1 STRUCTURAL ELEMENTS

1.1 Introduction

An important aspect of geomechanical analysis and design is the use of structural support to stabilize a soil or rock mass. The term *support* describes engineered materials used to restrict displacements in the immediate vicinity of an opening or excavation. In this section, support is divided into two types: reinforcement and surface support.

Reinforcement consists of tendons (i.e., cables) or bolts installed in holes drilled in the rock mass. Reinforcement acts to conserve inherent rock mass strength so that it becomes self-supporting. Two types of reinforcement model are provided in *UDEC*: *local* and *global* reinforcement. A local reinforcement model considers only the local effect of reinforcement where it passes through existing discontinuities. This reinforcement type is accessed with the **block struct reinforce** command; the model is described in Section 1.2.1.

A global reinforcement model considers the presence of the reinforcement along its entire length throughout the rock mass. Two types of global reinforcement are provided in *UDEC*. One type, cable reinforcement, only provides shear resistance along the cable. This model is accessed via the **block structure cable** command and is described in Section 1.2.2. The other type, rockbolt reinforcement, provides shear resistance along the length of the rockbolt, and resistance normal to the length of the bolt including bending resistance. The model is accessed with the **block structure rockbolt** command and is described in Section 1.2.3.

Surface support consists of concrete lining, steel sets, shotcrete, etc. that are placed on the surface of an excavation and, in many cases, act to truly support, in whole or part, the weights of individual blocks isolated by discontinuities or zones of loosened rock. Structural beam elements are created in *UDEC*, via the **block structure liner generate** command, to simulate surface support for both interior excavations (such as tunnels) and surface excavations (such as open cuts and natural slopes). Structural beam support is described in Section 1.3.1.

Surface support also includes one-dimensional support members that represent hydraulic or wooden props or packs. This support acts as a single degree-of-freedom member that is connected between two boundaries of an interior opening. The **block structure support** command accesses this logic. See Section 1.3.2.

A general guide to material properties for structural elements is given in Section 1.4, and specific modeling considerations when using structural elements are discussed in Section 1.5. In particular, note that, because *UDEC* is a two-dimensional program, the three-dimensional effect of regularly spaced elements is accommodated by scaling their material properties in the out-of-plane direction. The scaling is accomplished by assigning spacing as a property for the structural elements. This procedure is explained in Section 1.5.1.

In all cases, the commands necessary to define each of the structural element types are quite simple, but they invoke a very powerful and flexible structural logic. This logic is developed with the same finite-difference algorithms as the rest of the code (as opposed to a matrix-solution approach), allowing the structure to accommodate large displacements and to be applied for dynamic as well as static analysis. Example applications for each of the structural support types are provided at the end of the section describing each type. *.

---

* The data files in this section are stored in the folder “ITASCA\UDEC700\Datafiles\structures” with the extension “.DAT.” A project file is also provided for each example. For the *GIIC*, open the project file by clicking on the FILE / OPEN PROJECT menu item and select the project file name (with “.PRJ” extension). Then click on the *Project Options* icon at the top of the *Project Tree Record*, select *Rebuild unsaved states*. For the GUI, open the project file by clicking on the FILE / OPEN PROJECT menu item and select the project file name (with “.UDPRJ” extension). Then click on the *Project* tab and select the “Master.dat” and run it

1.2 Reinforcement

There are several different types of reinforcement designed to operate effectively in a range of ground conditions. One type is represented by a reinforcing bar or bolt fully encapsulated in a strong, stiff resin or grout. This system is characterized by the relatively large axial resistance to extensions that can be developed over a relatively short length of the shank of the bolt, and by the high resistance to shear that can be developed by an element penetrating a slipping joint. A second type of reinforcement system, represented by cement-grouted cables or rockbolts, offers little resistance to joint shear, and development of full-axial load may require deformation of the grout over a substantial length of the reinforcing element. These two types of reinforcement are identified, respectively, as local reinforcement and global (or spatially extensive) reinforcement. The characteristic behavior of these two rock reinforcement systems has been incorporated into *UDEC*. Local reinforcement can be applied to both rigid and deformable blocks. Global reinforcement can only be applied to deformable blocks. Both reinforcement systems simulate a row of equally spaced reinforcement in the out-of-plane direction.*

*1.2.1 Local Reinforcement at Joints (*block struct reinforce *Command)*

The local reinforcement formulation considers only the local effect of reinforcement where it passes through existing discontinuities. The formulation results from observations of laboratory tests of fully grouted untensioned reinforcement in good quality rocks with one discontinuity, which indicate that strains in the reinforcement are concentrated across the discontinuity (Bjurstrom 1974, and Pells 1974). This behavior can be achieved in the computational model by calculating, for each element, the forces generated by displacements across the discontinuity through which the element passes. The following description of this formulation is taken from Lorig (1985).

This formulation exploits simple force-displacement relations to describe both the shear and axial behavior of reinforcement across discontinuities. Large shear displacements are accommodated by considering the simple geometric changes that develop locally in the reinforcement near a discontinuity. Although the local reinforcement model can be used with either rigid blocks or deformable blocks, the representation is most applicable to cases in which deformation of individual rock blocks may be neglected in comparison with deformation of the reinforcing system. In such cases, attention may be reasonably focused on the effect of reinforcement near discontinuities.

---

* Note that reinforcement elements cannot be used to simulate a single vertical element because the structural element formulation does not apply to axisymmetric geometry.

1.2.1.1 Axial Behavior

Historically, axial testing of rock reinforcement has focused on pull-out tests for two reasons:

(1) ease of experimentation and interpretation of results; and

(2) provision of axial restraint (the main function of reinforcement in the prevailing conceptual models).

Consequently, a relatively good understanding of axial force-displacement relations has been achieved. The axial force-displacement relation typically used in the representation of rock reinforcement is shown in Figure 1.1.

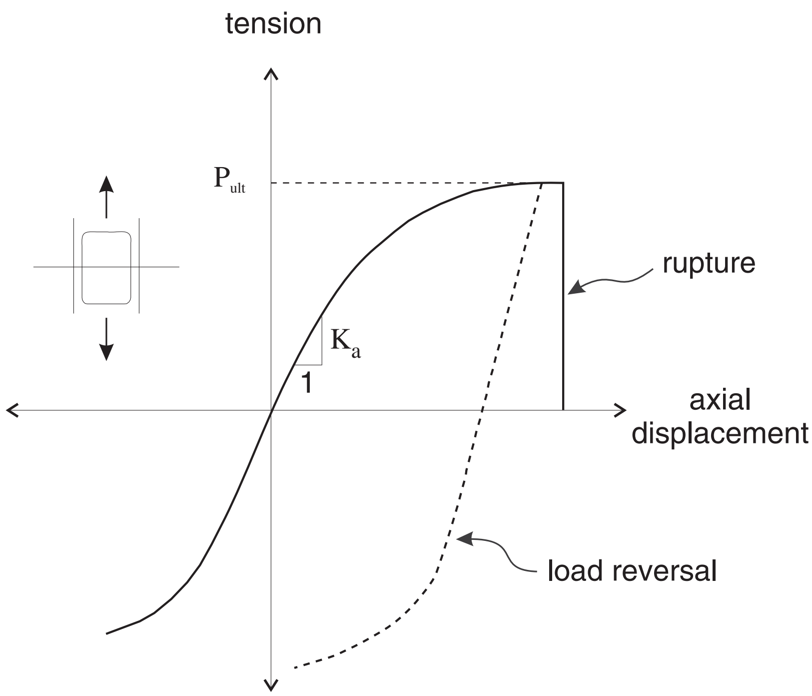


Figure 1.1 Axial behavior of local reinforcement systems

Figure 1.1 indicates an identical response in tension and compression. This may not be the case for all reinforcing systems.

If pull-test results are not available, the following theoretical expression given by Gerdeen et al. (1977) may be used to estimate the axial stiffness, K_a, for fully bonded solid reinforcing elements:

$$K_a = \pi\ k\ d_1 \tag{1.1}$$

where d_1 = reinforcement diameter;

k = $[\frac{1}{2}\ G_g\ E_b\ /\ (d_2\ /\ d_1 - 1)]^{1/2}$;

G_g = grout shear modulus;
E_b = Young's modulus of reinforcement material; and
d_2 = hole diameter.

Comparisons with finite element analyses (Gerdeen et al. 1977) indicate that Eq. (1.1) tends to slightly overestimate axial stiffnesses.

The ultimate axial capacity of the reinforcement depends on a number of factors, including strength of the reinforcing element, bond strength, hole roughness, grout strength, rock strength and hole diameter. In the absence of results of physical tests, empirical relations may be used to estimate the ultimate anchorage strength, P_{ult}. One such relation for the design of cement-grouted reinforcement is given by Littlejohn and Bruce (1975):

$$P_{ult} = 0.1\ \sigma_c\ \pi\ d_2\ L \tag{1.2}$$

where σ_c = uniaxial compressive strength of massive rocks (100% core recovery) up to a maximum value of 42 MPa, assuming that the compressive strength of the cement grout is equal to or greater than 42 MPa; and
L = bond length.

1.2.1.2 Shear Behavior

Recognition that reinforcement also acts to modify the shear stiffness and strength of discontinuities has led to laboratory shear testing of reinforced discontinuities. Experimental results and theoretical investigations indicate that shearing along a discontinuity induces bending stresses in the reinforcement that decay very rapidly with distance into the rock from the shear surface. Typically, within one to two reinforcing element diameters, the bending stresses are insignificant.

The shear force-displacement relation typically used to represent shear behavior is shown in Figure 1.2. The figure shows representative responses for reinforcement at various altitudes with respect to the traversed discontinuity and direction of shear.

If the results of physical tests are not available, the shear stiffness, K_s, may be estimated using the following expression from Gerdeen et al. (1977).

$$K_s = E_b\ I\ \beta^3 \tag{1.3}$$

where β = $[K\ /\ (4\ E_b\ I)]^{1/4}$;
K = $2E_g\ /\ (d_2/d_1 - 1)$;
I = second moment of area of the reinforcement element; and
E_g = Young's modulus of the grout.

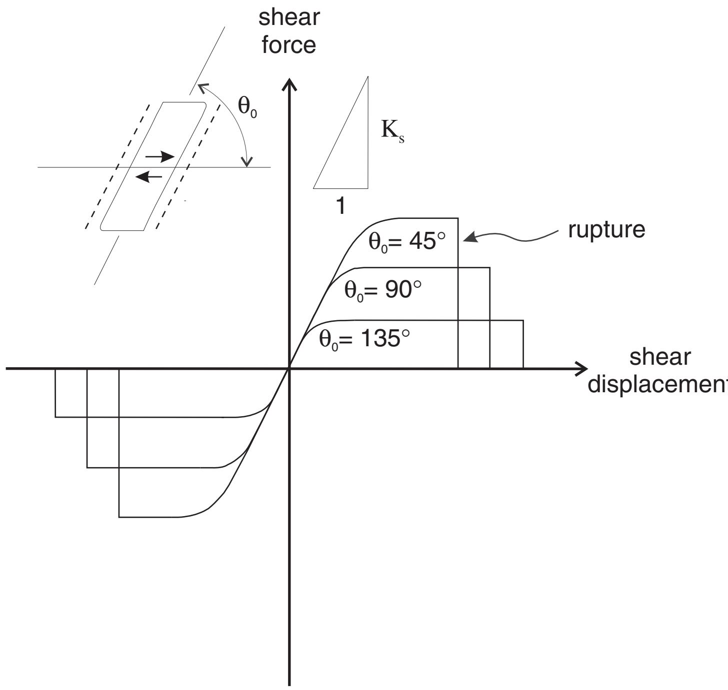


Figure 1.2 Shear behavior of reinforcement system

Empirical relations can be used to estimate the maximum shear force, $F_{s,b}^{max}$, for a reinforcement element at various orientations with respect to a transgressed discontinuity and direction of shear. For example, Bjurstrom (1974) used the results of shear tests of ungrouted reinforcement perpendicular to a discontinuity in granite to develop the expression

$$F_{s,b}^{max} = 0.67\, d_1^2\, (\sigma_b\, \sigma_c)^{1/2} \tag{1.4}$$

where σ_b = yield strength of reinforcement.

In their assessment of maximum shear resistance, St. John and Van Dillen (1983) applied the results of Azuar et al. (1979). The latter found that the maximum shear force was about half the product of the uniaxial tensile strength of the reinforcement and its cross-sectional area for reinforcement perpendicular to the discontinuity. The force increased to 80-90% of that product for reinforcement, inclined with the direction of shear. Shear displacements causing rupture were reported after approximately two reinforcement diameters for the perpendicular case, and one diameter for the inclined case. St. John and Van Dillen interpreted differences between strength and amount of displacement before rupture in terms of the extent of crushing of rock around the reinforcement.

1.2.1.3 Numerical Formulation

The model in *UDEC* assumes that, during shear displacement along a discontinuity, the reinforcement deforms as shown in Figure 1.3. The short length of reinforcement, which spans the discontinuity and changes orientation during shear displacement, is referred to as the active length. The assumed geometric changes were originally suggested in a derivation by Haas (1976) for conventional point-anchored reinforcement and adopted by Fuller and Cox (1978) in considering fully grouted reinforcement.

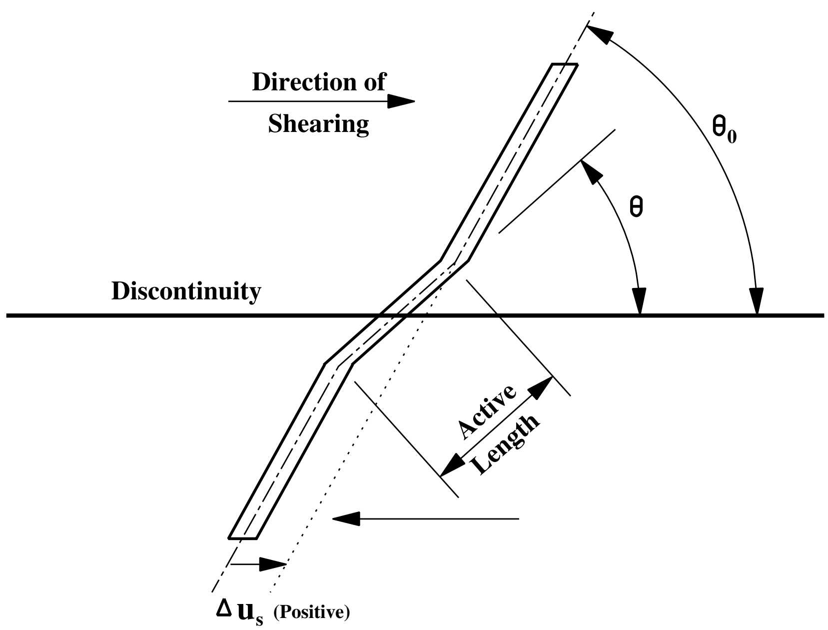


Figure 1.3 Assumed reinforcement geometry after shear displacement, Δu_s

It is assumed that the active length changes orientation only as a direct geometric result of shear and normal displacements at the discontinuity. Methods for estimating the active length are presented in the next section. The model may be considered to consist of two springs located at the discontinuity interface and oriented parallel and perpendicular to the reinforcement axis, as shown in Figure 1.4. Following shear displacement, the axial spring is oriented parallel to the active length, while the shear spring remains perpendicular to the original orientation, as shown in Figure 1.4. Similar geometric changes follow displacements normal to the discontinuity.

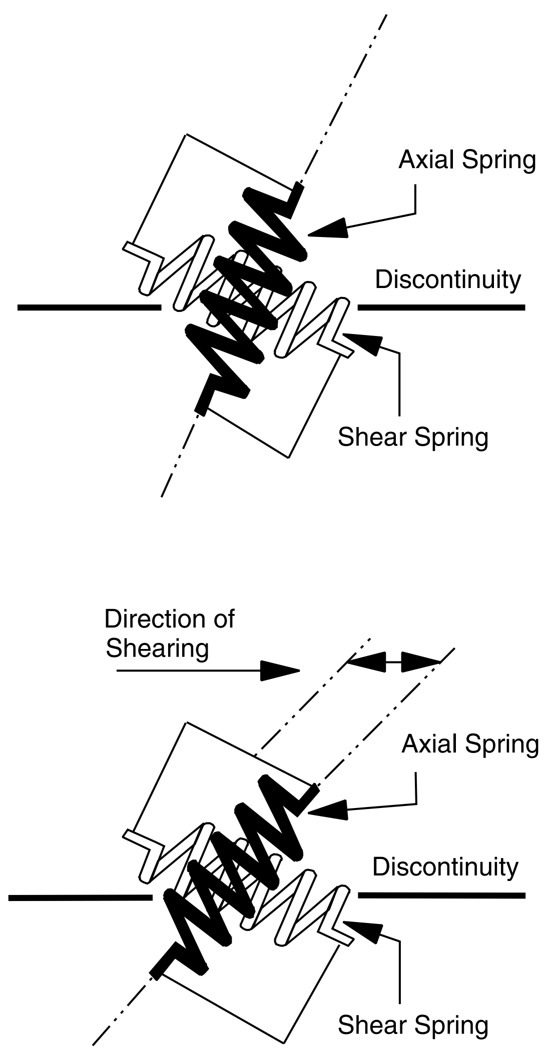


Figure 1.4 Orientation of shear and axial springs representing reinforcement prior to and after shear displacement

The force-displacement models used in *UDEC* to represent axial and shear behavior are continuous, nonlinear algorithms written in terms of stiffness (axial or shear), the ultimate load capacity and a yield function. The yield function describes the force-displacement path followed in approaching the ultimate capacity.

The force-displacement relation that describes the axial response is given by the equation

$$\Delta F_a = K_a |\Delta u_a| \ f(F_a) \tag{1.5}$$

where ΔF_a is an incremental change in axial force;
Δu_a is an incremental change in axial displacement;
K_a is the axial stiffness; and
$f(F_a)$ is a function describing the path by which the axial force, F_a approaches the ultimate (or bounding) axial force $F^{max}_{a,b}$.

The function

$$f(F_a) = \left[|F^{max}_{a,b} - F_a| \, \frac{(F^{max}_{a,b} - F_a)}{[F^{max}_{a,b}]^2} \right]^{e_a} \tag{1.6}$$

is used to represent the axial yield curve. From Eq. (1.5), the axial force "seeks" the bounding force in an asymptomatic manner. The axial stiffness exponent, e_a, controls the rate at which the bounding force is reached. If $e_a = 0$, then the axial stiffness remains constant.

In computing the incremental axial displacement of the active length, it is necessary to account for crushing of the grout and/or rock near the discontinuity as shear displacement causes the reinforcement to bear against one side of the hole. In the present model, a reduction factor, r_f, is applied to incremental axial displacements arising from changes in orientation of the active length to account for the crushing. The reduction factor is computed from the expression

$$r_f = |u_{\text{axial}}| \, (u_s^2 + u_n^2)^{-1/2} \tag{1.7}$$

where u_{axial} = summation of axial displacement increments (i.e., discontinuity displacement increments resolved at each configuration in the direction of the active length);
u_s = total discontinuity shear displacement; and
u_n = total discontinuity normal displacement.

Note that no reduction (r_f = 1.0) is applied for cases in which there is no change in orientation of the active length.

The shear force-displacement relation is described in incremental form by the expression

$$\Delta F_s = K_s |\Delta u_s| \; f(F_s) \tag{1.8}$$

where ΔF_s is an incremental change in shear force;
Δu_s is an incremental change in shear displacement;
K_s is the shear stiffness; and
$f(F_s)$ is a function describing the path by which the shear force, F_s, approaches the ultimate or bounding shear force, $F^{max}_{s,b}$.

The function

$$f(F_s) = \left[|F_{s,b}^{max} - F_s| \frac{(F_{s,b}^{max} - F_s)}{[F_{s,b}^{max}]^2} \right]^{e_s} \tag{1.9}$$

is used to represent the shear yield curve. From Eq. (1.8), the shear force seeks the bounding force in an asymptotic manner. The shear stiffness exponent, e_s, controls the rate at which the bounding force is reached. If $e_s = 0$, then the shear stiffness remains constant.

The maximum shear force, $F_{s,b}^{max}$, changes for various orientations of the active length. The following equation is used to adjust the maximum shear force.

$$F_{s,b}^{\max} = \frac{F_s^{\max} \left[\, 1 + [\text{sign}(\cos\theta_0, \, \Delta u_s) \cdot \cos(\theta_0)] \, \right]}{2} \tag{1.10}$$

where $F_s^{\max} = \pi \; d_1^2 \; \sigma_b/4$; and

Δu_s = incremental change in shear displacement.

The term $\text{sign}(\cos\theta_0, \Delta u_s)$ assigns the sign of Δu_s to $\cos(\theta_0)$. The maximum shear force, $F_{s,b}^{max}$, decreases from a maximum at $\theta_0 = 0°$ to 50% of F_s^{max} at $\theta_0 = 90°$ (see Figure 1.2), which is consistent with the results of Azuar et al. (1979).

The force-displacement relations described above are used to determine forces arising in the springs from incremental displacements at the end points of the active length. The resultant shear and axial forces are resolved into components parallel and perpendicular to the discontinuity, as shown in Figure 1.5. Forces are then applied to the neighboring blocks.

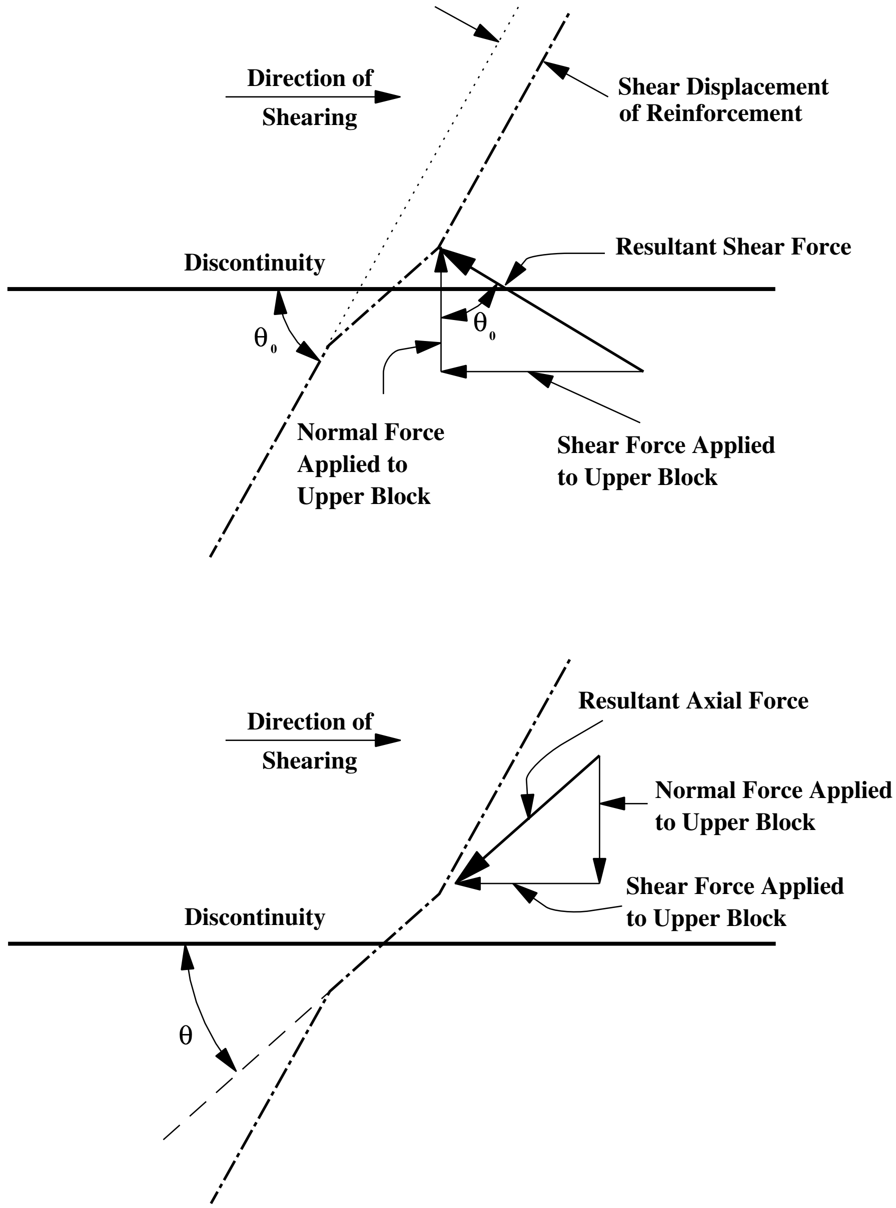


Figure 1.5 Resolution of reinforcement shear and axial forces into components parallel and perpendicular to discontinuity

1.2.1.4 Estimation of Active Length

An estimate of the active length is required to define the assumed local deformation illustrated in Figure 1.3. It has been shown that the active length extends approximately one to two reinforcing element diameters on either side of the discontinuity. In the absence of experimental data, results of theoretical analysis may be used to define the active length. For example, in defining the elastic shear stiffness, K_s, Gerdeen et al. (1977) also determine a quantity, l, called the load transfer length, or “decay length.” If ρ_{max} is the proportion of maximum deflection in the reinforcement, the relation between it and the load transfer length may be expressed by

$$e^{-\beta l} = \rho_{max} \tag{1.11}$$

For example, the point at which the deflection decays to 5% of its maximum value is

$$e^{-\beta l} = 0.05$$

or

$$l = 3/\beta.$$

This approach was developed for reinforcement oriented perpendicular to the shear plane. Dight (1982) presents a theoretical analysis for determining the distance from the shear plane to maximum moment that corresponds with the location of the plastic hinge in the reinforcement element. This approach places no restrictions on the orientation of the reinforcement with respect to the shear plane. A significant result of this analysis is that the distance of the plastic hinge from the shear plane does not appear to vary greatly with shear displacement, especially for displacements greater than 10 mm (0.4 in) for typical reinforcement systems. This observation is in agreement with the assumed geometry changes described earlier.

1.2.1.5 Local Reinforcement Properties

The local reinforcement elements used in *UDEC* require several input parameters:

(1) axial stiffness [force/length];

(2) axial stiffness exponent;

(3) ultimate axial capacity [force];

(4) axial failure strain;

(5) shear stiffness [force/length];

(6) shear stiffness exponent;

(7) ultimate shear capacity [force];

(8) 1/2 active length;

(9) reversal factor; and

(10) spacing.

The axial stiffness, axial stiffness exponent and ultimate axial capacity are usually determined to best fit pull-out tests, as described in Section 1.2.1.1. By default, the axial stiffness exponent is zero, so the axial force-displacement relation follows a constant axial stiffness until the ultimate axial capacity is reached. A limiting axial strain can also be defined; if not specified, the axial strain is unlimited.

The shear stiffness, shear stiffness exponent, ultimate shear capacity and 1/2 the active length can also be back-calculated from experimental testing, as discussed in Sections 1.2.1.2 and 1.2.1.4. By default, the shear stiffness exponent is zero, and a constant shear stiffness is assumed until the ultimate shear capacity is reached.

A reversal factor can also be specified. This controls the slope of the shear force-displacement relation when the direction of the shear force is reversed. Reasonable values for the reversal factor vary between 0 and 1. If the factor equals 1, then the slope upon reversal is the same as that upon initial loading.

1.2.1.6 Summary of Commands Associated with Local Reinforcement Elements

All of the commands associated with local reinforcement elements are listed in Table 1.1. See Help in *UDEC* for a detailed explanation of these commands.

Table 1.1 Keywords associated with **block struct reinforce** *command*

create	keyword		
	begin	*x1y1*	
	end	*x2y2*	
	material	*n*	
delete		*<range>*	
property	**mat** *mat*	keyword	
		active-half-length	*value*
		rupture-tension-strain	*value*
		spacing	*value*
		stiffness-axial	*value*
		stiffness-axial-exponent	*value*
		siffness-shear	*value*
		stiffness-shear-exponent	*value*
		yield-shear-force	*value*
		yield-tension-force	*value*
list			

1.2.1.7 Example Application – Reinforced Slope

The rock slope shown in Figure 1.6 is 40 m high, with foliation planes parallel to the slope face. The planes dip at an angle of 76°, and have a 4 m spacing. Two nearly horizontal joints intersect the slope face at a dip angle of 2.5°. The friction angle of all joints is 6°. The slope is not stable, and fails in a reverse-toppling mode, as shown in Figure 1.7.

The slope is stabilized by adding two horizontal lines of local reinforcement. The mobilized axial forces in the reinforcement are shown in Figure 1.8. The input commands for this example are listed in Example 1.1.

Example 1.1 Reinforced slope

```
model new
model title 'Reinforced Slope'
block config cell 10 10
block tolerance corner-round-length 0.5
block tolerance minimum-edge-length 1
block create polygon 0 0 0 50 80 50 80 0
block joint-region id 1 30.0 10.0 39.1 50.0 61.0 50.0 52.0 10.0
block cut joint-set angle 76.0001 spacing 4 origin 30 10 range jregion 1
block cut crack 0 10 32 10
block cut crack 30 12.5 65 14
block cut crack 35 30 70 31.5
block delete range pos-x 0 30 pos-y 10 50
block joint-delete
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E8 ...
  stiffness-normal 1E8 friction 5.7 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E8 ...
  stiffness-normal 1E8 friction 5.7
block change material 1
block property material 1 density 2E3
block fix range pos-x 0,80 pos-y 10,20
model gravity 0.0 -9.81
model save 'reinf1.sav'
;
; no support
block cycle 3000
model save 'reinf2.sav'
;
model restore 'reinf1.sav'
;
; add reinforcement
block structure reinforce create material 1 begin 30.0 40.0 end 60.0 40.0
```

```
block structure reinforce create material 1 begin 30.0 20.0 end 60.0 20.0
block struct reinforce property material 1 stiffness-axial 1E8 ...
  active-half-length 1 stiffness-shear 1E8 rupture-tension-strain 1E30 ...
  yield-tension 1E6 yield-shear 1E6
block solve ratio 1.0E-5
model save 'reinf3.sav'
```

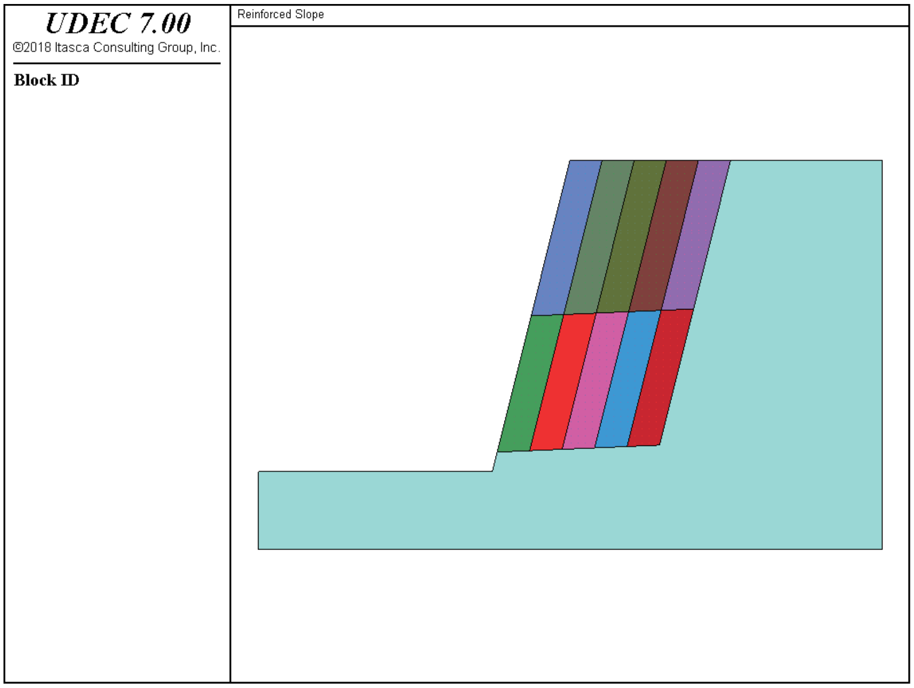


Figure 1.6 Slope with steeply dipping foliation planes

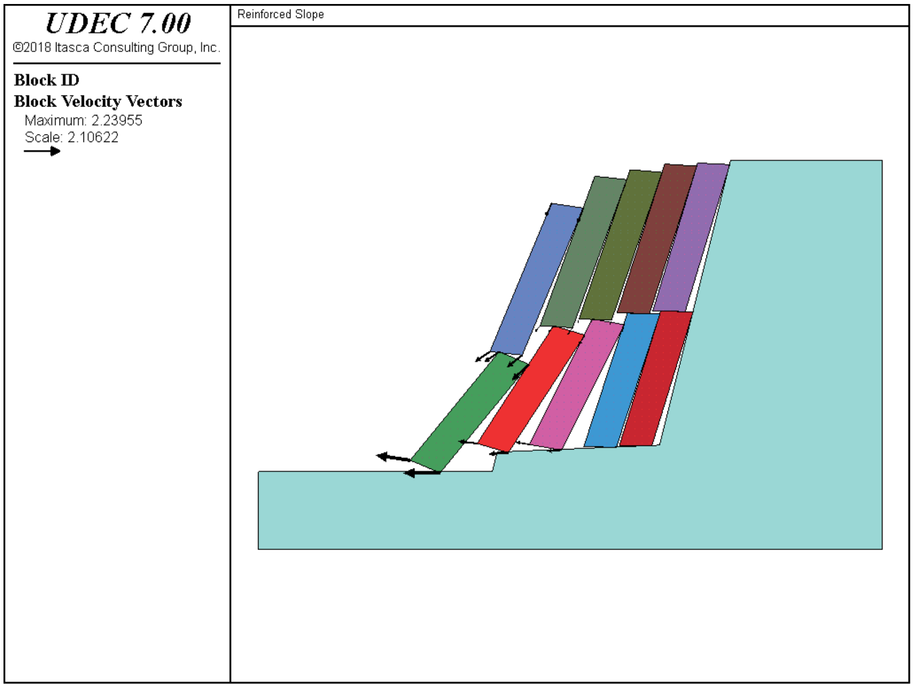


Figure 1.7 Slope failure by reverse toppling

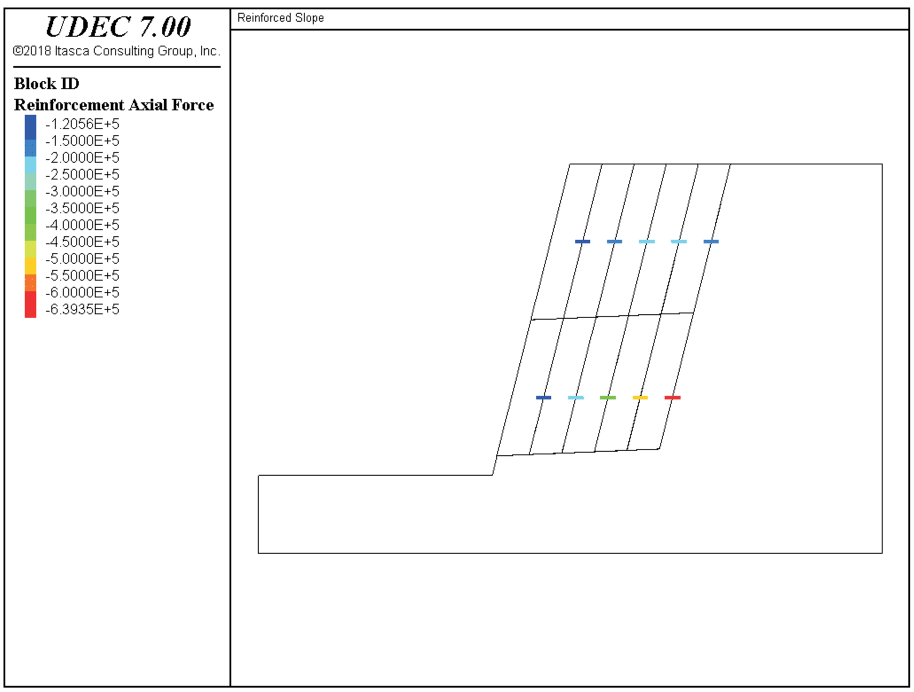


Figure 1.8 Stabilization of slope by reinforcement

1.2.2 Global Shearing-Resistant Reinforcement (block struct cable Command)

In assessing the support provided by rock reinforcement, it is often necessary to consider not only the local restraint provided by reinforcement where it crosses discontinuities, but also the restraint to intact rock that may experience inelastic deformation in the failed region surrounding an excavation. Such situations arise in modeling inelastic deformations associated with failed rock and/or reinforcement systems (e.g., cable bolts) in which the bonding agent (grout) may fail in shear over some length of the reinforcement.

Cable elements in *UDEC* allow the modeling of a shearing resistance along their length, as provided by the shear resistance (bond) between the grout and either the cable or the host medium. The cable is assumed to be divided into a number of segments of length L, with nodal points located at each segment end. The mass of each segment is lumped at the nodal points, as shown in Figure 1.9. Shearing resistance is represented by spring/slider connections between the structural nodes and the block zones in which the nodes are located.

Deformable blocks must be specified to use the cable logic, and the blocks must be made deformable before the cable elements are installed. Each structural node is associated with a finite difference zone for calculation of shear forces between the cable and the zones.

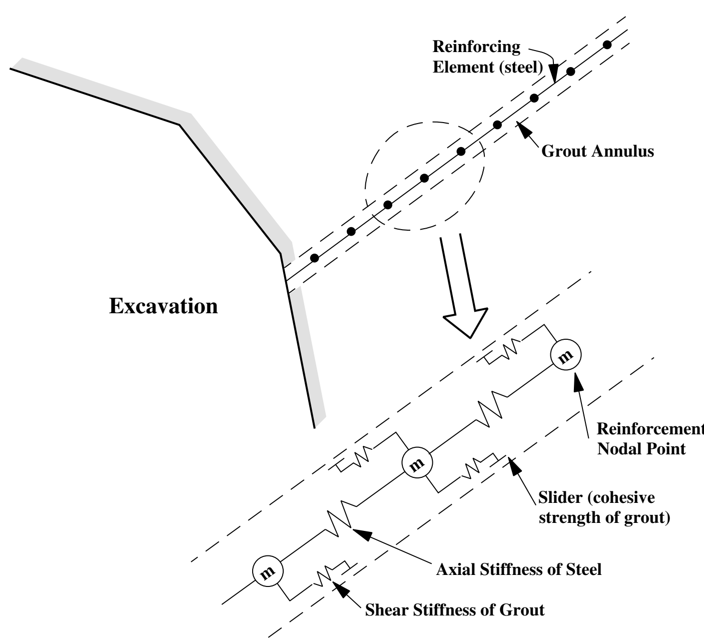


Figure 1.9 Conceptual mechanical representation of fully bonded reinforcement which accounts for shear behavior of the grout annulus

1.2.2.1 Axial Behavior

The axial behavior of conventional reinforcement systems may be assumed to be governed entirely by the reinforcing element itself. The reinforcing element is usually steel, and may be either a bar or cable. Because the reinforcing element is slender, it offers little bending resistance (particularly in the case of cables), and is treated as a one-dimensional member subject to uniaxial tension or compression. A one-dimensional constitutive model is adequate for describing the axial behavior of the reinforcing element. In the present formulation, the axial stiffness is described in terms of the reinforcement cross-sectional area, A, and the Young's modulus, E.

The incremental axial force, ΔF^t, is calculated from the incremental axial displacement by

$$\Delta F^t = -\frac{EA}{L}\,\Delta u^t \tag{1.12}$$

$$\begin{aligned} \text{where}\quad \Delta u^t &= \Delta u_i t_i \\ &= \Delta u_1 t_1 + \Delta u_2 t_2 \\ &= (u_1^{[b]} - u_1^{[a]})t_1 + (u_2^{[b]} - u_2^{[a]})t_2 \end{aligned}$$

$u_1^{[a]}$, $u_1^{[b]}$, etc. are the displacements at the cable nodes associated with each cable element. Subscript $_1$ corresponds to the x-direction, and subscript $_2$ to the y-direction. The superscripts $^{[a]}$, $^{[b]}$ refer to the nodes. The direction cosines t_1, t_2 refer to the tangential (axial) direction of the cable segment.

A tensile yield force limit (**yield-tension**) and a compressive yield force limit (**yield-compression**) can be assigned to the cable. Accordingly, cable forces that are greater than the tensile or compressive limits (Figure 1.10) cannot develop. If either **yield-tension** or **yield-compression** is not specified, the cable will have zero strength for loading in that direction.

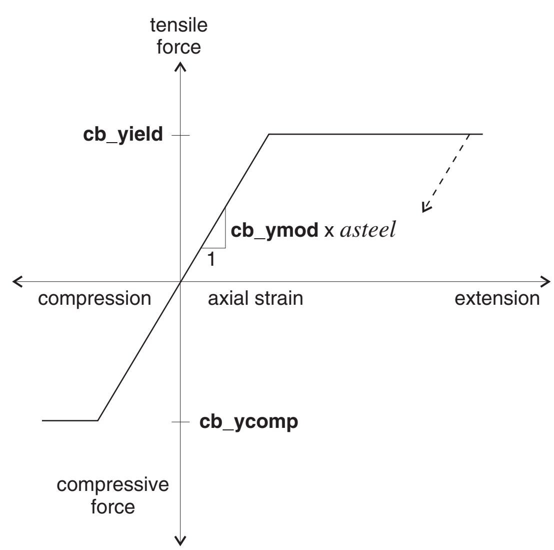


Figure 1.10 Cable material behavior for cable elements

In evaluating the axial forces that develop in the reinforcement, displacements are computed at nodal points along the axis of the reinforcement, as shown in Figure 1.9. Out-of-balance forces at each nodal point are computed from axial forces in the reinforcement, as well as shear forces contributed through shear interaction along the grout annulus. Axial displacements are computed based on integration of the laws of motion using the computed out-of-balance axial force and a mass lumped at each nodal point.

1.2.2.2 Shear Behavior of Grout Annulus

The shear behavior of the grout annulus is represented as a spring/slider system located at the nodal points shown in Figure 1.9. The shear behavior of the grout annulus during relative displacement between the reinforcing/grout interface and the grout/rock interface is described numerically by the grout shear stiffness **grout-stiffness** in (Figure 1.11).

The maximum shear force that can be developed per length of element, F_s^{max}/L, is limited by the cohesive strength of the grout (property keyword **grout-strength**). The limiting shear-force relation is depicted by the diagram in Figure 1.11.

Calculation of the relative displacement at the grout/rock interface uses an interpolation scheme to compute the displacement of the rock in the cable axial direction at the cable node. Each cable node is assumed to exist within an individual *UDEC* triangular zone (hereafter referred to as host zone). The interpolation scheme uses weighting factors that are based on the distance to each of the gridpoints of the host zone. The calculation of the weighting factors is based on satisfying moment equilibrium.

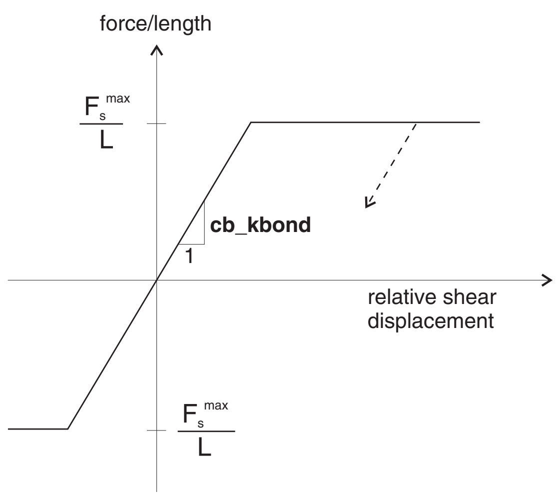


Figure 1.11 Grout material behavior for cable elements

For example, in computing the axial displacement of the grout/rock interface, the following interpolation scheme is used. Consider reinforcement passing through a constant-strain finite difference triangle making up part of the intact rock, as shown in Figure 1.12(a). The incremental x-component of displacement (Δu_{xp}) at the nodal point is given by

$$\Delta u_{xp} = W_1 \ \Delta u_{x1} + W_2 \ \Delta u_{x2} + W_3 \ \Delta u_{x3} \tag{1.13}$$

where Δu_{x1}, Δu_{x2}, Δu_{x3} are the incremental gridpoint displacements; and

W_1, W_2, W_3 are weighting factors.

A similar expression is used for y-component displacements. The weighting factors W_1, W_2, W_3 are computed from the position of the nodal point within the triangle:

$$W_1 = A_1/A_T \tag{1.14}$$

where A_T is the total area of the finite-difference triangle; and

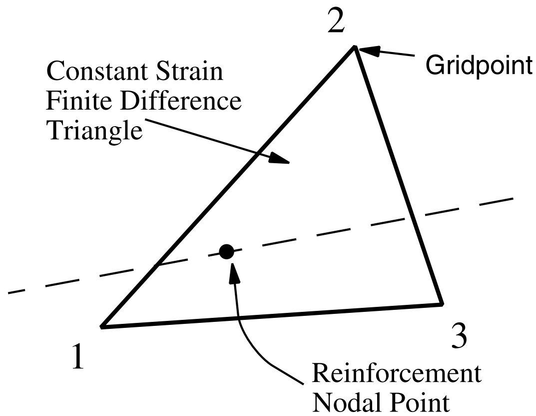
A_1 is the area of the triangle in Figure 1.12(b).

Incremental x- and y-displacements (Eq. (1.13)) are used at each calculation step to determine the new local reinforcing orientation. The axial component of displacement of the grout/rock interface is computed from the current orientation of the reinforcing segment.

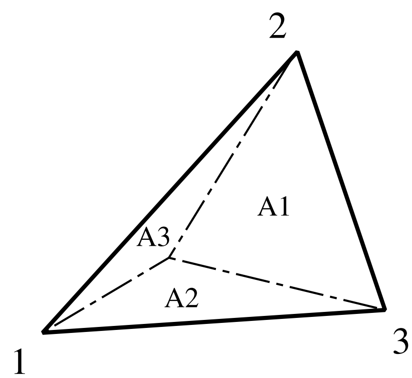
Forces generated at the grout/rock interface (F_{xp}, F_{yp}) are distributed back to gridpoints according to the same weighting factors used previously:

$$\begin{aligned} F_{x1} &= W_1 \ \cdot \ F_{xp} \\ F_{x2} &= W_2 \ \cdot \ F_{xp} \\ F_{x3} &= W_3 \ \cdot \ F_{xp} \end{aligned} \tag{1.15}$$

where F_{x1}, F_{x2} and F_{x3} are forces applied to the gridpoints.



(a) typical reinforcing element passing through a triangular sub-zone



(b) areas used in determining weighting factors used to compute displacement of grout/medium interface

Figure 1.12 Geometry of triangular finite difference zone and transgressing reinforcement used in distinct element formulation

1.2.2.3 Normal Behavior at Grout Interface

As explained above, an interpolated estimate of gridpoint velocity is made at each cable node. The velocity component *normal* to the average axial cable direction is transferred directly to the node (i.e., the cable node is "slaved" to the gridpoint motion in the normal direction). The node exerts no normal force on the grid if the cable segments on either side of the node are colinear. However, if the segments make an angle with each other, then a proportion of their axial forces will act in the mean normal direction. This net force acts on both the gridpoint and the cable node (in opposite directions). Thus, an initially straight cable can sustain normal loading if it undergoes finite deflection.

1.2.2.4 Cable Element Properties

The cable elements used in *UDEC* require several input parameters:

(1) cross-sectional area of cable;

(2) mass density for cable;

(3) elastic Young's modulus for cable;

(4) tensile yield strength [force] of the cable;

(5) compressive yield strength [force] of the cable;

(6) extensional failure strain for the cable;

(7) stiffness of the grout [force/cable length/displacement];

(8) cohesive capacity of the grout [force/cable length]; and

(9) spacing.

Note that property numbers are assigned to cable elements with the **block struct cable material-steel** ***mats*** material property number for the cable material, and with the **block struct cable material-grout** ***matg*** material property number for the grout. Each different cable can then be assigned geometric and material properties by specifying the **block struct cable property** command with the appropriate property keywords following the cable material property number and the grout material property number. For example,

```
bl str cable prop mat=2 grout-stiff=1e9 grout-strength=2e5
```

assigns a cable bond stiffness value of 10^9 and a cable bond strength value of 2×10^5 to property number 2, which is defined (using the **block struct cable material-grout** command) to be the property number for the grout.

The area, density, modulus and yield-force resistance of the cable are usually readily available from handbooks, manufacturer's specifications, etc. A limiting extensional strain can also be defined for the cable; if not specified, the extensional strain is unlimited.

The properties related to the grout are more difficult to estimate. The grout annulus is assumed to behave as an elastic, perfectly plastic solid. As a result of an incremental relative shear displacement, Δu^t, between the tendon surface and the borehole surface, the incremental shear force, ΔF^t, mobilized per length of cable is related to the grout stiffness, K_{bond}:

$$\Delta F^t = K_{\text{bond}} \, \Delta u^t \tag{1.16}$$

K_{bond} can be estimated from pull-out tests. Alternatively, the stiffness can be calculated from a numerical estimate for the elastic shear stress, τ_G, obtained from an equation describing the shear stress at the grout/rock interface (St. John and Van Dillen 1983):

$$\tau_G = \frac{G}{(D/2+t)} \quad \frac{\Delta u}{\ln(1+2t/D)} \tag{1.17}$$

where G = grout shear modulus;
D = reinforcing diameter; and
t = annulus thickness.

Consequently, the grout shear stiffness, K_{bond}, is simply given by

$$K_{\text{bond}} = \frac{2\pi\ G}{\ln\ (1+2t/D)} \tag{1.18}$$

In many cases, the following expression has been found to provide a reasonable estimate of K_{bond} for use in *UDEC*:

$$K_{\text{bond}} \simeq \frac{2\pi\ G}{10\ ln(1+2t/D)} \tag{1.19}$$

The one-tenth factor helps to account for the relative shear displacement that occurs between the host-zone gridpoints and the borehole surface. This relative shear displacement is not accounted for in the present formulation.

The maximum shear force per cable length in the grout is the bond cohesive strength. The value for bond cohesive strength can be estimated from the results of pull-out tests conducted at different confining pressures or, should such results not be available, the maximum force per length may be approximated from the peak shear strength (St. John and Van Dillen 1983):

$$\tau_{\text{peak}} = \tau_I\, Q_B \tag{1.20}$$

where τ_I is approximately one-half of the uniaxial compressive strength of the weaker of the rock and grout, and Q_B is the quality of the bond between the grout and rock (Q_B = 1 for perfect bonding).

Neglecting frictional confinement effects, S_{bond} may then be obtained from

$$S_{\text{bond}} = \pi(D+2t)\ \tau_{\text{peak}} \tag{1.21}$$

Failure of reinforcing systems does not always occur at the grout/rock interface. Failure may occur at the reinforcing/grout interface, as is often true for cable reinforcing. In such cases, the shear stress should be evaluated at this interface. This means that the expressions $(D + 2t)$ are replaced by (D) in Eq. (1.21).

The calculation of cable-element properties is demonstrated by the following example. A 25.4-mm (1 inch) diameter locked-coil cable was installed at 2.5-m spacing perpendicular to the plane of analysis. The reinforcing system is characterized by the properties

cable diameter (D)	25.4 mm
hole diameter $(D + 2t)$	38 mm
cable modulus (E)	98.6 GPa
cable ultimate tensile capacity	0.548 MN
grout compressive strength	20 MPa
grout shear modulus (G)	9 GPa

Two independent methods are used in evaluating the maximum shear force in the grout. In the first method, the bond shear strength is assumed to be one-half the uniaxial compressive strength of the grout. If the grout-material compressive strength is 20 MPa and the grout is weaker than the surrounding rock, the grout shear strength is then 10 MPa.

In the second method, reported pull-out data are used to estimate the grout shear strength. The report presents results for 15.9-mm (5/8 inch) diameter steel cables grouted with a 0.15-m (5.9 inch) bond length in holes of varying depths. The testing indicated capacities of roughly 70 kN. If a surface area of 0.0075 m^2 (0.15 m $\times$ 0.05 m) is assumed for the cables, then the calculated maximum shear strength of the grout is

$$\frac{70 \text{ x } 10^3 \text{ N}}{0.0075 \text{ m}^2} = 9.33 \text{ x } 10^6 \text{ N/m}^2 = 9.33 \text{ MPa}$$

This value agrees closely with the 10 MPa estimated above, and either value could be used. Assuming that failure occurs at the cable/grout interface, the maximum bond force per length is (using Eq. (1.21), with $D + 2t$ replaced by D)

$$S_{bond} = \pi \text{ (0.0254 m) (10 MPa)} = 800 \text{ kN/m}$$

The bond stiffness, K_{bond}, is estimated from Eq. (1.19). For the assumed values shown above, a bond stiffness of $1.5 \times 10^{10} \; \frac{N/m}{m}$ is calculated.

Values for K_{bond}, S_{bond}, E and tensile yield force are divided by 2.5 to account for the 2.5-m spacing of cables perpendicular to the modeled cross-section (see Section 1.5.1).

The final input properties for *UDEC* are

cross-sectional-area	5×10^{-4} m^2
grout-stiffness	6×10^{9} N/m/m
grout-strength	3.2×10^{5} N/m
young	40 GPa
yield-tension	2.2×10^{5} N

Mass scaling is performed automatically to adjust the cable mass to achieve a timestep that coincides with that calculated for the *UDEC* model without cables. If the grout stiffness or axial modulus of the cable element is very high, it may be necessary to reduce the timestep (using the **block mechanical timestep-factor** command) to avoid numerical instability errors.

1.2.2.5 Pretensioning Cable Elements

Cable elements may be pretensioned in *UDEC* by fixing a pretension force (**block struct cable fix-tension** ***value***), cycling to equilibrium, and then releasing the force (**block struct cable fix-tension**). A positive value represents tension.

In practice, pretensioned elements may be fully grouted, or they may be left ungrouted over part of their length. In either case, some form of anchorage is provided at the ends of the cable during pretensioning. This process can be simulated in *UDEC* by the application of several commands. First, the **block struct cable create** command is used to define the geometry for the cable, the property numbers for the anchored ends and the ungrouted section (the **material-first-node** and **material-last-node** keywords may be used to do this), and then the **block struct cable fix-tension** ***value*** command to set the pretension force. **block struct cable property** commands need to be used to define the different properties that exist at the anchored ends and the middle section. The ungrouted section is initially left free, so **grout-strength** should be 0.0 for that property. The model may be cycled at this point to allow the pretension force to distribute to the rock mass. The procedure for subsequent "grouting" of the free length is to simply change the **grout-strength** values for the free section to appropriate values for a grouted section. The **block struct cable free-tension** command is then issued to allow the cable axial force to respond to changes in the rock.

1.2.2.6 Estimating the Maximum Length for Cable Element Segments

If the segment lengths for a cable element are too long, failure by grout slippage cannot occur because the cable material yield force will be reached before the shear resistance along the grout can be mobilized. It is recommended that at least two or three segments be provided along the "development length" of a cable in order to account for the possibility of grout slippage. The development length is the length of bonded reinforcing required to develop the axial capacity of the cable. The development length, l_d, can be estimated from the grout bond strength, S_{bond}, and the cable yield force capacity, F_y, using the expression

$$l_d = \frac{F_y}{S_{\text{bond}}} \tag{1.22}$$

1.2.2.7 Connecting Cable Elements to Beam Elements and to Other Cable Elements

A cable element can be connected to a structural (beam) element, for example, to simulate a cable bolt connected to a tunnel lining. The **connect** keyword, specified as part of a **block struct cable create** command, will position the cable node closest to a structural (beam) element node to coincide with that beam node. The structural elements must be defined first. The connection to the structural element node is not allowed to fail. The cable node connected to the structural element node will not appear in the list of the cable element nodes. The node ID number for the missing cable node will be the ID number of the structural element node to which the cable is connected.

A new cable can be connected to the end of an existing cable by adding the keyword **extend** to the **block struct cable create** command for the new cable. The end node of the new cable closest to an existing cable end node will be connected to the existing cable, and have the same node ID as the existing node.

1.2.2.8 Summary of Commands Associated with Cable Elements

All of the commands associated with local reinforcement elements are listed in Table 1.2. See Help in *UDEC* for a detailed explanation of these commands.

Table 1.2 Keywords associated with **block structure cable** *command*

change		keyword	
		mat-first-node	*mat*
		mat-last-node	*mat*
		mat-steel	*mats*
		mat-grout	*matg*
create		keyword	
		begin	*x1 y1*
		connect	
		end	*x2 y2*
		extend	
		group	*name*
		material-first-node	*mat*
		material-last-node	*mat*
		material-grout	*matg*
		material-steel	*mats*
		segments	*n*
delete		*<range>*	
list		keyword	
		element	
		group	
		node	
		property	
property	**mat** *matg*	keyword	
		grout-stiffness	*value*
		grout-strength	*value*
property	**mat** *mats*	keyword	
		cross-sectional area	*value*
		density	*value*
		rupture-tension-strain	*value*
		spacing	*value*
		thermal-expansion	*value*
		yield-compression	*value*
		yield-tension	*value*

1.2.2.9 Example Application – Pull-Test for a Grouted Cable Anchor

The most common method for determining cable bolt properties is to perform pull-out tests on small segments of grouted cables in the field. Typically, segments from 10 to 50 cm in length are grouted into boreholes. The ends of these segments are pulled with a jack mounted to the surface of the tunnel, and connected to the cable via a barrel-and-wedge type anchor. The force applied to the cable and the deformation of the cable are plotted to produce an axial force-deflection curve. From this curve, the peak shear strength of the grout bond is normally determined and converted to a strength in tons/m cable length.

In this example we simulate a pull-test on a single 15.2-mm diameter cable. The cable material properties are

cable area	181 mm^2
cable bond length	0.5 m
cable modulus (E)	98.6 GPa
cable ultimate tensile capacity	0.232 MN

We select two properties for the grout:

grout bond stiffness	1.12×10^7 N/m/m
grout cohesive strength	1.75×10^5 N/m

These values are representative of a relatively weak grout (e.g., see Hyett et al. 1992).

We apply a load to the cable by gluing a small block to the end of the cable; we can pull the cable by pulling the block. The cable is attached to the small block by assigning a high grout shear-stiffness and shear-strength to the cable node embedded in the small block. The grout properties are changed for the one cable node by changing the material number assigned to this node. The material number integer is changed by adding the keyword phrase **material-first-node** *2* to the end of the **block structure cable create** command. This changes the material property number for the first node of this cable to 2.

FISH function **pullf** is used to monitor the pull force on the cable (pull force per cable length). The pull force is determined from the sum of reaction forces that develop on the block as the cable is pulled. Example 1.2 presents the data file for this model.

Example 1.2 Simulation of a pull-test for a grouted cable anchor

```
model new
model title 'cable pull test'
block tolerance corner-round-length 1E-3
block tolerance minimum-edge-length 2E-3
block create polygon 0 0 0 1 1 1 1 0
```

```
block cut crack 0.5 0 0.5 1
block cut crack 0.4 0 0.4 1
block cut crack 0.4 0.4 0.5 0.4
block cut crack 0.4 0.5 0.5 0.5
block delete range pos-x 0 0.4 pos-y 0 1
block delete range pos-x 0.4 0.5 pos-y 0 0.4
block delete range pos-x 0.4 0.5 pos-y 0.5 1
block zone gen quad 0.13 0.4 range pos-x 0.5 1 pos-y 0 1
block zone gen quad 0.11 range pos-x 0.4 0.5 pos-y 0.4 0.5
block zone group 'block'
block zone cmodel assign elastic density 2.5E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E11 ...
  stiffness-normal 1E11 friction 30 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E11 ...
  stiffness-normal 1E11 friction 30
model save 'pulltest1.sav'
;
;
block structure cable create begin 0.48 0.45 end 0.98 0.45 segments 12 ...
  material-steel 1 material-grout 1 material-first-node 2
block struct cable property material 1 cross-sectional-area 1.81E-4 ...
  density 7.5E3 rupture-tension-strain 1E30 yield-compression 1E10 ...
  yield-tension 2.32E5 young 9.86E10 grout-stiffness 1.12E7 ...
  grout-strength 1.75E5 spacing 1
block struct cable property material 2 cross-sectional-area 1.81E-4 ...
  density 7.5E3 rupture-tension-strain 1E30 yield-compression 1E10 ...
  yield-tension 2.32E5 young 9.86E10 grout-stiffness 1E10 ...
  grout-strength 1E10 spacing 1
model save 'pulltest2.sav'
;
;
;Name:find_block
fish define find_block
  ib_rock = block.near(.5,.5)
  ib_block = block.near(.7,.5)
end
@find_block
;
fish define pullf
  sum = 0.0
  x_plus = x_loc + x_tol
  ib = block.head
  loop while ib # 0
```

```
    if ib = ib_rock then
      ig = block.gp(ib)
      loop while ig # 0
        ; index of boundary corner associated with gridpoint
        ibou=block.gp.boundary.corner(ig)
        if ibou # 0 then
          if block.boundary.type.x(ibou) = 4 then
            x_pos = block.gp.pos.x(ig)
            if x_pos <= x_plus then
              sum = sum + block.boundary.force.x(ibou)
            endif
          else
            sum = sum + block.gp.force.x(ig)
          endif
        endif
        ig = block.gp.next(ig)
      endloop
    endif
    ib = block.next(ib)
  endloop
  pullf = sum / 0.5
  x_disp = -0.05 * global.step * block.mechanical.timestep
end
fish set @x_loc=0.5
fish set @x_tol=0.02
@pullf
history @pullf
history @x_disp
history interval 50
model save 'pulltest3.sav'
;
;
block hide  range pos-x 0.4 0.5 pos-y 0.4 0.5
block gridpoint apply velocity-x 0 ...
  range pos-x 0.499 0.501 pos-y -1E-2 1.01
block show
block gridpoint apply velocity-x -5E-3 ...
  range pos-x 0.39 0.41 pos-y 0.39 0.52
model save 'pulltest4.sav'
;
;

block cycle 220000
model save 'pulltest5.sav'
```

Figure 1.13 displays the axial force distribution in the cable at the end of the test. The total pull force versus displacement results from the test are shown in Figure 1.14. As this figure shows, the peak load is very similar to the input value for grout cohesive strength. The loading slope also compares closely with the input value for grout bond stiffness. This is because the grout is much softer than the cable material. If the stiffness of the grout is increased, this slope will vary from the grout stiffness because the loading slope reflects the combined deflection due to both strain in the cable and the relative displacement of the grout. In general, the bond stiffness can be back-calculated by adjusting this stiffness to fit *UDEC* results to a pull-test force-displacement curve.

The cable shear-bond strength will, in general, increase with increasing effective pressure acting on the cable. The pressure dependency is not accounted for directly in the present formulation. However, it is possible to add this dependency through the use of a *FISH* routine. Alternatively, rockbolt elements can be used to simulate this pressure dependency (see Section 1.2.3.2).

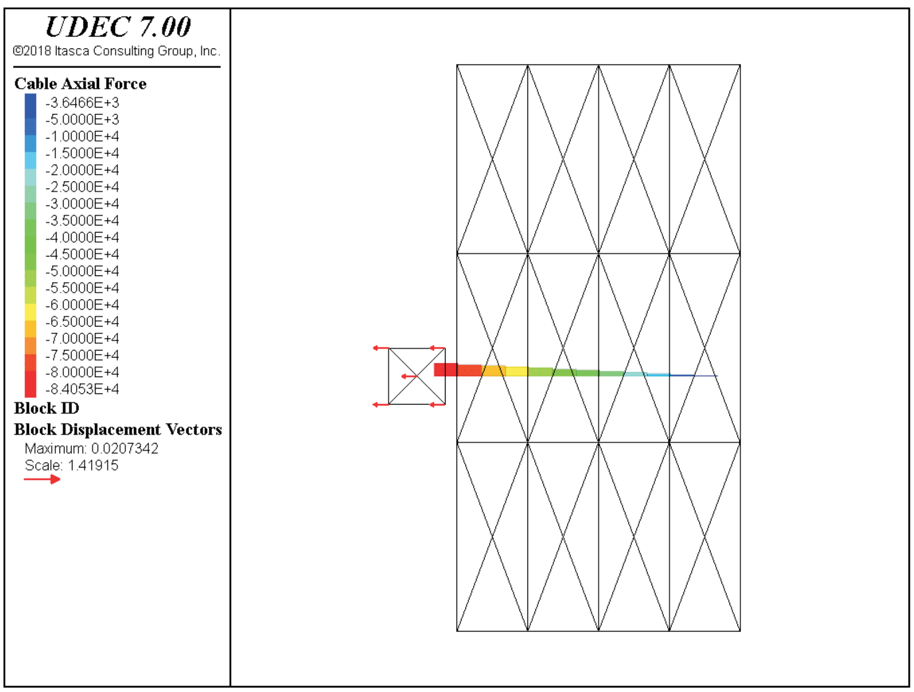


Figure 1.13 Axial force and displacement vectors for pull-test

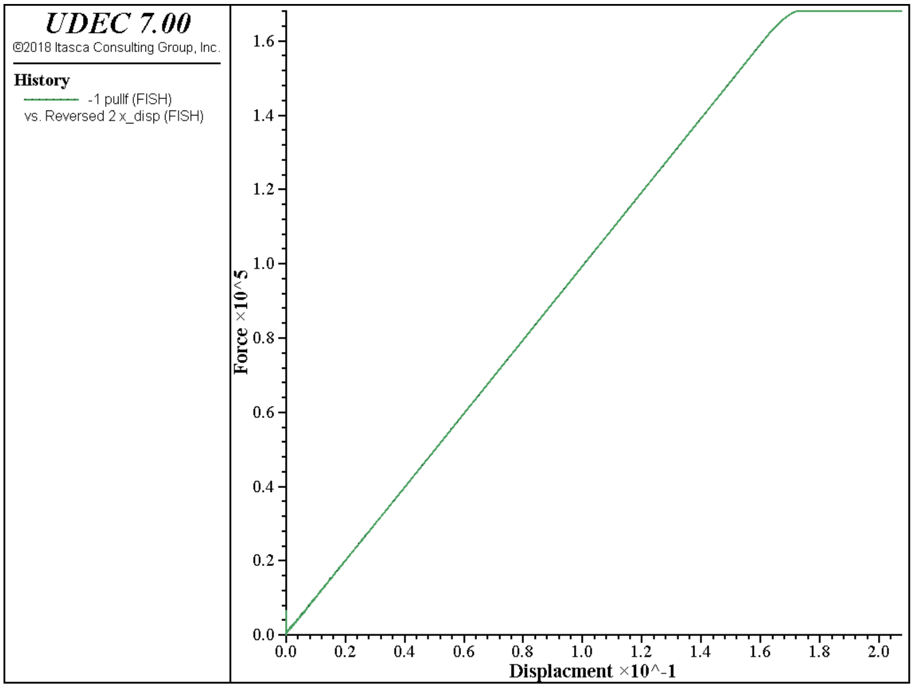


Figure 1.14 Cable grout shear force versus displacement at node in small block

*1.2.3 Global Shearing- and Bending-Resistant Reinforcement (***block struct rockbolt*** Command)*

The rockbolt element in *UDEC* is different from the cable bolt element in that it provides bending-resistant behavior.* Rockbolts are two-dimensional elements with 3 degrees of freedom (two displacements and one rotation) at each end node. The formulation for the rockbolt element is identical to that for beams, as described in Section 1.3.1.1. Rockbolt elements can yield in the axial direction, and can also simulate bolt breakage based upon a user-defined tensile failure strain limit.

Rockbolts interact with *UDEC* via shear and normal coupling springs. The coupling springs are nonlinear connectors that transfer forces and motion between the rockbolt elements and the gridpoints associated with the block zone in which the rockbolt nodes are located. The formulation is similar to that for cable elements. The behavior of the shear coupling springs is identical to the representation for the shear behavior of grout, as described for cable elements in Section 1.2.2. The behavior of the normal coupling springs includes the capability to model load reversal. The normal coupling springs are primarily intended to simulate the effect of the medium squeezing around the rockbolt.

---

* The rockbolt model was developed in collaboration with Geocontrol S.A., Madrid, Spain for application to analyses in which nonlinear effects of confinement, grout or resin bonding, or tensile rupture are important.

The formulations for the rockbolt segments and shear and normal coupling springs are described below.

1.2.3.1 Behavior of Rockbolt Segments

A rockbolt element segment is treated as a linearly elastic material that may yield in the axial direction in both tension and compression. The behavior is identical to that prescribed for cable elements, as depicted in Figure 1.10.

Inelastic bending is simulated in rockbolts by specifying a limiting plastic moment. The present formulation in *UDEC* assumes that rockbolt elements behave elastically until they reach the plastic moment. This assumption is reasonably valid for rockbolt sections, because the difference between the moment necessary to produce the yield stress and the moment that results in yielding across the entire section is small. The section at which the plastic moment occurs can continue to deform without inducing additional resistance after it reaches this limit. The plastic-moment capacity sets the limit for the internal moments of structural-element segments for rockbolts.

In addition, segments may break and separate at the nodes. Rockbolt breakage is simulated based upon a user-defined tensile failure strain limit (**tension-failure-strain**). A strain measure, based upon adding the axial and bending plastic strains, is evaluated at each rockbolt node. The axial plastic strain, ε_{pl}^{ax}, is accumulated based on the average strain of rockbolt element segments using the node. The bending plastic strain is averaged over the rockbolt, and then accumulated. The total plastic tensile strain, ε_{pl}, is then calculated by

$$\varepsilon_{pl} = \Sigma\varepsilon_{pl}^{ax} + \Sigma\frac{d}{2}\frac{\theta_{pl}}{L} \tag{1.23}$$

where d = rockbolt diameter;
L = rockbolt segment length; and
θ = average angular rotation over the rockbolt.

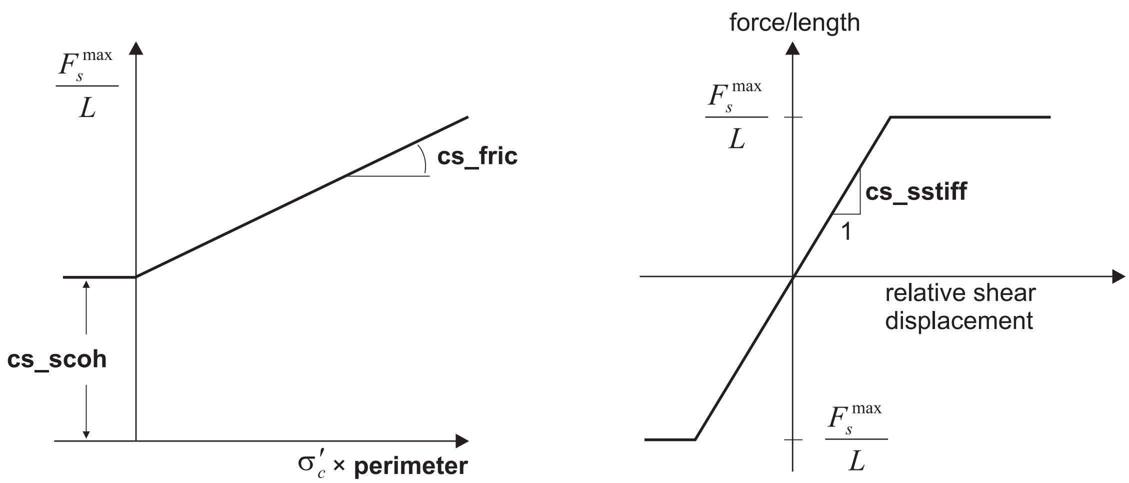
If this strain exceeds the limit **tension-failure-strain**, the forces and moment in this rockbolt segment are set to zero, and the rockbolt is assumed to have failed.

1.2.3.2 Behavior of Shear Coupling Springs

The shear behavior of the rockbolt/gridpoint interface is represented as a spring-slider system at the rockbolt nodal points. The system is similar to that illustrated for the cable/gridpoint interface in Figure 1.9. The shear behavior of the interface during relative displacement between the rockbolt nodes and the gridpoints is described numerically by the coupling spring shear stiffness (**coupling-stiffness-shear** in Figure 1.15(b)).

$$\frac{F_s}{L} = cs_{\text{sstiff}}\,(u_p - u_m) \tag{1.24}$$

where F_s = shear force that develops in the shear coupling spring (i.e., along the interface between the rockbolt element and the gridpoint);
cs_{sstiff} = coupling spring shear stiffness (**coupling-stiffness-shear**);
u_p = axial displacement of the rockbolt;
u_m = axial displacement of the medium (soil or rock); and
L = contributing element length.



a) Shear strength criterion b) Shear force versus displacement

Figure 1.15 Material behavior of shear coupling spring for rockbolt elements

The maximum shear force that can be developed along the rockbolt/gridpoint interface is a function of the cohesive strength of the interface and the stress-dependent frictional resistance along the interface. The following relation is used to determine the maximum shear force per length of the rockbolt:

$$\frac{F_s^{\text{max}}}{L} = cs_{\text{scoh}} + \sigma_c' \times \tan(cs_{\text{sfric}}) \times \text{perimeter} \tag{1.25}$$

where cs_{scoh} = cohesive strength of the shear coupling spring (**coupling-cohesion-shear**);
σ_c' = mean effective confining stress normal to the rockbolt element;
cs_{sfric} = friction angle of the shear coupling spring (**coupling-friction-shear**); and
perimeter = exposed perimeter of the element (**perimeter**).

The effective confining stress acting on the rockbolt is based on the change in stress since installation. Stresses in the zone around the rockbolt are stored when the element is installed, and as calculation progresses, the effective confining stress around the element is calculated as the change in stress from the installation state.

The mean effective confining stress normal to the element is defined by the equation

$$\sigma_c' = -\left(\frac{\sigma_{nn} + \sigma_{zz}}{2} + p\right) \tag{1.26}$$

where p = pore pressure;
σ_{zz} = out-of-plane stress;
σ_{nn} = $\sigma_{xx}\, n_1^2 + \sigma_{yy}\, n_2^2 + 2\sigma_{xy}\, n_1\, n_2$; and
n_i = unit vectors relative to the local rockbolt segment axes.

The limiting shear-force relation is depicted by the diagram in Figure 1.16(a). The input properties are shown in bold type on this figure.

A user-defined table (**coupling-confining-table**) can be specified to give a correction factor for the effective confining stress, in cases of non-isotropic stress, as a function of a deviatoric stress ratio. By default, the confining stress is given by Eq. (1.26). By specifying a table with **coupling-friction-table**, factors are applied to the value of σ_m to account for non-isotropic stresses.

Softening as a function of shear displacement for the shear coupling-spring cohesion and friction angle properties can be prescribed via the user-defined tables **coupling-cohesion-table** and **coupling-friction-table**. The tables relate these properties to relative shear displacement.

The same interpolation scheme as that employed for the cable elements is used to calculate the displacement of the block grid in the rockbolt axial direction at the rockbolt node.

1.2.3.3 Behavior of Normal Coupling Springs

The normal behavior of the rockbolt/gridpoint interface is represented by a linear spring with a limiting normal force that is dependent on the direction of movement of the rockbolt node. The normal behavior during the relative normal displacement between the rockbolt nodes and the gridpoint is described numerically by the coupling spring normal stiffness (**coupling-stiffness-normal** in Figure 1.16(b)):

$$\frac{F_n}{L} = cs_{\text{nstiff}}\ (u_p^n - u_m^n) \tag{1.27}$$

where F_n = normal force that develops in the normal coupling spring (i.e., along the interface between the rockbolt element and the gridpoint);
cs_{nstiff} = coupling spring normal stiffness (**coupling-stiffness-normal**);
u_p^n = displacement of the rockbolt normal to the axial direction of the rockbolt;
u_m^n = displacement of the medium (soil or rock) normal to the axial direction of the rockbolt; and
L = contributing element length.

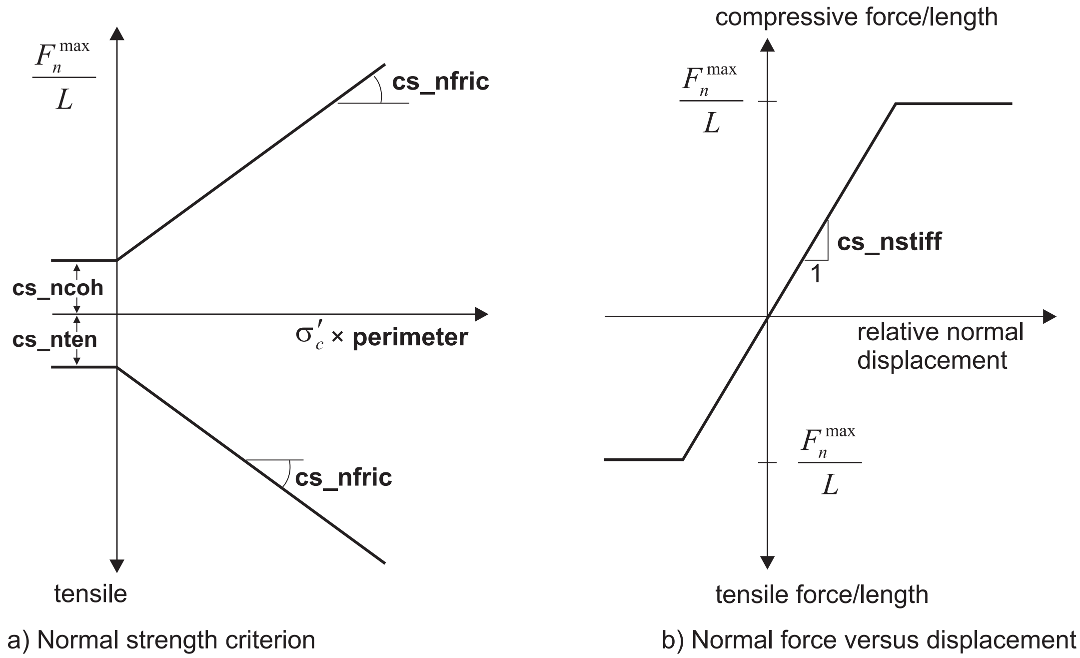


Figure 1.16 Material behavior of normal coupling spring for rockbolt elements

A limiting normal force can be prescribed to simulate the localized three-dimensional effect of the rockbolt pushing through the grid (e.g., a soil being squeezed around a single rockbolt). The limiting force is a function of a normal cohesive strength and a stress-dependent frictional resistance between the rockbolt and the gridpoint. The following relation is used to determine the maximum normal force per length of the rockbolt:

$$\frac{F_n^{max}}{L} = cs_{ncoh} + \sigma_c' \times \tan(cs_{nfric}) \times \text{perimeter} \tag{1.28}$$

where cs_{ncoh} = cohesive strength of the normal coupling spring (**coupling-cohesion-normal**), which is dependent on the direction of loading;

σ_c' = mean effective confining stress normal to the rockbolt element;

cs_{nfric} = friction angle of the normal coupling spring (**coupling-friction-normal**); and

perimeter = exposed perimeter of the element (**perimeter**).

The mean effective confining stress normal to the element is defined by Eq. (1.26).

1.2.3.4 Rockbolt-Element Properties

The rockbolt elements in *UDEC* require several input parameters (rockbolt property keywords are shown in parentheses):

(1) cross-sectional area (**area** or **radius**) [length^2] of the rockbolt;

(2) second moment of area (**moi**) [length^4] (commonly referred to as the moment of inertia) of the rockbolt;

(3) density (**density**) [mass/volume] of the rockbolt (optional – used for dynamic analysis and gravity loading);

(4) elastic modulus (**young**) [stress] of the rockbolt;

(5) spacing (**spacing**) [length] (optional – if not specified, rockbolts are considered to be continuous in the out-of-plane direction);

(6) plastic moment (**plastic-moment**) [force–length] (optional – if not specified, the moment capacity is assumed to be infinite);

(7) tensile yield strength (**yield-tension**) [force] of the rockbolt (if not specified, the tensile yield strength is zero);

(8) compressive yield strength (**yield-compression**) [force] of the rockbolt (if not specified, the compressive yield strength is zero);

(9) tensile failure strain limit of the rockbolt (**tension-failure-strain**);

(10) exposed perimeter (**perimeter**) [length] of the rockbolt (i.e., the length of the rockbolt surface that is in contact with the medium);

(11) stiffness of shear coupling spring (**coupling-stiffness-shear**) [force/rockbolt length/displ.];

(12) cohesive strength of shear coupling spring (**coupling-cohesion-shear**) [force/rockbolt length];

(13) frictional resistance of the shear coupling spring (**coupling-friction-shear**) [degrees];

(14) number of table relating cohesion of shear coupling spring to relative shear displacement (**coupling-cohesion-table**);

(15) number of table relating friction angle of shear coupling spring to relative shear displacement (**coupling-friction-table**);

(16) number of table relating confining stress factor to deviatoric stress (**coupling-confining-table**);

(17) stiffness of normal coupling spring (**coupling-stiffness-normal**) [force/rockbolt length/displ.];

(18) cohesive (and tensile) strength of normal coupling spring (**coupling-cohesion-normal**) [force/rockbolt length];

(19) frictional resistance of the normal coupling spring (**coupling-friction-normal**) [degrees]; and

(20) thermal expansion coefficient (**thermal-expansion**) (optional – used for thermal analysis).

The radius of the rockbolt element cross-section can also be prescribed instead of the area and moment of inertia. The area and moment of inertia will then be calculated automatically.

The exposed perimeter of a rockbolt element and the properties of the coupling springs should be chosen to represent the behavior of the rockbolt/medium interface commensurate with the problem being analyzed. The rockbolt/rock interaction can be expressed in terms of a shear response along the length of the bolt as a result of axial loading and/or in terms of a normal response when the direction of loading is perpendicular to the rockbolt axis.

1.2.3.5 Commands Associated with Rockbolt Elements

All of the commands associated with rockbolt elements are listed in Table 1.3. This includes the commands associated with the generation of rockbolts, and those required to plot and print rockbolt-element variables. See Help in *UDEC* for a detailed explanation of these commands.

Table 1.3 Keywords associated with **block struct rockbolt** *command*

rockbolt	keyword			
	create	keyword		
		begin	*<keyword>*	
			node	*n*
			x y	
		connect		
		connect-distance	*d*	
		end	keyword	
			node	*n*
			x y	
		group	*name*	
		material	*np*	
		segment	*ns*	
		table	*n*	
	delete	*<n1 n2>*		
	list	keyword	keyword	
		element		
			displacment	
			force	
			geometry	
		group		
		interface		
		node		
			displacment	
			force	
			state	
		property		

Table 1.3 Keywords associated with **block struct rockbolt** *command (continued)*

rockbolt	keyword			
	node *n**	keyword		
		create	*x y*	
		fix-x		
		fix-y		
		fix-r		
		free-x		
		free-y		
		free-r		
		initial	keyword	
			disp-x	*value*
			disp-y	*value*
			vel-x	*value*
			vel-y	*value*
			vel-r	*value*
		force	*fx fy m*	
		mat		
		pin		
	prop	*np*	keyword	
			area	*value*
			coupling-cohesion-normal	*value*
			coupling-cohesion-shear	*value*
			coupling-cohesion-table	*n*
			coupling-confining-table	*n*
			coupling-friction-normal	*value*
			coupling-friction-shear	*value*
			coupling-friction-table	*n*
			coupling-stiffness-normal	*value*
			coupling-stiffness-shear	*value*
			density	*value*
			moi	*value*
			perimeter	*value*
			plastic-moment	*value*
			radius	*value*
			spacing	*value*
			tension-failure-strain	*value*
			thermal-expansion	*value*
			yield-compression	*value*
			yield-tension	*value*
			young	*value*

1.2.3.6 Example Application – Rockbolt Pullout Tests

The most common method for determining rockbolt properties is to perform pullout tests on small segments of rockbolts in the field. Typically, segments of 50 cm in length, or longer, are grouted into boreholes. The ends of these segments are pulled with a jack mounted to the surface of the tunnel, and connected to the rockbolt via a barrel-and-wedge type anchor. The force applied to the rockbolt and the deformation of the rockbolt are plotted to produce an axial force-deflection curve. From this curve, the peak shear strength of the grout bond is determined. The results of simulated pullout on one-half meter segments are illustrated in this example.

The data file in Example 1.3 contains several variations of a single rockbolt pull-test. The rockbolt end node is pulled at a small, constant y-direction velocity, as indicated in Figure 1.17. A *FISH* function **ff** is used to sum the reaction forces and monitor nodal displacement generated during the pull-tests.

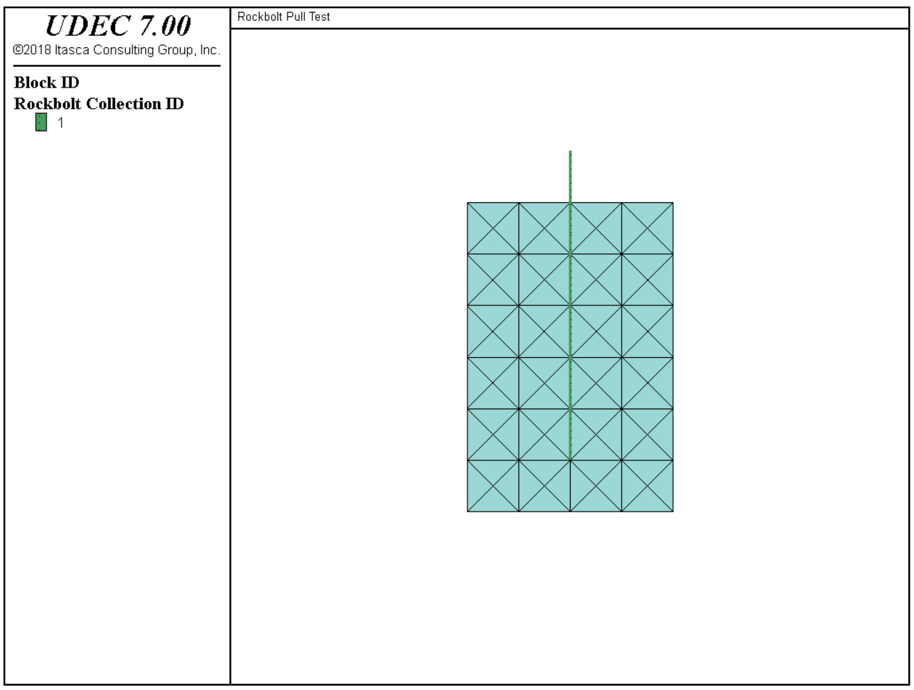


Figure 1.17 Rockbolt element in grid; velocity applied at top end node

Example 1.3 Rockbolt pullout tests

```
model new
model title 'Rockbolt Pull Test'
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone gen quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 range position-y 0.59 0.61
;
block structure rockbolt create begin 0.2 0.1 end 0.2 0.7 segment 12 ...
  material 1
block structure rockbolt property 1 young 200e9 ...
  cross-sectional-area 5e-4 coupling-cohesion-shear 1.0e5 ...
  coupling-stiffness-shear 2.0e7 perimeter 0.08
block structure rockbolt property 1 yield-tension 2.25e5 ...
  moi 2e-8 density 0.001 yield-compression 2.23e5
;struct prop 1 cs_nstiff 1e10 cs_ncoh 2e6 cs_nfric=45
block structure rockbolt node 13 fix-y
block structure rockbolt node 13 initial velocity-y 8e-2
model save 'p1.sav'
;
; --- Fish functions ---
; ff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_end_node
   _inode = block.structure.node.head
   _end_node = 0
   loop while _end_node = 0
      _yp = block.struct.bolt.node.pos.y(_inode)
; node 13
      if _yp > 0.69 then
         _end_node = _inode
      endif
      _inode = block.struct.bolt.node.next(_inode)
   end_loop
end
@_find_end_node
;
fish define pullf
; node 13
   nadd = _end_node
```

```
    dd = block.struct.bolt.node.disp.y(nadd)
    ;ff_elem = fmem(index()+$SELFAX)
    ff_elem = block.struct.bolt.node.force.y(nadd)
    ffnode = block.struct.bolt.node.force.y(nadd)
    ffbou = 0.0
    loop jj (1,5)
       xx = (jj-1) * 0.1
       ig1 = block.gp.near(xx,0.6)
       ibou1 = block.gp.boundary.corner(ig1)
       fb1 = block.boundary.force.y(ibou1)
       ffbou = ffbou+fb1
    endloop
    pullf = ffbou
 end
 fish history @pullf
 fish history @dd
 fish history @ff_elem
 block  smallstrain
 model save 'p2.sav'
 ;
 ;  --- pull out tests - single 25 mm rockbolt (20 mm deformation)
 ;
 ;  --- default behavior ---
 model restore 'p2.sav'
 block mechanical history unbalanced-maximum
 block cycle 20000
 model save 'p3.sav'
 ;
 ;  --- cohesion softening ---
 model restore 'p2.sav'
 block structure rockbolt property 1 coupling-cohesion-table 1
 table 1 add 0 1e5 0.01 1e4 ;change in cohesion with relative shear displ.
 block cycle 20000
 model save 'p4.sav'
 ;
 ;
 ;   --- confinement = 5 MPa (in-plane) ---
 model restore 'p2.sav'
 block structure rockbolt property 1 young 200e9 ...
   cross-sectional-area 5e-4 coupling-stiffness-shear 2.00e7 perimeter 0.08
 block structure rockbolt property 1 yield-tension 2.25e5
 block structure rockbolt property 1 moi 2e-8
 block structure rockbolt property 1 coupling-friction-shear 45
 block structure rockbolt property 1 coupling-cohesion-shear 0.0
 block cycle 1
 block insitu  stress -5e6 0 0 stress-ZZ 0
```

```
block gridpoint apply velocity-x 0 range position-x -0.01 0.01
block gridpoint apply velocity-x 0 range position-x 0.39 0.41
block cycle 20000
model save 'p5.sav'
;
;
;   --- confinement = 5 MPa (in-plane) with conf. str. table ---
model restore 'p2.sav'
block structure rockbolt property 1 young 200e9 ...
  cross-sectional-area 5e-4 coupling-stiffness-shear 2.00e7 perimeter 0.08
block structure rockbolt property 1 yield-tension 2.25e5
block structure rockbolt property 1 moi 2e-8
block structure rockbolt property 1 coupling-friction-shear 45
block structure rockbolt property 1 coupling-cohesion-shear 0.0
;   define table for confining stress correction factor
table 1 add 0 0.5 0.3 0.48 0.5 0.45 0.6 0.39 0.68 0.36
block structure rockbolt property 1 coupling-friction-table 1
;   note : (snn-szz)/(snn+szz) is 1 : cfac = 0.36
block cycle 1
block insitu  stress -5e6 0 0 stress-ZZ 0
block gridpoint apply velocity-x 0 range position-x -0.01 0.01
block gridpoint apply velocity-x 0 range position-x 0.39 0.41
block cycle 20000
model save 'p6.sav'
;
;
;   --- tensile rupture ---
model restore 'p2.sav'
block structure rockbolt property 1 young 200e9 ...
  cross-sectional-area 5e-4 coupling-cohesion-shear 1.00e5 ...
  coupling-stiffness-shear 2.00e7 perimeter 0.08
block structure rockbolt property 1 yield-tension 1.0e5
block structure rockbolt property 1 moi 2e-8
block structure rockbolt property 1 coupling-friction-shear 45
block structure rockbolt property 1 tension-failure-strain 5e-2
block cycle 1
block insitu  stress -5e6 0 0 stress-ZZ 0
block gridpoint apply velocity-x 0 range position-x -0.01 0.01
block gridpoint apply velocity-x 0 range position-x 0.39 0.41
block cycle 17000
model save 'p7.sav'
;
return
```

In the first test, confining stress dependence on the rockbolt shear bond strength is neglected. The resulting axial force-deflection plot is shown in Figure 1.18. The peak force is approximately 50 kN.

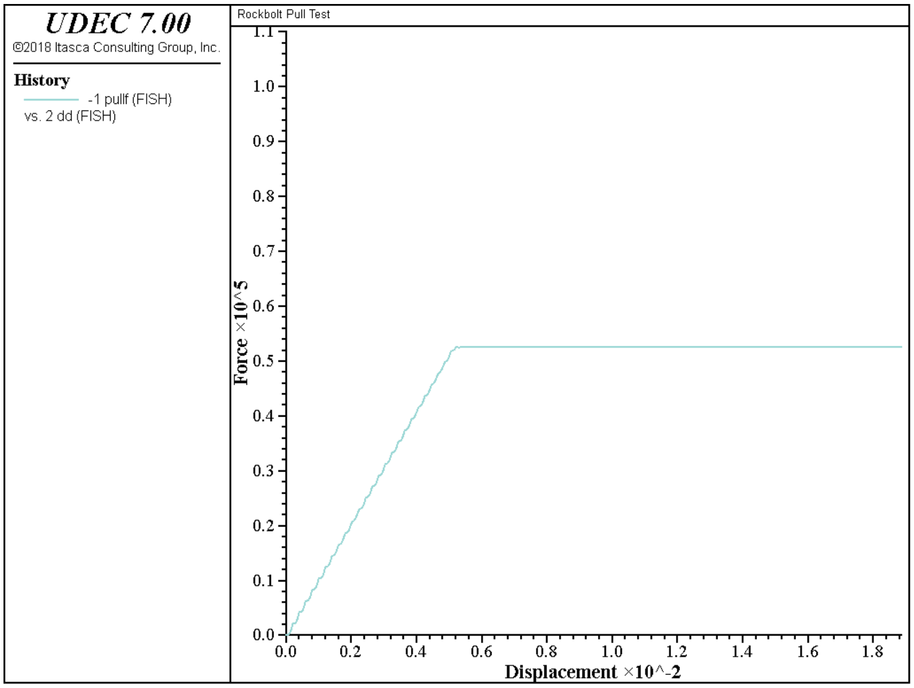


Figure 1.18 Rockbolt pull force (N) versus rockbolt axial displacement (meters) for a single 25 mm grouted rockbolt

In the second test, displacement weakening of the shear bond strength is introduced using the **coupling-cohesion-table** property. The displacement weakening relation to shear displacement is defined in table 100. The results are shown in Figure 1.19.

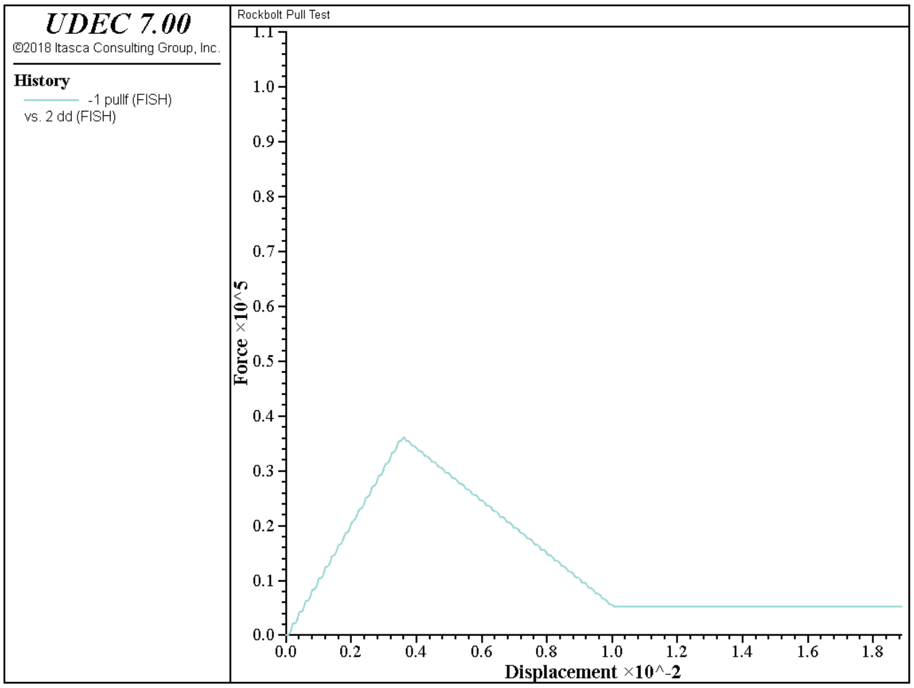


Figure 1.19 Rockbolt pull force (N) versus rockbolt axial displacement (meters) for a single 25 mm grouted rockbolt – with displacement weakening

The rockbolt shear bond strength will, in general, increase with increasing effective pressure acting on the rockbolt. A linear law is implemented in *UDEC*, whereby the rockbolt shear strength is defined as a constant (**coupling-cohesion-shear**) plus the effective pressure on the rockbolt multiplied by the rockbolt perimeter (**perimeter**) times the tangent of the friction angle (**coupling-friction-shear**). The pressure dependence is activated automatically by issuing the rockbolt properties **perimeter** and **coupling-friction-shear**.

In the third test, a 5 MPa in-plane confining stress is applied *after* the rockbolt is installed. Note that one calculational step is taken, in order to assign the rockbolt properties before the confining stress is applied. The results are shown in Figure 1.20.

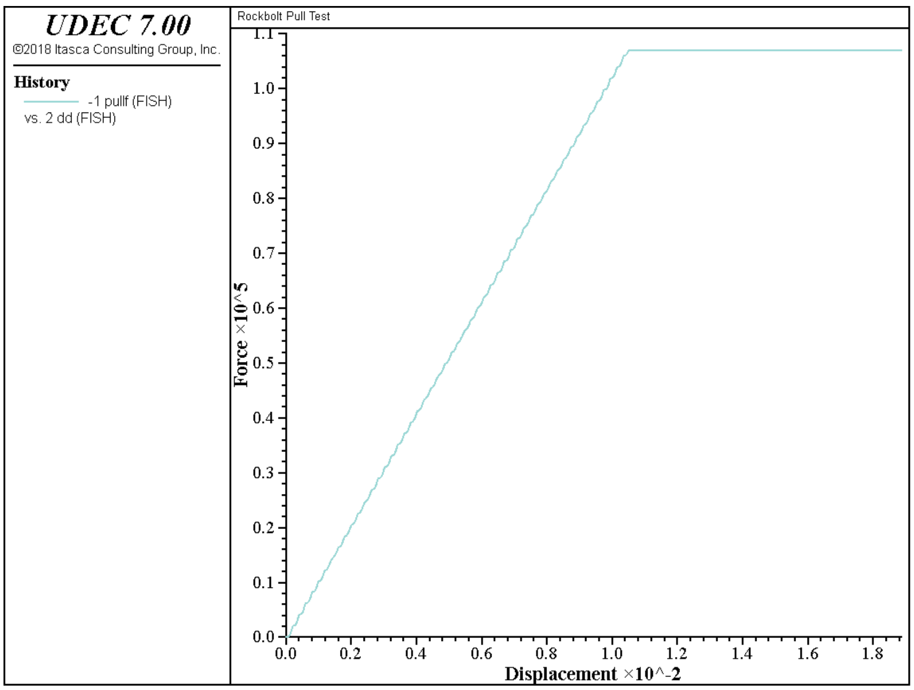


Figure 1.20 Rockbolt pull force (N) versus rockbolt axial displacement (meters) for a single 25 mm grouted rockbolt – with 5 MPa in-plane confinement and zero out-of-plane confinement

In the fourth test, the property **coupling-confining-table** is used to define the confining stress applied to the rockbolt, accounting for the reduced affect of the out-of-plane stress and the in-plane stress normal to the bolt. Table 1 is used to apply the reduction factor. The results are shown in Figure 1.21. Note that the pullout resistance is reduced compared to the previous case (compare Figure 1.21 to Figure 1.20).

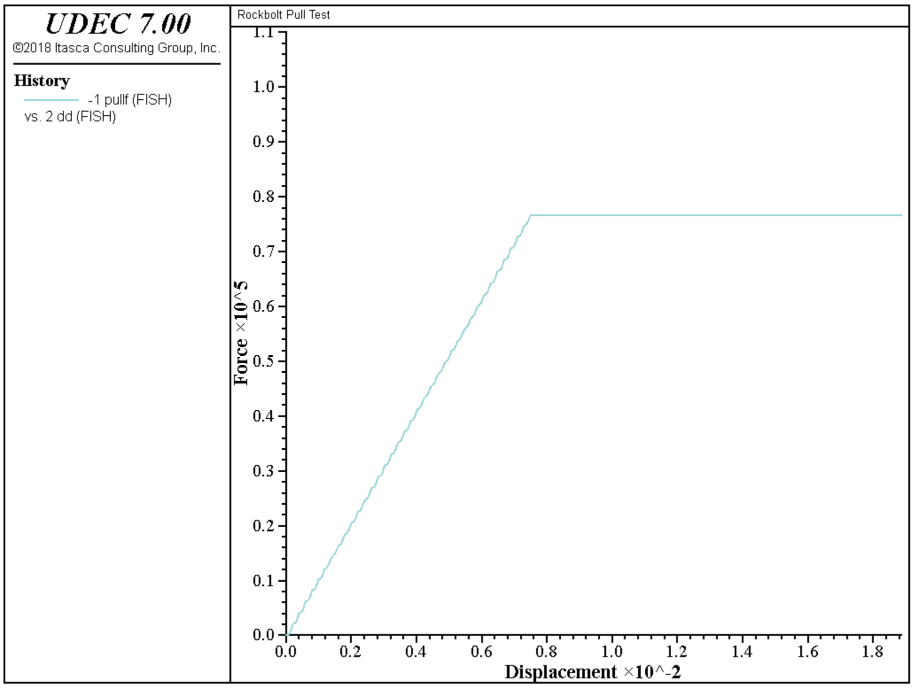


Figure 1.21 ***Rockbolt pull force (N) versus rockbolt axial displacement (meters) for a single 25 mm grouted rockbolt – with 5 MPa in-plane confinement plus a reduction factor and zero out-of-plane confinement***

In the fifth test, **yield-tension** is used to define the limiting axial yield force (100 kN) of the bolt, and **tension-failure-strain** is used to define the plastic strain (0.05) at which the bolt ruptures. The results are shown in Figure 1.22.

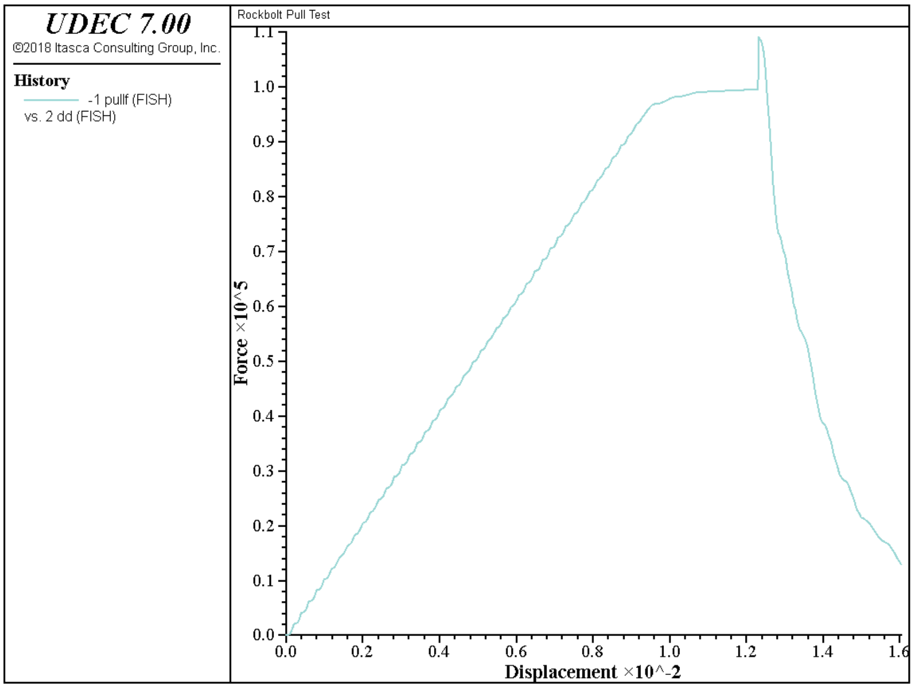


Figure 1.22 Rockbolt pull force (N) versus rockbolt axial displacement (meters) for a single 25 mm grouted rockbolt – with tensile rupture

1.2.3.7 Example Application – Rockbolt Shear Tests

Two shear tests are performed in this example. The tests use the same model as the pullout tests. In this case, though, a horizontal velocity is applied to the top rockbolt node. The data file is listed in Example 1.4. Note that normal coupling spring properties are now included.

Example 1.4 Rockbolt shear tests

```
model new
model title 'rockbolt Shear Test'
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.3 0.6 0.3 0
block zone gen quad 0.12
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-x 0 range pos-x -0.01 0.31 pos-y -0.01 0.01
block gridpoint apply velocity-y 0 range pos-x -0.01 0.31 pos-y -0.01 0.01
block gridpoint apply velocity-x 0 range pos-x -0.01 0.01 pos-y -0.01 0.61
block gridpoint apply velocity-x 0 range pos-x 0.29 0.31 pos-y -0.01 0.61
block structure rockbolt create begin 0.15 0.1 end 0.15 0.625 ...
  segment 25 material 4
block structure rockbolt property 4 young 200e9 ...
  cross-sectional-area 5e-4 coupling-cohesion-shear 1.00e5 ...
  coupling-stiffness-shear 2.00e7 perimeter 0.08
block structure rockbolt property 4 yield-tension 2.25e5
block structure rockbolt property 4 moi 2e-8
block structure rockbolt property 4 coupling-stiffness-normal 1e10 ...
  coupling-cohesion-normal 2e6 coupling-friction-normal 45
block structure rockbolt property 4 density 1000.0
;
block structure beam node 26 fix-x
block structure beam node 26 initial velocity-x 8e-2
model save 'shear1.sav'
;
;
; --- Fish functions ---
; ff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_node
   _inode = block.structure.node.head
   _this_node = 0
   loop while _this_node = 0
      _yp = block.struct.bolt.node.pos.y(_inode)
      _ydis = math.abs(_yp - _ynode)
```

```
      if _ydis < .01 then
         _this_node = _inode
      endif
      _last_node = _inode
      _inode = block.struct.bolt.node.next(_inode)
      if _inode = 0 then
         _this_node = _last_node
      endif
   end_loop
   _find_node = _this_node
end

fish define _nodes
   _ynode = 0.625
   _node26 = _find_node
   _ynode = 0.6040
   _node25 = _find_node
   _ynode = 0.583
   _node24 = _find_node
end
@_nodes

fish define shear_force
; node 26
   nadd = _node26
   dd = block.struct.bolt.node.disp.x(nadd)
   ffnode = block.struct.bolt.node.force.x(nadd)
   dx26 = block.struct.bolt.node.disp.x(nadd)
   dy26 = block.struct.bolt.node.disp.y(nadd)
   dr26 = block.struct.bolt.node.disp.rot(nadd)
   fx26 = block.struct.bolt.node.force.x(nadd)
; node 25
   nadd = _node25
   dx25 = block.struct.bolt.node.disp.x(nadd)
   dy25 = block.struct.bolt.node.disp.y(nadd)
   dr25 = block.struct.bolt.node.disp.rot(nadd)
   fx25 = block.struct.bolt.node.force.x(nadd)
; node 24
   nadd = _node24
   dx24 = block.struct.bolt.node.disp.x(nadd)
   dy24 = block.struct.bolt.node.disp.y(nadd)
   dr24 = block.struct.bolt.node.disp.rot(nadd)
   fx24 = block.struct.bolt.node.force.x(nadd)
;
   ffbou = 0.0
   loop jj (1,7)
```

```
      yy = (jj-1) * 0.1
      ig1 = block.gp.near(0.0,yy)
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boundary.force.x(ibou1)
      ffbou = ffbou+fb1
   endloop
   loop jj (1,7)
      yy = (jj-1) * 0.1
      ig1 = block.gp.near(0.3,yy)
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boundary.force.x(ibou1)
      ffbou = ffbou+fb1
   endloop
   loop jj (1,2)
      xx = (jj) * 0.1
      ig1 = block.gp.near(xx,0.0)
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boundary.force.x(ibou1)
      ffbou = ffbou+fb1
   endloop
   shear_force = ffbou
end
; --- Histories ---
hist interval 100
fish history @shear_force
fish history @dd
fish history @ffbou
fish history @ffnode
block mechanical history unbalanced-maximum
block gridpoint history displacement-y 0.2 0.6
fish history @dx26
fish history @dy26
fish history @dr26
fish history @fx26
fish history @dx25
fish history @dy25
fish history @dr25
fish history @fx25
fish history @dx24
fish history @dy24
fish history @dr24
fish history @fx24
model save 'shear2.sav'
;
; --- shear test ---
block cycle 30000
```

```
model save 'shear3.sav'
;
; --- bolt ruptures
model restore 'shear2.sav'
block structure rockbolt property 4 plastic-moment 5e3 ...
tension-failure-strain 1e-2
history interval 70
block cycle 30000
model save 'shear4.sav'
;
return
```

Figure 1.23 shows the plot of shear force versus shear displacement for a non-yielding bolt. Figure 1.24 shows the rockbolt geometry at the end of the test. The large displacement of the rockbolt near the rock surface is a result of the failure of the normal coupling springs, which simulates the crushing of the rock.

In the second test, **plastic-moment** is specified to define a limiting moment (5000 N-m) of the bolt, and **tension-failure-strain** is set to define a limiting plastic strain (0.01) at which the bolt ruptures. The results are shown in Figures 1.25 and 1.26.

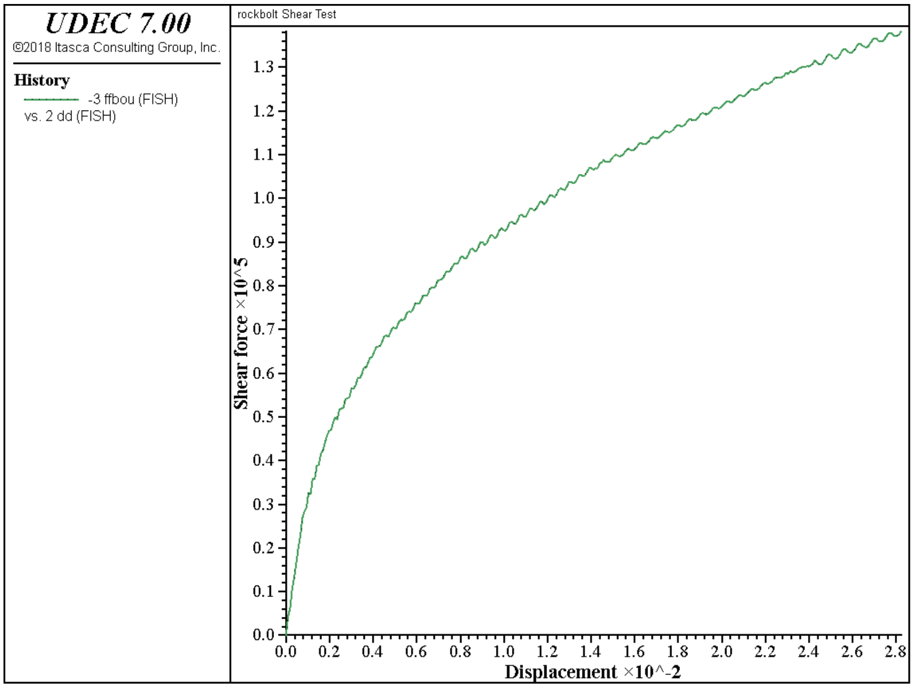


Figure 1.23 Rockbolt shear force (N) versus rockbolt shear displacement (meters) for a single 25 mm grouted rockbolt

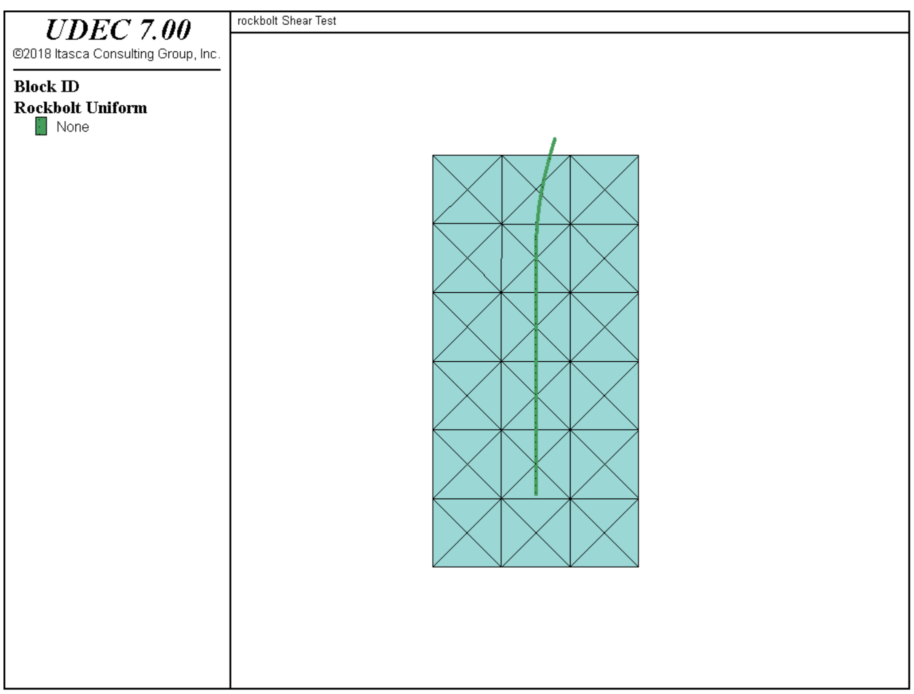


Figure 1.24 Deformed shape of 25 mm diameter rockbolt at end of shear test

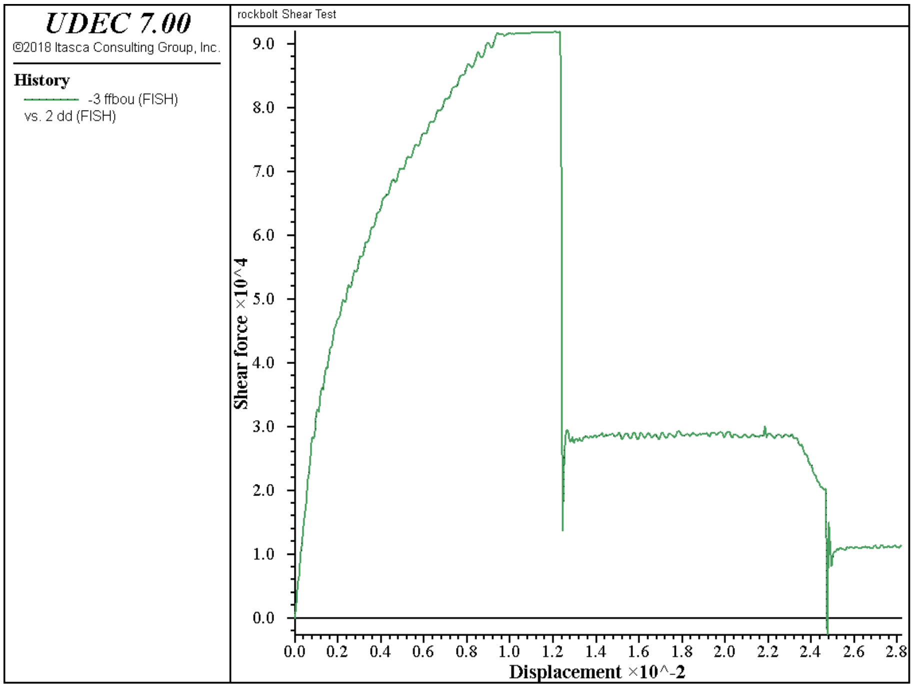


Figure 1.25 Rockbolt shear force (N) versus rockbolt shear displacement (meters) for a single 25 mm grouted rockbolt – with tensile rupture

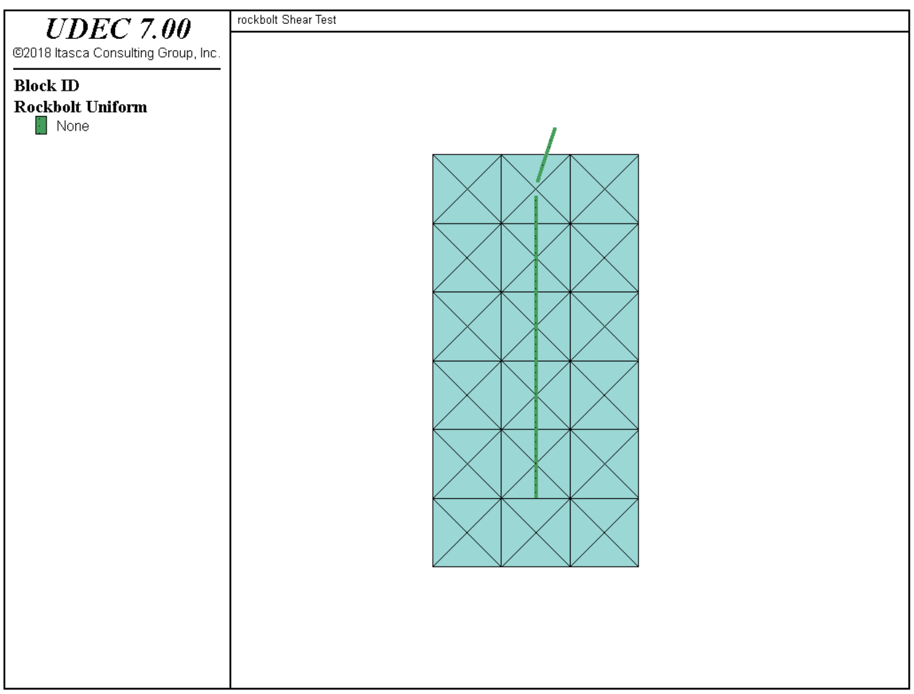


Figure 1.26 Deformed shape of 25 mm diameter rockbolt following rupture at end of shear test

1.3 Surface Support

Two types of structural elements are available in *UDEC* to simulate support placed on exposed rock surfaces. The first type consists of two-dimensional structural (beam) elements. Surface linings for tunnels and exposed slopes are typically thin, and their characteristic response to bending usually cannot be neglected. The beam element formulation provides an effective method to include bending effects. The beam elements are attached to a rock surface via spring connections oriented both radially and tangentially with respect to the support structure.

The second type of structural element consists of one-dimensional support members that are attached to two boundaries of an interior surface. The support member has no independent degrees of freedom, but simply imposes forces on the surfaces to which it is connected. Support members are intended to model props or packs that are placed to support an underground opening.

The characteristic behaviors of these two support types are described in the following sections.

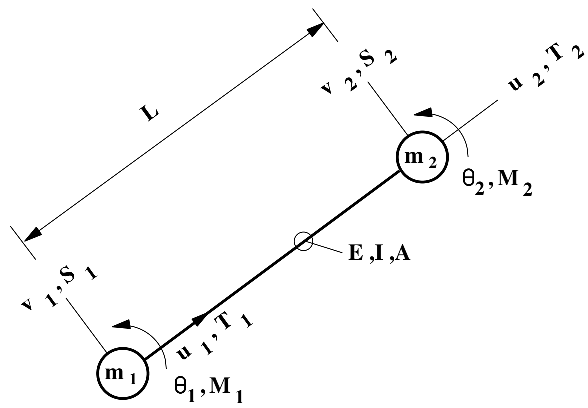
*1.3.1 Structural Beam Elements (***block structure liner create** *Command)*

The structural element method is well-documented in structural engineering texts. The use of beam elements in two-dimensional linear analysis of excavation support is reported by Dixon (1971), Brierley (1975) and Monsees (1977), among others. Paul et al. (1983) present analyses using beam elements that include nonlinear behavior. Analysis of any support structure is initiated by discretization of the structure into a number of elements whose response to axial, transverse and flexural loads can be represented in matrix form, as shown in Figure 1.27.

The rock-structure interface is represented by springs connected between the structural element nodes and the *UDEC* model. The structural beam elements can be connected to rigid blocks or deformable blocks.

1.3.1.1 Structural (Beam) Element Formulation

Either an implicit or explicit formulation may be used in analyzing the behavior of a support structure composed of beam elements and interface stiffnesses. In the first formulation (implicit), a global stiffness matrix is formed for the entire structure. The size of the stiffness matrix is reduced by deleting free nodes (i.e., those nodes that are not located at the rock-support interface). This is possible because these nodes are not subjected to directly imposed external loads or displacements by the surrounding medium. The resultant efficiency, however, limits straightforward application of this formulation to quasi-static problems involving linear elastic behavior. This formulation does not provide information about failure mechanisms or ultimate capacities of interior supports. However, factors of safety based on lining stresses should be conservative since they do not take into account the highly indeterminate nature of a lining in contact with the rock. A detailed description of this formulation, its use with distinct elements and numerous other examples are presented by Lorig (1984).



Structural Element Sign Convention

$$
\begin{bmatrix} T_1 \\ S_1 \\ M_1 \\ T_2 \\ S_2 \\ M_2 \end{bmatrix}
= \frac{E}{L}
\begin{bmatrix}
A & & & & & \\
0 & \frac{12\,I}{L^2} & & \text{SYM.} & & \\
0 & \frac{6\,I}{L} & 4\,I & & & \\
-A & 0 & 0 & A & & \\
0 & -\frac{12\,I}{L^2} & -\frac{6\,I}{L} & 0 & \frac{12\,I}{L^2} & \\
0 & -\frac{6\,I}{L} & 2\,I & 0 & -\frac{6\,I}{L} & 4\,I
\end{bmatrix}
\begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{bmatrix}
$$

Figure 1.27 Local stiffness matrix for structural element representation of excavation support

In the second formulation (explicit), local stiffness matrices are used following division of the structure into segments with the distributed mass of the structure “lumped” at nodal points, as shown in Figure 1.28. Forces generated in support elements are applied to the lumped masses, which move in response to unbalanced forces and moments in accordance with the equations of motion. This formulation has two desirable characteristics: (1) slip between support and excavation periphery is modeled in a manner identical to block interaction along a discontinuity; and (2) large displacements with nonlinear material behavior are readily accommodated. These capabilities are illustrated in Figure 1.29, in which a roof block loads and displaces a hypothetical four-element interior structural support.

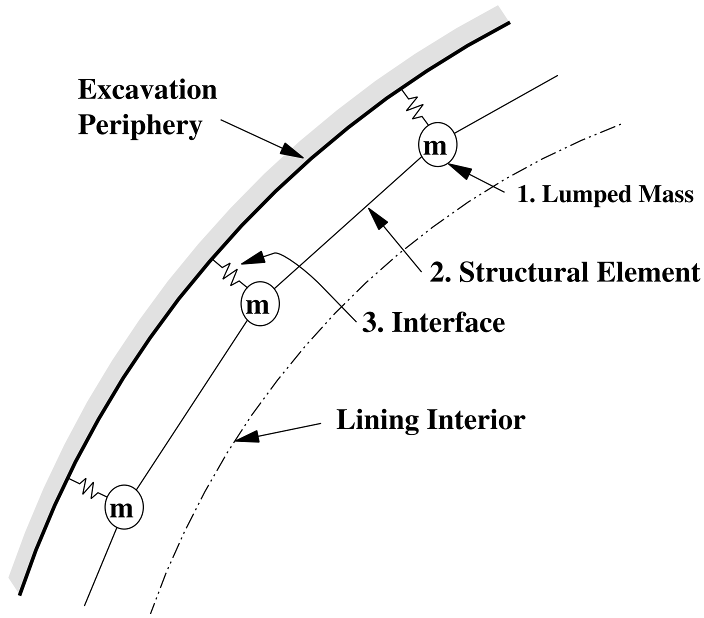


Figure 1.28 Lumped mass representation of structure used in explicit formulation

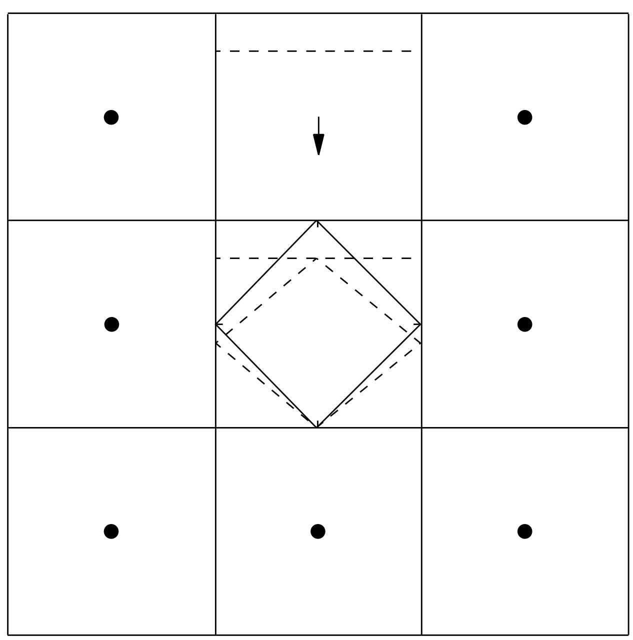


Figure 1.29 Demonstration of interface slip and large displacement capabilities of explicit structural element formulation

The structural (beam) elements include an elastic-plastic material model that incorporates bending resistance, limiting bending moments and yield strengths of the beam material.* The material model can simulate inelastic behavior that is representative of common surface-lining materials. This includes materials (such as steel) that behave in a ductile manner, as well as non-reinforced and reinforced cementatious materials (such as concrete and shotcrete) that can exhibit either brittle or ductile behavior. Note that shear failure is *not* included in the material model.

The behavior of the material model can be shown on a moment-thrust interaction diagram, such as that given in Figure 1.30. Moment-thrust diagrams are commonly used in the design of concrete columns. These diagrams illustrate the maximum force that can be applied to a typical section for various eccentricities. The ultimate failure envelopes for non-reinforced and reinforced cementatious materials are similar. However, reinforced materials have a residual capacity that remains after failure at the ultimate load. Non-reinforced cementatious materials typically have no residual capacity.

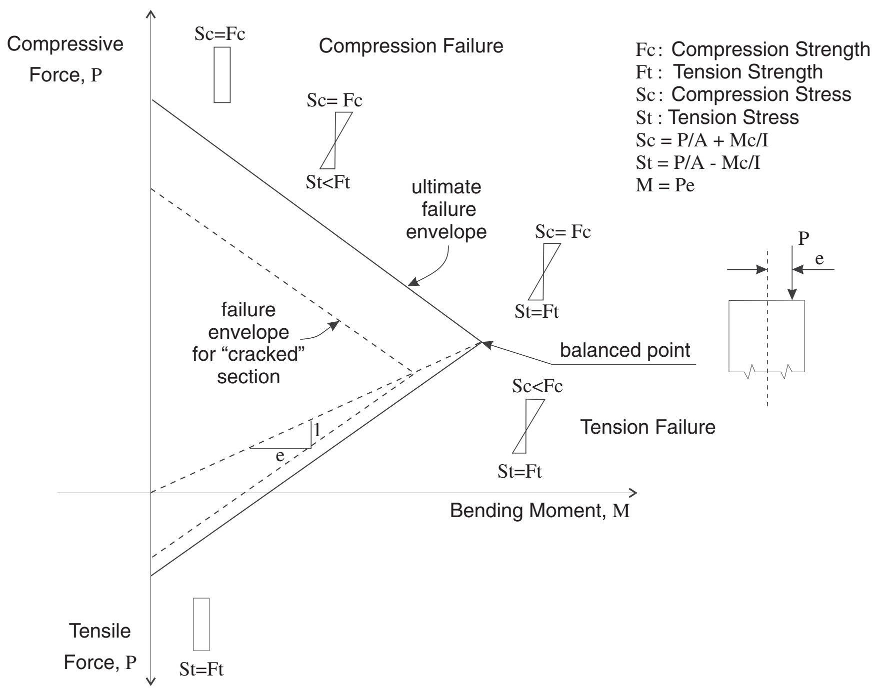


Figure 1.30 Typical moment-thrust diagram

Interaction diagrams can be constructed by specifying the section geometry and compressive and tensile strengths (in terms of stress) for the material. The thickness and compressive and tensile strengths are input (with property keywords **yield-compression** and **yield-tension**, respectively), and the model uses this information to determine the ultimate capacity for various eccentricities (*e* in

* The inelastic material behavior model was developed with funding from the Norwegian Geotechnical Institute, NGI, Oslo, Norway, for application to the analysis of fiber-reinforced shotcrete.

Figure 1.30). As the calculation progresses, the axial forces and moments in the structural elements are compared to the ultimate capacity. When a node reaches the ultimate capacity, a "fracture" flag is set, indicating that all future evaluations for that node will use the "cracked" failure envelope and the residual strength capacity (specified with the property keyword **yield-compression-cracked** for residual compressive strength and **yield-tension-residual** for residual tensile strength).

The diagram in Figure 1.30 is defined by the three points (1, 2 and 3) shown in the figure. The axial force, P, and moment, M, at these three points are calculated:

at Point 1

$$\begin{aligned} P_1 &= F_c A \\ M_1 &= 0 \end{aligned} \tag{1.29}$$

at Point 2

$$\begin{aligned} P_2 &= F_t A \\ M_2 &= 0 \end{aligned} \tag{1.30}$$

at Point 3

$$\begin{aligned} P_3 &= \frac{(F_c + F_t)}{2} A \\ M_3 &= I \frac{(F_c - F_t)}{h} \end{aligned} \tag{1.31}$$

where F_c and F_t are the initial compressive and tensile strengths of the material, A is the cross-sectional area, I is the moment of inertial and h is the thickness of the liner section.

The diagram in Figure 1.30 is developed assuming that the liner is initially uncracked. For unreinforced concrete or shotcrete, some cracking in the liner section may be permissible. The effect of cracking is approximated in the liner model by setting $F_t = 0$ and introducing an extra point, Point 4, which extends the ultimate failure envelope when a tensile crack exists. The crack depth, h_c, is generally limited to half the total section thickness, and can be related to the eccentricity, e_4, when the compressive stress reaches the compressive strength, F_c, by the equation

$$e_4 = \frac{h}{6} + \frac{h_c}{3} \tag{1.32}$$

The axial force and moment at the extra point (Point 4) are then calculated as

$$\begin{aligned} P_4 &= \frac{1}{2} F_c A (1 - \frac{h_c}{h}) \\ M_4 &= \frac{1}{12} F_c A h (1 + \frac{h_c}{h} - 2(\frac{h_c}{h})^2) \end{aligned} \tag{1.33}$$

The crack depth ratio, h_c/h, is assigned to beam elements as a property via the **crack-depth-ratio** keyword.

A parabolic shape of the failure envelope is often defined for the moment-thrust diagram in unreinforced liner design. The bending diagram can be input directly as a lookup table in *UDEC*. The table stores the P-M diagram, and then is assigned as the failure envelope by specifying the table number using the **moment-thrust-table** property keyword.

An example application given in Section 1.3.1.6 illustrates the inelastic material behavior of the structural (beam) element, and the construction of moment-thrust diagrams for the different cases of beam material behavior described above.

1.3.1.2 Structural Element Generation

Structural elements are generated as a liner along a surface that is a boundary of a domain. The domain can be either an internal region in the model or the exterior (outer domain) boundary. Two methods are provided in *UDEC* to generate structural element liners.

Liner Generated by Specifying Liner/Block Interface Contact Points – In this method, a liner is created by defining a starting point and ending point of the interface contact points for the liner along the block internal or external boundary using the command

```
block struct liner create by-end-points begin xb, yb end xe, ye ...
  mat-beam value mat-int value
```

The starting and ending point coordinates (x_b, y_b) and (x_e, y_e) are positioned, by default, at midpoints between block boundary corners (for rigid blocks) or at midpoints between boundary gridpoints (for deformable blocks). Additional structural nodes can be added along the liner surface by using the **length-max** keyword, which sets the maximum size for structural element segments along the liner surface.

The keyword **length-min** can be used to prevent structural nodes along the liner surface from being too close to one another. By default, **length-min** is set to the minimum block edge length.

Alternatively, all of the interface contact points can be generated individually by using the **block struct beam create by-table *nt*** command to generate the liner. In this case, a table is first created using the **table** command in which the x, y pairs of the table correspond to the interface contact points for the liner/block. Points in the table should lie within a tolerance of the rounding length to the block edge, in order for these points to be recognized as interface contact points for the liner.

The liner can be generated one segment at a time by using the **block struct beam create single begin** x_b, y_b **end** x_e, y_e command to generate each beam segment along the liner surface. Also, structural nodes can be generated individually by using the **lock struct bem node** command, and then beam segments can be created by connecting nodes with the command **block struct beam create single begin node *n1* end node *n2***.

Liner Generated by "Spraying" – In the second method, interface contact points for the liner/block interaction are "sprayed" from a central point located within the domain. (The central point must be *within* the domain that contains the surface to receive the lining.) The extent of the lined region is controlled by specifying (1) the angle of the first contact point (a positive angle is measured counterclockwise about the central point from the positive x-axis), and (2) the total angle of the surface to receive the lining (a positive angle is measured counterclockwise about the central point from the angle of the first contact point). See Figure 1.31 for an illustration of these locations. If these limits are not specified, the entire boundary surface for the domain will be lined.

An average number of contact points for the lining is selected by the user. Contact points will be generated to support all block edges along the surface. The total number of contact points generated can be greater than the average number specified.

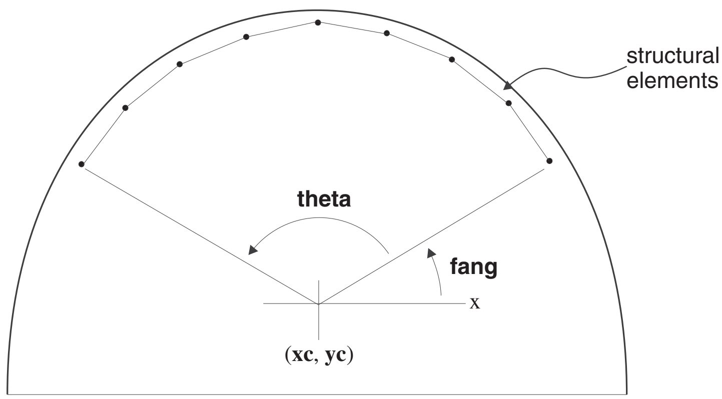


Figure 1.31 Parameters to define structural element locations

Layering Structural Element Segments – Multiple sections of lining can be created along the same surface. For example, two different linings can be applied in layers to the same surface. Interaction will occur at the interface between the linings. There is no restriction to the number of layers that can be applied. Structural element contacts will be created between the layers. Different material properties can be assigned to the contacts between two layered segments by using the **material-interface** keyword with the **block struct liner create by-angle** command to specify a separate material number for properties between two layers.

The user must check for proper placement of structural element layers to be sure that they will interact. There must be at least one structural node of the new layer overlapping the previous layer for the layers to interact. Overlapping layers are detected if the new layer is within a space that is 0.4 times the thickness of the previous layer. The overlap space can be controlled with the **connect-distance** keyword.

Connecting Structural Element Segments – Structural nodes of different segments that are located at the same position can be connected by specifying the **connect** keyword at the end of the **block struct liner create** command. The structural element segments can only connect at end nodes. The ends must be located within the rounding length tolerance, to connect. The result of connecting two structural element segments is one segment. There will be a skip in the node ID numbers because one node will be deleted at the connection.

Cable nodes can also be connected to structural element nodes. See Section 1.2.2.7.

Deleting Structural Elements – Structural element segments can be deleted at any time in the calculation process by specifying the command **block struct beam delete** followed by a range that includes the centroid of the element segment. For example, to delete structural elements along a circular excavation boundary, use the command

```
block struct beam delete range annulus center (0,0) radius 9.9 10.1
```

1.3.1.3 Structural Element Properties

The structural beam elements used in *UDEC* require three sets of input parameters: geometry parameters, constitutive model and material number parameters, and liner/block interface and structural element properties. These parameters are listed below.

Geometry Parameters – Geometry parameters generate the geometry and specify the general conditions of the structural element lining. This set of parameters can be given in two different forms.

If the lining is generated by specifying individual liner/block contact points along a boundary surface, the input parameters are

(1) x- and y-coordinates of starting and ending points along the lining surface;

(2) maximum length of a structural element segment;

(3) minimum length of a structural element segment; and

(4) material number for element and interface properties.

If the lining is generated by spraying the liner/block contact points within a domain, the parameters are

(1) x- and y-coordinates of a point within the domain that contains the lined surface;

(2) average number of interface contact points;

(3) material number for element and interface properties;

(4) angle of first contact point (positive angle is measured counterclockwise from the positive x-axis);

(5) total angle of structural element lining (positive angle is measured counterclockwise from the first contact point); and

(6) minimum length of a structural element segment.

Constitutive Model and Material Number Parameters – The second set of parameters specifies (or changes) the element or liner/block interface material conditions. These parameters can be used to assign different conditions and properties to different portions of the lining. The parameters include

(1) interface constitutive model;

(2) material number for element properties; and

(3) material number for interface contact properties.

Liner/Block Interface and Structural Element Properties – The third set specifies the material properties for the element and the interface. These properties are

(1) elastic modulus [stress];

(2) Poisson's ratio;

(3) density;

(4) tensile yield strength [stress];

(5) residual tensile yield strength [stress];

(6) compressive yield strength [stress];

(7) residual compressive yield strength (stress);

(8) ratio of crack depth to initial cross-section thickness;

(9) P-M diagram lookup table;

(10) interface normal stiffness [stress/unit displacement];

(11) interface shear stiffness [stress/unit displacement];

(12) interface cohesion [stress];

(13) interface friction [degrees];

(14) interface dilation [degrees];

(15) interface tensile strength [stress];

(16) element shape;

(17) element second moment of inertia;

(18) element thickness;

(19) element cross-sectional area;

(20) element width; and

(21) element spacing.

Note that property numbers are assigned to structural elements and interfaces with the **block struct liner/beam create ... material-beam material-interface** command. Property numbers can be changed locally for structural element material, and for interface material, by using the **block struct beam change material-beam *n*** material property number for the structural element material, and the **block struct change material-interface *n*** material property number for the interface material. Each different structural element and interface point can then be assigned geometric and material properties by specifying the **block struct beam property** command with the appropriate property keywords following the structural-element material property number and the interface material property number. For example,

```
bl str beam change mat-int 2
bl str beam prop mat=2 coupling-stiff-nor=1e9 coup-coh=2e5
```

assigns an interface stiffness value of 10^9, and an interface, cohesive strength value of 2×10^5 to property number 2. Note that the interface contacts behave according to the **model 5** constitutive model (i.e., the properties are specified in terms of stress and displacement).

If the structural element has unit dimension in the out-of-plane direction, then only the thickness is required; the area, moment of inertia and shape factor are calculated automatically. For rectangular shapes, the shape factor is 5/6. Figure 1.32 lists shape factors and inertial moments for various shapes.

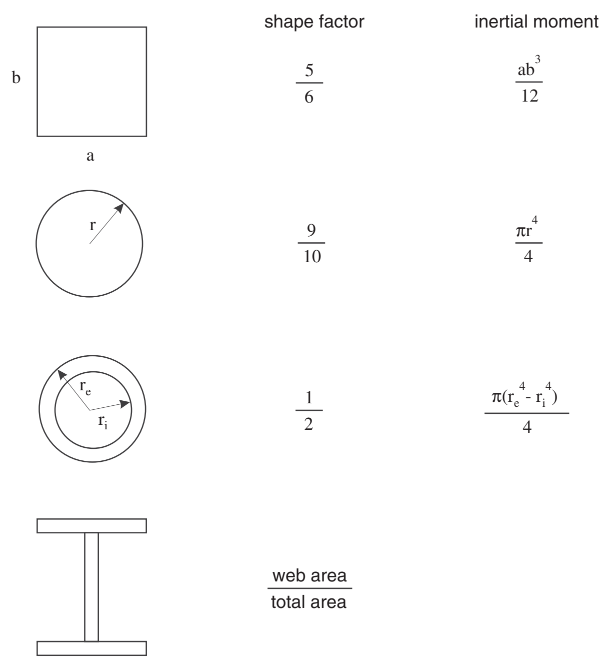


Figure 1.32 Shape factors and inertial moments for different shapes

Structural element properties are easily calculated or obtained from handbooks. Composite systems, such as reinforced concrete, should be based on the transformed section. Note that the structural element formulation is a *plane-stress* formulation. If the element is representing a structure that is continuous in the direction perpendicular to the analysis plane (e.g., a concrete tunnel lining), the value specified for elastic modulus should be divided by $(1 - \nu^2)$ to account for plane-strain conditions.

Note that the mass density is required for the structural element formulation.

1.3.1.4 End Conditions and Applied Pressure

The supplemental command **block struct beam node** ***n*** provides options for describing the end conditions of beam nodes. The options include

(1) free or fixed x- and$y$-displacements or rotations;

(2) applied velocities; and

(3) applied loads or moments.

These options are given by the following qualifying keywords following the node number ***n***.

fix-x fix-y fix-r

This option allows beam node ***n*** to have fixed x- and/or y-velocities or fixed angular velocities.

free-x free-y free-r

This removes the constraint set by the **fix** keyword. (The default condition is free.)

initial keyword

Certain node variables can be assigned initial values. The following keywords apply.

displacement**-x** ***value***

x-displacement for beam nodes

displacement**-y** ***value***

y-displacement for beam nodes

velocity**-r**otation ***value***

rotational velocity for beam nodes

velocity**-x** ***value***

x-velocity for beam nodes

velocity**-y** ***value***

y-velocity for beam nodes

force ***fx, fy, mom***

This allows the user to apply x- and/or y-direction forces or moments to node ***n*** for beams.

material ***n***

Changes material-interface number for nodes

pin fixes location of nodes

The logic used in solving for forces on structural (beam) elements when applied as tunnel liners in saturated conditions is not capable of determining the fluid pressure difference between the inside and outside of the liner. The command **block struct liner apply pressure inside** ***p*** **outside** ***p*** allows the specification of the pressure difference so that the liner loads include hydrostatic forces. The keyword **inside** is the pressure inside the tunnel, and the keyword **outside** is the pressure at the block/liner interface. This pressure is applied to the block.

If gravity is acting and the water density is specified, a pressure gradient may be applied to **inside** and **outside**. These values are taken as the pressure at a given reference location, and the hydrostatic gradient is superimposed. The command **block struct liner apply pressure reference-location** ***x y*** is used to set the reference location. If the reference location is not given, no gradient is applied. Fluid properties and gravity must be set for the calculation to work.

1.3.1.5 Summary of Commands Associated with Structural Elements

All of the commands associated with structural elements are listed in Table 1.4. See Help in *UDEC* for a detailed explanation of these commands.

Table 1.4 Keywords associated with **block struct liner** *command*

apply	keyword		
	pressure	keyword	
		inside	*p*
		outside	*p*
		ref-loc	*x y*
create	keyword		
	by-angle	keyword	(spray contacts)
		angle-begin	*value*
		angle-span	*value*
		center	*x y*
		connect	
		connect-dist	*value*
		group	*name*
		length-max	*value*
		length-min	*value*
		mat-beam	*n*
		mat-int	*n*
		points	*np*
	by-end-points	keyword	(specify contacts)
		begin	*x y*
		connect	
		connect-dist	*value*
		end	*x y*
		group	*name*
		length-max	*value*
		length-min	*value*
		mat-beam	*n*
		mat-int	*n*
delete	*<range>*		
list	keyword		
	element	<keyword>	
		<**displacement**>	
		<**force**>	
		<**geometry**>	
	enclosed-area		
	group		
	interface		
	property		

Table 1.4 Keywords associated with **block struct liner** *command (continued)*

	node	<keyword>	
		<**disp**>	
		<**force**>	
		<**state**>	
node *n*	keyword		
	fix-x		
	fix-y		
	fix-r		
	force	*fx fy m*	
	free-x		
	free-y		
	free-r		
	initial	keyword	
		dispacement-x	*value*
		dispacement-y	*value*
		velocity-x	*value*
		velocity-y	*value*
		velocity-r	*value*
	material	*mat*	
	pin		

Table 1.4 Keywords associated with **block struct liner** *command (continued)*

PROPERTY	**mat** *n*	keyword	
		coupling-cohesion	*value*
		coupling-dilation	*value*
		coupling-friction	*value*
		coupling-kn	*value*
		coupling-ks	*value*
		coupling-tensile	*value*
		st_area	*value*
		st_density	*value*
		st_inertia	*value*
		st_pmtable	*n*
		st_prat	*value*
		st_rcrack	*value*
		st_scresid	*value*
		st_shape	*value*
		st_spacing	*value*
		st_thermal-expansion	*value*
		st_thick	*value*
		st_width	*value*
		st_yield-compression	*value*
		st_yield	*value*
		st_ymod	*value*
		st_yresid	*value*

Table 1.5 Keywords associated with **block struct beam** *command*

change		keyword	
		material-beam	*n*
		material-interface	*n*
		model-interface	*n*
create	keyword		
	by-table *n*	keyword	
		connect	
		connect-dist	*value*
		group	*name*
		mat-beam	*n*
		mat-int	*n*
		no-layer	
	single	keyword	
		begin	*x y*
		connect	
		connect-dist	*value*
		end	*x y*
		group	*name*
		mat-beam	*n*
		mat-int	*n*
		no-layer	
delete			*<range>*
list	keyword		
	element	<keyword>	
		<**displacement**>	
		<**force**>	
		<**geometry**>	
	group		
	interface		
	node	<keyword>	
		<**disp**>	
		<**force**>	
		<**state**>	
	property		

Table 1.5 Keywords associated with **block struct beam** *command (continued)*

node *n*	keyword		
	fix-x		
	fix-y		
	fix-r		
	force	*fx fy m*	
	free-x		
	free-y		
	free-r		
	initial	keyword	
		dispacement-x	*value*
		dispacement-y	*value*
		velocity-x	*value*
		velocity-y	*value*
		velocity-r	*value*
	material	*mat*	
	pin		

Table 1.5 Keyowrds associated with **block struct beam** *commands (continued)*

property	**mat** *n*	keyword	
		coupling-cohesion	*value*
		coupling-dilation	*value*
		coupling-friction	*value*
		coupling-stiff-normal	*value*
		coupling-stiff-shear	*value*
		coupling-tension	*value*
		crack-depth-ratio	*value*
		cross-sectional-area	*value*
		density	*value*
		moi	*value*
		moment-thrust-table	*n*
		poisson	*value*
		shape-factor	*value*
		shear	*value*
		spacing	*value*
		thermal-expansion	*value*
		thickness	*value*
		width	*value*
		yield-compression	*value*
		yield-compression-crack	*value*
		yield-tension	*value*
		yield-tension-residual	*value*
		young	*value*

1.3.1.6 Example Application – Inelastic Material Behavior of a Cantilever Beam

A simple cantilever beam test is performed with a structural (beam) element to illustrate the inelastic material behavior of the beam and compare the axial force and moment response of the beam to a failure envelope presented in a moment-thrust diagram.

The beam is composed of a single beam segment that is fixed from translation and rotation at one end. A constant velocity in the axial direction and a constant rotational velocity are applied at the other end of the beam. The moment and axial force are monitored during the test.

The beam material represents a 25 cm thick unreinforced shotcrete lining material. The cantilever beam specimen is 2.9 m long and 1 m wide. The properties of the material are

density	2100 kg / m^3
Poisson's ratio	0.2
elastic modulus	30.5 GPa
compressive yield strength	16.67 MPa
residual compressive strength	16.0 MPa
tensile yield strength	2.2 MPa
residual tensile strength	1.6 MPa

Four simulations are run to illustrate the behavior of the inelastic material. In each case, a constant translational velocity and a rotational velocity are applied, and the inelastic material response is monitored and plotted on the P-M diagram. The translational velocity produces a compressive axial force in the beam. In the first case, the residual strengths are set equal to the initial strengths. In the second case, the actual residual strengths are specified. In the third case, the tensile yield strength is set to zero, and a crack depth ratio, h_c, of 0.33 is prescribed. In the fourth case, a parabolic failure surface is input to define the inelastic material response.

The data file for the four cases is listed in Example 1.5.

Example 1.5 Inelastic material behavior of a cantilever beam

```
model new
;Project Record Tree export
;File:crliner.dat
model Title 'Thrust-Moment Diagram'
;Name:_liner
;Input:_syc/float/16.67e6/compressive strength
;Input:_scr/float/16.67e6/residual compressive strength
;Input:_syi/float/2.2e6/tensile strength
;Input:_syr/float/2.2e6/residual tensile strength
;Input:_rcrk/float/0.0/crack depth ratio
;Input:_pmtab/int/0/P-M diagram table
fish define _liner
  ; geom
  ; thickness (height)
  _thi = 0.25
  ; width
  _wid = 1.0
  _moi = _wid*_thi^3/12.
  _are = _wid*_thi
  ; change sign of tensile strength (note: compression positive)
  _syim = -_syi
  _syrm = -_syr
  _scrp = _scr
  _Pc = _syc*_are
  _Mc = 0.
  _Pb = (_syim+_syc)*_are/2.
  _Mb = _moi*(_syc-_syim)/_thi
  _Pt = _syim*_are
  _Mt = 0.
  ; parabolic P-M diagram table
  _hpm = _Mb
  _kpm = _Pb
  _apm = -(_Pc-_Pb)^2/_hpm
  _np = 100
  _yinc = (_Pc-_Pt)/float(_np)
  _yinc = (_Pt-_Pc)/float(_np)
  loop n (0,_np)
    ;_y = _Pt + float(n)*_yinc
    _y = _Pc + float(n)*_yinc
    _x = (_y-_kpm)*(_y-_kpm)/_apm + _hpm
    table.y(10,n+1) =  _x   ; M
    table.x(10,n+1) = -_y   ; P
  endloop
```

```
  ;
  ; create diagram
  if _pmtab > 0
    ; swap P and M for plotting purpose
    _ts = table.size(10)
    loop n (1,_ts)
      table.x(1,n) = table.y(10,n)
      table.y(1,n) = -table.x(10,n)
    endloop
  endif
  if _pmtab = 0
    ns = 1
    table.x(1,ns) = _Mc
    table.y(1,ns) = _Pc
    ns = ns + 1
    table.x(1,ns) = _Mb
    table.y(1,ns) = _Pb
    if _rcrk > 0.
      ns  = ns+1
      _P4 = 0.5*_syc*_are*(1.-_rcrk)
      _M4 = _syc*_are*_thi*(1.+_rcrk-2.*_rcrk^2)/12.
      table.x(1,ns) = _M4
      table.y(1,ns) = _P4
    endif
    ns = ns + 1
    table.x(1,ns) = _Mt
    table.y(1,ns) = _Pt
  endif
end
;
block tolerance corner-round-length 0.0001
block tolerance minimum-edge-length 0.0002
block create polygon 0 0 0 0.1 0.1 0.1 0.1 0
block cut crack 0 0.05 0.1 0.05
block change material 1
block property material 1 density 1E3
block contact property mat 1 stiffness-normal 1 stiffness-shear 1
block fix range pos-x 0,0.1 pos-y 0,0.1
block hide  range
model save 'crliner1.sav'
;
;
; Case 1 --- initial compressive and tensile strengths
fish set @_syc=16.67e6 @_scr=16.67e6 @_syi=2.2e6 @_syr=2.2e6 @_rcrk=0.0
fish set @_pmtab=0
@_liner
```

```
block structure beam create single begin 0.1 0.1 end 3.0 0.1 mat-beam 1
block structure beam property material 1 density 2.1E3 ...
  cross-sectional-area @_are moi @_moi thickness @_thi ...
  yield-compression @_syc yield-tension @_syi ...
  yield-tension-residual @_syr yield-compression-cracked @_scr ...
  young 30.5e9 poisson 0.2 shape-factor 0.83333 spacing 1 width 1 ...
  coupling-stiffness-normal 1 coupling-stiffness-shear 1
block struct beam node 1 fix-x fix-y fix-r
block struct beam node 2 fix-x fix-y fix-r
block struct beam node 2 init vel-x -1e-7 init vel-r 2e-6
model save 'crliner2.sav'
;
fish define _samp_pm
  _epnt = block.structure.element.head
  _P = -block.struct.beam.force.axial(_epnt)
  _M = block.struct.beam.moment2(_epnt)
  _ns = _ns + 1
  table.x(2,_ns) =  math.abs(_M)
  table.y(2,_ns) =  _P
end
fish define _load
  _nstep = 50
  _np    = 200
  loop n (0,_np)
    command
      block cycle @_nstep
    endcommand
    _samp_pm
  endloop
end
block  smallstrain on
@_load
model save 'crliner3.sav'


model restore 'crliner1.sav'
; Case 2 --- residual compressive and tensile strengths
fish set @_syc=16.67e6 @_scr=16.0e6 @_syi=2.2e6 @_syr=1.6e6 @_rcrk=0.0
fish set @_pmtab=0
@_liner
block structure beam create single begin 0.1 0.1 end 3.0 0.1 mat-beam 1
block structure beam property material 1 density 2.1E3 ...
  cross-sectional-area @_are moi @_moi thickness @_thi ...
  yield-compression @_syc yield-tension @_syi ...
  yield-tension-residual @_syr yield-compression-cracked @_scr ...
  crack-depth-ratio @_rcrk young 30.5e9 poisson 0.2 ...
```

```
  shape-factor 0.83333 spacing 1 width 1 coupling-stiffness-normal 1 ...
  coupling-stiffness-shear 1
block struct beam node 1 fix-x fix-y fix-r
block struct beam node 2 fix-x fix-y fix-r
block struct beam node 2 init vel-x -1e-7 init vel-y 2e-6
model save 'crliner4.sav'
;
fish define _samp_pm
  _epnt = block.structure.element.head
  _P = -block.struct.beam.force.axial(_epnt)
  _M = block.struct.beam.moment2(_epnt)
  _ns = _ns + 1
  table.x(2,_ns) =  math.abs(_M)
  table.y(2,_ns) =  _P
end
fish define _load
  _nstep = 50
  _np    = 200
  loop n (0,_np)
    command
      block cycle @_nstep
    endcommand
    _samp_pm
  endloop
end
block  smallstrain on
@_load
model save 'crliner5.sav'
;
;
model restore 'crliner1.sav'
;Case 3 --- crack depth ratio = 0.33
fish set @_syc=16.67e6 @_scr=16.67e6 @_syi=0.0 @_syr=0.0 @_rcrk=0.33
fish set @_pmtab=0
@_liner
block structure beam create single begin 0.1 0.1 end 3.0 0.1 mat-beam 1
block structure beam property material 1 density 2.1E3 ...
  cross-sectional-area @_are moi @_moi thickness @_thi ...
  yield-compression @_syc yield-tension @_syi ...
  yield-tension-residual @_syr yield-compression-cracked @_scr ...
  crack-depth-ratio @_rcrk young 30.5e9 poisson 0.2 ...
  shape-factor 0.83333 spacing 1 width 1 coupling-stiffness-normal 1 ...
  coupling-stiffness-shear 1
block struct beam node 1 fix-x fix-y fix-r
block struct beam node 2 fix-x fix-y fix-r
block struct beam node 2 init xvel -1e-7 init rvel 2e-6
```

```
model save ’crliner6.sav’
;
fish define _samp_pm
  _epnt = block.structure.element.head
  _P = -block.struct.beam.force.axial(_epnt)
  _M = block.struct.beam.moment2(_epnt)
  _ns = _ns + 1
  table.x(2,_ns) =  math.abs(_M)
  table.y(2,_ns) =  _P
end
fish define _load
  _nstep = 50
  _np    = 200
  loop n (0,_np)
    command
      block cycle @_nstep
    endcommand
    _samp_pm
  endloop
end
block  smallstrain on
@_load
model save ’crliner7.sav’
;
;
model restore ’crliner1.sav’
;Case 4 --- input parabolic failure envelope
fish set @_syc=16.67e6 @_scr=16.67e6 @_syi=0.0 @_syr=0.0 @_rcrk=0.0
fish set @_pmtab=10
@_liner
block structure beam create single begin 0.1 0.1 end 3.0 0.1 mat-beam 1
block structure beam property material 1 density 2.1E3 ...
  cross-sectional-area @_are moi @_moi thickness @_thi ...
  yield-compression @_syc yield-tension @_syi ...
  yield-tension-residual @_syr yield-compression-cracked @_scr ...
  crack-depth-ratio @_rcrk young 30.5e9 ...
  poisson 0.2 shape-factor 0.83333 spacing 1 width 1 ...
  coupling-stiffness-normal 1 coupling-stiffness-shear 1
;block struct beam prop mat 1 moment-thrust-table @_pmtab
block struct beam prop mat 1 moment-thrust-table 1
block struct beam node 1 fix-x fix-y fix-r
block struct beam node 2 fix-x fix-y fix-r
block struct beam node 2 init vel-x -1e-7 init vel-r 2e-6
model save ’crliner8.sav’
;
fish define _samp_pm
```

```
    _epnt = block.structure.element.head
    _P = -block.struct.beam.force.axial(_epnt)
    _M = block.struct.beam.moment2(_epnt)
    _ns = _ns + 1
    table.x(2,_ns) =  math.abs(_M)
    table.y(2,_ns) =  _P
  end
  fish define _load
    _nstep = 50
    _np    = 200
    loop n (0,_np)
      command
        block cycle @_nstep
      endcommand
      _samp_pm
    endloop
  end
  block  smallstrain on
  @_load
  model save 'crliner9.sav'
```

In the first run, the initial compressive and tensile strength properties are assigned to the structural element material, and residual strengths are set equal to the initial strengths. Based upon equations Eqs. (1.29) through (1.31), the values for the three points of the moment-thrust diagram based upon the initial strengths are calculated to be

$$\textit{at Point 1}\ \ P_1 = 4167.5\text{kN}$$

$$M_1 = 0$$

$$\textit{at Point 2}\ \ P_2 = -550.0\text{kN}$$

$$M_2 = 0$$

$$\textit{at Point 3}\ \ P_3 = 1808.8\text{kN}$$

$$M_3 = 98.3\text{kN}$$

Note that, by definition, compressive stresses are positive, so $F_c = 16.67\times10^6$ and $F_t = -2.2\times10^6$.

Figure 1.33 plots the moment-thrust diagram for this case using these three points. The axial loads and moments calculated from the *UDEC* model are also shown. The crosses on the figure indicate the loading path that the *UDEC* model follows, and shows a good fit to the analytical results for the failure envelope.

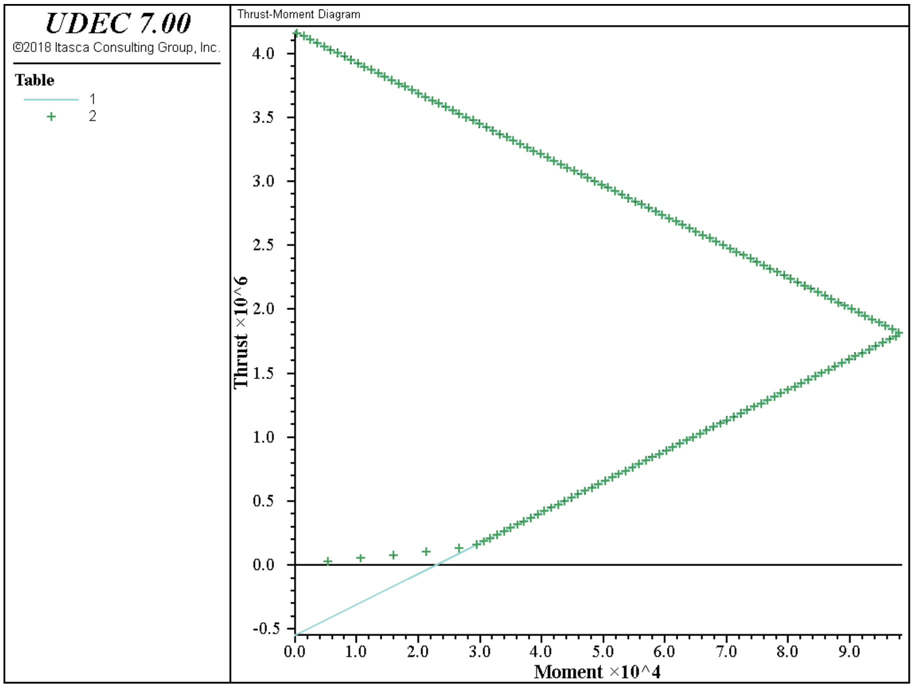


Figure 1.33 Moment-thrust diagram – at initial compressive and tensile strengths

For the second case, the actual residual compressive and tensile strengths are specified. In this simulation, when the stresses reach the initial ultimate strengths, the strength capacity is reduced to the residual value from that point on. Figure 1.34 displays the results for this case, and shows that the axial forces and moments are reduced to a "cracked" failure envelope.

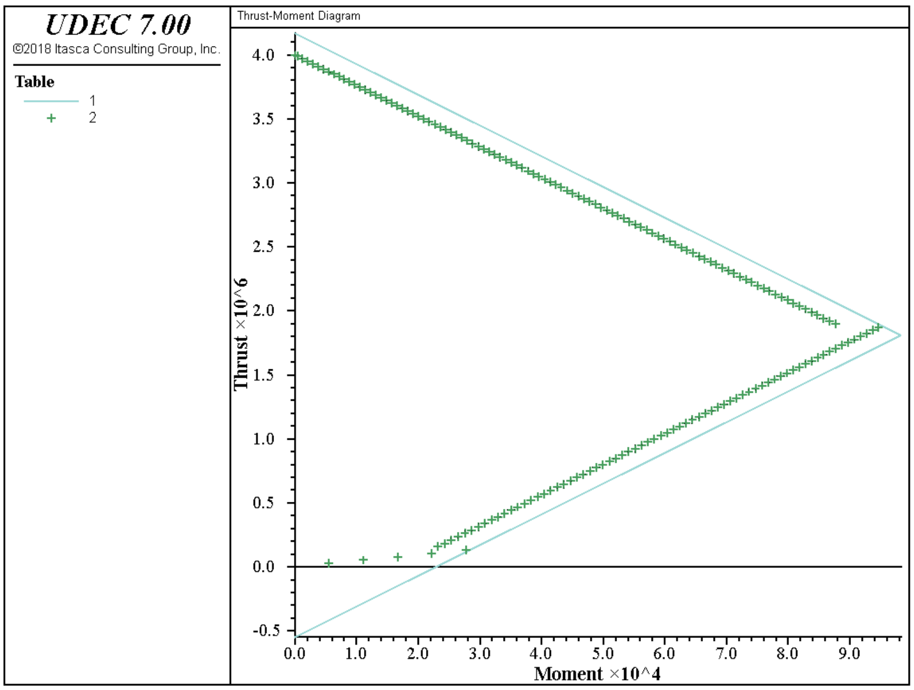


Figure 1.34 Moment-thrust diagram – include residual compressive and tensile strengths

For the third case, the effect of cracking is explicitly included by specifying a crack depth for the section. The initial and residual tensile yield strengths are set to zero (i.e., P_2 = 0), and a crack depth ratio, h_c/h, of 0.33 is input. This results in a fourth point in the moment thrust diagram, for which the axial force, P_4, and moment, M_4, are calculated (using Eq. (1.33)) to be

$$\text{at Point 4 } P_4 = 1396.1\text{kN}$$

$$M_4 = 96.6\text{kN}$$

Figure 1.35 compares the analytical results to the *UDEC* calculation and shows a good agreement for this case.

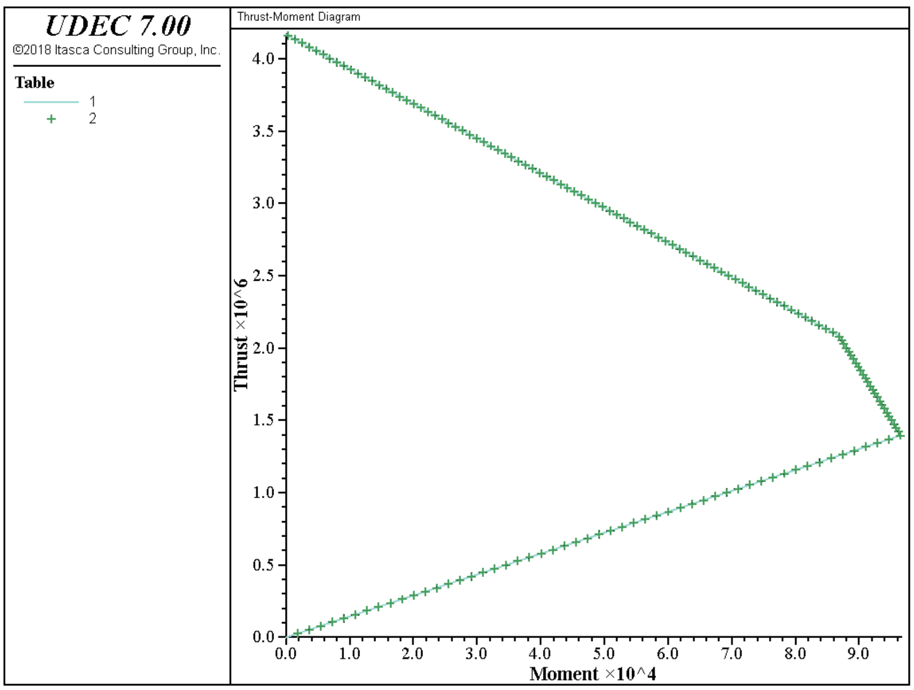


Figure 1.35 Moment-thrust diagram – with zero tensile strength and crack depth ratio = 0.33

In the fourth case, a parabolic failure envelope is input to define the inelastic response. The equation of the parabola is defined to pass through the three points (Point 1, Point 2 and Point 3) as calculated for the first case. The parabolic equation has the form

$$M = M_3 \left[1 - \left(\frac{P - P_3}{P_1 - P_3} \right)^2 \right] \tag{1.34}$$

Values for P and M that prescribe the failure envelope based on this equation are calculated in the *FISH* function **_liner**, and stored in table 10. The failure envelope is plotted in the P-M diagram shown in Figure 1.36. This figure also plots the *UDEC* results. This failure surface is invoked for the beam material in the *UDEC* model via the **moment-thrust-table** property keyword.

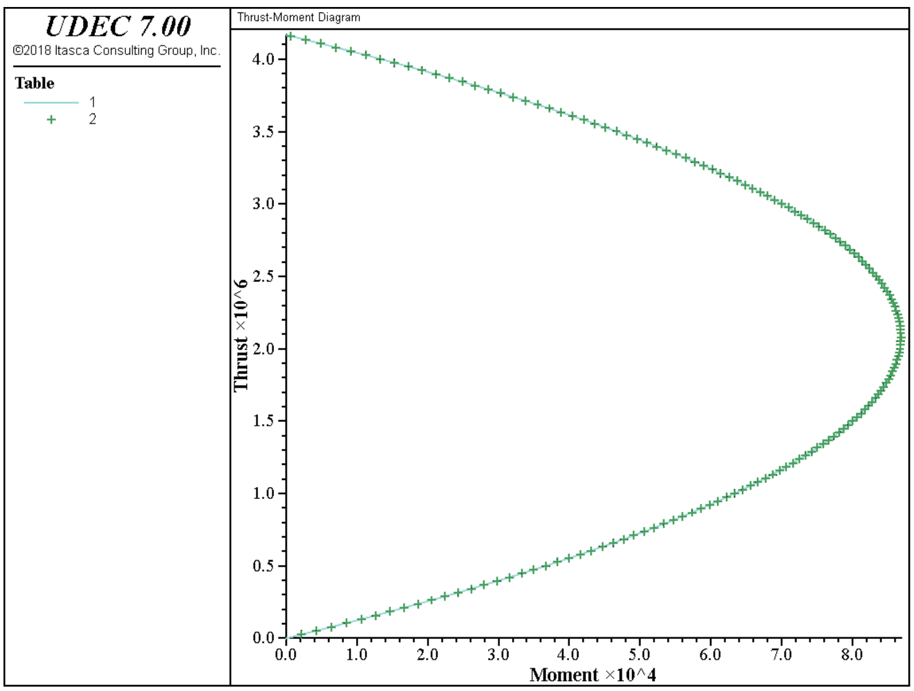


Figure 1.36 Moment-thrust diagram – input P-M diagram table

1.3.1.7 Example Application – Support of a Wedge in a Tunnel Roof

A simple test is presented to demonstrate the capability of the structural element formulation to simulate failure and the post-failure strength of a tunnel lining. The example is a 6 m × 6 m square tunnel containing a wedge in the roof. The wedge weighs 54 kN. The geometry is shown in Figure 1.37.

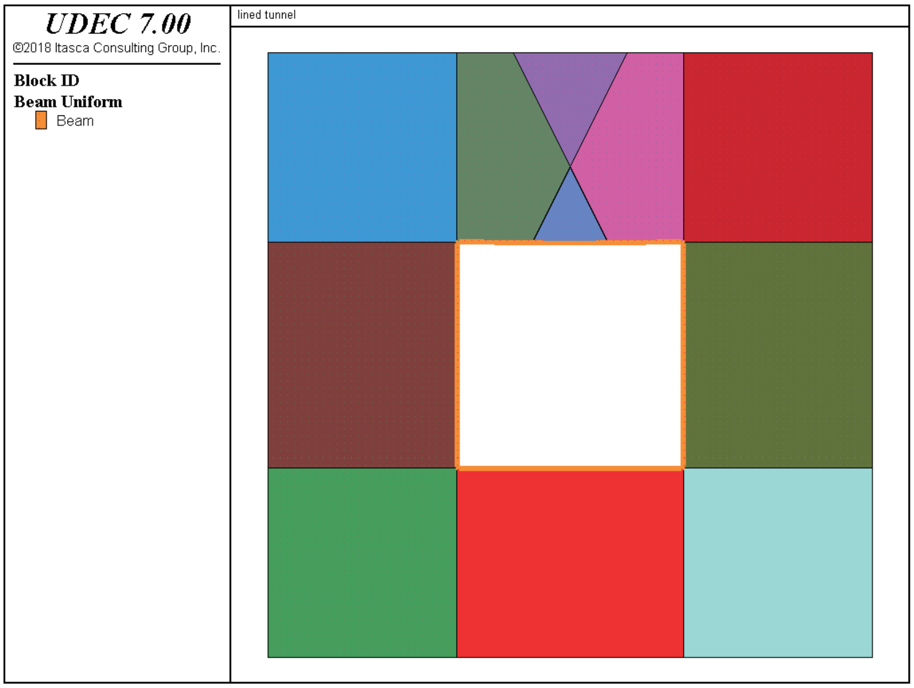


Figure 1.37 Lined tunnel with wedge in roof

The displacement of the wedge and the moments in the tunnel lining are compared for four conditions of the structural elements:

(1) liner with high strength (elastic analysis);

(2) liner with yield strength and residual strength equal to 18 MPa;

(3) liner with yield strength equal to 18 MPa and residual strength equal to 16.7 MPa; and

(4) double layer of lining with high strength (elastic analysis).

A rigid block analysis is performed; the material properties associated with the rock and joints are

density	2700 kg / m^3
joint normal stiffness	10.0 GPa / m
joint shear stiffness	10.0 GPa / m
joint friction	30°

A continuous liner is generated along the tunnel surface. The liner contains 240 structural nodes (specified by setting the keyword **length-max** = 0.1 in the **block struct liner create by-end-points** command). The properties of the liner are as follows.

Liner Material

Young's modulus	21.0 GPa		
Poisson's ratio	0.15		
	Case 1	Case 2	Case 3
tensile yield strength	40.0 MPa	18.0 MPa	18.0 MPa
residual yield strength	40.0 MPa	18.0 MPa	16.7 MPa
compressive yield strength	40.0 MPa	40.0 MPa	40.0 MPa

Rock/Liner Interface

normal stiffness	1.0 GPa / m
shear stiffness	1.0 GPa / m
friction	50°

The *UDEC* data file is given in Example 1.6.

Example 1.6 Lined tunnel with wedge in roof

```
model new
;  create tunnel with wedge block in roof
model title 'lined tunnel'
block tolerance corner-round-length 0.05
block tolerance minimum-edge-length 0.1
block create polygon -8 -8 -8 8 8 8 8 -8
block cut crack -3 -8 -3 8
block cut crack 3 -8 3 8
block cut crack -8 3 8 3
block cut crack -8 -3 8 -3
block delete range pos-x -3 3 pos-y -3 3
block cut crack -1 3 1.5 8
block cut crack 1 3 -1.5 8
;
```

```
;  rock properties
block change material 1
block property material 1 density 2.7E-3
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 30 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 30
block fix all range position-x -8 8 position-y -8 0
block fix all range position-x -8 8 position-y 5 8
model save 'w1.sav'
;
;
;  structural liner and properties
block structure liner create by-end-points ...
  begin -3 -1.51485 end -3 -1.51485 ...
  length-maximum 0.1 material-beam 1
block structure beam property material 1 density 2.5E-5 ...
  poisson 0.15 yield-compression 40 yield-tension 40 young 2.1E4 ...
  yield-tension-residual 40 cross-sectional-area 0.1 ...
  moi 8.33333E-5 shape-factor 0.83333 spacing 1 thickness 0.1 ...
  width 1 coupling-friction 50 coupling-cohesion 0 ...
  coupling-tension 0 coupling-dilation 0 ...
  coupling-stiffness-normal 1E3 coupling-stiffness-shear 1E3
;
;  gravity load
model gravity 0 -10
;
block gridpoint history displacement-y 0.0 3.0
block gridpoint history velocity-y 0.0 3.0
;
;  Case 1 : elastic analysis
block solve cycle 100000000 ratio 1.0E-7
model save 'w2.sav'
;
;
;  Case 2 : yield strength = residual strength = 18 MPa
block structure beam property material 1 density 2.5E-5 poisson 0.15 ...
  yield-compression 40 yield-tension 18 young 2.1E4 ...
  yield-tension-residual 18 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 50 coupling-stiffness-normal 1E3 ...
  coupling-stiffness-shear 1E3
block solve ratio 1.0E-7
model save 'w3.sav'
```

```
;
;
;  Case 3 : yield strength = 18 MPa residual strength = 16.7 MPa
model restore 'w2.sav'
;
block structure beam property material 1 density 2.5E-5 poisson 0.15 ...
  yield-compression 40 yield-tension 18 young 2.1E4 ...
  yield-tension-residual 16.7 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 50 coupling-stiffness-normal 1E3 ...
  coupling-stiffness-shear 1E3
block solve ratio 1.0E-7
model save 'w4.sav'
;
;
;  Case 4 : elastic analysis with double layer of liner
model restore 'w1.sav'
;
block structure liner create by-end-points ...
  begin -3 0.9802 end -3 0.9802 length-maximum .1 material-beam 1
block structure liner create by-end-points ...
  begin -3 0.9802 end -3 0.9802 length-maximum .1 material-beam 1
block structure beam property material 1 density 2.5E-5 poisson 0.15 ...
  yield-compression 40 yield-tension 40 young 2.1E4 ...
  yield-tension-residual 40 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 50 coupling-cohesion 0 coupling-tension 0 ...
  coupling-dilation 0 coupling-stiffness-normal 1E3 ...
  coupling-stiffness-shear 1E3
;
;  gravity load
model gravity 0 -10
;
block gridpoint history displacement-y 0.0 3.0
block gridpoint history velocity-y 0.0 3.0
;
block solve cycle 10000000 ratio 1.0E-7
model save 'w5.sav'
```

The results of the elastic analysis, Case 1, are compared to the analytic solution for bending of a beam with fixed ends. The analytical results are

maximum displacement = 0.026 m

maximum bending moment = 36 kN-m

The analytical solution does not include the weight of the liner, so the structural element density in the model is set to a very low value to remove the influence of this weight on the *UDEC* solution.

The *UDEC* results are plotted in Figure 1.38. The maximum bending moment (35.0 kN-m) and maximum displacement (0.028 m) are very close to the analytical solution.

Note that the sense of the structural moment plot is determined by the order in which the beam elements are generated. The sense of the moment plot can be changed by reversing it in the plot item attributes.

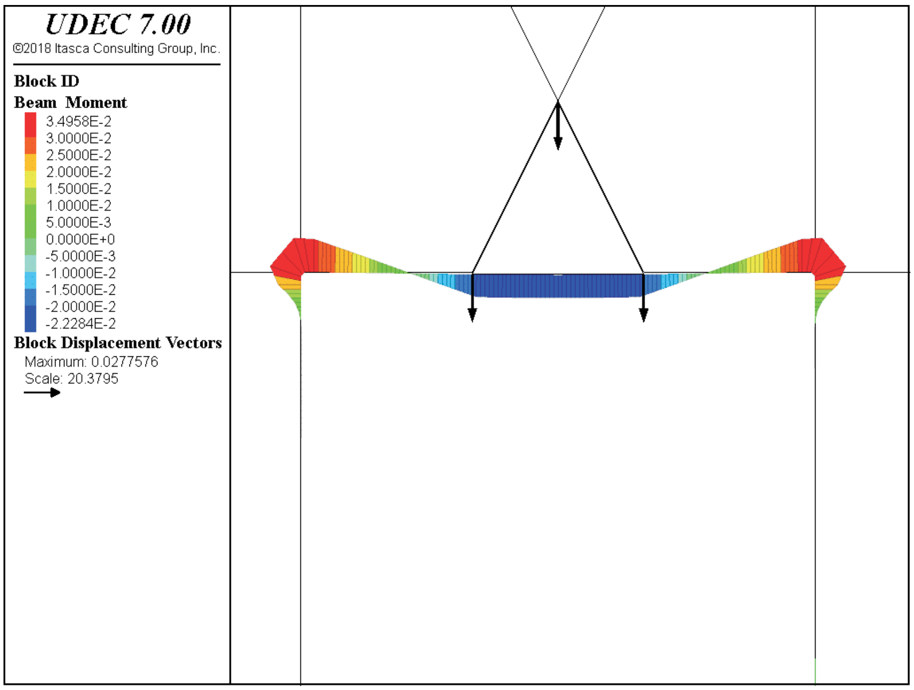


Figure 1.38 Case 1: elastic solution

The results for the Case 2 and Case 3 plastic analyses are shown in Figures 1.39 and 1.40, respectively. The structural nodes at the top corners of the lining fail in tension for both of these cases. In Case 2, the maximum displacement is now greater, and the maximum moment smaller, than for the elastic case because of the plastic hinges that form at the corners. In Case 3, the displacement is even higher, and the moment lower, because of the residual strength at the plastic hinges.

For Case 4, two layers of structural element lining are created, each with a thickness of 0.1 m. Figure 1.41 illustrates the two layers as if they have an actual thickness; The structural nodes for each layer are actually at the same locations for the calculation. The result for this case is shown in Figure 1.42. The maximum moment for each layer, and the maximum displacement, are approximately one third of the values calculated for the single layer (compare to Figure 1.38).

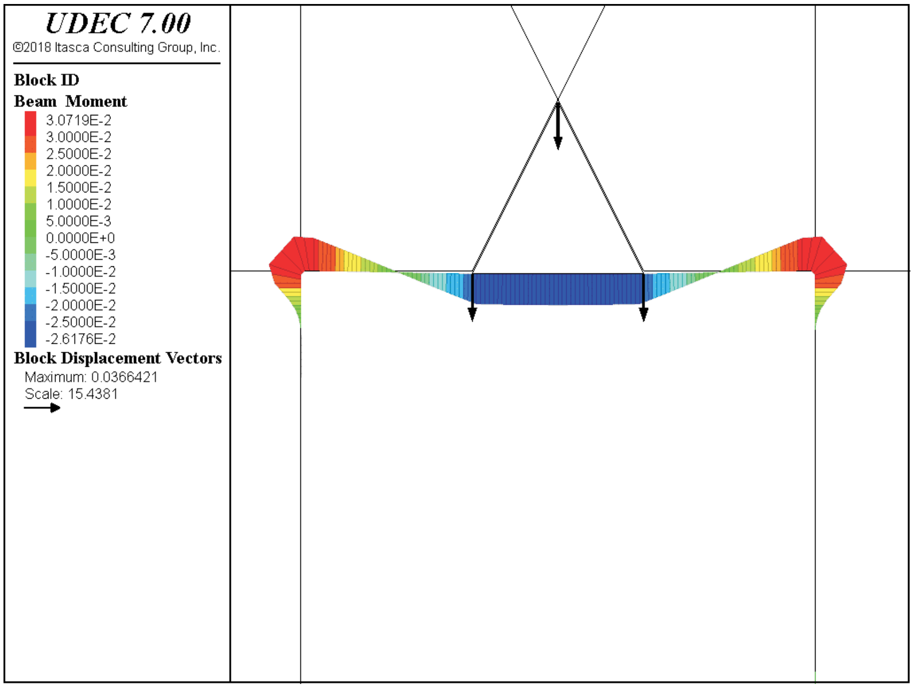


Figure 1.39 Case 2: yield strength = residual strength = 18 MPa

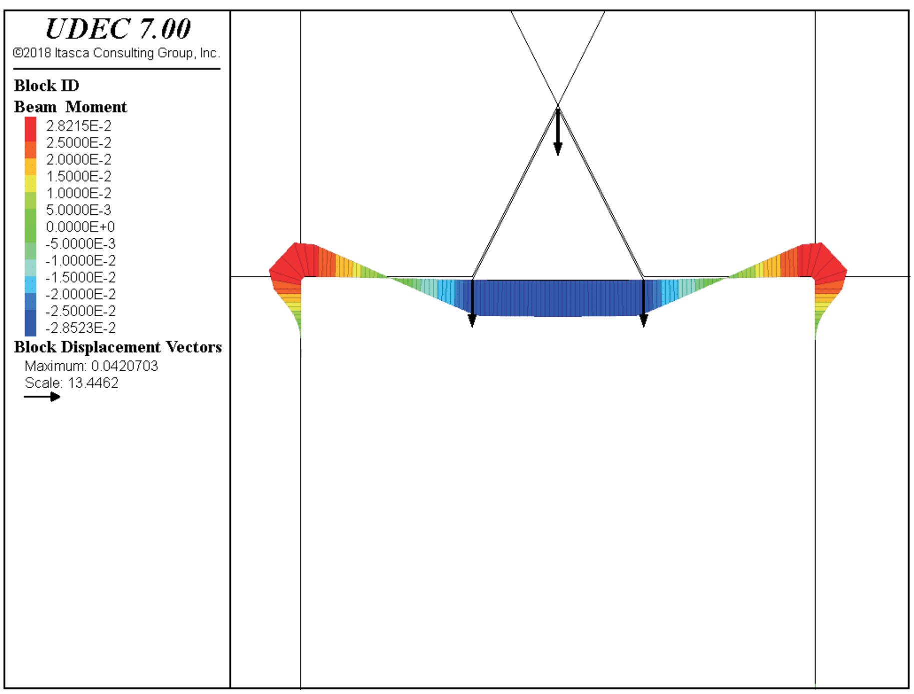


Figure 1.40 Case 3: yield strength = 18 MPa, residual strength = 16.7 MPa

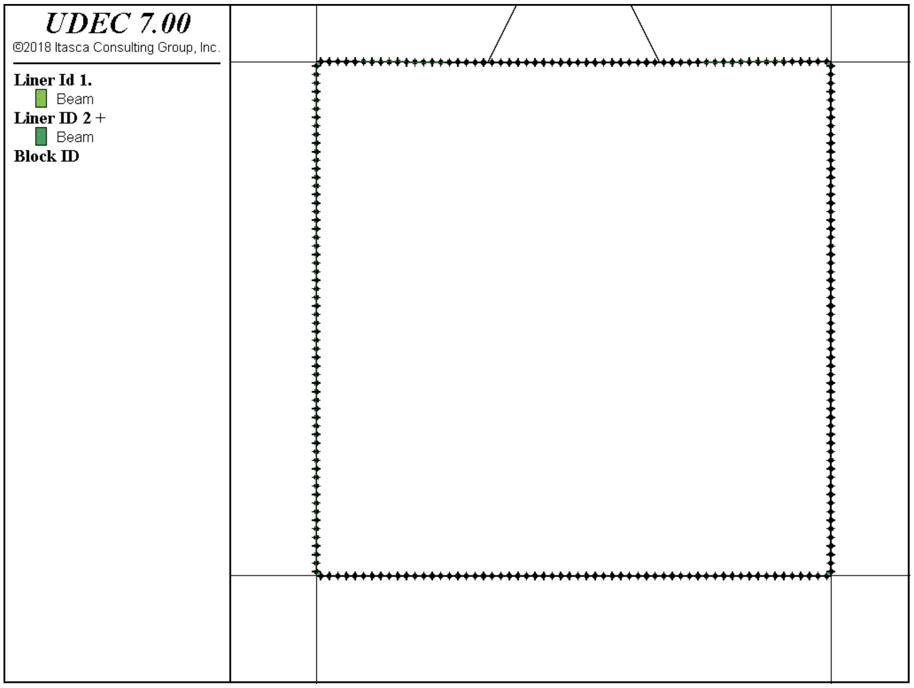


Figure 1.41 Case 4: two layers of lining

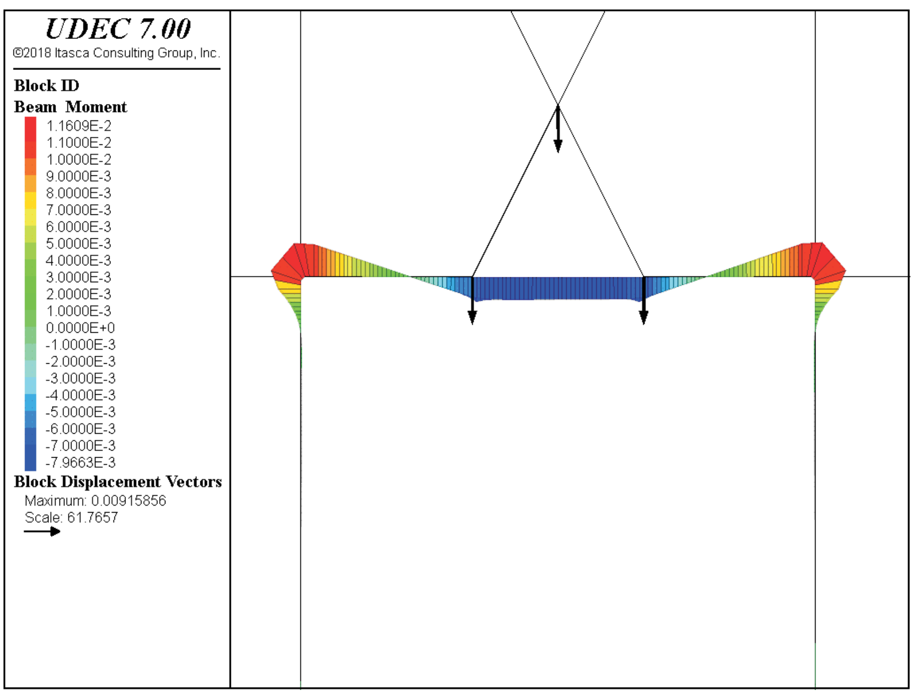


Figure 1.42 Case 4: elastic solution for two layers of lining

1.3.1.8 Example Application – Circular Tunnel Excavation with Interior Support

Conceptual understanding of the interaction between support loading and rock mass unloading is often explained in terms of a reaction curve for the rock medium and a stress-displacement curve for the support system. These curves are also called characteristic lines, characteristic curves or rock response curves. In the conceptual model used in this approach, the problem is reduced to consideration of a plane perpendicular to the tunnel axis, and all variables (i.e., stress, strain and displacement) vary only with radial distance from the tunnel. The ground reaction curve is frequently shown to consist of two parts: a descending portion and an ascending portion (as shown in Figure 1.43). The descending portion of the ground reaction curve generally consists of two distinct parts: an elastic part and an inelastic part (explained as follows). The model assumes that the excavation of the tunnel is simulated by quasi-statically unloading the boundary of the excavation. Upon unloading, the system responds elastically until the elastic limit is reached. Further unloading causes propagation of a plastic or "failed" zone around the excavation. If gravity is neglected, and the rock mass is assumed to behave as an isotropic, homogeneous, time-independent continuum, the descending portion of the ground reaction curve can be determined analytically based on material properties.

For the example shown here, it is assumed that strength properties are sufficient, such that only elastic behavior need be considered. The expression for radial displacement in an elastic material is given by

$$u_i = \frac{r_i}{2G} \left(P_o - P_i \right) \tag{1.35}$$

where r_i, u_i = tunnel radius and radial displacement;

P_o = in-situ isotropic stress ($P_o = P_{xx} = P_{yy}$);

G = shear modulus; and

P_i = internal pressure.

Expressions have also been developed to describe the support stiffness of a variety of supports, assuming that the support reaction is radially symmetric. For example, the stiffness of a blocked steel set is given by Daemen (1975) as

$$\frac{1}{K_{ss}} = \frac{u_i}{P_i} = \frac{S r_i^2}{EA} + \frac{S r_i^4}{EI} \left[\frac{\theta\,(\theta + sc)}{2s^2} - 1 \right] + \frac{2 S r_i\, \theta\, t_B}{A_B\, E_B} \tag{1.36}$$

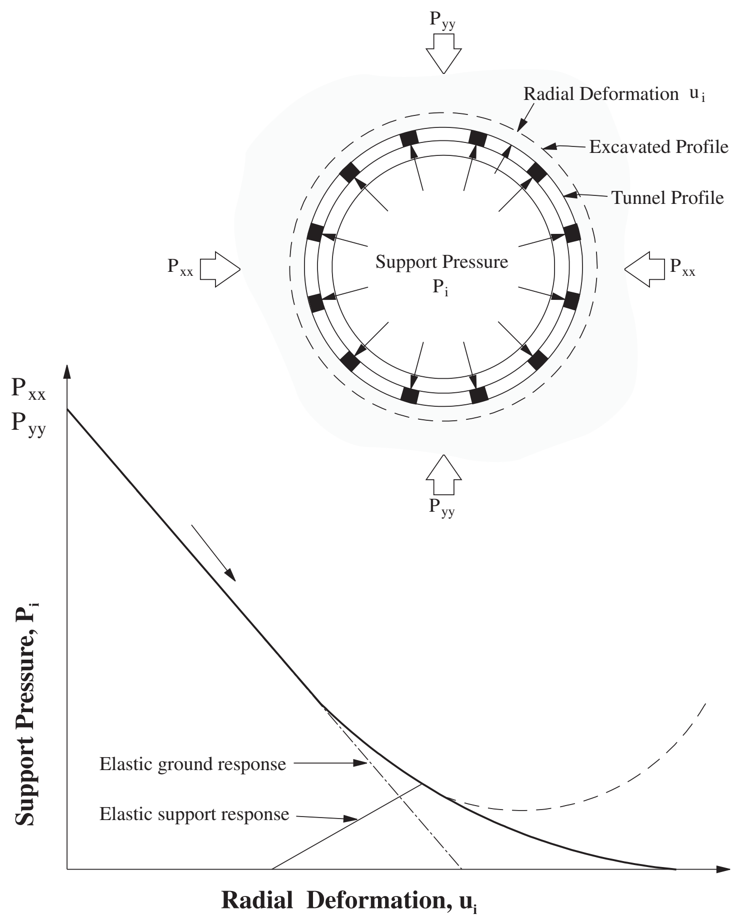


Figure 1.43 Conceptual representation of support reaction and ground reaction curves

where K_{ss} = stiffness of blocked steel set;

A, E, I = steel cross-sectional area, elastic modulus and moment of inertia, respectively;

S = steel set spacing;

2θ = angle between blocking points;

$n = \pi/\theta$ = number of blocking points;

s = $\sin\theta$;

c = $\cos\theta$;

E_B, t_B = elastic modulus and thickness of blocks; and

A_B = cross-sectional area of blocks.

Four calculations are made to establish a simple reaction curve for this example:

(1) tunnel excavation without support (far-field stress constant);

(2) tunnel excavation without support (far-field boundary fixed);

(3) tunnel excavation with support (far-field stress constant); and

(4) tunnel excavation with support (far-field boundary fixed).

The two assumptions concerning far-field boundaries (i.e., constant stress and fixed) are required to "bound" the numerical solution because the analytical solution assumes infinite far-field boundaries. The problem parameters used to describe the ground reaction are

$$
\begin{aligned}
r_i &= 1 \text{ m} \\
G &= 1 \text{ MPa} \\
P_o &= 10 \text{ MPa}
\end{aligned}
$$

The parameters used to describe the structural lining stiffness are

$$
\begin{aligned}
A &= 0.1 \text{ m}^2 \\
E &= 2.57 \text{ GPa} \\
I &= 8.33 \times 10^{-5} \text{m}^4 \\
S &= 1 \text{ m} \\
n &= 24
\end{aligned}
$$

The blocking is described in the *UDEC* model by the interface normal stiffness, kn_{if} (force / displacement), between the structure (i.e., steel set) and the rock. In this case, the last term in Eq. (1.36) is replaced by $(2Sr_i)/kn_{if}$, where

$$
kn_{if} = \frac{A_B \, E_B}{t_B} \quad (kn_{if} = 1000 \text{ MN / m})
$$

The usual assumption made in analyzing blocked steel sets is that no shear force is transferred between the rock mass and the steel set. Consequently, friction and cohesion values were not specified (default value is zero).

In setting up the numerical problem, the problem domain was divided into quadrants and concentric rings to facilitate zoning. All discontinuities were assigned as construction joints, using the **block contact join by-contact** command. The net result is that the joints are essentially "transparent," and do not affect the final result. The zone discretization is shown in Figure 1.44.

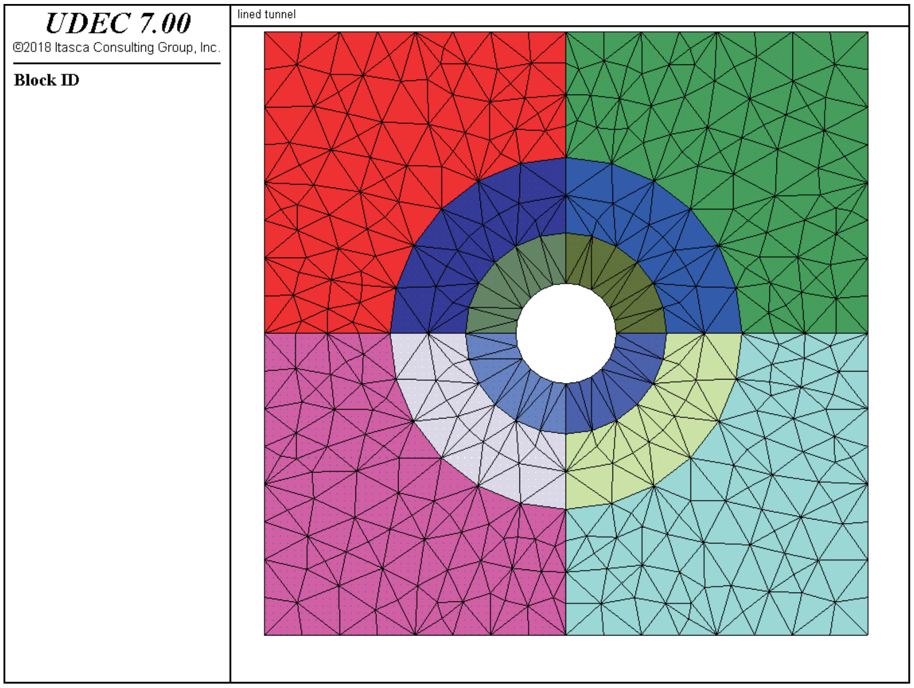


Figure 1.44 Zoning for UDEC model of circular tunnel excavation

The *UDEC* data file is given in Example 1.7.

Example 1.7 Circular excavation with interior support

```
model new
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon -6 -6 -6 6 6 6 6 -6
block cut crack -10 0 10 0
block cut crack 0 -10 0 10
block cut arc 0 0 1 0 360 24
block cut arc 0 0 2 0 360 24
block cut arc 0 0 3.5 0 360 24
block zone gen quad 0.8 range annulus center 0 0 rad 3.5 2
block zone gen quad 0.6 range annulus center 0 0 rad 2 1
block zone gen edge 1.2
block delete range annulus center 0 0 rad 0 1
block zone group 'block'
block zone cmodel assign elastic density 0.003 bulk 2E3 ...
  shear 1E3 range group 'block'
block contact join by-contact
block edge apply stress -10.0 0.0 -10.0
block insitu  stress -10.0 0.0 -10.0
```

```
block gridpoint history displacement-x 1.0 0.0
block gridpoint history displacement-x -1.0 0.0
block gridpoint history displacement-y 0.0 1.0
block gridpoint history displacement-y 0.0 -1.0
model save 'support1.sav'
;
;
model restore 'support1.sav'
model title ...
'Circular Excavation with Interior Support - Unlined - Stress Boundary'
block solve ratio 1.0E-5
model save 'support2.sav'
;
;
model restore 'support1.sav'
model title ...
'Circular Excavation with Interior Support - Unlined - Fixed Boundary'
block gridpoint apply velocity-x 0
block gridpoint apply velocity-y 0
block solve ratio 1.0E-5
model save 'support3.sav'
;
;
model restore 'support1.sav'
model title ...
'Circular Excavation with Interior Support - Lined - Stress Boundary'
block structure liner create by-end-points ...
  begin -0.97809 -0.12878 end -0.97809 -0.12878 ...
  length-maximum 1.0E9 length-minimum 0.02 material-beam 1
block structure beam property material 1 density 2.5E-3 poisson 0.2 ...
yield-compression 1E10 yield-tension 1E10 young 2.57E3 ...
cross-sectional-area 0.1 moi 8.33333E-5 shape-factor 0.83333 spacing 1 ...
thickness 0.1 width 1 coupling-tension 100 ...
coupling-stiffness-normal 1E3 coupling-stiffness-shear 1
block solve ratio 1.0E-5
model save 'support4.sav'
;
;
model restore 'support1.sav'
model title ...
  'Circular Excavation with Interior Support - Lined - Fixed Boundary'
block structure liner create by-end-points ...
  begin -0.97809 -0.12878 end -0.97809 -0.12878 ...
  length-maximum 1.0E9 length-minimum 0.02 material-beam 1
block structure beam property material 1 density 2.5E-3 poisson 0.2 ...
yield-compression 1E10 yield-tension 1E10 young 2.57E3 ...
```

```
cross-sectional-area 0.1 moi 8.33333E-5 shape-factor 0.83333 spacing 1 ...
thickness 0.1 width 1 coupling-tension 100 ...
coupling-stiffness-normal 1E3 coupling-stiffness-shear 1
block gridpoint apply velocity-x 0
block gridpoint apply velocity-y 0
block solve ratio 1.0E-5
model save 'support5.sav'
```

The results of the analyses are compared in Figure 1.45. As expected, the analytical solution falls between the numerical solutions obtained, assuming different boundary conditions. In analyzing the results, the internal pressure, P_i, supplied by the support can be determined in one of two ways. The internal pressure is given by the thrust or axial force in the structural elements divided by the external radius of the support (i.e., 1 m), or by the radial force in the "blocking" divided by its tributary area ($= 2\pi\ r_i\ S/n$). Both methods will yield the same value for P_i supplied by the support.

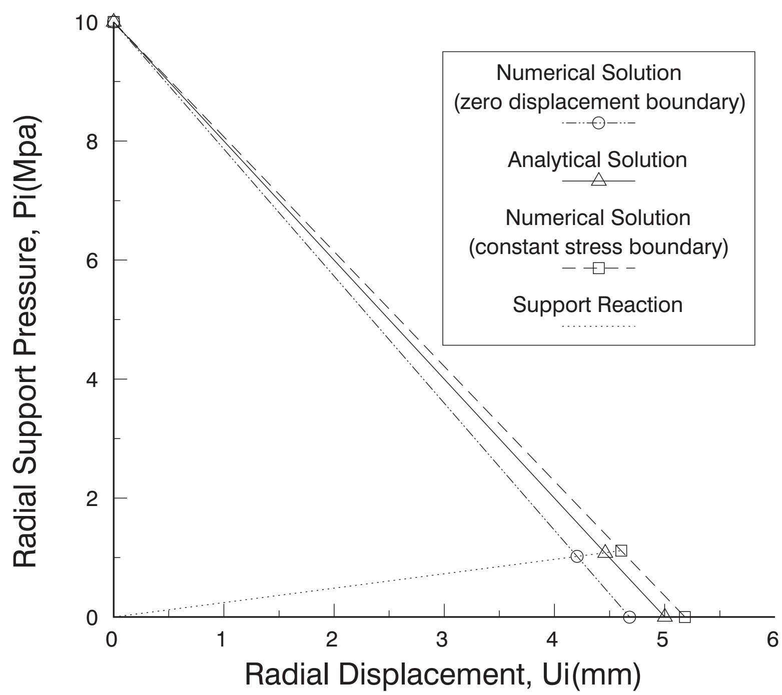


Figure 1.45 Comparison of ground reaction/support reaction lines

1.3.1.9 Example Application – Shotcrete Lined Tunnel

Structural elements can be placed automatically around a tunnel periphery, even if the surface is highly irregular. The following example demonstrates this feature for the analysis of tunnel support in jointed rock.

A tunnel is excavated in rock containing four joint sets. The orientations of the joint sets are summarized in Table 1.6.

Table 1.6 Joint set orientations

Joint Set	Dip (°)	Trace Length (m)	Gap Length (m)	Spacing (m)
	mean/max.dev.	mean/max.dev.	mean/max.dev.	mean/max.dev.
Set 1	45.0/5.0	20.0/2.0	1.0/0.1	0.8/0.1
Set 2	25.0/5.0	10.0/1.0	1.0/0.1	0.6/0.1
Set 3	135.0/5.0	5.0/0.2	0.2/0.0	0.7/0.1
Set 4	95.0/5.0	2.5/0.2	0.2/0.0	0.5/0.1

The rock is assumed to be rigid relative to the joints. The material properties associated with the rock and joints are

density	2500 kg/m^3
joint normal stiffness	1.0 GPa/m
joint shear stiffness	1.0 GPa/m
joint friction	45 °

For demonstration purposes, the tunnel is excavated instantaneously and lined with shotcrete. The shotcrete extends in a 300° arc, and covers all but the invert of the tunnel. Figure 1.46 shows the location of the shotcrete around the tunnel periphery.

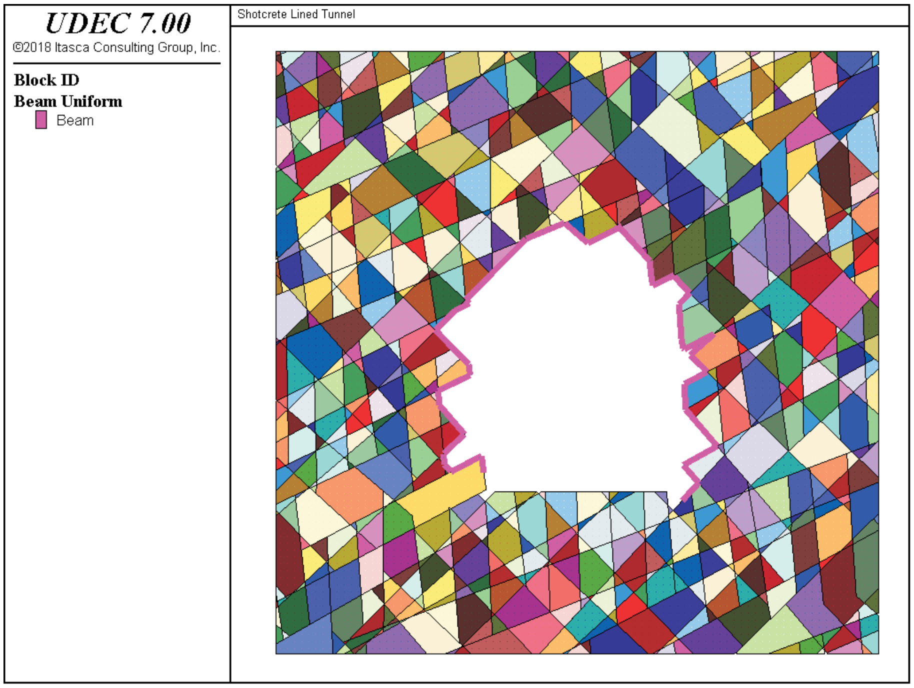


Figure 1.46 Shotcrete applied in 300° arc on tunnel periphery

The properties of the shotcrete are as follows:

Shotcrete	
density	2500 kg/m^3
Young's modulus	21.0 GPa
Poisson's ratio	0.15
tensile yield strength	20.0 MPa
residual yield strength	10.0 MPa
compressive yield strength	40.0 MPa
Rock/Shotcrete Interface	
normal stiffness	1.0 GPa/m
shear stiffness	1.0 GPa/m
friction	45 °

The data file for this example is given in Example 1.8.

Example 1.8 Shotcrete lined tunnel

```
model new
model title 'Shotcrete Lined Tunnel'
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
model random 1000
block create polygon 0 0 0 10 10 10 10 0
block cut joint-set angle 45 5 trace 20 2 gap 1 0.1 spacing 0.8 0.1
block cut joint-set angle 25 5 trace 10 1 gap 1 0.1 spacing 0.6 0.1
block cut joint-set angle 135 5 trace 5 0.2 gap 0.2 spacing 0.7 0.1
block cut joint-set angle 95 5 trace 2.5 0.2 gap 0.2 spacing 0.5 0.1
block cut crack 3.5 2.7 6.8 2.7
block joint-delete
block delete range area 0 .002
block delete range pos-x 3 7 pos-y 2.7 5
block delete range annulus center 5 5 rad 0 2
block delete range area 0.0020
block change material 1
block property material 1 density 2.5E3
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E9 ...
  stiffness-normal 1E9 friction 45 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E9 ...
  stiffness-normal 1E9 friction 45
block fix range pos-x 0,0.5  pos-y 0,10
block fix range pos-x 9.5,10 pos-y 0,10
block fix range pos-x 0,10   pos-y 0,0.5
block fix range pos-x 0,10   pos-y 9.5,10
block insitu  stress -1000000.0 0.0 -1000000.0 ...
  gradient-x 0.0 0.0 0.0 gradient-y 0.0 0.0 25000.0
model gravity 0 -10
; shotcrete lining
block structure liner create by-end-points ...
  begin 6.73665 2.54539 end 3.44999 3.00737 ...
  length-maximum 1.0E9 length-minimum 0.02 material-beam 1
block structure beam property material 1 density 2.5E3 poisson 0.15 ...
  yield-compression 100e6 yield-tension 100e6 young 2.1E10 ...
  yield-tension-residual 100e6 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 45 coupling-stiffness-normal 1E9 ...
  coupling-stiffness-shear 1E9
model save 'stun1.sav'
;
```

```
block solve ratio 1.0E-5
model save 'stun2.sav'
;
;
; change shotcrete properties to actual values with residual str = 20 MPa
block structure beam property material 1 density 2.5E3 poisson 0.15 ...
  yield-compression 40e6 yield-tension 20e6 young 2.1E10 ...
  yield-tension-residual 20e6 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 45 coupling-stiffness-normal 1E9 ...
  coupling-stiffness-shear 1E9
block solve ratio 1.0E-6
model save 'stun3.sav'
;
;
model rest 'stun2.sav'
;
; change shotcrete properties to actual values with residual str = 10 MPa
block structure beam property material 1 density 2.5E3 poisson 0.15 ...
  yield-compression 40e6 yield-tension 20e6 young 2.1E10 ...
  yield-tension-residual 10e6 cross-sectional-area 0.1 moi 8.33333E-5 ...
  shape-factor 0.83333 spacing 1 thickness 0.1 width 1 ...
  coupling-friction 45 coupling-stiffness-normal 1E9 ...
  coupling-stiffness-shear 1E9
block solve ratio 1.0E-6
model save 'stun4.sav'
```

Note that, for this example, it is not necessary to specify the shape factor and moment of inertia for the shotcrete because it has a uniform cross-section. These factors are calculated automatically.

When the **block struct liner create by-end-points** command is issued, nodes are created automatically in order to follow the irregular surface of the excavation.

Two cases are evaluated. First, a simulation is made with the residual yield strength of the shotcrete set equal to the initial yield strength of 20 MPa. Figure 1.47 shows the axial force distribution in the shotcrete, and Figure 1.48 shows the locations along the lining that have failed in tension for this case. The moment-thrust diagram for the shotcrete is shown in Figure 1.49. Note that in this figure, three failure surfaces are plotted for safety factors of 1.0, 1.2 and 1.4. The axial load and moment calculated for each liner segment are also denoted in this figure by asterisk markers. A few asterisks are plotted at or near the failure surface corresponding to a factor of safety equal to 1, indicating segments at or near failure.

In the second simulation, the residual strength is reduced to 10 MPa. Figure 1.50 shows that axial forces in the liner are reduced, as compared to Figure 1.47.

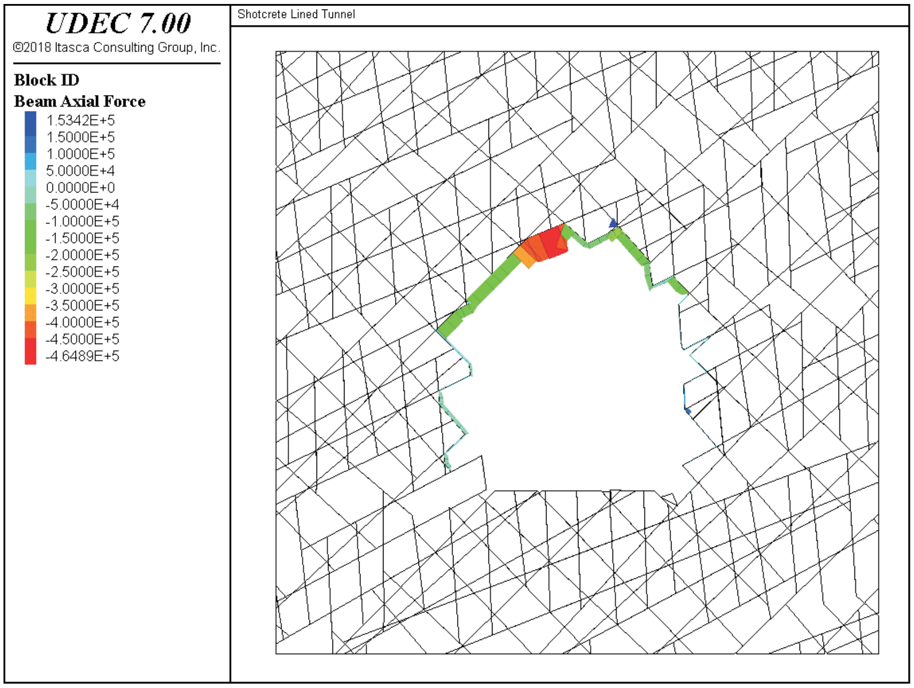


Figure 1.47 Axial force distribution in shotcrete lining – residual yield strength = 20 MPa

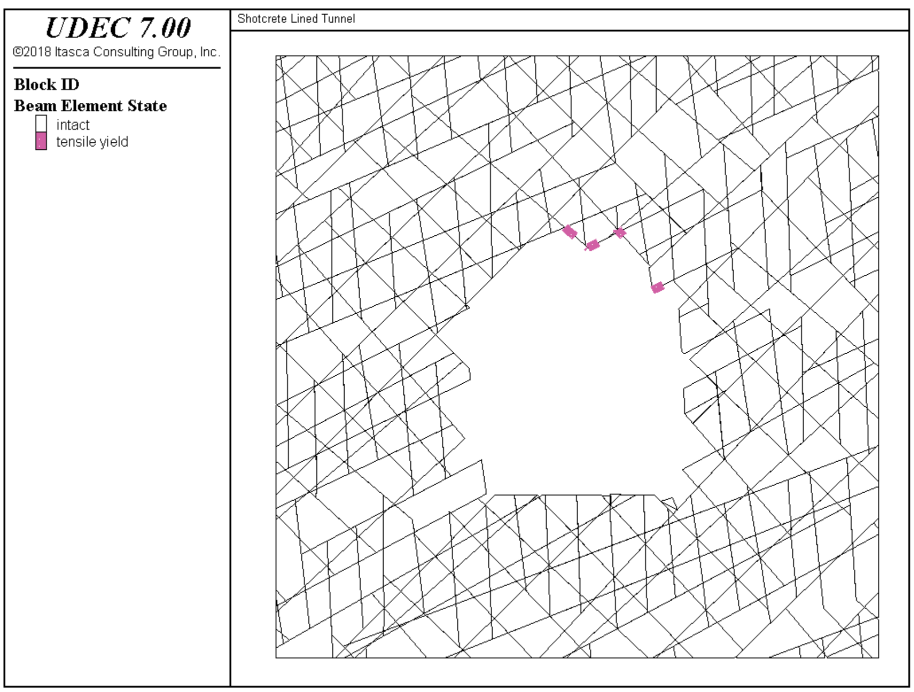


Figure 1.48 Tensile failure locations in shotcrete – residual yield strength = 20 MPa

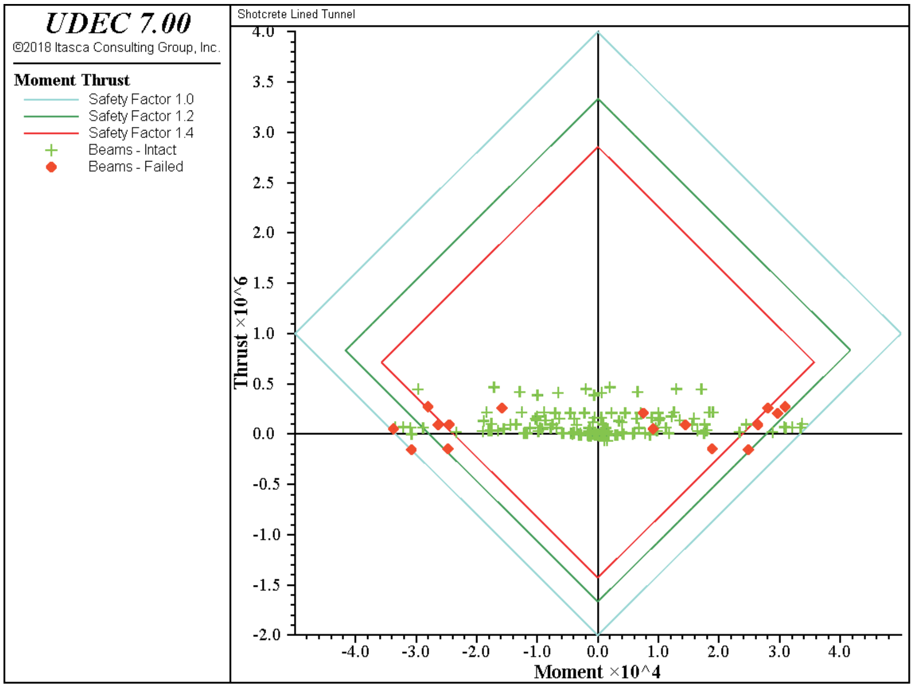


Figure 1.49 Moment-thrust diagram for tensile yield strength = 20 MPa and compressive yield strength = 40 MPa

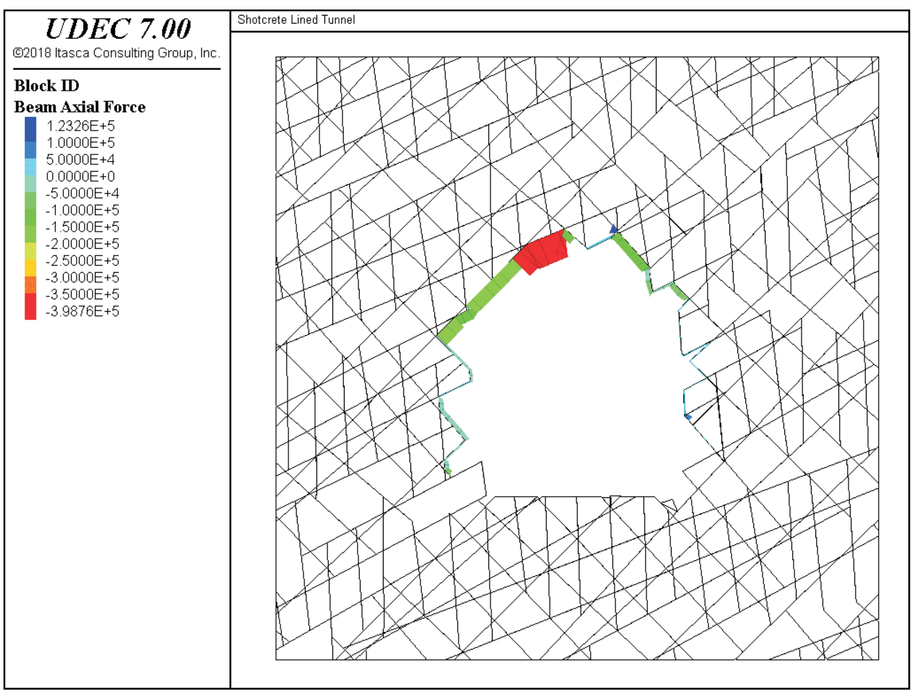


Figure 1.50 Axial force distribution in shotcrete lining – residual yield strength = 10 MPa

1.3.1.10 Example Application – Slope Stabilization

In this example, a jointed rock slope is stabilized by applying a shotcrete lining to the slope face. The problem geometry, shown in Figure 1.51, consists of a slope cut in a rock containing two continuous joint sets.

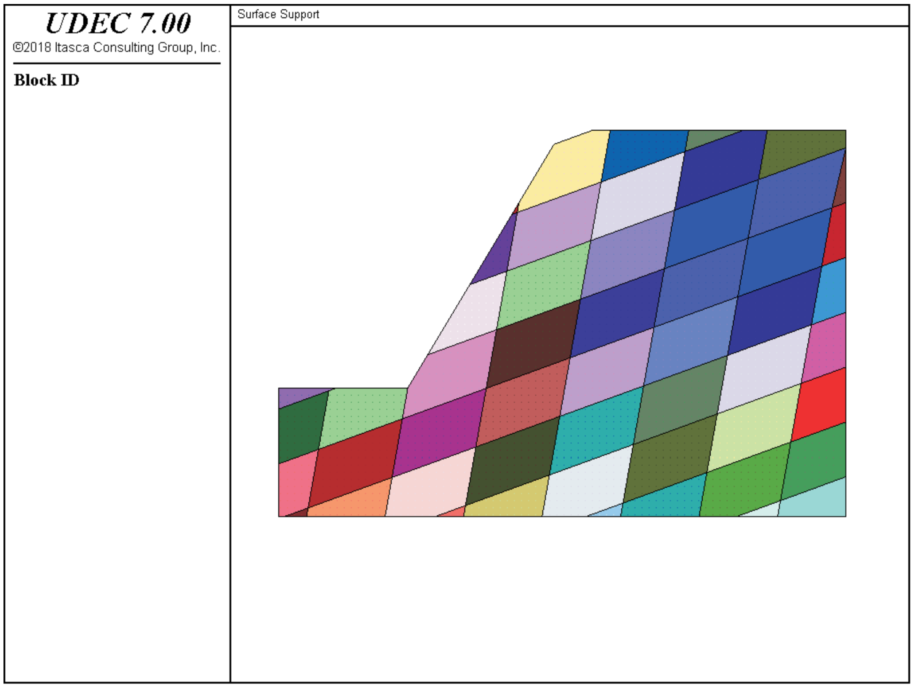


Figure 1.51 Slope cut in a jointed rock

The properties of the rock and joints are

density	2500 kg/m^3
rock bulk modulus	16.67 GPa
rock shear modulus	10.0 GPa
joint normal stiffness	10.0 GPa/m
joint shear stiffness	10.0 GPa/m
joint friction	15°

The slope is not stable for these conditions. This can be seen from the displacement vector plot in Figure 1.52. This result was obtained by first solving for the equilibrium state with a high joint friction angle, and then reducing the friction to the actual value.

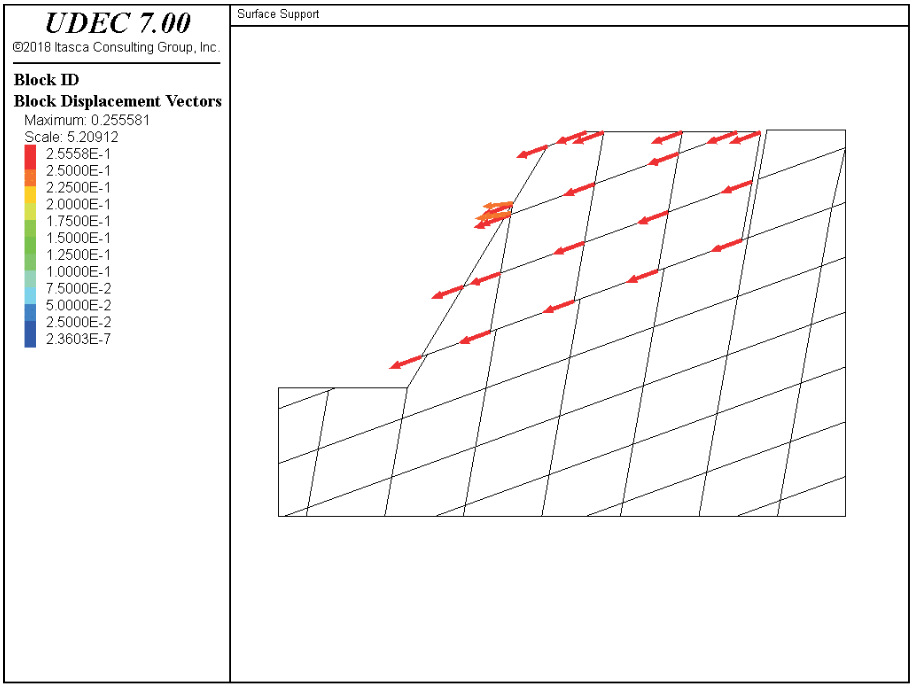


Figure 1.52 Unsupported slope is unstable

A slope lining can be placed on the slope using the structural element logic in *UDEC*. The lining is 0.1 m thick shotcrete with the following properties:

Shotcrete

density	2500 kg/m^3
Young's modulus	20.0 GPa
Poisson's ratio	0.15
tensile yield strength	3.0 MPa
residual yield strength	0.0 MPa
compressive yield strength	30.0 MPa

Rock/Shotcrete Interface

normal stiffness	100.0 GPa/m
shear stiffness	100.0 GPa/m
cohesive bond strength	2.0 MPa
tensile bond strength	1.0 MPa
friction	0

The lining is added at the equilibrium state with high joint friction. Now, when the friction is reduced, the slope is stable. This can be seen from Figure 1.53.

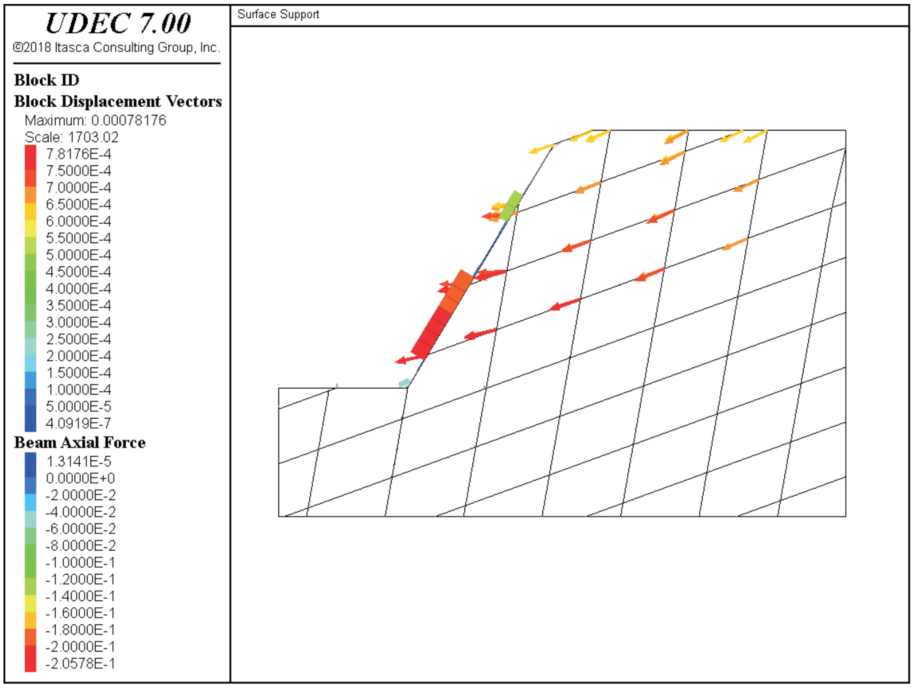


Figure 1.53 Slope is stabilized with shotcrete lining

The data file for this example is given in Example 1.9.

Example 1.9 Slope stabilization

```
model new
model title 'Surface Support'
block tolerance corner-round-length 0.1
block tolerance minimum-edge-length 0.2
block create polygon 0 -5 0 0 5 0 11 10 22 10 22 -5
block cut joint-set angle 20 spacing 2 origin 5 1
block cut joint-set angle 80 spacing 3 origin 5 0
block delete range area 0.1
block delete range pos-x 10.5 12 pos-y 9.5 10
block zone gen edge 10.0
block zone group 'block'
block zone cmodel assign elastic density 2.5E-3 bulk 1.6667E4 shear 1E4 ...
  range group 'block'
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 45 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 45
```

```
block insitu  stress -0.125 0.0 -0.25 gradient-x 0.0 0.0 0.0 ...
  gradient-y 0.0125 0.0 0.025
block gridpoint apply velocity-x 0 range pos-x -0.1 0.1 pos-y -5.1 0.1
block gridpoint apply velocity-x 0 range pos-x 21.9 22.1 pos-y -5.1 10.1
block gridpoint apply velocity-y 0 range pos-x -0.1 22.1 pos-y -5.1 -4.9
model gravity 0.0 -10.0
block solve ratio 1.0E-5
model save 'ls1.sav'
;
;
block contact group 'weak joint'
block contact cmodel assign area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 15 range group 'weak joint'
; new contact default
block contact cmodel default area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 15
block cycle 10000
model save 'ls2.sav'
;
;
model restore 'ls1.sav'
block contact group 'weak joint'
block contact cmodel assign area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 15 range group 'weak joint'
; new contact default
block contact cmodel default area stiffness-shear 1E4 ...
  stiffness-normal 1E4 friction 15
block structure liner create by-end-points begin 3.86309 7.78702E-5 ...
  end 12.2684 10 length-maximum 0.5 length-minimum 0.1 material-beam 1
block structure beam property material 1 density 2.5E-3 poisson 0.15 ...
   yield-compression 30 yield-tension 3 young 2E4 ...
   cross-sectional-area 0.1 moi 8.33333e-5 shape-factor 0.83333 ...
   spacing 1 thickness 0.1 width 1 coupling-cohesion 2 ...
   coupling-tension 1 coupling-stiffness-normal 1E5 ...
   coupling-stiffness-shear 1E5
block solve ratio 1.0E-5
model save 'ls3.sav'
```

*1.3.2 Support Members (***block struct support** *Command)*

Support members are intended to model hydraulic props, wooden props, sticks or wooden packs. In its simplest form, a support member is a spring connected between two boundaries. The spring may be linear, or it may obey an arbitrary relation between axial force and axial displacement, as prescribed from a table of values. The support member has no independent degrees of freedom; it simply imposes forces on the boundaries to which it is connected. A support member may also have a width associated with it. In this case, it behaves as if it were composed of several parallel members spread out over the specified width. The force-displacement behavior can also be made load-rate dependent, as described in Section 1.3.2.2.*

1.3.2.1 Standard Formulation

In the standard formulation for support elements, two options are available to describe the axial force-displacement relation. In one option, the relation is elastic-plastic and is defined by an axial stiffness and a compressive yield limit. In the other option, the relation between force and displacement is prescribed by a table of force/displacement values. These two options are illustrated in Figure 1.54.

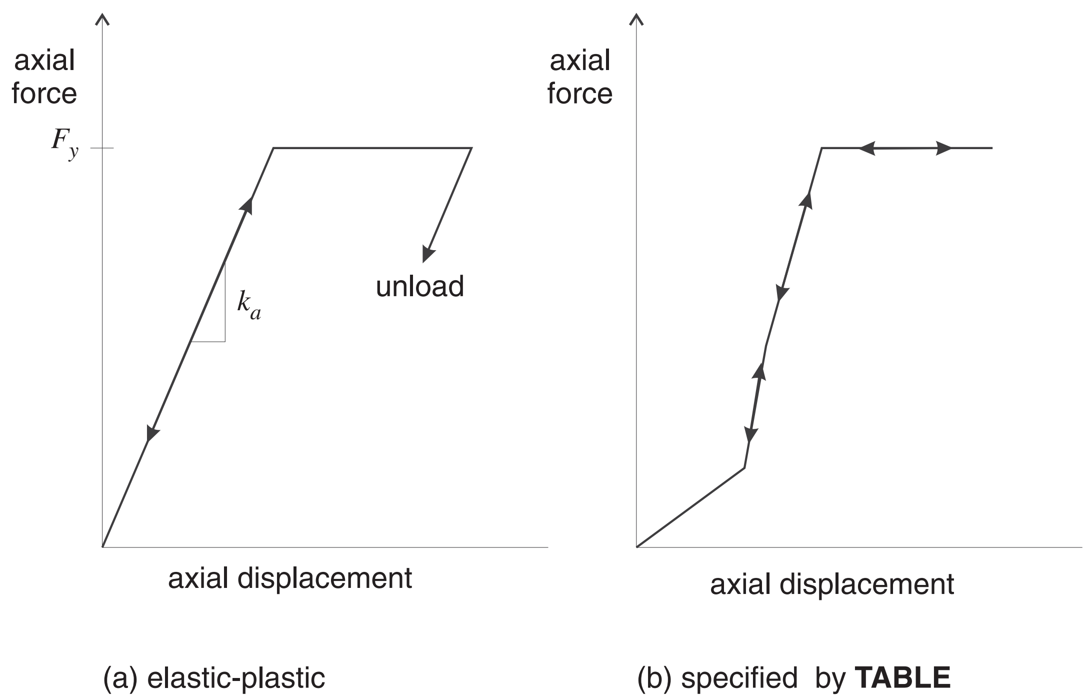


Figure 1.54 Force-displacement behavior for standard support model

---

* The load-rate dependent model was developed with funding and technical support from CSIR MiningTEK, Johannesburg, Republic of South Africa. The model is experimental and has only been partially tested. It should be used with caution.

The formulations for the two options differ in their relation of axial force to axial displacement. For the elastic-plastic option, the force/displacement relation is incremental; for the table option, the force in the support member is related to the total displacement of the member.

When a support member has nonzero width (i.e., it is divided into sub-members), the force in each sub-member is computed in one of two ways. For the elastic-plastic option, the force F in each sub-member is defined by

$$\Delta F = \frac{k_a}{(n+1)} \Delta u_a \tag{1.37}$$

$$\text{and} \quad F \leq \frac{F_y}{(n+1)}$$

where n = number of sub-members;
Δu = axial displacement increment of the sub-member;
ΔF = axial force increment;
k_a = axial stiffness; and
F_y = compressive force limit.

and for the table relation, the force is defined by

$$F = \frac{f(u)}{(n+1)} \tag{1.38}$$

where u = total displacement of the sub-members; and
$f(u)$ = table lookup function.

The total force exerted by the support is the sum of the sub-member forces.

These formulations also differ in their treatment of unloading. The elastic-plastic option unloads using the axial stiffness, k_a. The relation specified by a table unloads according to the specified table, as indicated in Figure 1.54.

1.3.2.2 Load-Rate Dependency

Several types of support elements used in reef mining (e.g., profile props, yielding props and sticks) have a force-displacement behavior that is dependent on the rate of loading. Measurements in the laboratory and underground indicate that the shape of the force-displacement curve is similar for different loading rates, but that the maximum force that the support can withstand varies with the loading rate.

The force-displacement behavior, including load-rate dependency, is considered to be composed of a rate-dependent curve superimposed on a static (slow loading rate) curve. The additional force due to the higher loading rate is calculated incrementally, based upon an increment in deformation and the proximity of the support force to the maximum force level. This is expressed as

$$\Delta F_{i+1} = \Delta F_i + C\left[1 - \frac{\Delta F_i}{\Delta F_{max}}\right](u_{i+1} - u_i) \tag{1.39}$$

where ΔF_{i+1} = new addition in support force;
ΔF_i = old addition in support force;
u_{i+1} = new deformation of support element;
u_i = old deformation of support element;
C = stiffness constant; and
ΔF_{max} = maximum support force addition.

The total force in the support element is calculated by adding the rate-dependent force increment to the static support force:

$$F_{total} = F_{static} + \Delta F_{i+1} \tag{1.40}$$

As the total rate-dependent force increment, ΔF, approaches the maximum force increment, ΔF_{max}, the increment in total support force is gradually reduced, and an asymptotically increasing support force is added to the static force. This ensures that the shape of the static force-displacement curve is replicated for higher loading rates. The stiffness constant, C, controls the rate at which the maximum force level is approached. The maximum force increment, ΔF_{max}, is estimated for a specific loading rate. The following expression is used to describe the maximum force.

$$\Delta F_{max} = \dot{u}^{\alpha} F_{max} - F_{max.static} \tag{1.41}$$

where $F_{max.static}$ = maximum static force;
$\dot{u}$ = loading rate;
F_{max} = maximum force at highest loading for the support element; and
α = constant depending on support type.

For large displacements of a prop, the support force will eventually go to zero as the prop fails. This effect is simulated in *UDEC* by having F_{max} decrease linearly for displacements larger than the displacement at which the maximum static force, $F_{max.static}$, is reached. The maximum support force addition for displacements greater than this limiting displacement is

$$\Delta F_{max.lim} = \Delta F_{max} - K_F(u - u_{lim}) \tag{1.42}$$

where $\Delta F_{max.lim}$ = maximum support force addition for $u > u_{lim}$;
u = displacement of the support unit;
u_{lim} = displacement at the maximum static force; and
K_F = constant for reducing maximum support force addition.

K_F defines the gradient from the point at which u_{lim} is reached to the point that ΔF_{max} is equal to zero.

Figure 1.55 illustrates the force-displacement behavior that is represented by Eqs. (1.39) through (1.42).

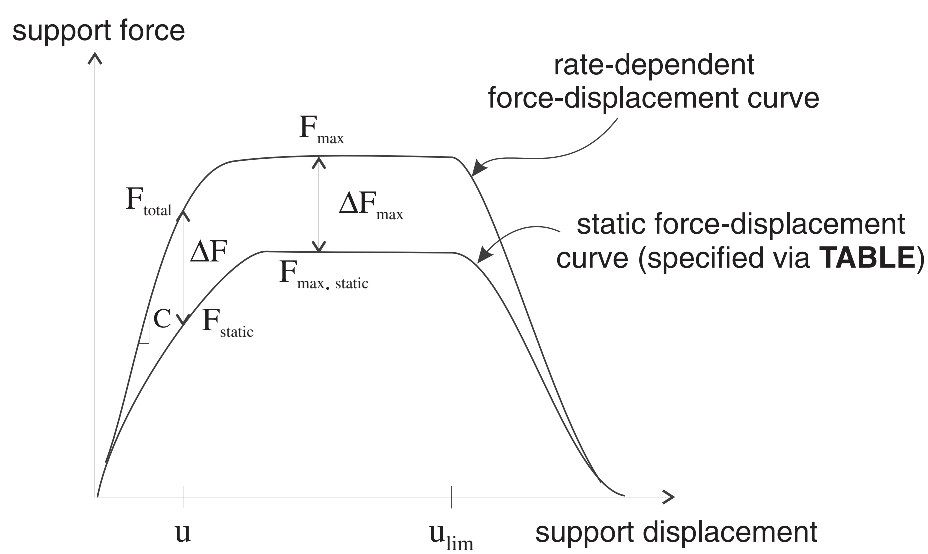


Figure 1.55 Force-displacement behavior including load-rate dependent support force

The support stiffness, C, in Eq. (1.39) can also be related to the maximum support force addition. The following relation is used in *UDEC*.

$$C = k(\Delta F_{max} + F_{max.static}) \tag{1.43}$$

where k = constant depending on support type.

A tensile limit can also be specified for the model. This is the maximum tensile force that the support element can withstand. If the tensile force in the support exceeds t_{max}, the support force is set to zero and the support is deleted.

The constant k, along with the parameters ΔF_{max}, α and t_{max}, and a table defining the static force-displacement relation, are the required input for the rate-dependent model.

1.3.2.3 Numerical Stability

For support members, the stiffness is *not* taken into account in the consideration of numerical stability, mainly because it is difficult to estimate the stiffness in advance for table lookups. If the support is stiffer than the rock, this may lead to numerical instability, but the reverse is likely to be the case in most problems. If numerical instabilities do occur, the timestep should be reduced using the **block mechanical timestep-factor** command.

1.3.2.4 Support Member Properties

The standard model for support elements in *UDEC* requires two input parameters:

(1) axial stiffness of the support member, k_a (force/displacement); and

(2) compressive yield strength (force) of the support member, F_y.

If the support member contains sub-elements, then the axial stiffness and yield strength are for the group of sub-members.

Alternatively, the relation between axial force and axial displacement can be specified by a lookup table. However, a table is not recommended if the standard support is subjected to significant unloading. When a table is to be specified, the axial stiffness, k_a, should be entered as a negative integer. The table number is the absolute value of this integer. Table values are specified with the **table** command.

For the load-rate dependent model, the user must supply four model parameters plus a table of the static force-displacement relation. The four parameters are

(1) exponent constant, α;

(2) maximum support force addition, ΔF_{max};

(3) support stiffness constant, k; and

(4) tensile strength (force), t_{max}.

The four parameters are usually determined from laboratory or underground testing. CSIR (1993) provides some preliminary results for k, ΔF_{max} and α for three support types. The values are summarized in Table 1.7.

Table 1.7 Support properties for profile props, sticks and yielding props (CSIR 1993)

Support Type	α		ΔF_{max} (kN)		k
			loading rate (m/s)		
		2×10^{-7}	2×10^{-4}	1.0	
profile props	0.0638	272	418	723	27.6
sticks	0.0197	74	85	100	20.0
yielding props	0.0481	200	240	600	40.0

The force-displacement behavior specified by a table for the rate-dependent model must be concave, and must end at a zero support force.

Support elements typically offer almost no resistance to tensile loading. However, specifying a small tensile strength, t_{max}, for a support element instead of setting the strength to zero is advisable. This ensures that the element is not deleted accidentally during the initial transient phase of the calculation.

1.3.2.5 Summary of Commands Associated with Support Members

All of the commands associated with support members are listed in Table 1.8. See Help in *UDEC* for a detailed explanation of these commands.

Table 1.8 Keywords associated with **block struct support** *command*

create	keyword		
	angle	*value*	
	group	*name*	
	location	*x1 y1*	
	material	*n*	
	segment	*n*	
	type	keyword	
		standard	
		rate-dependent	
	width	*value*	
delete	<**range**>		
delete-auto			
list	**keyword**		
	group		
	property		
property	**mat** *n*	keyword	
		load-rate-exponent	*value*
		maximum-force-addition	*value*
		spacing	*value*
		stiffness-constant	*value*
		stiffness-axial	*-nt*
		yield-compression	*value*
		yield-tension	*value*
table	*n*	*x1 y1* <*x2 y2*>	<*x3 y3*> . . .

1.3.2.6 Example Application – Support of Faulted Ground

This example problem illustrates the use of support members in faulted ground. Figure 1.56 shows the location of the vertical fault and support members. The support members have the force-displacement relation shown in Figure 1.57. The specified support "yields" at 40 MN, as shown.

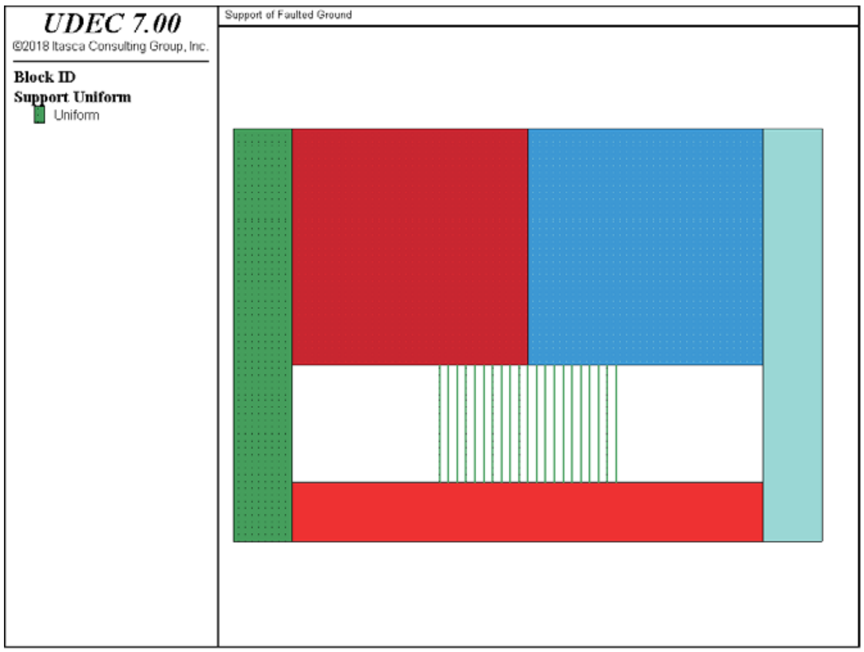


Figure 1.56 Support members before loading

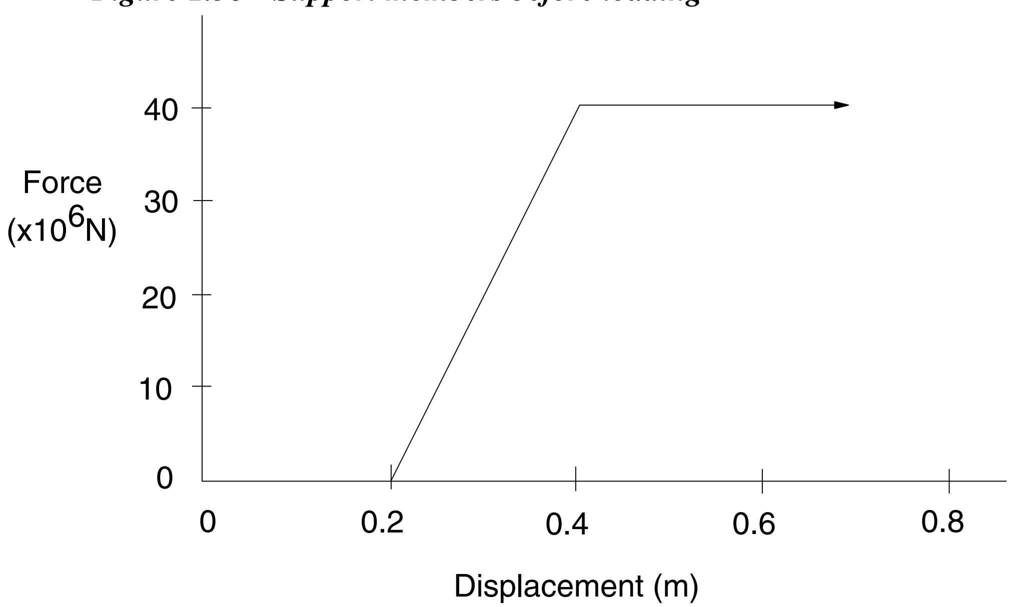


Figure 1.57 Force-displacement relation for support in example problem

Example 1.10 contains the data file for this example.

Example 1.10 Support of faulted ground

```
model new
model title 'Support of Faulted Ground'
block tolerance corner-round-length 0.1
block tolerance minimum-edge-length 0.2
block create polygon -1 0 -1 7 9 7 9 0
block cut crack 0 0 0 7
block cut crack 8 0 8 7
block cut crack 0 1 8 1
block cut crack 0 3 8 3
block cut crack 4 3 4 7
block delete range pos-x 0 8 pos-y 1 3
block zone gen quad 1.1 range pos-x 0 8 pos-y 0 7
;
block zone group 'block'
block zone cmodel assign mohr-c density 1E3 bulk 1.5E8 shear 5E7 ...
  coh 2.5E5 range group 'block'
block change material 1
block property material 1 density 1E3
block contact group 'joint'
block contact cmodel assign area stiffness-shear 1E8 ...
  stiffness-normal 1E8 range group 'joint'
; new contact default
block contact cmodel default area stiffness-shear 1E8 stiffness-normal 1E8
block fix range pos-x -1,0 pos-y 0,7
block fix range pos-x  8,9 pos-y 0,7
block gridpoint apply velocity-x 0 range pos-x -0.1 8.1 pos-y -0.1 0.1
block gridpoint apply velocity-y 0 range pos-x -0.1 8.1 pos-y -0.1 0.1
block hide  range pos-x 4 8 pos-y 3 7
block gridpoint apply velocity-x 0 range pos-x -0.1 4.1 pos-y 6.9 7.1
block gridpoint apply velocity-y 0 range pos-x -0.1 4.1 pos-y 6.9 7.1
block show
block hide  range pos-x 0 4 pos-y 3 7
block gridpoint apply velocity-y -0.2 range pos-x 3.9 8.1 pos-y 6.9 7.1
block show
model save 'sup1.sav'
;
;
block structure support create location 4 2 width 3 segment 20 material 3
block structure support property material 3 stiffness-axial -1
table 1 add 0 0 0.2 0 0.4 0.4e7 10.0 0.4e7
block cycle 5000
model save 'sup2.sav'
```

Figure 1.58 shows the deformed position of the supports after the upper surface on the right side of the model has displaced downward approximately 0.6 m.

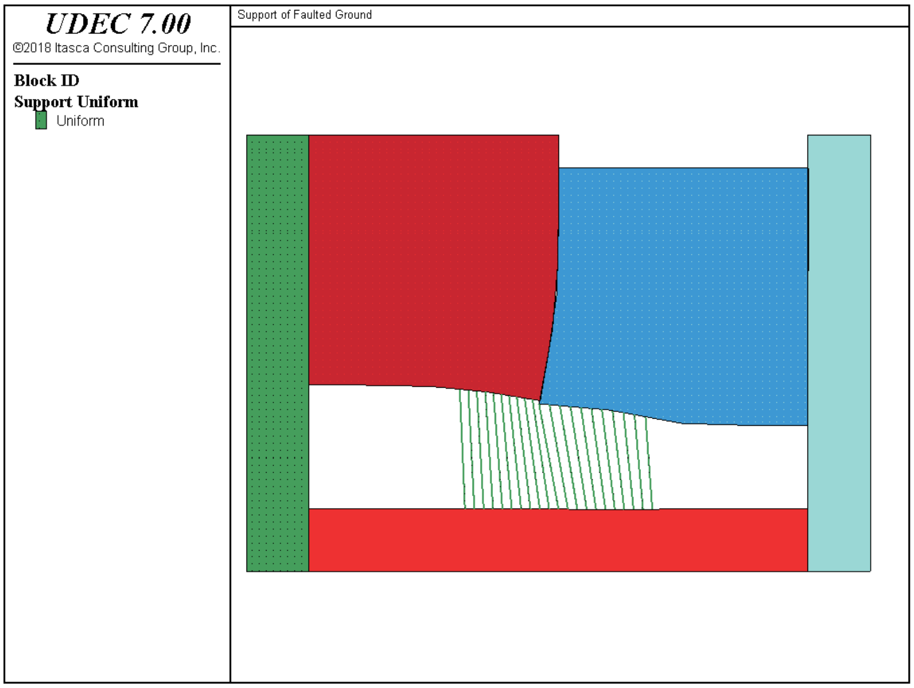


Figure 1.58 Support members after loading

1.3.2.7 Example Application – Load-Rate Dependent Support

The rate-dependent support model is used to simulate the behavior of profile props subjected to loading and unloading. Each prop is composed of five sub-elements and has a width of 2 m. The model configuration is shown in Figure 1.59.

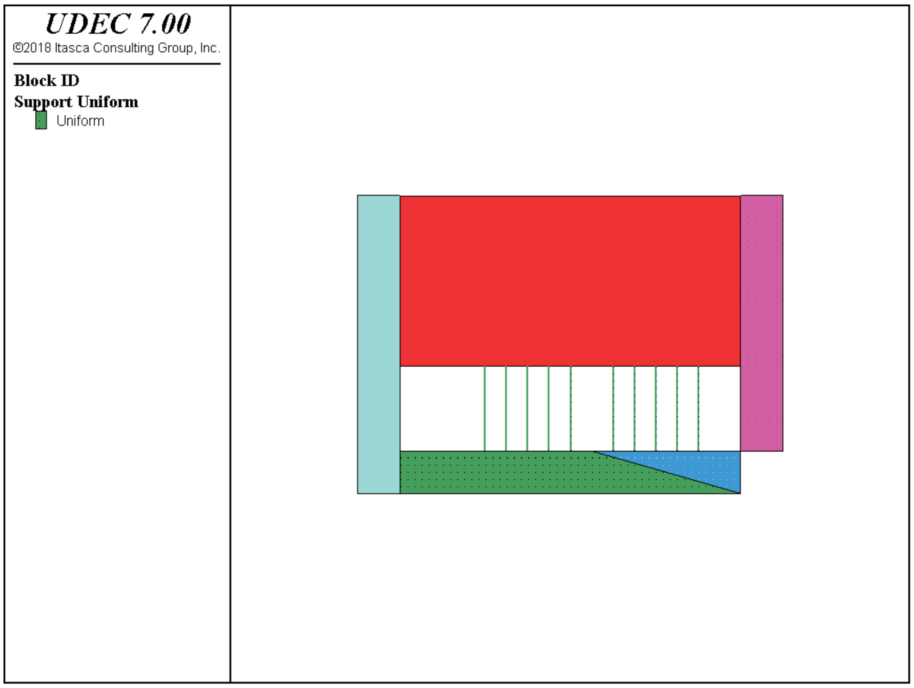


Figure 1.59 Model test for rate-dependent support members

There are two stages to this analysis. First, the model is subjected to gravity loading and brought to an equilibrium state. Then, the bottom-right triangular block is freed. The prop connected to this block unloads when this block moves. The axial force and displacement in the props are monitored during both stages. Example 1.11 contains the data file for this example.

Example 1.11 Load-rate dependent support

```
Model new
block tolerance corner-round-length 0.1
block tolerance minimum-edge-length 0.2
block create polygon -1 0 -1 7 9 7 9 0
block cut crack 0 0 0 7
block cut crack 8 0 8 7
block cut crack 0 1 8 1
block cut crack 0 3 8 3
block cut crack 8 1 9 1
block cut crack 4.5 1 8 0
```

```
block delete range pos-x 0 8 pos-y 1 3
block delete range pos-x 8 9 pos-y 0 1
block change material 1
block property material 1 density 1E3
;
; assign friction for vertical and inclined joints
block contact group ’joint10’
block contact group ’joint5’ range pos-x -0.1 0.1 pos-y -0.1 7.1
block contact group ’joint5’ range pos-x 7.9 8.1 pos-y -0.1 7.1
block contact cmodel assign area stiffness-shear 1E8 ...
  stiffness-normal 1E8 friction 10 range group ’joint10’
block contact cmodel assign area stiffness-shear 1E8 ...
  stiffness-normal 1E8 friction 5 range group ’joint5’
; new contact default
block contact cmodel default area stiffness-shear 1E8 ...
  stiffness-normal 1E8 friction 10
fix range pos-x -1,0 pos-y 0,7
fix range pos-x 8,9  pos-y 0,7
fix range pos-x 0,8  pos-y 0,1
block insitu  stress -10000.0 0.0 -100000.0
model gravity 0 -10
model save ’lrsup1.sav’
;
;
; install two supports
block structure support create location 3 2 segment 5 width 2.0 ...
  material 3 type rate-dependent
block structure support create location 6 2 segment 5 width 2.0 ...
  material 3 type rate-dependent
block structure support property material 3 load-rate-exponent 0.0638
block structure support property material 3 maximum-force-addition 723000
block structure support property material 3 stiffness-constant 27.6
block structure support property material 3 yield-tension 1000
block structure support property material 3 stiffness-axial -1
table 1 add 0 0 50e-3 472000 200e-3 472000 250e-3 0
;
fish define sup_forc1
  ;     force/disp. hist of non-deleted support
  ;     force/disp. hist of deleted supportp
  sup_forc1 = block.struct.support.force.normal(iad_sup_1)
  sup_disp1 = block.struct.support.disp.normal(iad_sup_1)
  sup_forc2 = block.struct.support.force.normal(iad_sup_2)
  sup_disp2 = block.struct.support.disp.normal(iad_sup_2)
end
fish define _sup
  iad_sup_1 = block.struct.support.head
```

```
  iad_sup_2 = block.struct.support.next(iad_sup_1)
end
@_sup
@sup_forc1
history interval  1
fish history @sup_forc1
fish history @sup_disp1
fish history @sup_forc2
fish history @sup_disp2
;
; equilibrate
block solve ratio 1.0E-5
model save 'lrsup2.sav'

;
;
;  allow support to be deleted
block struct support auto-delete on
block free range pos-x 4.5,8 pos-y 0,1
block cycle 2000
model save 'lrsup3.sav'
```

The load/displacement response of one sub-member of the left prop during the gravity-loading stage is shown in Figure 1.60. The response of the right prop is similar. The force-displacement behavior is dependent on the prescribed properties for a profile prop, taken from Table 1.7. The initial loading of the prop is nonlinear, followed by linear unloading and reloading as the force in the sub-member approaches a static value of approximately 32 kN.

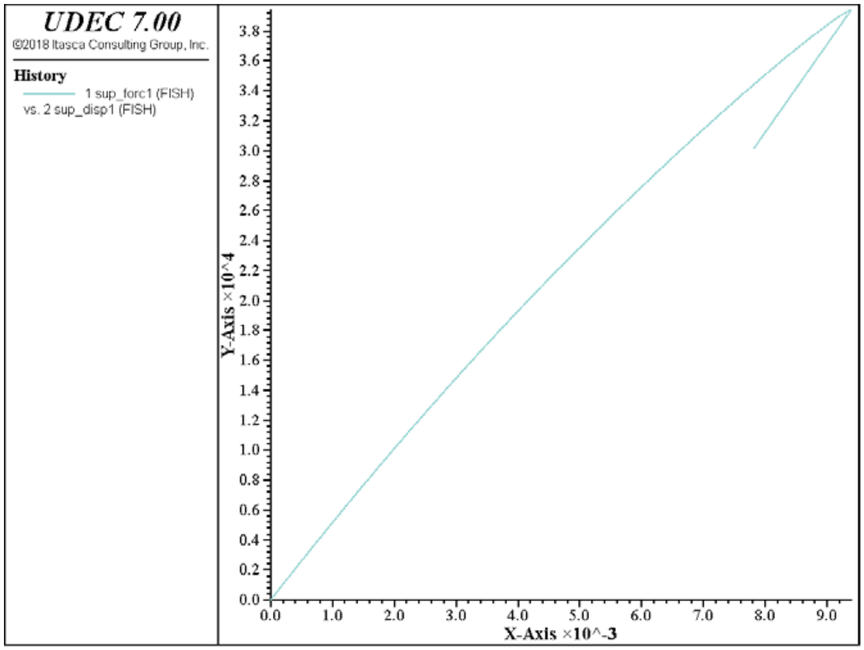


Figure 1.60 Axial force versus displacement response for left prop

When the floor block is freed in the second stage, it begins to slide and the right prop unloads. By specifying the command **block struct support delete-auto**, the support will be deleted when the tensile force exceeds the specified tensile limit, **yield-tension** (**block struct support property** command). The unloading of the right prop is indicated by the load/displacement plot in Figure 1.61. The load is transferred to the left prop, as shown in Figure 1.62.

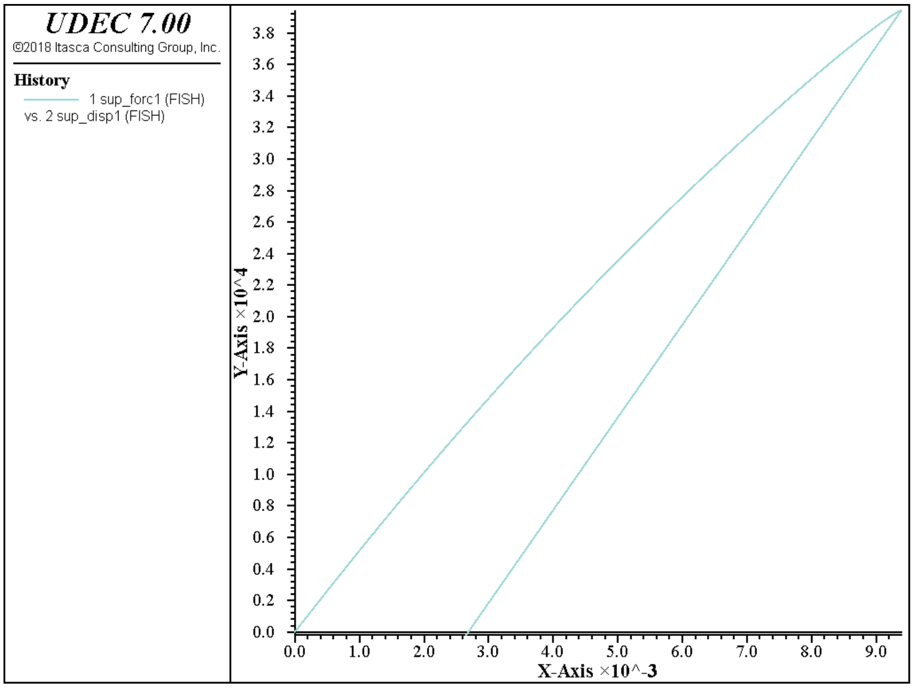


Figure 1.61 Axial force versus displacement response for right prop unloading

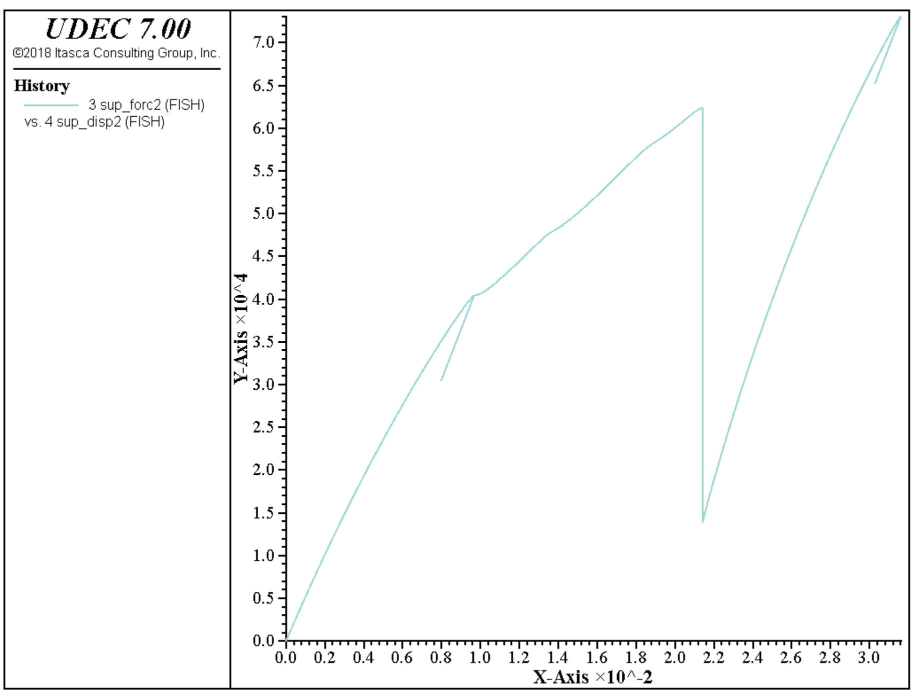


Figure 1.62 Axial force versus displacement response in left prop for right prop unloading

1.4 Material Properties

Property numbers are assigned to local reinforcement, cables, beam elements, rockbolts, and support members with the **property** keyword. Note that all quantities must be given in an equivalent set of units (see Table 1.9). For reinforcement elements, cable elements, rockbolts and support members, stiffness has units of [force/displacement] and strength has units of [force]. For structural (beam) elements, stiffness has units of [stress/displacement] and strength has units of [stress]. Also, for cable elements, rockbolts and structural (beam) elements, the code does take into account the weight of the structure when calculating loads.

Table 1.9 Systems of units – structural elements

Property	Unit	SI				Imperial	
Area	$length^2$	m^2	m^2	m^2	cm^2	ft^2	in^2
Bond Stiffness	force/length/disp	N/m/m	kN/m/m	MN/m/m	Mdynes/cm/cm	lb_f/ft/ft	lb_f/in/in
Bond Strength	force/length	N/m	kN/m	MN/m	Mdynes/cm	lb_f/ft	lb_f/in
Density	mass/volume	kg/m^3	10^3 kg/m^3	10^6 kg/m^3	10^6 g/cm^3	$slugs/ft^3$	$snails/in^3$
Elastic Modulus	stress	Pa	kPa	MPa	bar	lb_f/ft^2	psi
Failure Strain	—						
Moment of Inertia	$length^4$	m^4	m^4	m^4	cm^4	ft^4	in^4
Plastic Moment	force-length	N-m	kN-m	MN-m	Mdynes-cm	ft-lb_f	in-lb_f
Stiffness*	force/disp	N/m	kN/m	MN/m	Mdynes/cm	lb_f/ft	lb_f/in
Stiffness**	stress/disp	Pa/m	kPa/m	MPa/m	bar/cm	lb_f/ft^3	lb_f/in^3
Yield Strength*	force	N	kN	MN	Mdynes	lb_f	lb_f
Yield Strength**	stress	Pa	kPa	MPa	bar	lb_f/ft^2	psi

where
1 bar = 10^6 dynes / cm^2 = 10^5 N / m^2 = 10^5 Pa,
1 atm = 1.013 bars = 14.7 psi = 2116 lb_f / ft^2 = 1.01325 × 10^5 Pa,
1 slug = 1 lb_f– s^2 / ft = 14.59 kg,
1 snail = 1 lb_f– s^2 / in, and
1 gravity = 9.81 m / s^2 = 981 cm / s^2 = 32.17 ft / s^2.

* Refers to axial, normal and shear stiffnesses and strength related to the material of reinforcement elements, cable elements, rockbolts or support members.

** Refers to axial, normal and shear stiffnesses and strength related to the material of structural (beam) elements and structure/block interfaces.

1.5 Modeling Considerations

1.5.1 2D/3D Equivalence

Reducing 3D problems to 2D problems with regularly spaced structural elements involves averaging the effect in 3D over the distance between the elements. Donovan et al. (1984) suggest that linear scaling of material properties is a simple and convenient way of distributing the discrete effect of elements over the distance between elements in a regularly spaced pattern.

The relation between actual properties and scaled properties can be demonstrated by considering the strength properties for regularly spaced rockbolts. The actual maximum normal force per length of the rockbolt is defined by Eq. (1.25). Internally, *UDEC* uses the expression

$$\frac{(F_n^{\max})^s}{L} = (cs_{\text{ncoh}})^s + p' \times \tan(cs_{\text{nfric}}) \times (\text{perimeter})^s \tag{1.44}$$

where $(F_n^{\max})^s$ is the (scaled) maximum normal force per unit model thickness calculated by *UDEC*. (The superscript ^{s} does not denote a power.) We want the total force calculated by *UDEC* over a spacing, S, to be the same as the actual force. The actual maximum normal force is then

$$F_n^{\max} = (F_n^{\max})^s \times S \tag{1.45}$$

and the actual normal force is

$$F_n = (F_n)^s \times S \tag{1.46}$$

The relation between the actual force and the *UDEC* force can be satisfied by substituting Eq. (1.45) and the following relations into Eq. (1.44).

$$(cs_{\text{ncoh}})^s = \frac{cs_{\text{ncoh}}}{S} \tag{1.47}$$

$$(\text{perimeter})^s = \frac{\text{perimeter}}{S} \tag{1.48}$$

The actual normal stress on the rockbolt, σ_n, is calculated by dividing the actual force by the actual effective area (perimeter $\times L$):

$$\sigma_n = \frac{(F_n)^s \times S}{\text{perimeter} \times L} \tag{1.49}$$

Note that the choice to scale perimeter is arbitrary, because only the product $\tan(cs_{\text{nfric}}) \times$ perimeter is relevant. Alternatively, the friction term could be scaled.

It is important to remember that the forces (and moments) for structural elements that are calculated by *UDEC* are *scaled* forces (and moments). The actual forces and moments can be calculated by multiplying the *UDEC* forces and moments by *S*. *FISH* access to *UDEC* values for forces and moments access *scaled* values, and thus should be multiplied by the appropriate spacing value to determine the actual values.

A scaling property (**spacing**) is provided for the structural elements to scale properties, and account for a spaced pattern of beams, cables, reinforcement, rockbolts, and supports. When the spacing property is specified, the actual properties of the structural element are input. The scaled properties are then calculated automatically by dividing the actual properties by the spacing, *S*. When the calculation is complete, the *actual* forces and moments in the spaced structural elements are then determined automatically (by multiplying by the spacing) for presentation in output results.

The following lists summarize the structural element properties that are scaled when the spacing property is specified to simulate regularly spaced structural elements.

For reinforcement elements, several properties are scaled:

(1) axial stiffness;

(2) ultimate axial capacity;

(3) shear stiffness; and

(4) ultimate shear capacity.

For cable elements, several properties are scaled:

(1) elastic modulus of the cable;

(2) tensile yield strength of the cable;

(3) compressive yield strength of the cable;

(4) stiffness of the grout; and

(5) cohesive strength of the grout.

For rockbolt elements, several properties are scaled:

(1) elastic modulus of the rockbolt;

(2) plastic moment of the rockbolt;

(3) tensile yield strength of the rockbolt;

(4) compressive yield strength of the rockbolt;

(5) stiffness of the shear coupling spring;

(6) cohesive strength of the shear coupling spring;

(7) stiffness of the normal coupling spring;

(8) cohesive strength of the normal coupling spring; and

(9) exposed perimeter of the rockbolt.

For beam elements, several properties are scaled:

(1) elastic modulus of the element;

(2) tensile yield strength of the element;

(3) residual tensile yield strength of the element;

(4) cohesive yield strength of the element;

(5) residual compressive yield strength of the element;

(6) interface normal stiffness;

(7) interface shear stiffness;

(8) interface cohesion; and

(9) interface tensile strength.

For support elements, two properties are scaled:

(1) axial stiffness of the support member; and

(2) compressive yield strength of the support member.

The spacing property also applies to gravity loads, which are calculated using the cross-sectional area and the scaled structure density. Any pretensioning that is specified to cable elements (using the command **block struct cable fix-tension**) is scaled if **cb_spacing** has been specified given.

If loading is applied using the **block struct beam node *n* load** command, these loads are *not* scaled when the spacing property is assigned. The loads should be scaled by dividing by *S*.

The following example illustrates the simulation of regularly spaced structural elements. In this case, vertical rockbolts at an equal spacing of 2 m are subjected to axial loading. The actual elastic modulus of the rockbolt is 10 GPa, and the actual stiffness of the shear coupling spring is 1 GN/m/m. The cohesive strength of the shear coupling spring is set to a high value to prevent shear failure for this simple example. A vertical axial loading of 2 MN is applied at the top of the pile, and the pile spacing is set to 2 m. Example 1.12 lists the commands for this example.

The model is run for both the case in which **spacing** is specified, and the case in which it is not. In the second case, the input values for elastic modulus and shear coupling spring stiffness are scaled (by dividing by 2). Note that for both cases, the applied vertical load is scaled (**block struct beam node 2 load 0.0, 1000000.0, 0.0**).

Figure 1.63 displays the result for the first case. When **spacing** is given, the actual axial forces are displayed in the pile axial force plot. Figure 1.47 shows the results for the second case. When **spacing** is not given, but the input properties are scaled, the axial force plot displays the scaled values for axial force. The axial forces in Figure 1.64 must be multiplied by 2 to obtain the actual values.

Example 1.12 Axial loading of rockbolts at 2 m spacing

```
;  cs_spacing = 2
round 5E-3
edge 1E-2
block 0,0 0,5 5,5 5,0
gen edge 1.0
group zone 'elastic'
zone model elastic density 1E3 bulk 1E9 shear 3E8 range group 'elastic'
boundary xvelocity 0 range -0.1,5.1 -0.1,0.1
boundary yvelocity 0 range -0.1,5.1 -0.1,0.1
boundary xvelocity 0 range -0.1,0.1 -0.1,5.1
boundary xvelocity 0 range 4.9,5.1 -0.1,5.1
struct rockbolt begin 2.5,5.0 end 2.5,2.5 seg 5 prop 1
struct prop 1 cs_scoh 1E20 cs_sstiff 1E9 e 1E10 density 4000 radius 1 &
 spacing 2 yield 1E20 ycomp 1E20
struct node 1  load 0.0,1000000.0 0.0
solve ratio 1.0E-5
save sp1.sav

new
; divide properties by 2
round 5E-3
edge 1E-2
block 0,0 0,5 5,5 5,0
gen edge 1.0
group zone 'elastic'
zone model elastic density 1E3 bulk 1E9 shear 3E8 range group 'elastic'
```

```
boundary xvelocity 0 range -0.1,5.1 -0.1,0.1
boundary yvelocity 0 range -0.1,5.1 -0.1,0.1
boundary xvelocity 0 range -0.1,0.1 -0.1,5.1
boundary xvelocity 0 range 4.9,5.1 -0.1,5.1
struct rockbolt begin 2.5,5.0 end 2.5,2.5 seg 5 prop 1
struct prop 1 cs_scoh 1E20 cs_sstiff 5E8 density 4E3 e 5E9 radius 1 yield &
 1E20 ycomp 1E20
struct node 1  load 0.0,1000000.0 0.0
solve ratio 1.0E-5
save sp2.sav
```

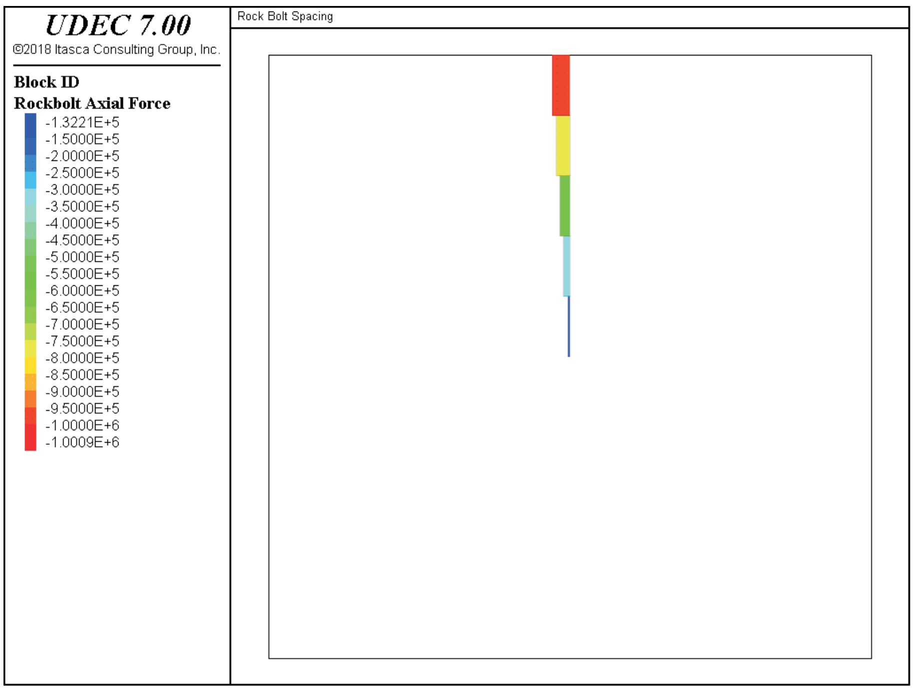


Figure 1.63 Actual axial forces in vertically loaded rockbolt at 2 m spacing (spacing given)

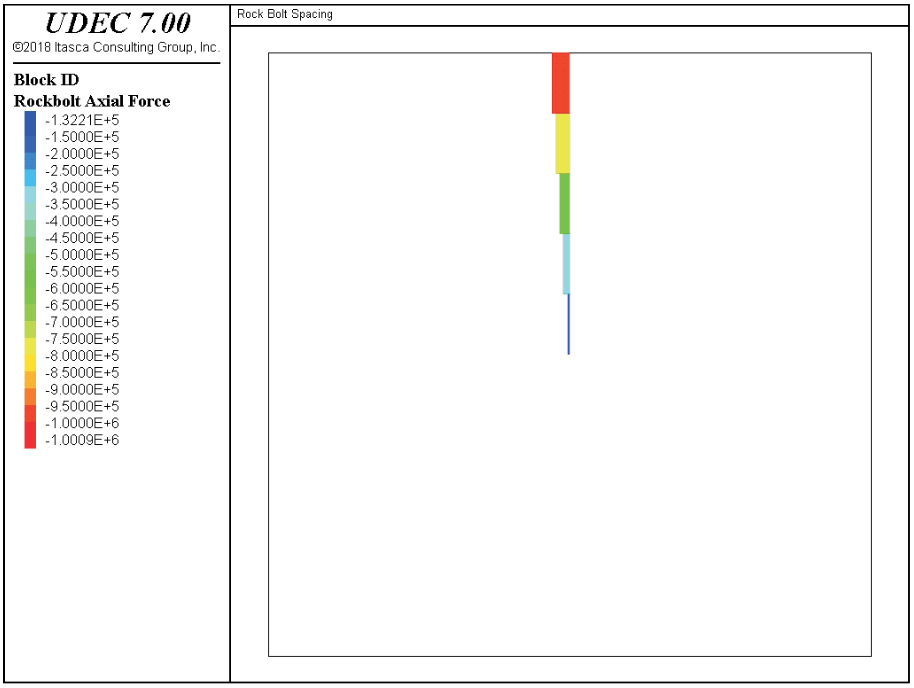


*Figure 1.64 Scaled axial forces in vertically loaded rockbolt at 2 m spacing (***spacing** *not given)*

1.5.2 Symmetry Conditions

Structural elements that lie on a line of symmetry should be assigned full properties for modulus and stiffness. Property values for cross-sectional area and yield strength should be reduced by 50% compared to the same property values for elements not on the symmetry line. Loads applied to structural elements on symmetry lines should also be reduced by 50% compared to the same loads applied away from the symmetry line.

1.5.3 Equilibrium Conditions

The user must decide when the model has reached an equilibrium state. Equilibrium for problems involving structural elements can be determined by all the usual criteria (e.g., histories and velocity fields). However, if beam elements or rockbolts are used, an additional equilibrium criterion is available. At equilibrium, beam or rockbolt element segments that share a common node will have equal and opposite moments. This can be confirmed with the **block struct beam list** or **block struct rockbolt list** command.

For certain types of structural-element problems (e.g., pull-tests on cables or rockbolts) a significant portion of the model region may develop nonzero components of velocity at the final state of solution. The default mechanical damping algorithm in *UDEC* can have difficulty damping this motion properly, because the mass-adjustment process requires velocity sign-changes (see Section 1.2.7 in

Theory and Background). An alternative form of damping is available for this type of problem. This damping, known as combined damping or "creep-type" damping, is described in the **Creep Material Models** volume. Combined damping is invoked for the *UDEC* model with the **block mechanical damping combined** command.

1.5.4 Sign Convention

Axial forces in all structural elements are positive in compression. Shear forces follow the opposite sign convention as that given for zone shear stresses (illustrated in Figure 2.46 in the **User's Guide**). Axial displacements for cable elements, rockbolt elements, beam elements and support members are positive for loading in compression. Axial displacements for reinforcement elements are positive for loading in tension. Normal forces at structural element interface contacts are positive in compression; normal displacements are positive for loading in tension. Moments at the end of beam and rockbolt elements are positive in the counterclockwise direction. Translational displacements at nodes are positive in the direction of the positive coordinate axes, and angular displacements are positive in the counterclockwise direction.

1.6 Selecting Input parameters For Different Types of Rockbolts

1.6.1 Introduction

Introduction There are three structural element types in *UDEC* that can be used to simulate rock and soil structural supports such as cables and bolts. These support types differ by increasing complexity. They are:

(1) local reinforcement;

(2) Cables; and

(3) Rockbolts.

The simplest form (local reinforcement) is implemented by creating a special contact at the points that a line crosses joints. This special contact simulates the normal and shear forces due to the presence of a bolt or dowel crossing the joint. The forces are not distributed back into the material as would be done by a real structural support. The next type (cables) is used to simulate a mechanically anchored or grouted support cable or rock bolt. This is implemented by applying elements and nodes back into the supported material. The connection between the nodes and the supported material is simulated by a material that can fail representing the failure of grout or the slippage of a mechanical anchor. The advantage of this support type over the local reinforcement is the way forces are distributed back into the supported material. This is a more realistic way to model the support. The cable logic is only axial and does not support shear resistance or bending. The idea is that the support comes only from the increased normal forces on a crossing joint and not by the dowel action of the bolt. The most complex type (rockbolts), also simulates a mechanically anchored or grouted cable or rockbolt. Despite the naming, the difference in the simulation of cables or rockbolts comes from the properties chosen. The rockbolt type is implemented by applying beam elements and nodes back into the supported material. The beam elements can provide shear and bending resistance. The connection between the nodes and the supported material can fail, representing the failure of grout or the slippage of a mechanical anchor and is sensitive to changes in the stress state that may occur post placement.

Figure 1.65 is a diagram (Stillborg 1994) representing the pullout loads on several types of rockbolts. The examples in this section are designed to closely match the results of these pull out tests. Because of the variability in application It is necessary to use pull out tests conducted under the actual local conditions when designing supports. The rockbolt logic in *UDEC* can approximate the behavior of all of these bolt types by variation in the properties specified in the command input. The main limitation in *UDEC* is that the failure mechanism is elastic-perfectly plastic with a strain limit. So, modeling bolt types that exhibit a curved post yield behavior will only be an approximate match. The most critical properties that result in the different bolt behavior types are:

(1) Bond shear stiffness (coupling-stiffness-shear);

(2) Bolt diameter;

(3) Shear strength of bond (coupling-stiffness-cohesion); and

(4) Rupture Strain.

The following sections will describe how the properties of the rockbolt can be tailored to simulate the actions of:

(1) Mechanically anchored rockbolt;

(2) Fully grouted rockbolt;

(3) Swellex rockbolt;

(4) Split set rockbolt; and

(5) Pre-tensioned bolts.

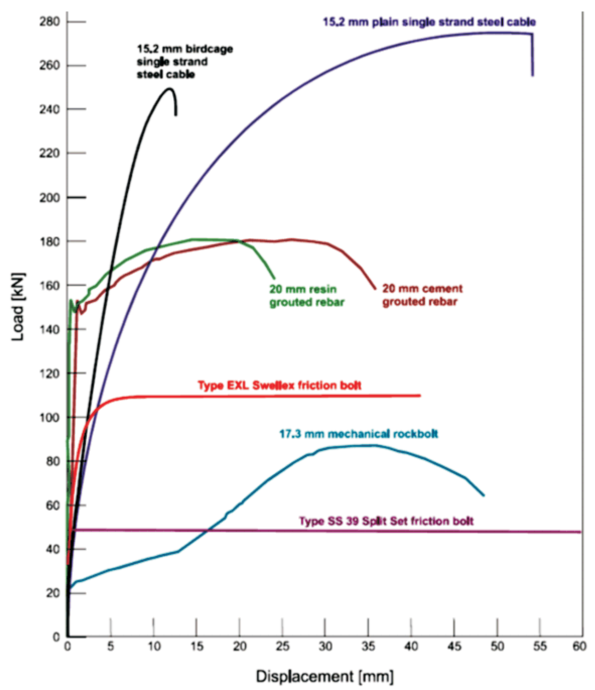


Figure 1.65 Results of pull tests by Stillborg (1994)

1.6.2 Grout Shear Strength

The input value for the coupling shear cohesion (CS_{scoh}) when modeling resin or cement grouted bolts may be taken directly from manufacturers data on grout shear strength or may be calculated from the grout UCS by:

$$CS_{\text{scoh}} = \frac{1}{2}\pi(D + 2t)(UCS)\ (Q_b) \tag{1.50}$$

where D = rod diameter;

t = grout annular thickness;

UCS = Grout unconfined compressive strength; and

Q_b = Quality of bond (0-1).

1.6.3 Rockbolt Segment Length

It is important that the number of segments specified for the rockbolt is sufficient to realistically distribute the axial loads in the supported material. A rule of thumb is to have 2-3 segments per development length. The development length is the length of the grouted section that is required to generate the yield strength of the steel. There are numerous formulae in the literature for this value. The formula for developement length (DL) used in UDEC (ignoring coupling shear friction which is a function of confinement) is:

$$DL = (a\sigma_b\ /(CS_{\text{scoh}}) \tag{1.51}$$

where a = bolt area;

σ_b = steel tensile yield stress; and

CS_{scoh} = coupling shear cohesion;

So, segment length is DL/3.

Another rule of thumb is to make the segment lengths approximately the same as the zone sizes so that each node will fall in a different zone.

1.6.4 Coupling Shear Stiffness

Load transfer from the rock to the steel bolt occurs through shear forces from the rock-grout and grout-steel bolt interfaces. The shear behavior of the bond is represented as a spring-slider system at the rockbolt nodes. The properties that describe the bond are its coupling shear stiffness (CS_{sstiff}) and its coupling shear cohesion and friction. The coupling shear stiffness determines the load applied to the rockbolt through the bond due to shear displacement between the rock and the steel bolt. The best way to detemine the coupling shear stifness is to match the elastic portion of the slope of a pull test. Alternatively, the shear coupling stiffness may be calculated using the equation developed by St. John and Van Dillen (1983) for a rigid member surrounded by an elastic annulus:

$$CS_{\text{sstiff}} = \frac{2\pi \; G}{\ln\left(1 + 2t/D\right)} \tag{1.52}$$

where G = grout shear modulus;
D = reinforcing diameter; and
t = annulus thickness.

1.6.5 Rupture Strain

The strain in UDEC is calculated over individual segments (not the entire rockbolt length). This may result in a calculated segment strain being larger than the average strain over the entire length. As a result, the input value of rupture strain is segment length dependent and may be much larger than what would be expected to get the rockbolt to break at the expected total extension.

1.6.6 Modeling a Fully Grouted Rockbolt

The fully grouted rockbolt consists of a steel rod cemented into the drill hole using a cementitious or resin grout. The typical bar diameters are 20 mm and 22 mm. Depending on the properties, this type of bolt may fail by tensile yielding in the steel or by grout shear failure. The information from the pullout tests from Stillborg as illustrated in Figure 1 indicate that the grout bond was stronger than the steel rod. The selection of properties are designed to accommodate this result. Table 1.10 shows the properties selected that will replicate the 20 mm resin grouted bolt pullout test in Figure 1.65. Note that the strength has been selected as the final rupture point on the curve (not the peak load). Figure 1.66 shows the axial load vs axial displacement from the *UDEC* model. Example 1.13 presents the data file for this model.

Table 1.10 Parameters for fully grouted rockbolt

Grout Young's modulus	0.05	GPa
Grout Poisson's ratio	0.25	
Grout shear modulus	0.02	GPa
Grout annular thickness	5	mm
Bolt diameter	20	mm
Tensile yield strength (steel)	517	MPa
Hole diameter	30	mm
UCS of resin	7.1	MPa
Steel Young's modulus	200	GPa
Tensile strain limit	0.25	
Perimeter	0.08	m
Area	3.14e−4	m^2
Coupling shear stiffness	3.1e8	N/m/m
Coupling shear cohesion	3.4e5	MPa
Tensile yield of bolt	162	KN
MOI	7.85e−9	m^4

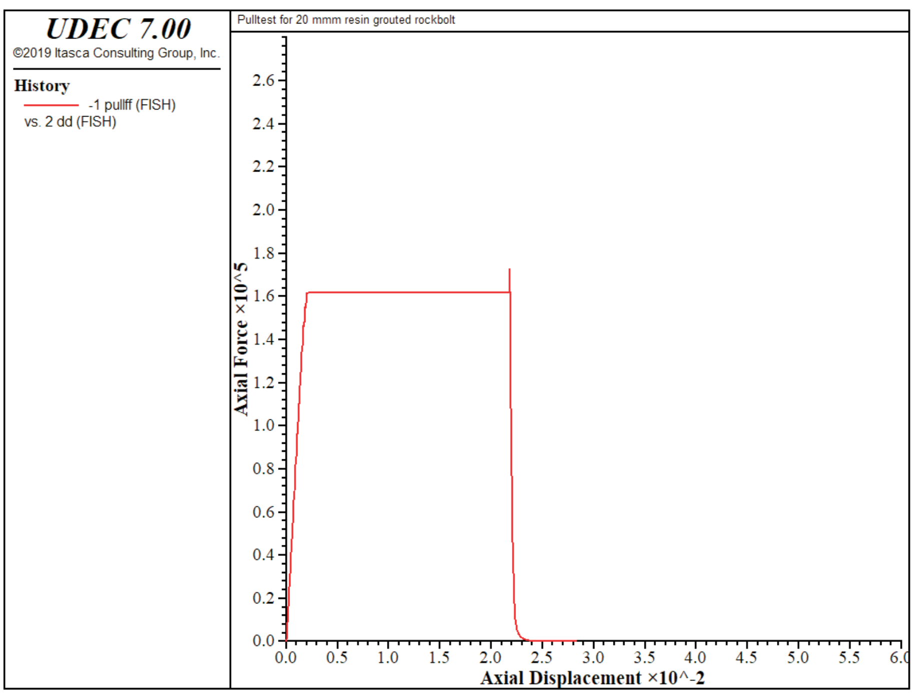


Figure 1.66 UDEC simulation of pullout test of 20 mm resin grouted rockbolt.

Example 1.13 Simulation of a pull-test for a fully grouted rockbolt

```
model new
;file: grouted.uddat
model title "Pulltest for 20 mmm resin grouted rockbolt"

fish define rockbolt_data
  y_mod_G = .05
  p_ratio_G = .25
  t = 5
  D = 20
  ult_s_t = 517
  _tfstrain = 0.25
  _cs_sfric = 45.0
  UCS_g = 7.2
  QB = 1
  y_mod_b = 200e9
  Peri = .08
;
  s_mod_G_ = y_mod_G / (2.0 * (1.0 + p_ratio_G))
  area_ = math.pi*(0.5*D*1e-3)^2
  cs_sstiff_ = ((2*math.pi*s_mod_G_) ...
               /(math.ln(1+(2*(t*1e-3)/(D*1e-3)))))*1e9
  cs_scoh_ = math.pi*(D*1e-3+2*t*1e-3)*(UCS_g*0.5*QB*1e6)
  St_yield_ =(ult_s_t*1e6)*area_
  sec_mom_a_ =0.25*math.pi*(0.5*(D*1e-3))^4
end
@rockbolt_data
;
;
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone generate  quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 ...
  range position-x -0.01 0.41 position-y 0.59 0.61
block structure rockbolt create begin 0.2 0.1 end 0.2 0.7 ...
  segment 12 material 1
block structure rockbolt property 1 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear @cs_scoh_ ...
  coupling-stiffness-shear @cs_sstiff_ perimeter @Peri ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric
```

```
block structure rockbolt property 1 density 0.001
block structure beam node 13 fix-y
block structure beam node 13 initial velocity-y 8e-2
;
;
; --- Fish functions ---
; pullff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_end_node
   _inode = block.structure.node.head
   _end_node = 0
   loop while _end_node = 0
      _yp = block.struct.bolt.node.pos.y(_inode)
; node 13
      if _yp > 0.69 then
         _end_node = _inode
      endif
      _inode = block.struct.bolt.node.next(_inode)
   end_loop
end
@_find_end_node
fish define pullff
; node 13
   nadd = _end_node
   dd = block.struct.bolt.node.disp.y(nadd)
   ffbou = 0.0
   loop jj (1,5)
      xx = (jj-1) * 0.1
      ig1 = block.gp.near(xx,0.6)
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boun.force.y(ibou1)
      ffbou = ffbou+fb1
   endloop
   pullff = ffbou
 Ult_load_bou=math.min(Ult_load_bou,ffbou)
end
;
fish history @pullff
fish history @dd
;
;

block cycle 30000
fish list @ult_load_bou
;
model save "resin_grouted.sav"
```

```
;
return
```

1.6.7 Modeling a Mechanically Anchored Rockbolt

The mechanically anchored rockbolt consists of a steel rod with a mechanical anchor. The typical bar diameters are 16 mm, 17 mm, 19 mm, 22 mm, and 25 mm. typically this type of bolt fails by slippage of the anchor. Inspecting the pullout test for the mechanically anchored rockbolt in Figure 1.65, it is apparent that the loads fall significantly short of the tensile yield strength of the steel rod (290 MPa). This indicates that the anchor is slipping. Table 1.11 shows the properties selected that will replicate the 17 mm mechanically anchored bolt pullout test in figure 1. The coupling spring shear stiffness property has been selected to give a realistic stiffness based on the shape of the pull-test curves. Unlike the fully grouted rockbolt, coupling properties vary along the length of the bolt. Usually the bolt is only bonded at the ends and coupling spring properties with zero strength values are used for the rest of the length. This requires the use of the comfontblock struct rockbolt change material-interface command to modify the coupling spring properties. For the purpose of the pull test example, only the anchor end coupling spring properties are non-zero. The coupling shear spring properties for the rest of the bolt are zero. It is important to note that the strength of the bond at the anchor points will be dependent on the rockbolt segment length. The coupling spring shear cohesion of the anchorage has been selected to reproduce the final rupture point on the curve (not the peak load). Figure 1.67 shows the axial load vs axial displacement from the *UDEC* model. Example 1.14 presents the data file for this model.

Table 1.11 Parameters for mechanically anchored rockbolt

Bolt diameter	17.3	mm
Ult. tensile strength steel	528	MPa
Hole diameter	20	mm
Steel Young's modulus	200	GPa
Tensile strain limit	0.25	
Perimeter	0.08	m
Area	$2.35e{-4}$	m^2
Coupling shear stiffness	2.2e9	N/m/m
Coupling shear cohesion	2.6e6	MPa
Tensile yield of bolt	124	KN
MOI	$4.4e{-9}$	m^4

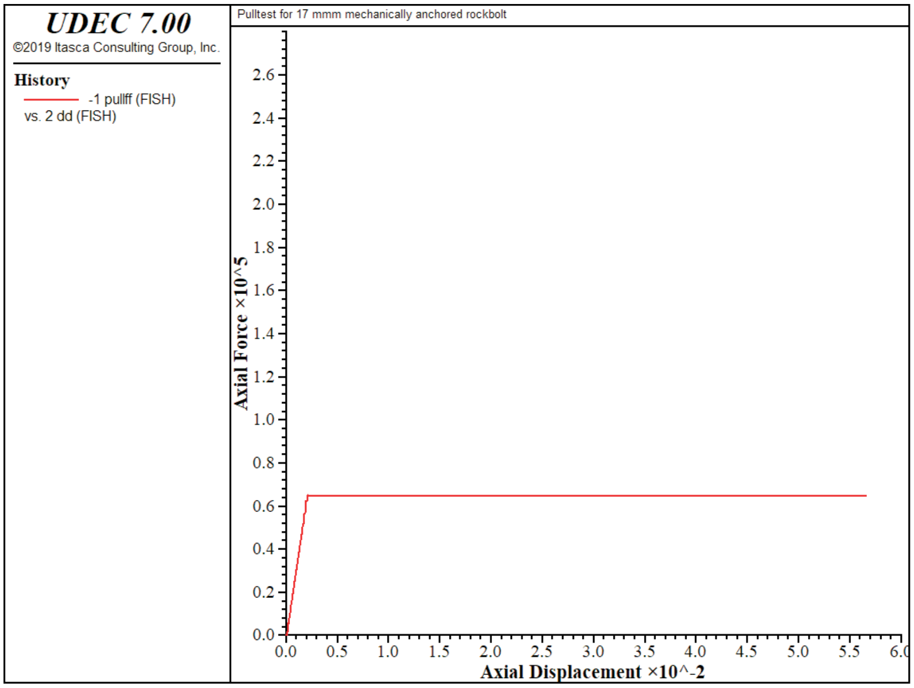


Figure 1.67 UDEC simulation of pull test of a 17 mm mechanically anchored rockbolt.

Example 1.14 Simulation of a pull-test for a mechanically anchored rockbolt

```
model new
;file: mechanical.uddat
model title "Pulltest for 17 mmm mechanically anchored rockbolt"

fish define rockbolt_data
  D = 17.3
  ult_s_t = 528
  _tfstrain = 0.25
  _cs_sfric = 45.0
  QB = 1
  y_mod_b = 200e9
  Peri = .08
  cs_sstiff_ = 2.2e9
  cs_scoh_ = 2.6e6

  area_ = math.pi*(0.5*D*1e-3)^2
  St_yield_ =(ult_s_t*1e6)*area_
  sec_mom_a_ =0.25*math.pi*(0.5*(D*1e-3))^4
end
@rockbolt_data
```

```
;
;
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone generate  quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 ...
  range position-x -0.01 0.41 position-y 0.59 0.61
block structure rockbolt create begin 0.2 0.1 end 0.2 0.7 ...
  segment 12 material 1
block structure rockbolt property 1 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear 0.0 ...
  coupling-stiffness-shear 0.0 perimeter @Peri ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric

;anchor properties
block structure rockbolt change mat-int 2 range pos-y 0.0 .12
block structure rockbolt property 2 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear @cs_scoh_ ...
  coupling-stiffness-shear @cs_sstiff_ perimeter @Peri ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric
block structure rockbolt property 1 density 0.001
block structure beam node 13 fix-y
block structure beam node 13 initial velocity-y 8e-2
;
;
; --- Fish functions ---
; pullff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_end_node
   _inode = block.structure.node.head
   _end_node = 0
   loop while _end_node = 0
      _yp = block.struct.bolt.node.pos.y(_inode)
; node 13
      if _yp > 0.69 then
         _end_node = _inode
      endif
      _inode = block.struct.bolt.node.next(_inode)
```

```
    end_loop
end
@_find_end_node
fish define pullff
; node 13
    nadd = _end_node
    dd = block.struct.bolt.node.disp.y(nadd)
    ffbou = 0.0
    loop jj (1,5)
       xx = (jj-1) * 0.1
       ig1 = block.gp.near(xx,0.6)
       ibou1 = block.gp.boundary.corner(ig1)
       fb1 = block.boun.force.y(ibou1)
       ffbou = ffbou+fb1
    endloop
    pullff = ffbou
 Ult_load_bou=math.min(Ult_load_bou,ffbou)
end
;
fish history @pullff
fish history @dd
;
;

block cycle 60000
fish list @ult_load_bou
;
model save "mechanical.sav"
;
return
```

1.6.8 Modeling a Split Set Rockbolt

The split set is a split cylinder of spring steel that is pressed into the drill hole. The split set derives its pull out strength through frictional resistance along the drill hole perimeter. Typical split set diameters are 33 mm, 39 mm, and 46 mm. Usually this type of bolt fails by slippage. Table 1.12 shows the properties selected that will replicate the 39 mm split set bolt pullout test in Figure 1.65. The coupling spring shear stiffness property is selected to give a realistic stiffness based on the shape of the pull-test curves. The coupling shear cohesion property of the anchorage has been selected to represent the pull out force shown in Figure 1.65. Figure 1.68 shows the axial load vs axial displacement from the *UDEC* model. Example 1.15 presents the data file for this model.

Table 1.12 Parameters for split set rockbolt

tube annular thickness	2.35	mm
Tube diameter	39	mm
Hole diameter	39	mm
Steel Young's modulus	200	GPa
Tensile strain limit	0.25	
Perimeter	0.08	m
Area	3.14e−4	m^2
Coupling shear stiffness	5.0e8	N/m/m
Coupling shear cohesion	3.3e5	MPa
Tensile yield of bolt	110	KN
MOI	7.85e−9	m^4

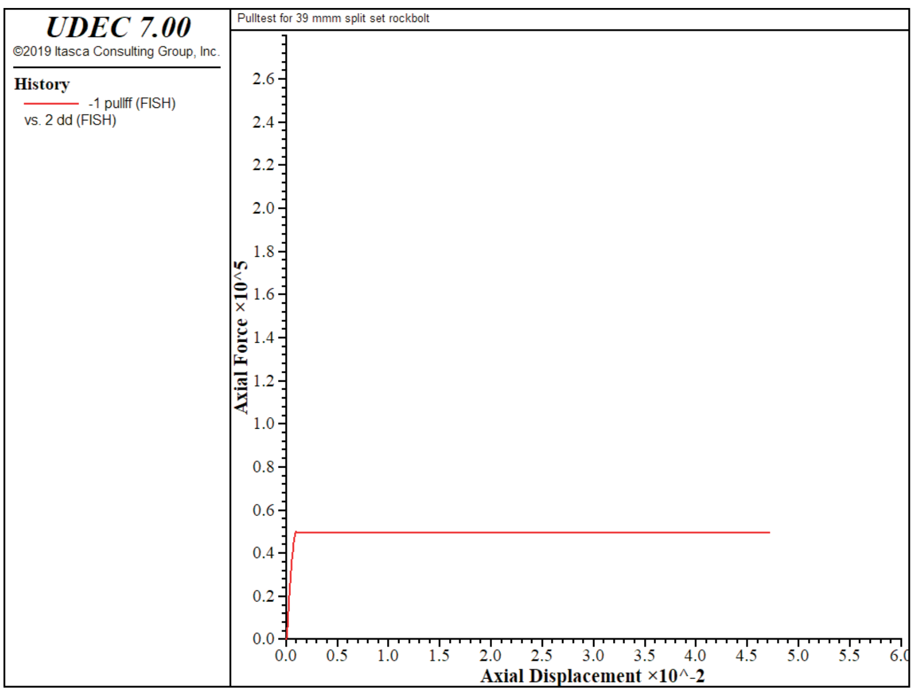


Figure 1.68 UDEC simulation of pull test of a 39 mm split set rockbolt.

Example 1.15 Simulation of a pull-test for a split set rockbolt

```
model new
model title "Pulltest for 39 mmm split set rockbolt"

fish define rockbolt_data
```

```
  t = 2.35
  D = 39
  ult_s_t = 517
  _tfstrain = 0.25
  _cs_sfric = 45.0
  y_mod_b = 200e9
  Peri = .08
  area_ = 1.4e-4
;
  cs_sstiff_ = 5.0e8
  cs_scoh_   = 1.0e5
;
  St_yield_ =(ult_s_t*1e6)*area_
  sec_mom_a_ = math.pi*(0.5*(D*1e-3))^3*(t*1e-3)
end
@rockbolt_data
;;
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone generate  quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 ...
  range position-x -0.01 0.41 position-y 0.59 0.61
block structure rockbolt create begin 0.2 0.1 end 0.2 0.7 ...
  segment 12 material 1
block structure rockbolt property 1 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear @cs_scoh_ ...
  coupling-stiffness-shear @cs_sstiff_ perimeter @Peri  ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric
block structure rockbolt property 1 density 0.001
block structure beam node 13 fix-y
block structure beam node 13 initial velocity-y 8e-2
;
;
; --- Fish functions ---
; pullff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_end_node
   _inode = block.structure.node.head
   _end_node = 0
   loop while _end_node = 0
```

```
      _yp = block.struct.bolt.node.pos.y(_inode)
; node 13
      if _yp > 0.69 then
         _end_node = _inode
      endif
      _inode = block.struct.bolt.node.next(_inode)
   end_loop
end
@_find_end_node
fish define pullff
; node 13
   nadd = _end_node
   dd = block.struct.bolt.node.disp.y(nadd)
   ffbou = 0.0
   loop jj (1,5)
      xx = (jj-1) * 0.1
      ig1 = block.gp.near(xx,0.6)
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boun.force.y(ibou1)
      ffbou = ffbou+fb1
   endloop
   pullff = ffbou
 Ult_load_bou=math.min(Ult_load_bou,ffbou)
end
;
fish history @pullff
fish history @dd
;
;

block cycle 50000
fish list @ult_load_bou
;
model save "split_set.sav"
;
return
```

1.6.9 Modeling a Swellex Rockbolt

The swellex rockbolt is a collapsed cylinder that is inserted into the drill hole and inflated with high pressure water. The swellex rockbolt derives its pull out strength through frictional resistance along the drill hole perimeter. Typical swellex diameters are 26 mm, 28 mm, 36 mm, 36.5 and 37 mm. These will expand into hole diameters ranging from 32 to 52 mm. Typically this type of bolt fails by slippage. In this particular pullout test the yield of the rockbolt matches the slippage strength. Table 1.13 shows the properties selected that will replicate the 26 mm swellex bolt pullout test in Figure 1.65. The coupling shear stiffness property was selected to give a realistic stiffness based on the elastic portion of the pull out test. The coupling shear cohesion property has been selected to represent the pull out value shown in Figure 1.65. Figure 1.69 shows the axial load vs axial displacement from the UDEC model. Example 1.16 presents the data file for this model.

Table 1.13 Parameters for swellex rockbolt

Tube annular thickness	2	mm
Original Tube diameter	41	mm
Compressed tube diameter	26	mm
Hole diameter	36	mm
Steel Young's modulus	200	GPa
Tensile strain limit	0.20	
Perimeter	0.08	m
Area	1.256e−4	m^2
Coupling shear stiffness	5.0e7	N/m/m
Coupling shear cohesion	2.3e5	MPa
Tensile yield of bolt	110	KN
MOI	7.85e−9	m^4

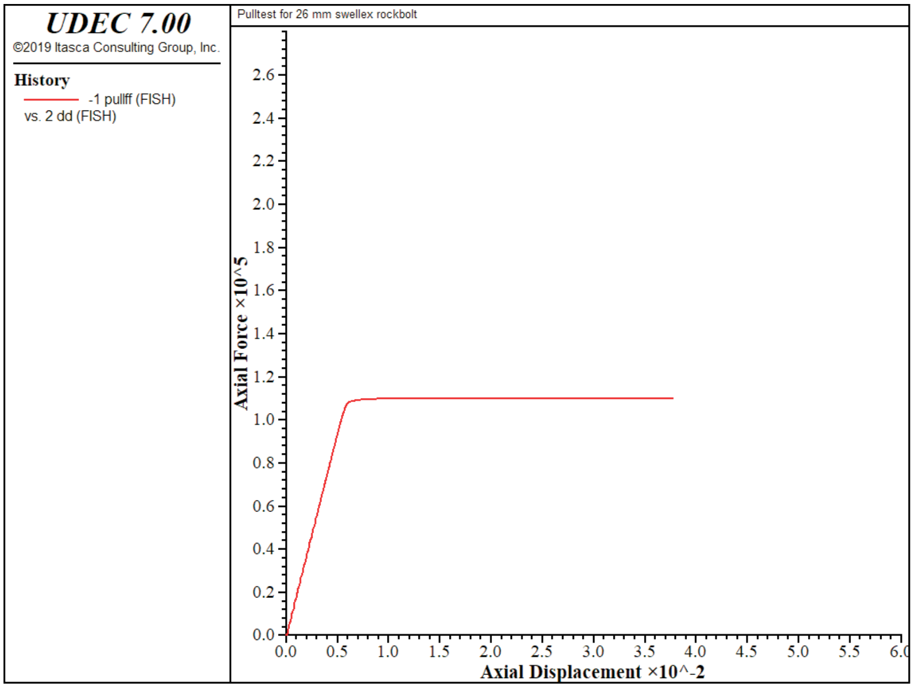


Figure 1.69 UDEC simulation of pull test of a 26 mm swellex rockbolt.

Example 1.16 Simulation of a pull-test for a swellex rockbolt

```
model new
;file: swellex.uddat
model title "Pulltest for 26 mm swellex rockbolt"

fish define rockbolt_data
  t = 2
  D = 36
  _tfstrain = 0.25
  _cs_sfric = 45.0
  y_mod_b = 200e9
  Peri = .08
  area_ = 1.256e-4
;
  cs_sstiff_ = 5.0e7
  cs_scoh_ = 2.3e5
  St_yield_ = 110e3
  sec_mom_a_ = math.pi*(0.5*(D*1e-3))^3*(t*1e-3)
end
@rockbolt_data
;
;
```

```
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone generate  quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 ...
  range position-x -0.01 0.41 position-y 0.59 0.61
block structure rockbolt create begin 0.2 0.1 end 0.2 0.7 ...
  segment 12 material 1
block structure rockbolt property 1 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear @cs_scoh_ ...
  coupling-stiffness-shear @cs_sstiff_ perimeter @Peri
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric
block structure rockbolt property 1 density 0.001
block structure beam node 13 fix-y
block structure beam node 13 initial velocity-y 8e-2
;
;
; --- Fish functions ---
; pullff : Pull force in bolt
; dd : Displacement of rockbolt end
fish define _find_end_node
   _inode = block.structure.node.head
   _end_node = 0
   loop while _end_node = 0
      _yp = block.struct.bolt.node.pos.y(_inode)
; node 13
      if _yp > 0.69 then
         _end_node = _inode
      endif
      _inode = block.struct.bolt.node.next(_inode)
   end_loop
end
@_find_end_node
fish define pullff
; node 13
   nadd = _end_node
   dd = block.struct.bolt.node.disp.y(nadd)
   ffbou = 0.0
   loop jj (1,5)
      xx = (jj-1) * 0.1
      ig1 = block.gp.near(xx,0.6)
```

```
      ibou1 = block.gp.boundary.corner(ig1)
      fb1 = block.boun.force.y(ibou1)
      ffbou = ffbou+fb1
   endloop
   pullff = ffbou
 Ult_load_bou=math.min(Ult_load_bou,ffbou)
end
;
fish history @pullff
fish history @dd
;
;

block cycle 40000
fish list @ult_load_bou
;
model save "swellex.sav"
;
return
```

1.6.10 Modeling Rockbolt Pretensioning

Originally *UDEC* had an input parameter for cables and rockbolts that allowed an initial tension load to be applied. This was done at the time of the bolt placement. Unfortunately, due to the fact that cables and rockbolts are much stiffer than most of the materials that they support, the pretension force would quickly dissipate. It became clear that pretensioning was a process rather than an input parameter. The ability to facilitate this process has been added to *UDEC* . The steps for pretensioning are (note this assumes that the properties specified for the ends are strong enough to support the pretension load) :

(1) Install cable or rockbolt with bonding only at the ends.

(2) Use block struct cable fix-tension 10000 or block struct rockbolt fix-tension 10000 to set a tensile force of 10000 in the structure.

(3) Cycle the model to equilibrium.

(4) Add bonding to the rest of the cable or rockbolt if it is to be fully grouted.

(5) Release the bolt to take loads by block struct cable free-tension or block struct rockbolt free-tension command.

Figure 1.70 shows the distribution of displacements due to the rockbolt pretensioning. Example 1.17 presents the data file for this model.

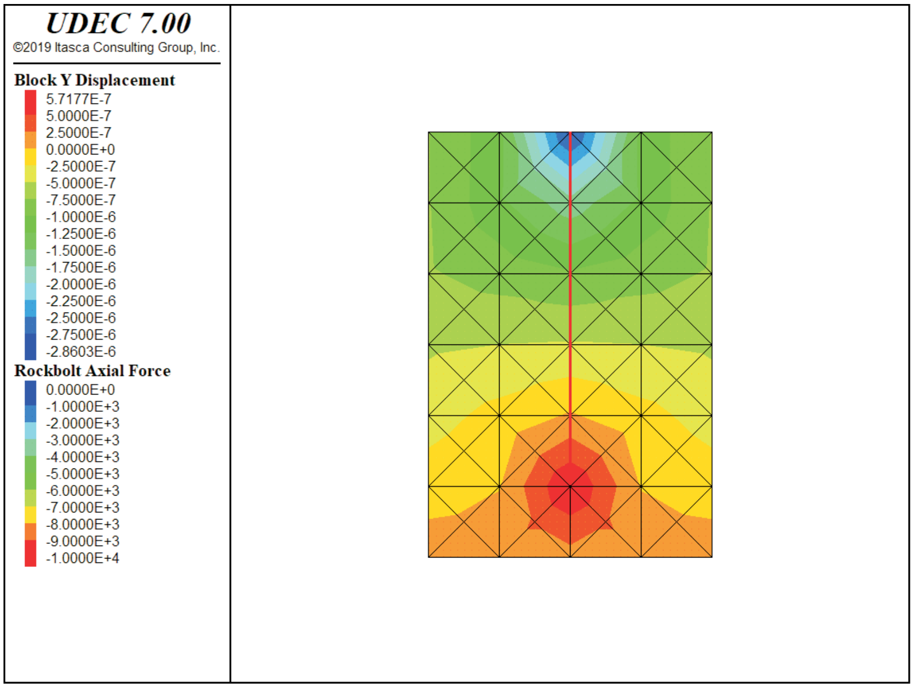


Figure 1.70 Rock displacements and rockbolt forces resulting from pretensioning a rockbolt.

Example 1.17 Simulation of a pull-test for a swellex rockbolt

```
model new
;file: pretension.uddat
model title "Pretensioning a rockbolt"
fish define rockbolt_data
  D = 17.3
  ult_s_t = 528
  _tfstrain = 0.25
  _cs_sfric = 45.0
  QB = 1
  y_mod_b = 200e9
  Peri = .08
  cs_sstiff_ = 2.2e9
  cs_scoh_ = 2.6e6

  area_ = math.pi*(0.5*D*1e-3)^2
  St_yield_ =(ult_s_t*1e6)*area_
  sec_mom_a_ =0.25*math.pi*(0.5*(D*1e-3))^4
end
@rockbolt_data
;
```

```
;
block tolerance corner-round-length 0.01
block tolerance minimum-edge-length 0.02
block create polygon 0 0 0 0.6 0.4 0.6 0.4 0
block zone generate  quad 0.11
block zone group 'block'
block zone cmodel assign elastic density 2E3 bulk 5E9 shear 3E9 ...
  range group 'block'
block gridpoint apply velocity-y 0 ...
  range position-x -0.01 0.41 position-y -.01 .01
block structure rockbolt create begin 0.2 0.1 end 0.2 0.6 ...
  segment 12 material 1
block structure rockbolt property 1 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear 0.0 ...
  coupling-stiffness-shear 0.0 perimeter @Peri ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric

;attach rockbolt ends only
block structure rockbolt change mat-int 2 range pos-y 0.0 .12
block structure rockbolt change mat-int 2 range pos-y .59 .61
;
block structure rockbolt property 2 young @y_mod_b ...
  cross-sectional-area @area_ coupling-cohesion-shear @cs_scoh_ ...
  coupling-stiffness-shear @cs_sstiff_ perimeter @Peri ...
  yield-tension @St_yield_ yield-compression @St_yield_ ...
  moi @sec_mom_a_ tension-failure-strain @_tfstrain ...
  coupling-friction-shear @_cs_sfric
block structure rockbolt property 1 density 0.001
block structure rockbolt fix-tension 10000
;
block solve
;
; grout rest of rockbolt
block structure rockbolt change mat-int 2
;
block solve

;
model save "pretension.sav"
;
return
```

1.7 References

Azuar, J. J., et al. "Le Renforcement des Massifs Rocheux par Armatures Passives (Rock Mass Reinforcement by Passive Rebars)," in ***Proceedings of the 4th ISRM Congress (Montreux, September 1979)***, Vol. 1, pp. 23-30. Rotterdam: A. A. Balkema and The Swiss Society for Soil and Rock Mechanics (1979).

Bjurstrom, S. "Shear Strength on Hard Rock Joints Reinforced by Grouted Untensioned Bolts," in *Proceedings of the 3rd International Congress on Rock Mechanics*, Vol. II, Part B, pp. 1194-1199. Washington, D.C.: National Academy of Sciences (1974).

Brierley, G. "The Performance during Construction of the Liner of a Large, Shallow Underground Opening in Rock." Ph.D. Thesis, University of Illinois at Urbana-Champaign (1975).

CSIR MiningTEK. Personal communication (1993).

Daemen, J. J. K. "Tunnel Support Loading Caused by Rock Failure." Ph.D. Thesis, University of Minnesota; also available as U.S. Army Corps of Engineers Report MRD-3-75 (1975).

Dight, P. M. "Improvements to the Stability of Rock Walls in Open Pit Mines." Ph.D. Thesis, Monash University (1982).

Dixon, J. D. "Analysis of Tunnel Support Structure with Consideration of Support-Rock Interaction," U.S. Dept. of Interior, Bureau of Mines Investigation, Report RI7526 (June 1971).

Donovan, K., W. E. Pariseau and M. Cepak. "Finite Element Approach to Cable Bolting in Steeply Dipping VCR Stopes," in ***Geomechanics Applications in Underground Hardrock Mining***, pp. 65-90. New York: Society of Mining Engineers (1984).

Fuller, P. G., and R. H. T. Cox. "Rock Reinforcement Design Based on Control of Joint Displacement – A New Concept," in ***Proceedings of the 3rd Australian Tunnelling Conference (Sydney, Australia, 1978)***, pp. 28-35. Sydney: Inst. of Engrs., Australia (1978).

Gerdeen, J. C., et al. "Design Criteria for Roof Bolting Plans Using Fully Resin-Grouted Nontensioned Bolts to Reinforce Bedded Mine Roof," U.S. Bureau of Mines, OFR 46(4)-80 (1977).

Haas, C. J. "Shear Resistance of Rock Bolts," *Trans. Soc. Min. Eng. AIME*, ***260***(1), 32-41 (1976).

Hyett, A. J., W. F. Bawden and R. D. Reichert. "The Effect of Rock Mass Confinement on the Bond Strength of Fully Grouted Cable Bolts," *Int. J. Rock Mech. Min. Sci. & Geomech. Abstr.*, ***29***(5), 503-524 (1992).

Littlejohn, G. S., and D. A. Bruce. "Rock Anchors – State of the Art. Part I: Design," *Ground Engineering*, ***8***(3), 25-32 (1975).

Lorig, L. J. "A Hybrid Computational Model for Excavation and Support Design in Jointed Media." Ph.D. Thesis, University of Minnesota (1984).

Lorig, L. J. "A Simple Numerical Representation of Fully Bonded Passive Rock Reinforcement for Hard Rocks," *Computers and Geotechnics*, ***1***, 79-97 (1985).

Monsees, J. E. "Station Design for the Washington Metro System," in ***Proceedings of the Engineering Foundation Conference – Shotcrete Support***, ACI Publication SP-54 (1977).

Paul, S. L., et al. "Design Recommendations for Concrete Tunnel Linings," University of Illinois, DOT Report No. DOT-TSC-UMTA-83-16 (1983).

Pells, P. J. N. "The Behaviour of Fully Bonded Rock Bolts," in ***Proceedings of the 3rd International Congress on Rock Mechanics***, Vol. 2, pp. 1212-1217 (1974).

St. John, C. M., and D. E. Van Dillen. "Rockbolts: A New Numerical Representation and Its Application in Tunnel Design," in ***Rock Mechanics – Theory - Experiment - Practice (Proceedings of the 24th U.S. Symposium on Rock Mechanics, Texas A&M University, June 1983)***, pp. 13-26. New York: Association of Engineering Geologists (1983).

Stillborg, B. "Professional Users Handbook for Rock Bolting," *Trans Tech Publications*, Germany, (1994)