness
D_s	: Shear stiffness
E	: Modulus of elasticity
I	: Moment of inertia

G : Shear modulus

A_s : Shear area

The shape functions of a 2-node beam element are assumed as follows:

$$\beta_y = N_1\beta_{y1} + N_2\beta_{y2} + N_3\Delta\beta_{y3}$$
$$N_1 = \frac{1-\xi}{2}, \quad N_2 = \frac{1+\xi}{2}, \quad N_3 = 1-\xi^2 \tag{4.4.4}$$

Using the shape functions in Eq. (4.4.4), the curvature in Eq. (4.4.2) can be redefined as,

$$\begin{aligned}
\kappa_y &= \frac{\partial N_1}{\partial x}\beta_{y1} + \frac{\partial N_2}{\partial x}\beta_{y2} + \frac{\partial N_3}{\partial x}\Delta\beta_{y3} \\
&= J^{-1}\left(\frac{\partial N_1}{\partial \xi}\beta_{y1} + \frac{\partial N_2}{\partial \xi}\beta_{y2} + \frac{\partial N_3}{\partial \xi}\Delta\beta_{y3}\right) \\
&= J^{-1}\begin{bmatrix} 0 & \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} \end{bmatrix}\begin{Bmatrix} w_1 \\ \beta_{y1} \\ w_2 \\ \beta_{y2} \end{Bmatrix} + J^{-1}\frac{\partial N_3}{\partial \xi}\Delta\beta_{y3} \\
&= \mathbf{B}_{b\beta}\mathbf{u} + B_{b\Delta\beta}\Delta\beta_{y3}
\end{aligned} \tag{4.4.5}$$

where,

$J = \dfrac{\partial x}{\partial \xi} = \dfrac{l}{2}$ (Jacobian operator)

Also the shear strain field is calculated as follows:

$$\overline{\gamma}_{xz} = \frac{V_y}{D_s} = \frac{1}{D_s}\frac{\partial M_y}{\partial x} = \frac{D_b}{D_s}\frac{\partial^2 \beta_y}{\partial x^2} \tag{4.4.6}$$

The angle of rotation, β_y , is now assumed as a second order polynomial equation of the element length.

$$\beta_y = \left(1-\frac{x}{l}\right)\beta_{y1} + \frac{x}{l}\beta_{y2} + 4\frac{x}{l}\left(1-\frac{x}{l}\right)\Delta\beta_{y3} \tag{4.4.7}$$

By substituting Eq. (4.4.7) into Eq. (4.4.6), the shear strain field becomes as follows:

$$\overline{\gamma}_{xz} = B_{s\Delta\beta}\Delta\beta_{y3} = -\frac{2}{3}\phi_3\Delta\beta_{y3}, \quad \phi_3 = \frac{D_b}{D_s}\frac{12}{l^2} = \frac{12EI}{GA_s l^2} \tag{4.4.8}$$

However, the shear strain field must satisfy the following constraint condition:

$$\int_0^l \left(\gamma_{xz} - \overline{\gamma}_{xz}\right) dx = 0 \tag{4.4.9}$$

Therefore, the conditional Eq. (4.4.10) can be established.

$$w_2 - w_1 + \frac{l}{2}\beta_{y1} + \frac{l}{2}\beta_{y2} + \frac{2}{3}l\left(1+\phi_3\right)\Delta\beta_{y3} = 0 \tag{4.4.10}$$

Now, the virtual angle of rotation at the center can be expressed in terms of the remaining nodal angles of rotation.

$$\begin{aligned}
&\Delta\beta_{y3} = \mathbf{Au} = \frac{1}{\frac{2}{3}l\left(1+\phi_3\right)}\begin{bmatrix} 1 & -\frac{l}{2} & -1 & -\frac{l}{2} \end{bmatrix}\begin{Bmatrix} w_1 \\ \beta_{y1} \\ w_2 \\ \beta_{y2} \end{Bmatrix} \\
&\mathbf{A} = \mathbf{A}_{\Delta\beta}^{-1}\mathbf{A}_w \\
&\mathbf{A}_{\Delta\beta} = A_{\Delta\beta} = \frac{2}{3}l\left(1+\phi_3\right), \quad \mathbf{A}_w = \begin{bmatrix} 1 & -\frac{l}{2} & -1 & -\frac{l}{2} \end{bmatrix}
\end{aligned} \tag{4.4.11}$$

From Eq. (4.4.5) and Eq. (4.4.8), the curvature and the shear strain field can be expressed as,

$$\begin{aligned}\kappa_y &= \mathbf{B}_{b\beta}\mathbf{u} + B_{b\Delta\beta}\mathbf{A}\mathbf{u} = \left(\mathbf{B}_{b\beta} + B_{b\Delta\beta}\mathbf{A}\right)\mathbf{u} = \mathbf{B}_b\mathbf{u} \\ \overline{\gamma}_{xz} &= B_{s\Delta\beta}\mathbf{A}\mathbf{u} = \left(\mathbf{B}_{s\Delta\beta}\mathbf{A}\right)\mathbf{u} = \mathbf{B}_s\mathbf{u}\end{aligned} \tag{4.4.12}$$

where,

$\mathbf{B}_b$: Curvature-Displacement matrix

$\mathbf{B}_s$: Shear strain field - Displacement matrix

The stiffness matrix is expressed as follows:

$$\begin{aligned}\mathbf{K} &= \mathbf{K}_b + \mathbf{K}_s \\ \mathbf{K}_b &= \int_0^l \mathbf{B}_b^T D_b \mathbf{B}_b dx, \quad \mathbf{K}_s = \int_0^l \mathbf{B}_s^T D_s \mathbf{B}_s dx\end{aligned} \tag{4.4.13}$$

The above integration can be algebraically calculated, and the final beam stiffness is as follows:

$$\mathbf{K} = \frac{EI}{1+\phi_3}\begin{bmatrix} \frac{12}{l^3} & -\frac{6}{l^2} & -\frac{12}{l^3} & -\frac{6}{l^2} \\ & \frac{4}{l}\left(1+\frac{\phi_3}{4}\right) & \frac{6}{l^2} & \frac{2}{l}\left(1-\frac{\phi_3}{2}\right) \\ & & \frac{12}{l^3} & \frac{6}{l^2} \\ \text{Sym.} & & & \frac{4}{l}\left(1+\frac{\phi_3}{4}\right) \end{bmatrix} \tag{4.4.14}$$

The above formulation of a beam element is based on the element coordinate system coinciding with the x, y & z axes. Therefore, it is necessary to transform the matrix into the global coordinate system prior to combining the stiffness matrices.

4.4.5 Internal Element Results

The analysis results of beam elements are the element member forces and cross sectional stresses.

4.4.5.1 Element Member Force Results

The sign convention for the element internal forces is given in Figure 4.4.2. The arrow directions represent the positive (+) directions.

Nx : Axial force in the ECS x-axis direction
Qy, Qz : Shear forces in the ECS y-axis & z-axis directions
Mx : Torsional moment about the ECS x-axis
My, Mz : Bending moments about the ECS y-axis & z-axis

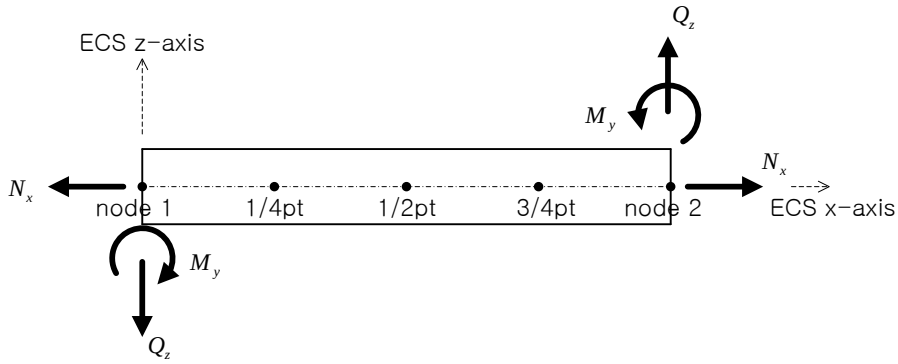
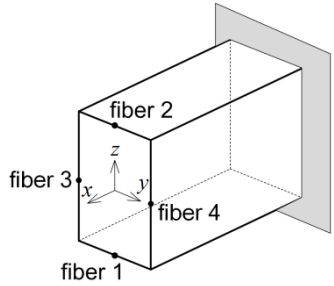


Figure 4.4.2 Element coordinate system of beam element and the sign convention for element internal forces and stresses

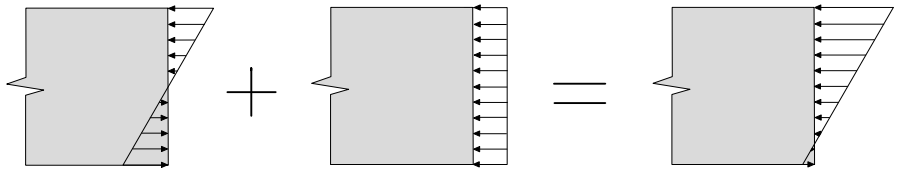
4.4.5.2 Element Stress Results

The sign convention for the internal stresses, except for the bending moment stresses, is the same as for the internal forces. The bending moment stresses are positive (+) for tension and negative (-) for compression. The stresses are produced at the outer-most fibers 1, 2, 3 and 4 as shown in Figure 4.4.3 (a). The normal stresses acting on the cross

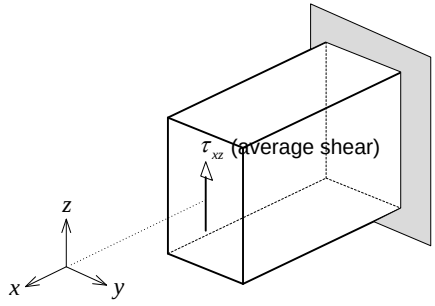
section of a beam are expressed as the combination of the bending and axial stresses as shown in Figure 4.4.3 (b). The shear stress in each direction is the average of the stresses across the section as shown in Figure 4.4.3 (c).



(a) Locations of stress output



(b) concept of combined bending and axial stresses



(c) shear stress

Figure 4.4.3 Beam element stresses

Fx	: Axial force
Fz	: Shear stress in the ECS z-axis direction
My	: Bending moment about ECS y-axis
Sax	: Axial stress in the ECS x-axis direction (σ_{xx})
Ssz	: Shear stress in the ECS z-axis direction (τ_{xz})
Bending Sbz+	: Positive stress due to bending moment (M_y) about the ECS y-axis [location 2]
Bending Sbz-	: Negative stress due to bending moment (M_y) about the ECS y-axis [location 1]
Combined Max	: Maximum & Minimum combined stresses among the combined stresses at 1, 2, 3 and 4
Combined 1	: Combined stress at 1 (axial stress + bending stress)
Combined 2	: Combined stress at 2 (axial stress + bending stress)

4.5 Interface Element

4.5.1 Overview

SoilWorks uses the interface element proposed by Goodman et al. to allow slippage between dissimilar materials or between elements having a significant difference in stiffness.

This type of modeling is possible by using general isoparametric elements with the selection of appropriate stiffness. However, the interface element enables the user to use a thin element without having to discretize the mesh.

The shear stress at the interface element is limited by the shear strength at the interface defined by the user. The yield criterion of the Coulomb friction is used to set the shear strength of the interface. If the shear stress exceeds the maximum shear strength, plastic flow takes place on the basis of the plastic theory.

Also, the presence of tensile stress in the interface element is checked. When the interface element is subjected to tension, the normal and shear stiffness are set to zero, and the interface is assumed to be incapable of resisting external forces simulating discontinuity.

The nodal coordinates of the interface element are the same for the two rows of nodes along the length dimension of the element. The number of degrees of freedom is identical to that of the plane elements connected at each node.

The interface element follows the general finite element formulation, but with the element thickness being equal to zero. When defining the interface element of zero thickness numerically, the penalty stiffness rule is applied. Large penalty stiffness may result in numerical instability, whereas very small penalty stiffness can produce inaccurate relative displacements of the interface element. Thus, an appropriate penalty stiffness value is required. SoilWorks recommends a penalty stiffness value of k = 10 x E/dv in which $\mathbf{E}$ is the smallest Young's modulus among the elements in the model, and $\mathbf{d}_v$ is the nonzero virtual thickness of the element. The nonzero virtual thickness of the element used to define the penalty stiffness generally ranges from 0.1 to 1. The nonzero thickness of the element is not the real thickness, but is used in the concept of a scale factor to control the unit and magnitude of the penalty stiffness. The reason for this is that the unit for the

elastic modulus is N/m^2, while the unit for the penalty stiffness is N/m^3.
For the purpose of numerical analysis, the relationship between the relative displacement, $\Delta\mathbf{u}$ and the traction, $\mathbf{t}$, on the basis of the penalty rule can be defined as the constitutive equation, Eq. (4.5.1).

$$\mathbf{t} = \mathbf{D} \cdot \Delta\mathbf{u} \tag{4.5.1}$$

$\mathbf{t}$, $\mathbf{D}$ and $\Delta\mathbf{u}$ are defined below, and the relative displacement and the traction force at an integration point are illustrated in Figure 4.5.1.

$$\mathbf{t} = \begin{Bmatrix} t_x \\ t_y \end{Bmatrix}, \quad \mathbf{D} = \begin{bmatrix} k_x & 0 \\ 0 & k_y \end{bmatrix}, \quad \Delta\mathbf{u} = \begin{Bmatrix} \Delta u_x \\ \Delta u_y \end{Bmatrix} \tag{4.5.2}$$

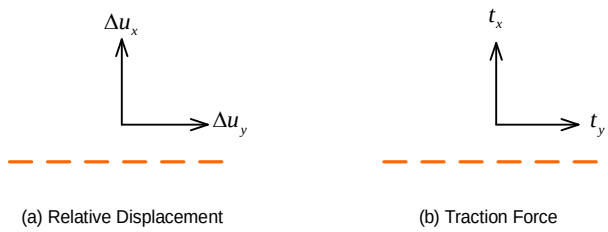


Figure 4.5.1 : Relative displacement and traction force on a linear interface element

where,

Dotted line: Interface

t_x : Normal traction [N/m^2]

t_y : Tangential traction [N/m^2]

Δu_x : Relative displacement in the normal direction [m]

Δu_y : Relative displacement in the tangential direction [m]

The relative displacement and the traction in Eq. (4.5.2) do not have any directional relationship. For example, the normal traction does not influence the tangential traction and vice versa.

4.5.2 ECS, Element Shape & Degrees of Freedom

SoilWorks provides linear interface elements of low order 4-node and high order 6-node elements as shown in Figure 4.5.3. The linear interface elements are used to describe opening/closing and sliding motions between plane elements or between plane and line elements.

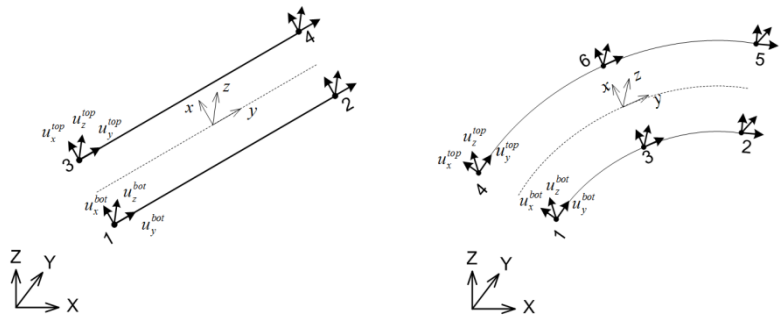


Figure 4.5.3 Linear Interface Elements

4.5.3 Element Properties

For the structural property of a linear interface element, a unit thickness of 1 is used in addition to the following stiffness coefficients:

k_x : Stiffness coefficient in the ECS x-axis direction (in normal direction)

k_y : Stiffness coefficient in the ECS y-axis direction (in tangential direction)

4.5.4 Finite Element Formulation

The tractions in ECS can be derived from the relative displacements in ECS, $\Delta\mathbf{u}_x$ and $\Delta\mathbf{u}_y$ and the stiffness matrix $\mathbf{D}$.

$$\begin{Bmatrix} t_x \\ t_y \end{Bmatrix} = \mathbf{D} \begin{Bmatrix} \Delta u_x \\ \Delta u_y \end{Bmatrix} \tag{4.5.3}$$

The stiffness matrix can be obtained from the strain energy equation of an interface element by a variational method:

$$\mathbf{K}_{inter} = \int_{\Gamma} \mathbf{B}_{inter}^{T} \mathbf{D} \mathbf{B}_{inter} d\Gamma \tag{4.5.4}$$

In addition, the internal force can be expressed as,

$$\mathbf{F}_{inter} = \int_{\Gamma} \mathbf{B}_{inter}^{T} \mathbf{t} \, d\Gamma \tag{4.5.5}$$

Expanding Eq. (4.5.4) and Eq. (4.5.5) into the form of numerical integration, the following stiffness matrix, Eq. (4.5.4) and the force matrix, (4.5.5) can be expressed.

$$\mathbf{K}_{inter} = \sum_{j=1}^{N_{ip}} \mathbf{B}_{inter}^{j}{}^{T} \mathbf{D} \mathbf{B}_{inter}^{j} \det \mathbf{J}^{j} \mathbf{W}^{j} \tag{4.5.6}$$

$$\mathbf{F}_{inter} = \sum_{j=1}^{N_{ip}} \mathbf{B}_{inter}^{j} \mathbf{t} \det \mathbf{J}^{j} \mathbf{W}^{j} \tag{4.5.7}$$

where,

N_{ip} : Number of integration points for the interface element

$\mathbf{B}_{inter}$: Matrix of the relationship between the relative displacement and element displacement at the time of calculating the internal force

The constitutive relationship of the linear interface element defining the stiffness matrix, $\mathbf{D}$ is identical to Eq. (4.5.2).

The global coordinates of a node of a linear interface element can be defined using the shape functions (N_i) below.

$$
\begin{aligned}
x^{bot} &= N_1^{bot} \cdot x_1^{bot} + N_2^{bot} \cdot x_2^{bot} \ (+N_5^{bot} \cdot x_5^{bot}) \\
x^{top} &= N_3^{top} \cdot x_3^{top} + N_4^{top} \cdot x_4^{top} \ (+N_6^{top} \cdot x_6^{top}) \\
y^{bot} &= N_1^{bot} \cdot y_1^{bot} + N_2^{bot} \cdot y_2^{bot} \ (+N_5^{bot} \cdot y_5^{bot}) \\
y^{top} &= N_3^{top} \cdot y_3^{top} + N_4^{top} \cdot y_4^{top} \ (+N_6^{top} \cdot y_6^{top})
\end{aligned}
\tag{4.5.8}
$$

The coordinates in the parentheses above represent a higher order element. Also the total displacements at a node can be expressed as,

$$
\begin{aligned}
u^{bot} &= N_1^{bot} \cdot u_1^{bot} + N_2^{bot} \cdot u_2^{bot} \ (+N_5^{bot} \cdot u_5^{bot}) \\
u^{top} &= N_3^{top} \cdot u_3^{top} + N_4^{top} \cdot u_4^{top} \ (+N_6^{top} \cdot u_6^{top}) \\
v^{bot} &= N_1^{bot} \cdot v_1^{bot} + N_2^{bot} \cdot v_2^{bot} \ (+N_5^{bot} \cdot v_5^{bot}) \\
v^{top} &= N_3^{top} \cdot v_3^{top} + N_4^{top} \cdot v_4^{top} \ (+N_6^{top} \cdot v_6^{top})
\end{aligned}
\tag{4.5.9}
$$

The isoparametric shape functions for the linear interface element are expressed as Eq. (4.5.10).

$$
\begin{aligned}
N_1^{bot}(\xi) &= N_3^{top}(\xi) = \frac{1}{2}(1-\xi) \\
N_2^{bot}(\xi) &= N_4^{top}(\xi) = \frac{1}{2}(1+\xi)
\end{aligned}
\tag{4.5.10}
$$

For the higher order interface element,

$$
\begin{aligned}
N_1^{bot}(\xi) &= N_3^{top}(\xi) = -\frac{1}{2}(1-\xi)\xi \\
N_2^{bot}(\xi) &= N_4^{top}(\xi) = \frac{1}{2}(1+\xi)\xi \\
N_5^{bot}(\xi) &= N_6^{top}(\xi) = \left(1-\xi^2\right)
\end{aligned}
\tag{4.5.11}
$$

(Nodes 5 and 6 are higher order nodes of the linear interface element.)

The integration points are placed at the interface in between the top and bottom elements in SoilWorks. Since the Newton-Cotes integration method is used, the integration points

exist at the nodes.
If the Gauss integration method is used for interface elements, the integration points are located within the elements. This leads to inability to accurately simulate the nodal behavior of the interface, and the result tends to oscillate. Therefore, the Newton-Cotes method is used to accurately simulate the behavior.

4.5.5 Internal Element Results

The results of interface elements are tractions, relative displacements and plastic relative displacements as follows:

- ✓ **Tractions**

  t_x : Traction in the ECS x-axis direction (in normal direction)

  t_y : Traction in the ECS y-axis direction (in tangential direction)

- ✓ **Relative Displacements**

  Δu_x : Relative displacement in the ECS x-axis direction (in normal direction)

  Δu_y : Relative displacement in the ECS y-axis direction (in tangential direction)

  Δu_x^p : Plastic relative displacement in the ECS x-axis direction (in normal direction)

  Δu_y^p : Plastic relative displacement in the ECS y-axis direction (in tangential direction)

4.6 Pile/Anchor/Nail/Rockbolt Elements

4.6.1 Overview

The pile element in SoilWorks is convenient to model a pile-soil interface because coincident nodal connectivity is no longer required. The pile element is created with a beam element embedded in the mother (ground) elements. Furthermore, linear interface elements are inserted in between the beam element nodes and the virtual nodes embedded in the ground elements to simulate the nonlinear behavior such as frictional slip between the pile and soil.

This element can be also used for anchors, nails and rockbolts. The difference between the general pile and anchor/nail/rockbolt elements is that only the material behaviors defined at the interfaces simply exist. In order to avoid the complexity of terminology, the elements will be referred to as pile elements hereafter.

The pile element in SoilWorks refers to a linear interface element, which inter-links the motions between the beam element and the ground as shown in Figure 4.6.1.

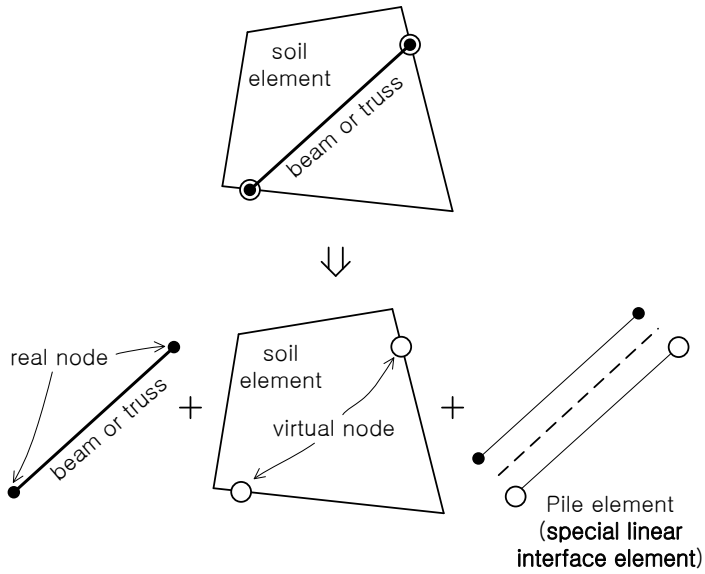


Figure 4.6.1 Pile element

4.6.2 Element Shape, ECS & Degrees of Freedom

The pile element is a special form of a linear interface element, which is created between the real nodes of a beam element and the virtual nodes of a soil element as shown in Figure 4.6.1. SoilWorks supports the 4-node low order linear interface element consisted of the two real and two virtual nodes.

The element local x-axis is defined in the direction from the i-end to j-end as shown in Figure 4.6.2. The ECS y-axis is defined in the direction of the global Z-axis from the ECS x-axis. The z-axis is defined using the right hand rule. If the element x-axis is parallel with the global Z-axis, the element y-axis is defined in the direction parallel with the global X-axis, and the element z-axis is defined using the right hand rule. The element DOF is given in Figure. 4.6.2.

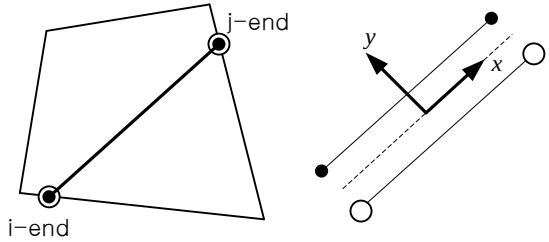


Figure 4.6.2 Coordinate System of Pile Element

4.6.3 Element Properties

In case a rigid body behavior is assumed for the pile element, the following material properties are required:

k_x : Stiffness coefficient in the ECS x-axis direction (in normal direction)

k_y : Stiffness coefficient in the ECS y-axis direction (in tangential direction)

Apart from the rigid body behavior, strength and stiffness can be assigned to vary with depth in the behavior of a pile element. Refer to the Material manual for details.

4.6.4 Finite Element Formulation

The pile elements can be embedded in the following low/high order plane strain elements:

- ✓ 3-node/6-node triangular elements
- ✓ 4-node/8-node rectangular elements

Pile elements can be modeled with the following linear elements:

- ✓ 2-node truss elements
- ✓ 2-node beam elements

The global coordinate system for interface elements is as follows:

$$\mathbf{X} = \{X \quad Z\} \tag{4.6.1}$$

The element local coordinate system for interface elements can be defined using the global coordinate system as follows:

$$\mathbf{x} = \{x_1(X_1, Z_1) \quad x_2(X_2, Z_2)\} \tag{4.6.2}$$

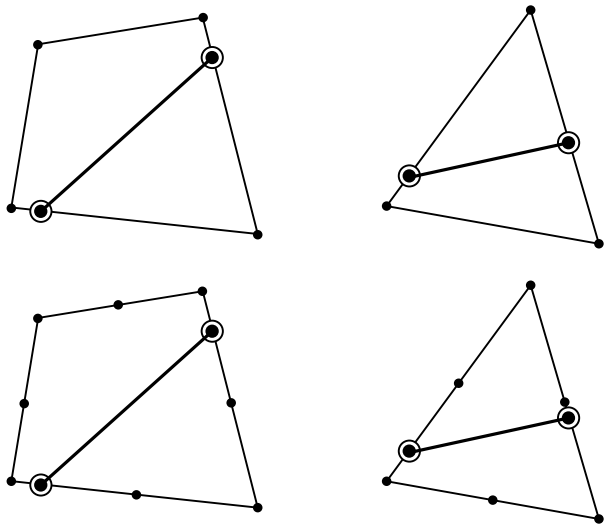


Figure: 4.3.3 Types of Interface Elements between Soil and Linear Elements

The global coordinate system can be expressed as,

$$\mathbf{X} = \mathbf{I} \tag{4.6.3}$$

The local coordinate system can be represented in terms of an orthogonal matrix as,

$$\mathbf{x} = \begin{bmatrix} X_1^1 & X_2^1 \\ X_1^2 & X_2^2 \end{bmatrix} \tag{4.6.4}$$

The nodal coordinates can be represented in the local coordinate system as follows:

$$\begin{aligned} \mathbf{x} &= \left\{ x_1\left(x_1^1, x_2^1\right) \quad x_2\left(x_1^2, x_2^2\right) \right\} \\ &= \begin{bmatrix} X_1^1 & X_2^1 \\ X_1^2 & X_2^2 \end{bmatrix} \left\{ x_1\left(X_1^1, X_2^1\right) \quad x_2\left(X_1^2, X_2^2\right) \right\}^T \end{aligned} \tag{4.6.5}$$

The first order shape function can be represented and written as follows:

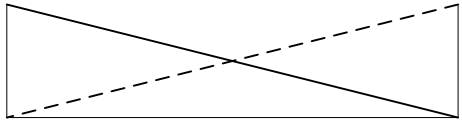


Figure 4.6.4 First Order Shape Function Representation

$$\begin{aligned} N_1 &= \frac{1}{2}(1-\xi) \\ N_2 &= \frac{1}{2}(1+\xi) \end{aligned} \tag{4.6.6}$$

The shape functions at a Gauss integration point is denoted by N_k^l, where k represents the node number, and l represents the integration point number. If there are two integration points, $l = 1, 2$. The nodal coordinates can be expressed as a_i^k, k being the node number and l being the number representing a degree of freedom. In case of a plane strain element, two translational degrees of freedom are used.

$$\mathbf{a}^l = \left\{ \sum_{k=1}^{3} a_1^k N_k^l, \sum_{k=1}^{3} a_2^k N_k^l \right\} \tag{4.6.7}$$

$\mathbf{a}^l$ is the coordinate of the Gauss integration point of a linear element in the element coordinate system. Since the coordinate of the integration point of a linear element is

known in the isoparametric coordinate system of a plane strain element, the coordinate system of the integration point of the linear element can be found in the element coordinate system of the plane strain element using the nodal coordinate of the plane strain element.

The coordinates in the isoparametric coordinate system are given by,

$$\boldsymbol{\alpha}^l = \left\{ \xi^l, \eta^l \right\} \tag{4.6.8}$$

If the base element is a 6-node triangular element, the shape function can be defined by,

$$^6 N_k^l \tag{4.3.8}$$

k represents the index of the shape function, and l represents the index of the integration point. If a 2-node pile element passes through a 6-node triangular element, the relative displacement-displacement matrix can be formulated as follows:

$$\mathbf{B}^l = \begin{bmatrix} -^6 N_1^l & 0 & \cdots & -^6 N_6^l & 0 & N_1^l & 0 & N_2^l & 0 \\ 0 & -^6 N_1^l & \cdots & 0 & -^6 N_6^l & 0 & N_1^l & 0 & N_2^l \end{bmatrix} \tag{4.3.9}$$

The gradient stiffness matrix of a plane strain-beam interface element is expressed as below.

$$\mathbf{K}_t = \sum_{l=1}^{np} \mathbf{B}^{l^T} \mathbf{T} \mathbf{B}^l W_l \det \mathbf{J}_l \tag{4.3.11}$$

where, $\mathbf{K}_t$ is the gradient stiffness, W_l is the weight and $\mathbf{T}$ is the relative displacement-friction matrix.

The types of material models used for pile elements include Linear Elastic Model and Nonlinear Elastic Model. The Relative Displacement-Friction relationship matrix is formulated as follows:

$$\mathbf{D} = \begin{bmatrix} k_x & 0 \\ 0 & k_y \end{bmatrix} \tag{4.3.12}$$

The normal stress (σ) and shear stress (τ) are associated with a constitutive equation for the normal strain and tangent strain. In case of a 3D structure, one normal strain and two tangent strains constitute the relationship.

$$\begin{Bmatrix} t_x \\ t_y \\ t_z \end{Bmatrix} = \mathbf{D} \begin{Bmatrix} \Delta u_x \\ \Delta u_y \\ \Delta u_z \end{Bmatrix} \tag{4.3.13}$$

4.6.5 Internal Element Results

The analysis results of pile elements are traction and relative displacement at the interface.

t_x : Traction in the ECS x-axis direction

t_y : Traction in the ECS y-axis direction

Δu_x : Relative displacement in the ECS x-axis direction

Δu_y : Relative displacement in the ECS y-axis direction

4.7 Pile Tip Bearing Element

4.7.1 Overview

The pile element was introduced on the basis of a linear element-soil interface in the preceding section. The pile element is complemented by adding a point-to-soil interface element at the pile tip. Figure 4.7.1 illustrates how a pile tip bearing is modeled in SoilWorks defining a point-to-soil interface in an element.

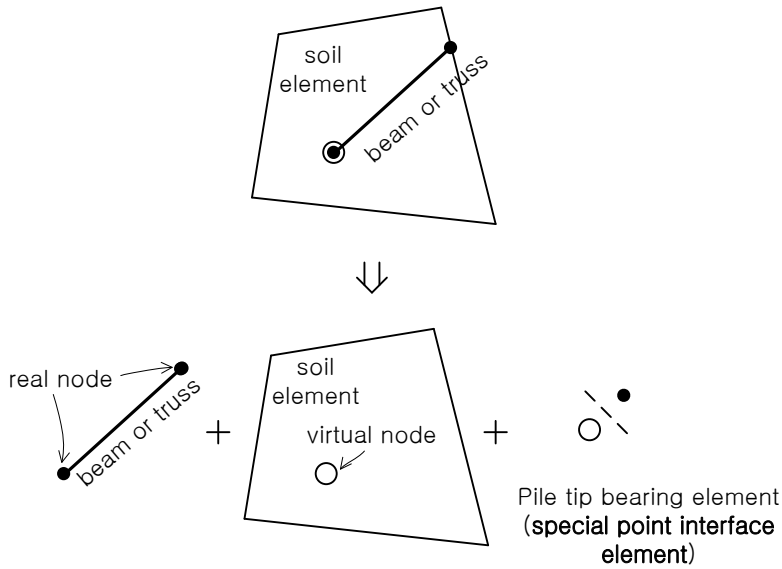


Figure 4.7.1 Conceptual composition of pile tip bearing element

4.7.2 Elements Shape, ECS & Degrees of Freedom

As shown in Figure 4.7.2, the pile tip bearing element can be used in 4-node and 8-node quadrilateral elements and 3-node and 6-node triangular elements.

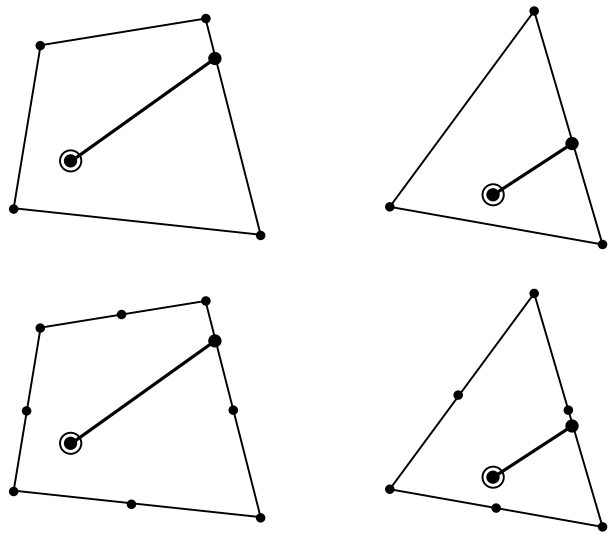
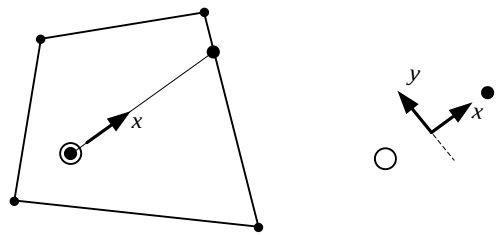


Figure 4.7.2 Pile tip bearing element used in various types of soil elements

Figure 4.7.3 illustrates the element coordinate system of a pile tip bearing element. The element local x-axis is defined in the axial direction of a beam or truss element, and the element local y-axis is defined in the direction normal to the local x-axis.



4.7.3 Coordinate System of Pile tip bearing element

4.7.3 Element Properties

In case a rigid body behavior is assumed for a pile tip bearing element, the following material properties are required:

k_x : Stiffness coefficient in the ECS x-axis direction

k_y : Stiffness coefficient in the ECS y-axis direction

Apart from the rigid body behavior of a pile tip bearing element, a strength degradation curve can be defined in SoilWorks. Refer to the Material manual for details.

4.7.4 Finite Element Formulation

The formulation for a pile tip bearing element is similar to that for a point interface element, and the pile tip bearing element is defined as a point interface element between a mother element and a point element. For the sake of convenience in the formulation, the mother element is assumed to be a two-dimensional element.

In order to define the translational displacement of the pile tip bearing element, the displacement of the mother element, $\mathbf{u}^i_{mot}$ and the displacement of the point element, $\mathbf{u}^i_{pt}$ need to be first defined.

$$\begin{aligned} \mathbf{u}^i_{mot} &= \left\{ u^i_{mot},\ v^i_{mot} \right\} \\ \mathbf{u}^i_{pt} &= \left\{ u^i_{pt},\ v^i_{pt} \right\} \end{aligned} \tag{4.7.1}$$

Since the axial transformation matrix of the pile tip element is one-dimensional, the same transformation matrix of the pile element is used for convenience without the user's input.

$$\mathbf{T}_{tip} = \mathbf{T}_{pile} \tag{4.7.2}$$

The nodal coordinates of the pile tip element can be defined in the isoparametric coordinate system of the mother element as expressed in Eq. (4.7.3) and shown in Figure 4.7.4.

$$\mathbf{a}_{tip}^{i} = \left\{ \xi_{iso}^{i},\ \eta_{iso}^{i} \right\} \tag{4.7.3}$$

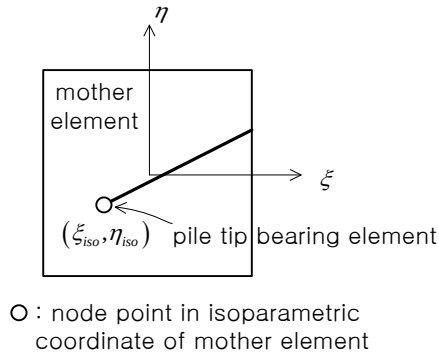


Figure 4.7.4: Nodal coordinates of a pile tip bearing element in the isoparametric coordinate system of a mother element

The shape function of a pile tip can be expressed by the difference between the shape function of the mother element and the shape function of the point element at the location of the pile tip.

$$\mathbf{N}_{tip}^{i} = \left[-N_{mot}^{i}\left(\xi_{iso}^{i},\ \eta_{iso}^{i}\right),\ N_{pt}^{i}\left(\xi_{pt}^{i}\right) \right] \tag{4.7.4}$$

The relative displacement of a pile tip element in the mother element coordinate system can be written as,

$$\Delta\mathbf{u}_{tip} = \mathbf{N}_{tip}^{i} \cdot \left\{ \mathbf{u}_{mot}^{i},\ \mathbf{u}_{pt}^{i} \right\}^{T} \tag{4.7.5}$$

The relative displacement in the pile tip element coordinate system can be obtained from the transformation matrix.

$$\Delta\hat{\mathbf{u}}_{tip} = \mathbf{T}_{tip} \cdot \Delta\mathbf{u}_{tip} \tag{4.7.6}$$

The stiffness matrix of a pile tip element can be expressed by the following equation:

$$\mathbf{K}_{ij} = \int_{V_{tip}} \left(\mathbf{T}_{tip}\mathbf{N}_{tip}\right)^T \cdot \mathbf{D} \cdot \left(\mathbf{T}_{tip}\mathbf{N}_{tip}\right) dV_{tip} \tag{4.7.7}$$

$\mathbf{D}$ is the constitutive matrix defined by the user-defined axial stiffness. The pile tip bearing element is a 1D element while the mother element is a 2D element. In order to match the dimension, the pile tip bearing element is defined in a 2X2 matrix in which the stiffness in non-axial directions is set to zero.

$$\mathbf{D} = \begin{bmatrix} k_n & 0 \\ 0 & 0 \end{bmatrix} \tag{4.7.8}$$

The constitutive equation for the traction, $\mathbf{t}_{tip}$ can be defined in terms of the relative displacement, $\Delta\mathbf{u}_{tip}$ as follows:

$$\begin{aligned} \hat{\mathbf{t}}_{tip} &= \mathbf{D} \cdot \Delta\hat{\mathbf{u}}_{tip} \\ \{t_x\} &= \mathbf{D}\{\Delta u_x\} \end{aligned} \tag{4.7.9}$$

4.7.5 Internal Element Results

The results of pile tip elements are tractions and relative displacements at the interface.

t_x	: Traction in the ECS x-axis direction
t_y	: Traction in the ECS y-axis direction
Δu_x	: Relative displacement in the ECS x-axis direction
Δu_y	: Relative displacement in the ECS y-axis direction

4.8 Geogrid Element

Geogrid materials are high polymers used in the environment of sands, soils, gravels, etc., which are closely related to geotechnical engineering works. They were initially used to prevent soil erosion and for filtering and subsequently used to separate, reinforce or drain soils. Recently they have been also used for waterproofing, preventing fissures, protecting underground structures and absorbing impact.

Geogrid structures are thin and flat, which do not retain flexural stiffness but retain axial stiffness. Geogrid structures cannot resist compression and can only resist tension.

The geogrid element used on a plane is a 2-node low order linear element, which can be used in the same way as the truss element.

Geogrid elements are represented as truss elements exhibiting the tension-only behavior. Since a unit width is assumed for the geometrical property of the truss element, only the thickness data is required.

4.9 Elastic Link

An elastic link element connects two nodes retaining the stiffness defined by the user without any physical properties. Figure 4.9.1 presents the direction of the ECS x-axis. An elastic link may be assigned tension-only or compression-only in which case only the stiffness in the ECS x-axis direction is accounted for. The element consists of two translational and one rotational stiffness parameters. The translational stiffness and rotational stiffness of an elastic link element are expressed in terms of force per unit length and moment per unit radian respectively.

The use of elastic link elements includes elastic bearings of a bridge structure connecting the bridge deck and the piers or soil boundary conditions retaining a compression-only characteristic. Upon selecting the rigid type option, the elastic link rigidly connects two nodes as the rigid link.

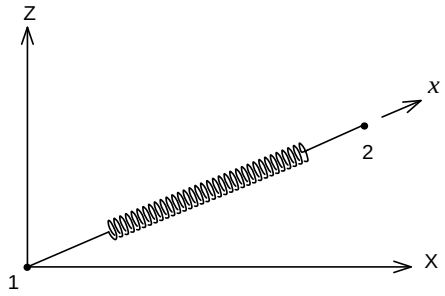


Figure 4.9.1 Coordinate system of an Elastic Link

4.10 Rigid Link

The rigid link element constrains relative geometric displacements or rotations between 2 or more nodes in a structure. A geometric constraint is prescribed to one or more degrees of freedom, which are subordinated to the degrees of freedom of a particular node. The particular reference node is called the master node, and the subordinated nodes are called the slave nodes. The inter-constraint of the master node and a slave node by a rigid link is expressed in Eq. (4.10.1).

$$\begin{aligned} U_{Xs} &= U_{Xm} \\ U_{Zs} &= U_{Zm} \\ R_{Ys} &= R_{Ym} \end{aligned} \tag{4.10.1}$$

where,

U_{Xs}, U_{Zs} : Translations of slave node in the GCS X and Z directions

U_{Xm}, U_{Zm} : Translations of master node in the GCS X and Z directions

R_{Ys} : Rotation of slave node about the GCS Y direction

R_{Ym} : Rotation of master node about the GCS Y direction

The fact that a rigid link element interconnects the degrees of freedom between two nodes to simulate the continuous soil behavior, rigid link elements are used at a stage before an interface element becomes activated.

The rigid link element always imposes the displacements of a slave node to coincide with those of the master node.

4.11 Point Spring/Damping

Point springs are used to define the elastic stiffness of adjoining structures or ground support conditions. Point springs are also used to prevent singular errors from occurring at the connecting nodes of elements with deficient degrees of freedom such as truss element, plane stress element, etc.

A point spring at a node can be expressed in three degrees of freedom (two translational and one rotational components) in GCS as shown in Figure 4.11.1. The translational stiffness and rotational stiffness are defined in terms of force per unit length and moment per unit radian respectively. When modeling soils with point springs, the spring stiffness can be calculated by the product of the modulus of subgrade reaction and the effective area of the corresponding node.

Point damping defines the spring damping of a node. The point damping is widely used in modeling viscous damping boundary conditions of soils and can be defined in three degrees of freedom (two translational and one rotational components) per node. Due to the characteristics of damping, the point damping can be used only in dynamic analysis, not in static analysis.

GCS is basically used for defining the point spring and point damping, but NCS may be used if it has been already defined.

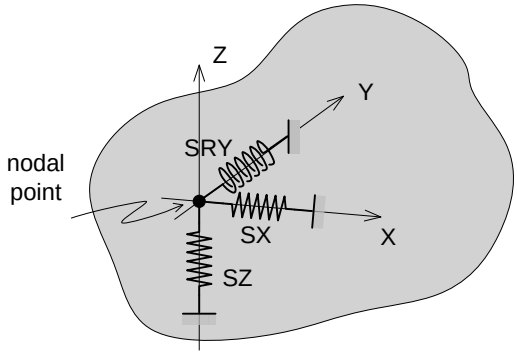


Figure 4.11.1 Coordinate System of Point Spring/Damping