NATIONAL TECHNICAL UNIVERSITY OF ATHENS

School of Civil Engineering – Geotechnical Department

Computational Methods in the Analysis of Underground Structures

Spring Term 2023 – 24

Lecture Series in Postgraduate Programs:

1. Analysis and Design of Structures (DSAK)
2. Design and Construction of Underground Structures (SKYE)

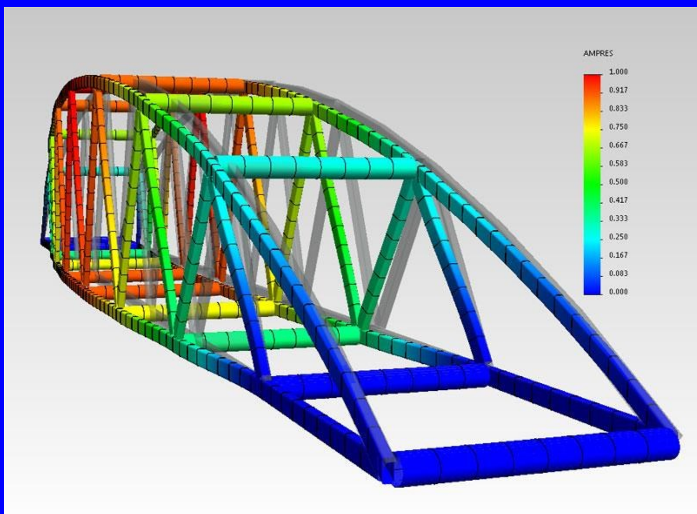
Instructor: Michael Kavvadas, Emer. Professor NTUA

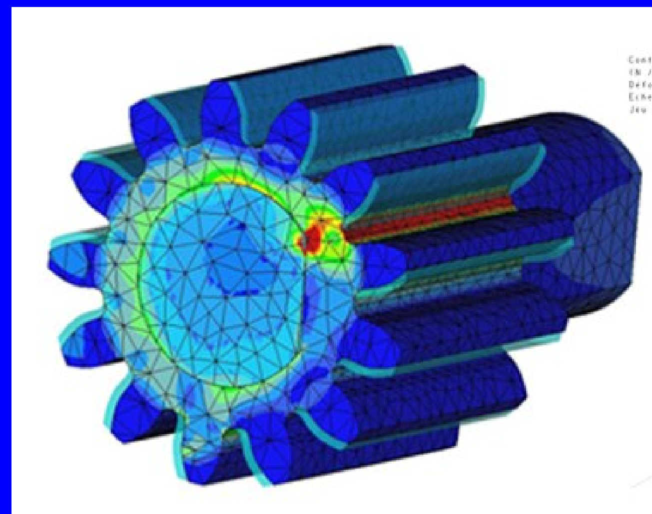
LECTURE 7: Numerical modelling of tunnel excavation and support using finite elements

26.03.2024

Numerical Methods in the Analysis of Structures
The Finite Element method

Finite Element Method (FEM): Numerical solution of differential equations. Since all problems in Continuum Mechanics (including geotechnical problems) are expressed via differential equations, they can be solved by FEM.

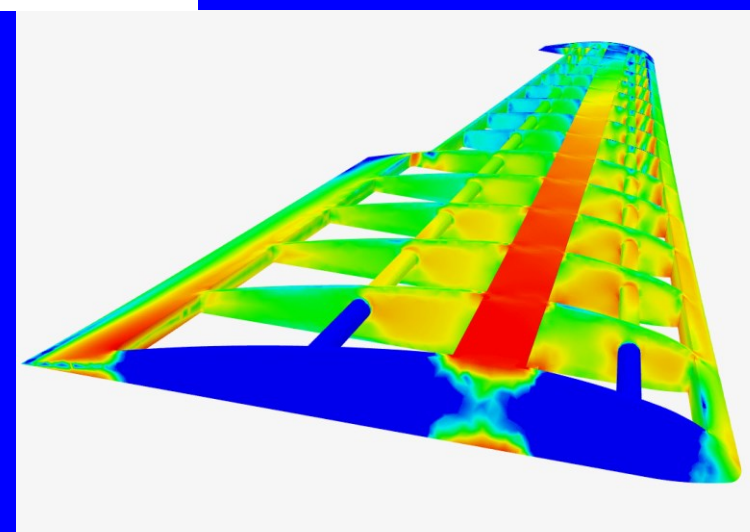




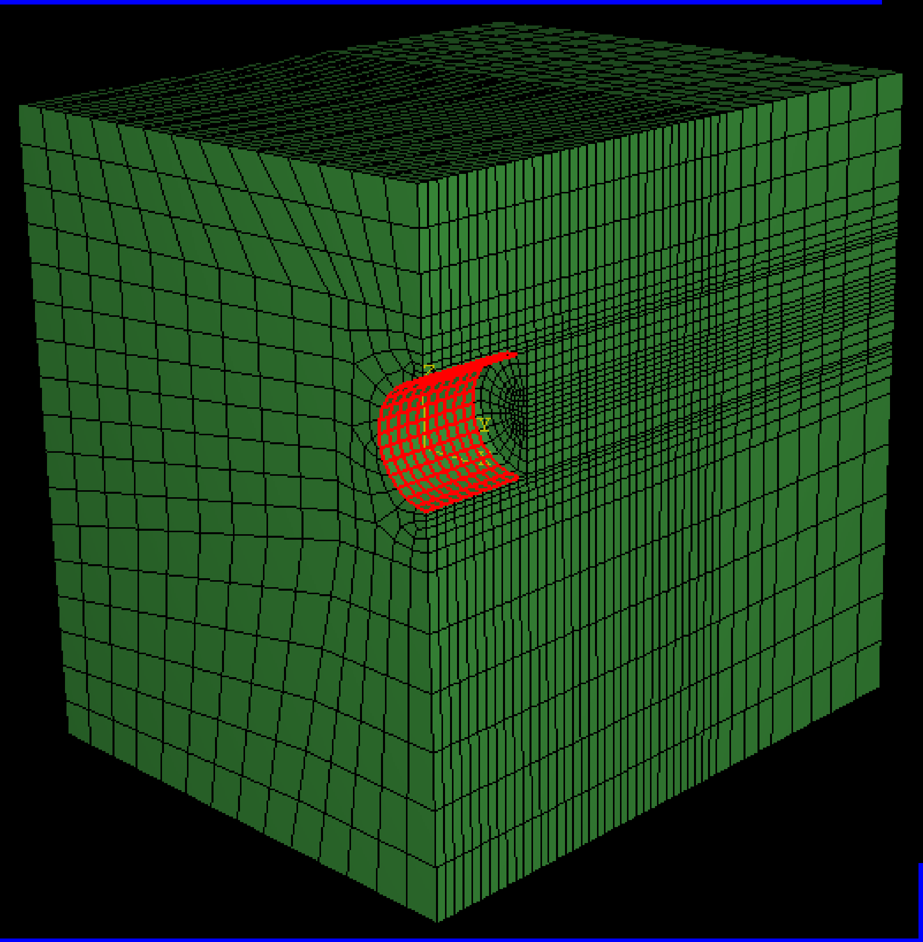
Applications of FEM

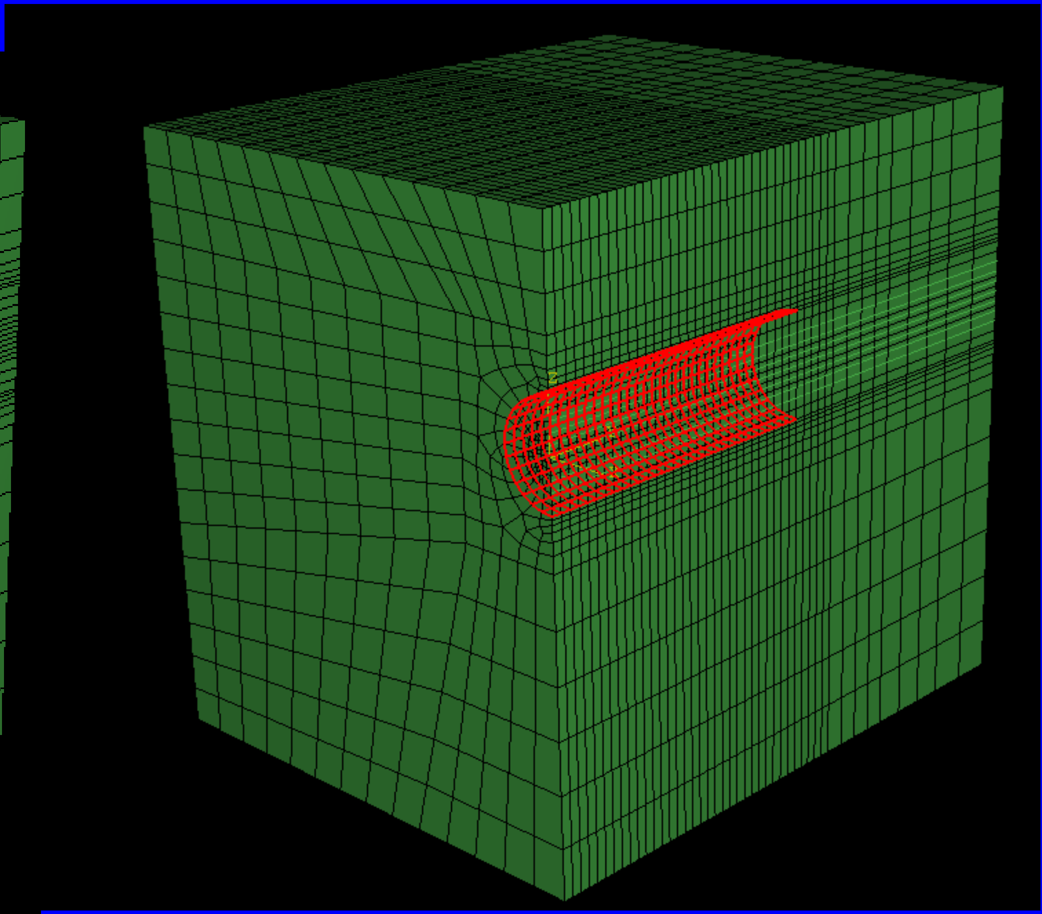
FEM initially developed for aeronautical applications, and later was extended to other mechanical and civil engineering problems.

The FEM is based on a discretization of the structure in “finite elements” which interact (via contacts). Each FE has a number of unknown variables (usually displacements) which are calculated in the solution.



The Finite Element Method in Tunnelling



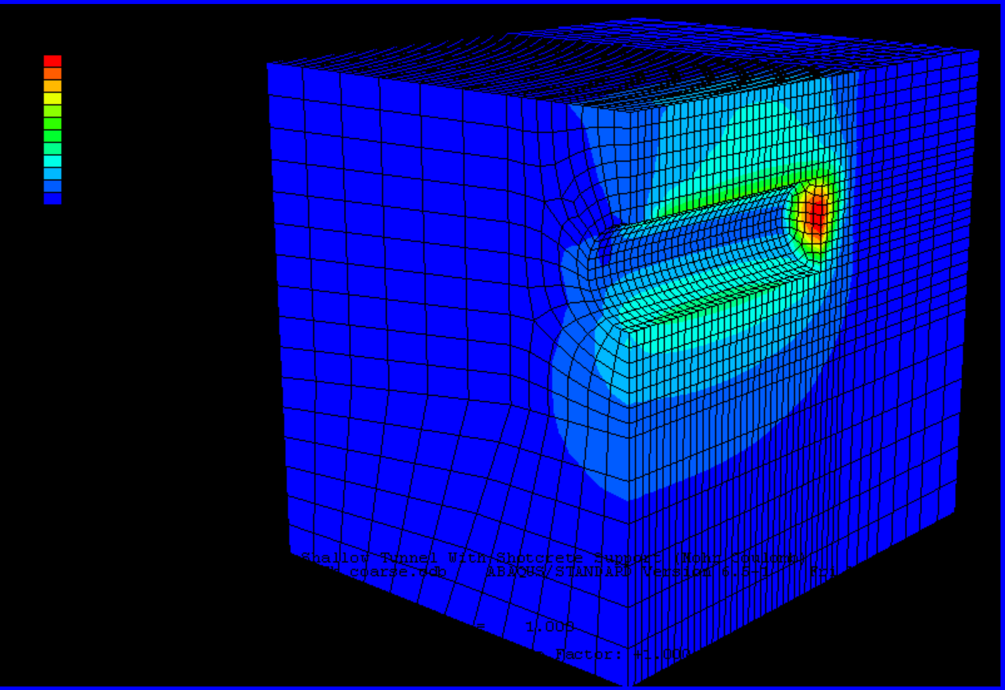


3D analysis of tunnel excavation using the FEM

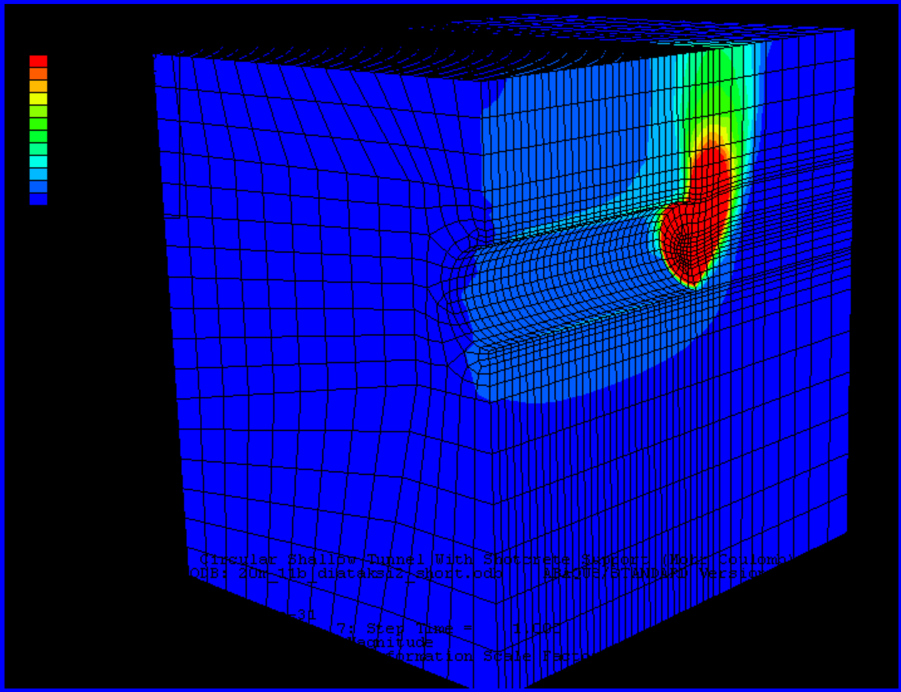
The Finite Element Method in Tunnelling

3D analysis of tunnel excavation using the FEM

Analysis results: Total displacement contours

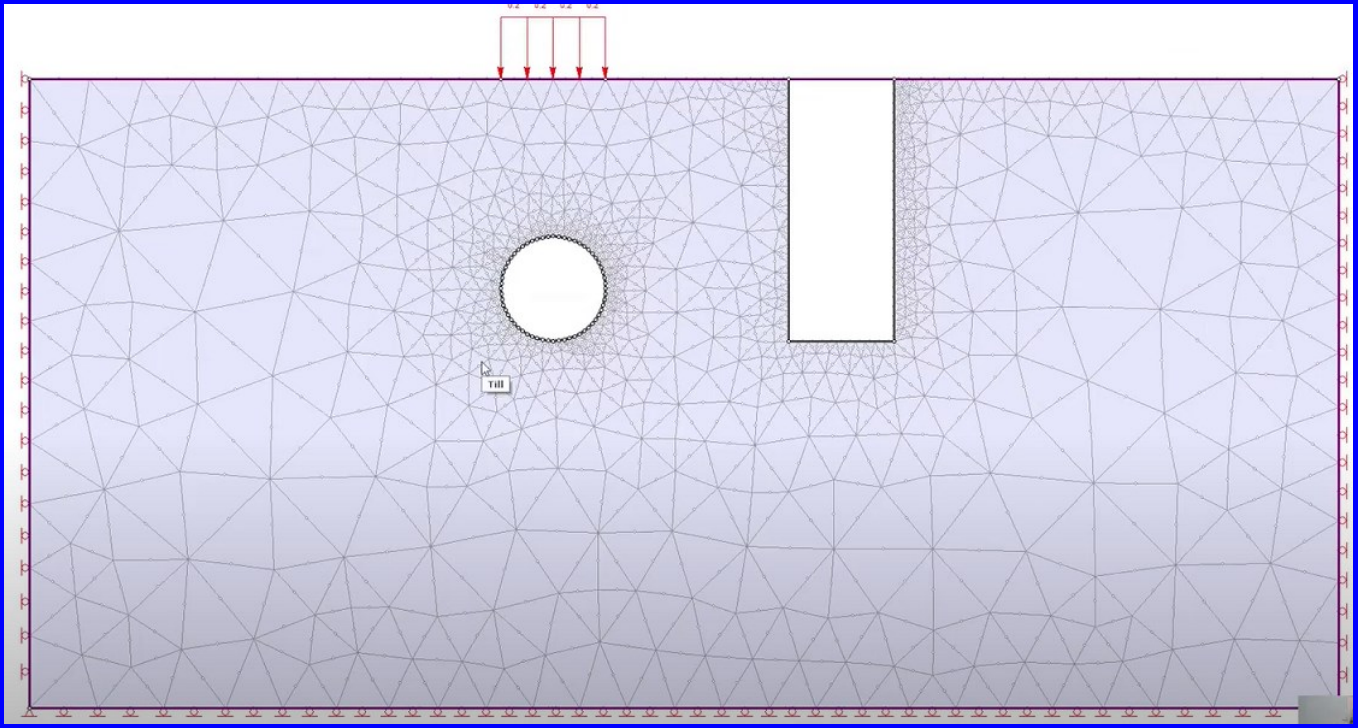


In relatively good ground



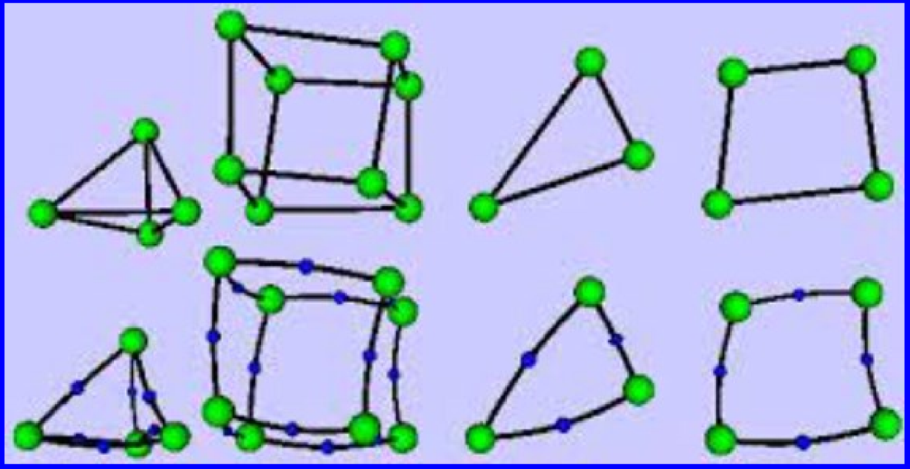
In relatively poor ground

The Finite Element Method

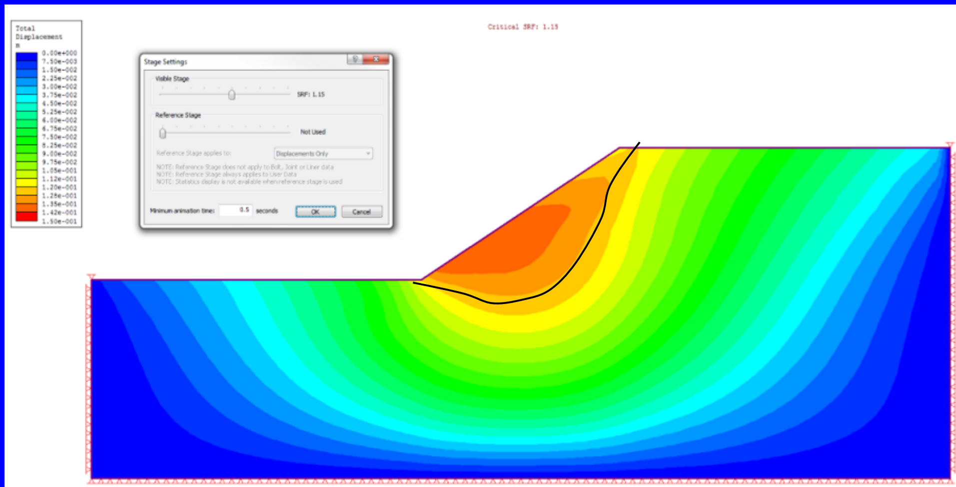


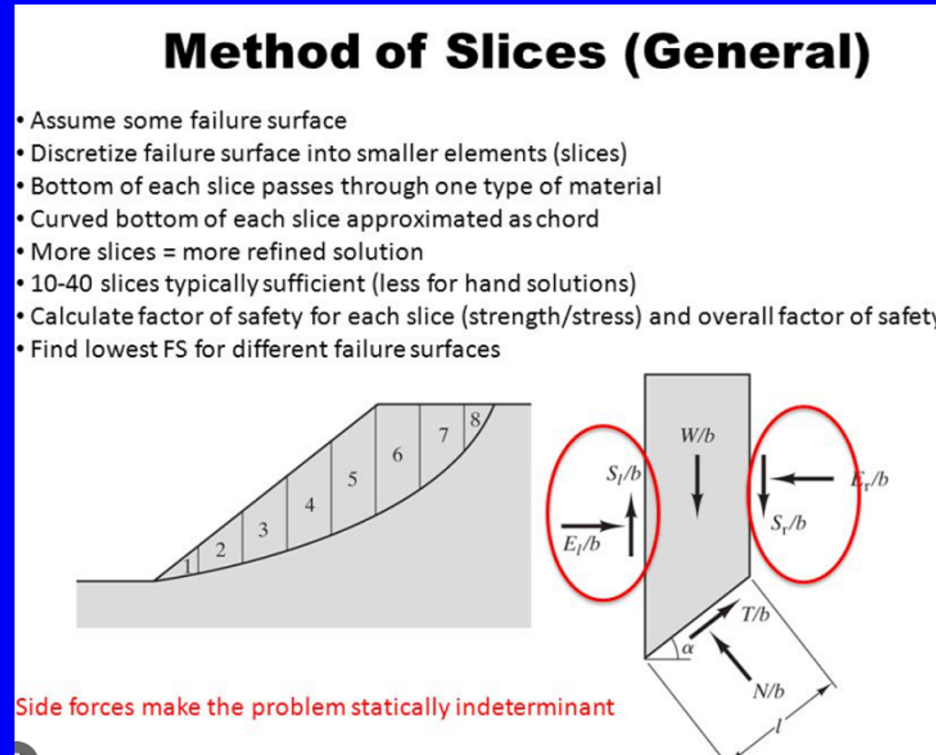
2D analysis of tunnel next to a deep excavation (software RS2).
Discretisation with triangular finite elements

Finite elements in 2D and 3D problems.
Each FE has the nodal displacements as variables.



Application of the FEM in slope stability analysis





Analysis with the FEM: Ground properties are gradually reduced, and ground displacements are calculated. Abrupt increase of displacements indicates slope stability failure → corresponding ground properties are limiting. Comparison with actual ground properties provides the margin of safety (Safety Factor SF).

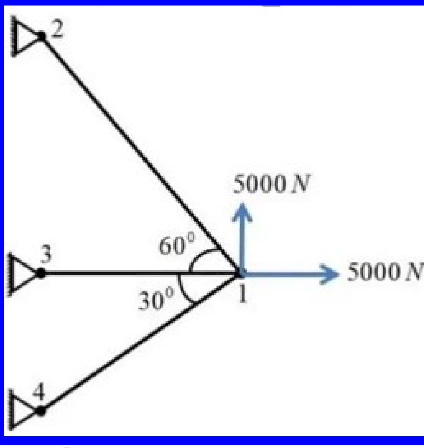
$c_{lim} = c / SF$, $\tan\varphi_{lim} = \tan\varphi / SF$

Limit equilibrium methods (e.g. method of slices): Many potential sliding surfaces are checked, and the surface providing minimum safety is the critical.

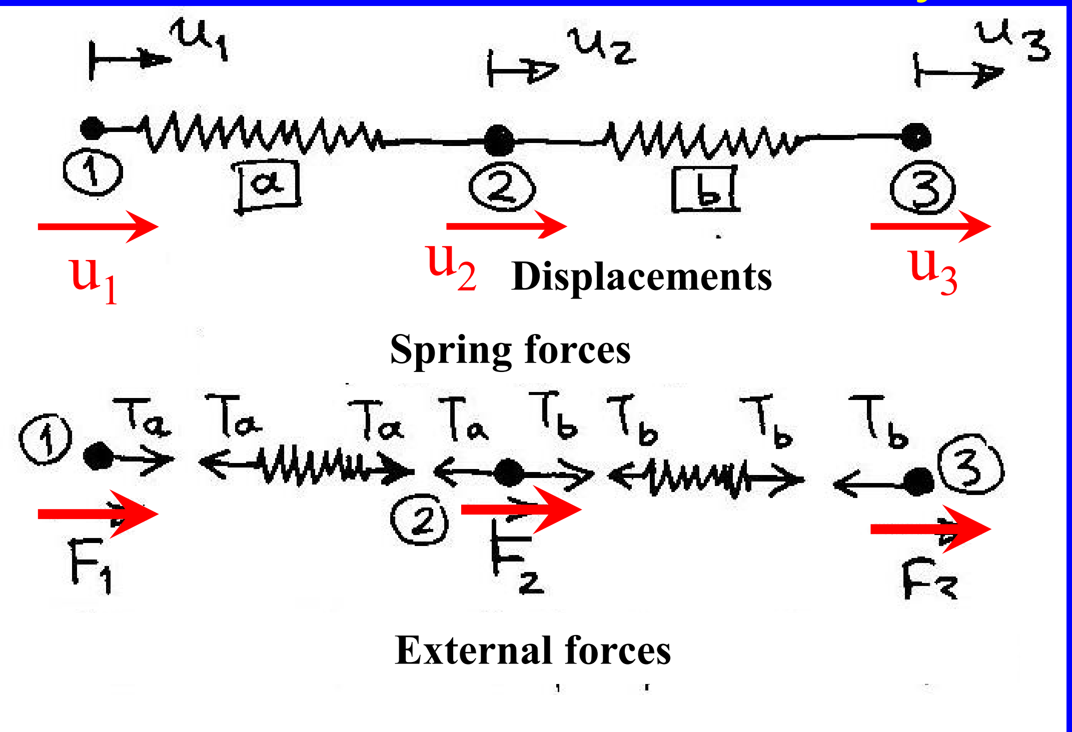
SF = overturning moment / stabilizing moment

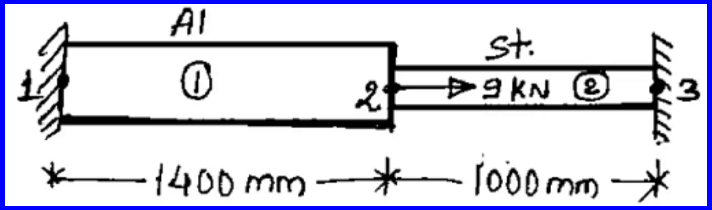
The Finite Element Method (FEM)

The FEM is a generalization of the "matrix method" in structural analysis



Matrix method in structural analysis:



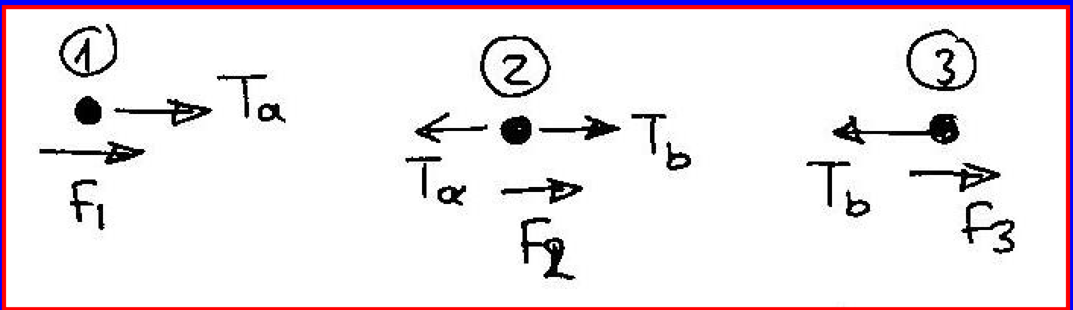


Force equilibrium at node i :

Internal + External forces = 0)

$$\left(\sum T\right) + F_i = 0$$

Force equilibrium at nodes 1, 2, 3 :

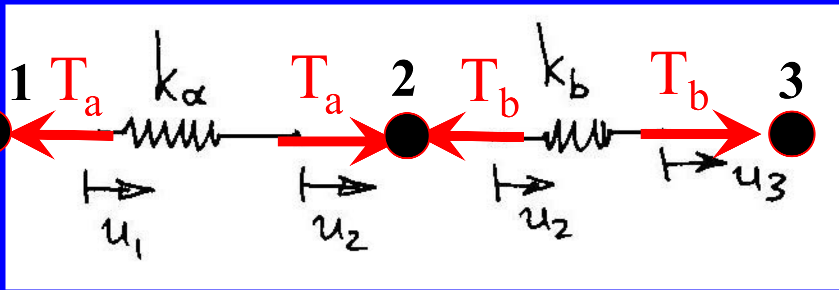


$$T_a + F_1 = 0$$

$$T_b - T_a + F_2 = 0$$

$$-T_b + F_3 = 0$$

Matrix method in structural analysis

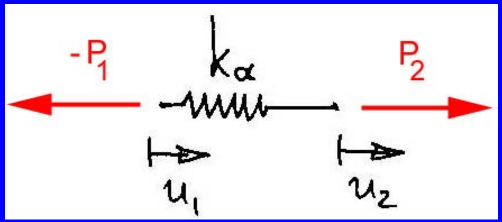


Force – Displacement relations:

$$T_a = k_a (u_2 - u_1)$$
$$T_b = k_b (u_3 - u_2)$$

Replacing at force equilibrium equations:

$$\left.\begin{aligned} T_a + F_1 &= 0 \\ T_b - T_a + F_2 &= 0 \\ -T_b + F_3 &= 0 \end{aligned}\right\} \Rightarrow \begin{cases} k_a u_1 - k_a u_2 = F_1 \\ -k_a u_1 + (k_a + k_b) u_2 - k_b u_3 = F_2 \\ -k_b u_2 + k_b u_3 = F_3 \end{cases}$$



Matrix form of equations: K U = F

$$\begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Element stiffness matrices:

$$\begin{bmatrix} k_a & -k_a \\ -k_a & k_a \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

$$\begin{bmatrix} k_b & -k_b \\ -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P_2 \\ P_3 \end{Bmatrix}$$

Matrix method in structural analysis

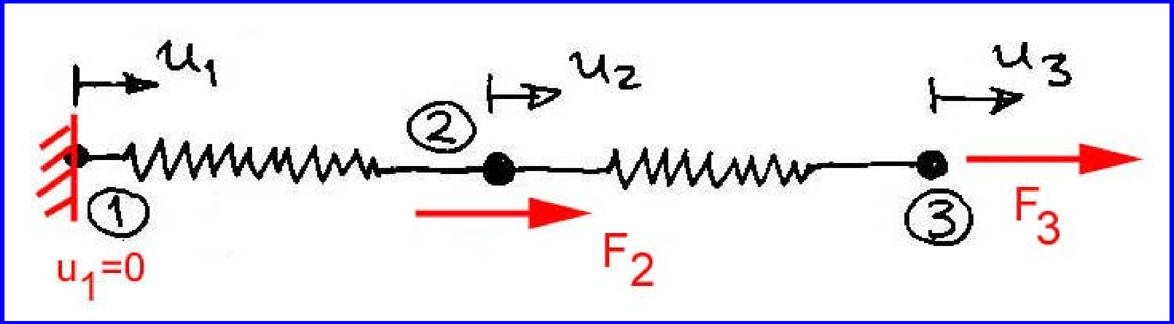
$$\mathbf{K\ U = F} \quad \Rightarrow \quad \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

With known external forces (**F**), and known stiffness (**K**), displacements (U) cannot be computed, because the stiffness matrix (K) cannot be inverted (its determinant is zero): $U = K^{-1}\ F$

The reason is that the homogeneous equation: K U = 0 has an infinite number of solutions (any vector U = { a a a }) i.e., any rigid body displacement.

Solution requires to remove rigid-body modes by applying suitable boundary conditions (i.e., fixing certain nodes).

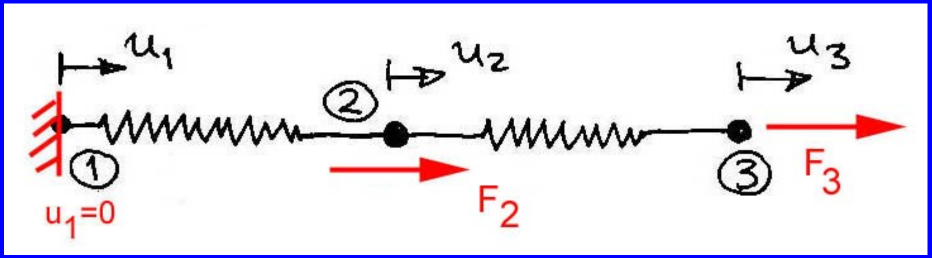
In the below system, a single boundary condition is required (e.g. $u_1 = 0$)



Matrix method in structural analysis

$$\mathbf{K}\,\mathbf{U} = \mathbf{F} \quad \Rightarrow \quad \begin{bmatrix} k_a & -k_a & 0 \\ -k_a & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Apply boundary condition $u_1 = 0$:



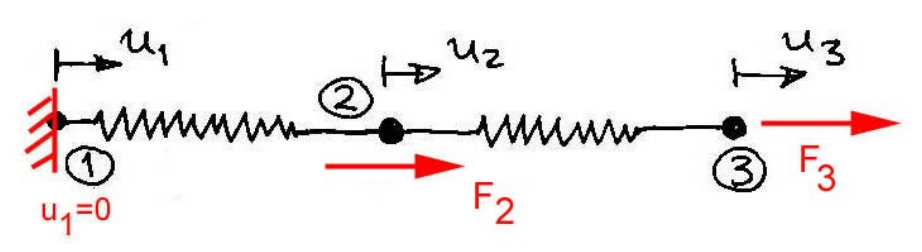
$$\begin{bmatrix} 1 & 0 & 0 \\ 0 & (k_a + k_b) & -k_b \\ 0 & -k_b & k_b \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_2 \\ F_3 \end{Bmatrix}$$

Solution, with inversion of the new stiffness matrix:

$$\mathbf{U} = \mathbf{K}^{-1}\,\mathbf{F} \quad \Rightarrow \quad \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & \frac{1}{k_a} & \frac{1}{k_a} \\ 0 & \frac{1}{k_a} & \left(\frac{1}{k_a} + \frac{1}{k_b}\right) \end{bmatrix} \begin{Bmatrix} 0 \\ F_2 \\ F_3 \end{Bmatrix}$$

Matrix method in structural analysis

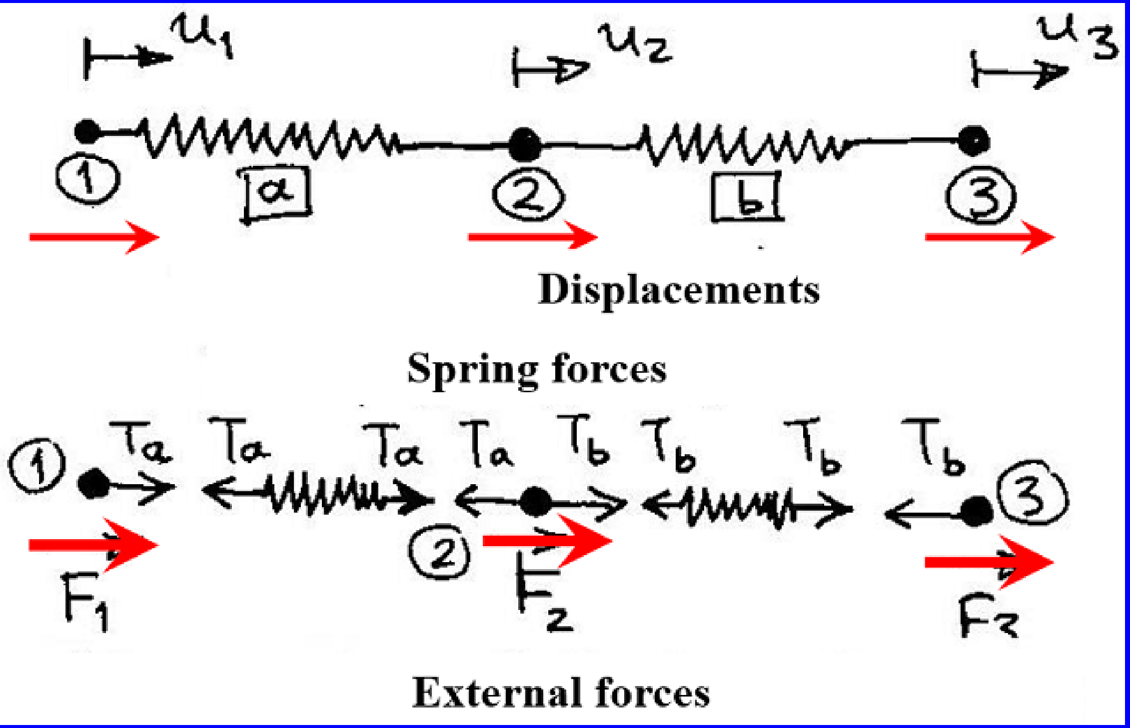
Compute reaction force at support (F_1) :



$$\mathbf{F} = \mathbf{K}\,\mathbf{U} \Rightarrow F_1 = k_a u_1 - k_a u_2 = -k_a u_2$$

Note: $F_1 = -\left(F_2 + F_3\right)$

Compute internal forces and strains:



$$T_a = k_a\left(u_2 - u_1\right)$$
$$T_b = k_b\left(u_3 - u_2\right)$$

$$\varepsilon_a = \left(u_2 - u_1\right)/\,L_a$$
$$\varepsilon_b = \left(u_3 - u_2\right)/\,L_b$$

The Finite Element Method

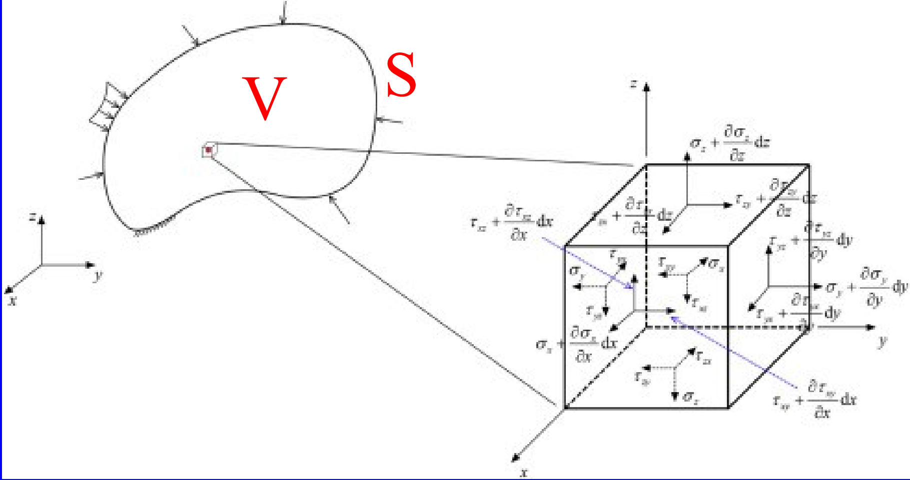
The Finite Element Method has "finite elements" (instead of the structural elements), which are connected with neighbouring elements at nodes.

- Each node has variables (usually displacements) to be determined by the solution
- Each element has a stiffness matrix (which is calculated from the geometrical and material properties of the element

Application sequence of the FEM:

1. Discretization of the structure in finite elements
2. Determination of the stiffness matrix (K_i) of each element
3. Form the global stiffness matrix (K) by inserting the (K_i) at suitable locations
4. Form the external force vector (F)
5. Apply boundary conditions on (K), i.e., modify (K).
6. Invert (K) and compute nodal displacements $U = K^{-1} F$
7. Calculate the reaction forces at supports: $F = K U$
8. Calculate internal forces (e.g. stresses) and strains of the elements

FEM in Geotechnical Applications



Governing equations:

$$\nabla \cdot \boldsymbol{\sigma} + \hat{\mathbf{f}} = \mathbf{0}$$

Equilibrium equations in volume V
($\boldsymbol{\sigma}$ = stress tensor, $\mathbf{f}$ = body force vector)

$$\mathbf{n} \cdot \boldsymbol{\sigma} = \hat{\mathbf{T}}$$

Stress boundary equations on surface S of V

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_{\mathbf{o}} + \Delta\boldsymbol{\sigma}$$

- $\boldsymbol{\sigma}$ = actual (total) stress
- $\boldsymbol{\sigma}_{\mathbf{o}}$ = initial (geostatic) stress
- $\Delta\boldsymbol{\sigma}$ = additional stress due to the applied actions

---

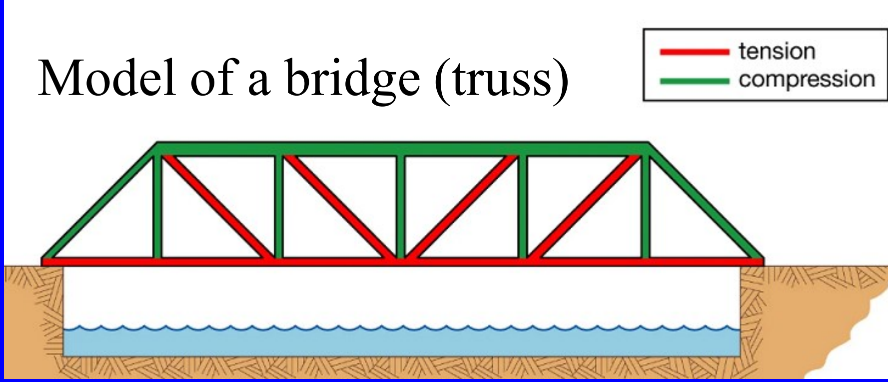
Equivalent description of equilibrium via the principle of Virtual Work:

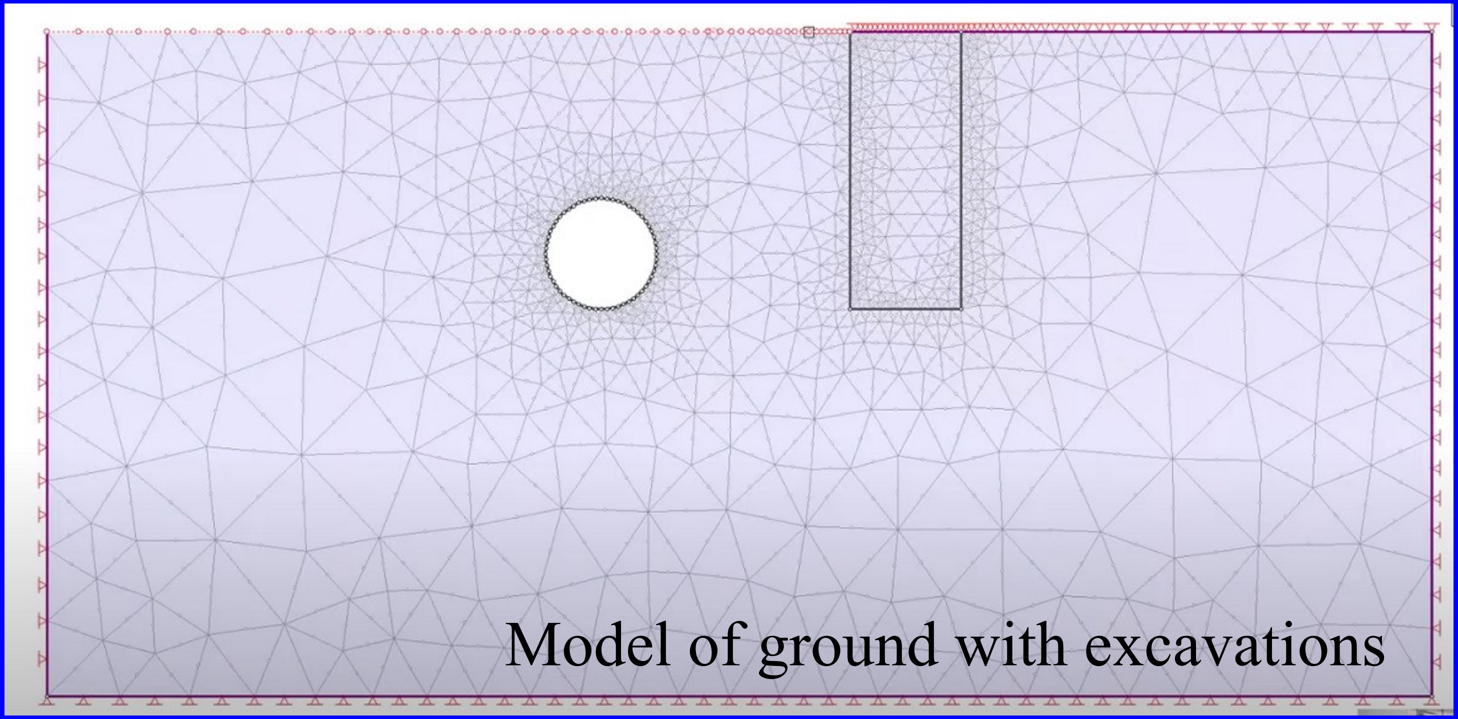
$$\int_V \boldsymbol{\delta\varepsilon}^{\mathbf{T}} \cdot \Delta\boldsymbol{\sigma}\, dV = \int_V \boldsymbol{\delta}\mathbf{u}^{\mathbf{T}} \cdot \hat{\mathbf{f}}\, dV + \int_S \boldsymbol{\delta}\mathbf{u}^{\mathbf{T}} \cdot \hat{\mathbf{T}}\, dS - \int_V \boldsymbol{\delta\varepsilon}^{\mathbf{T}} \cdot \boldsymbol{\sigma}_{\mathbf{o}}\, dV \Rightarrow \mathbf{K}\,\mathbf{U} = \mathbf{F}$$

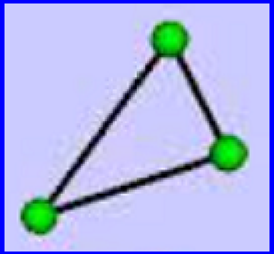
Additional stresses due to actions = { Body forces + Surface forces } - (initial stresses)

The Finite Element Method

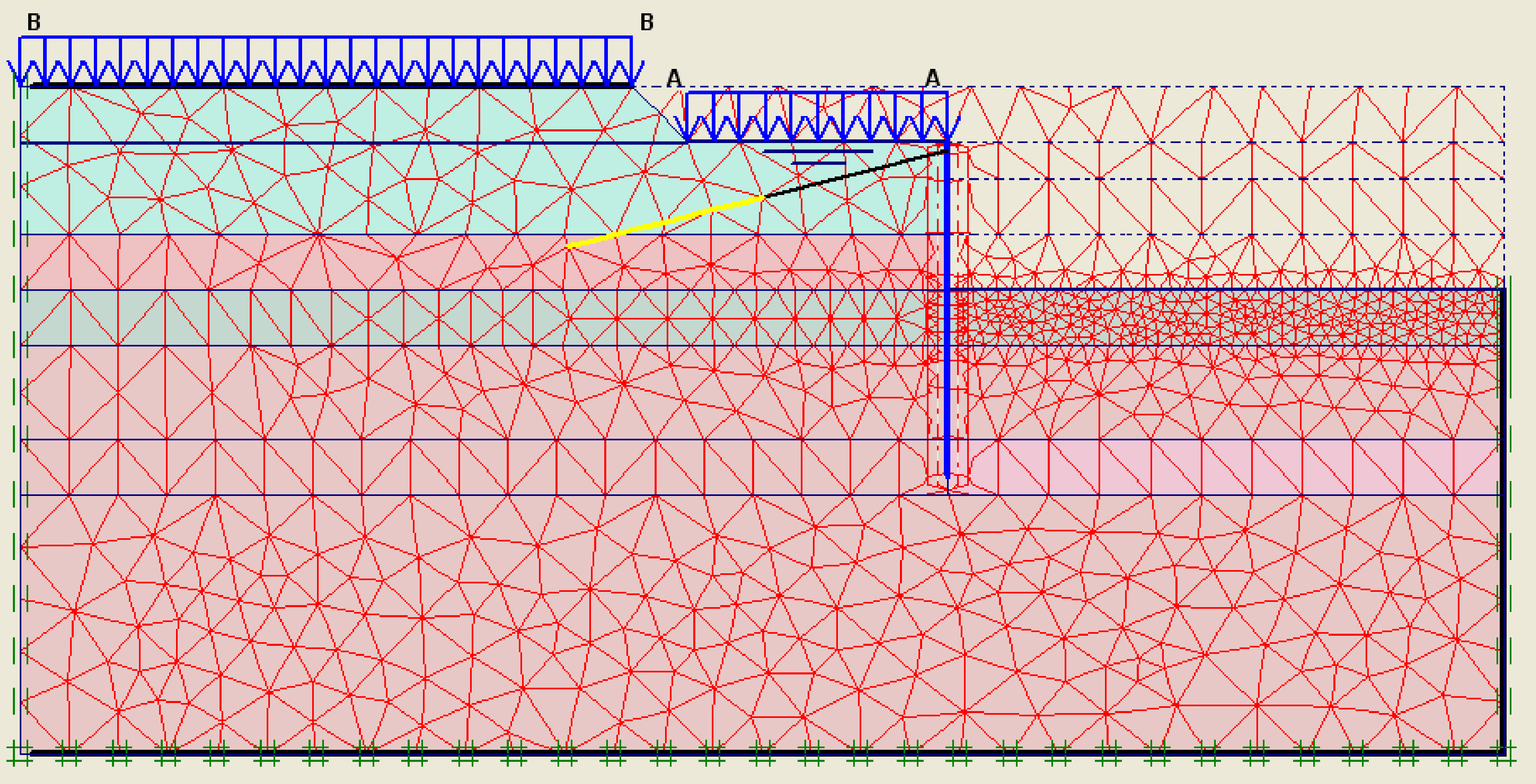
Problem discretization in Finite Elements





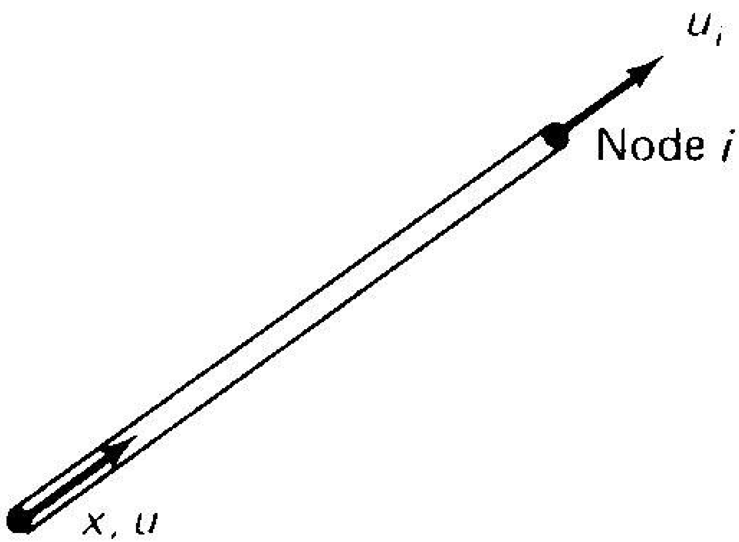


FE discretization in 2D ground problems



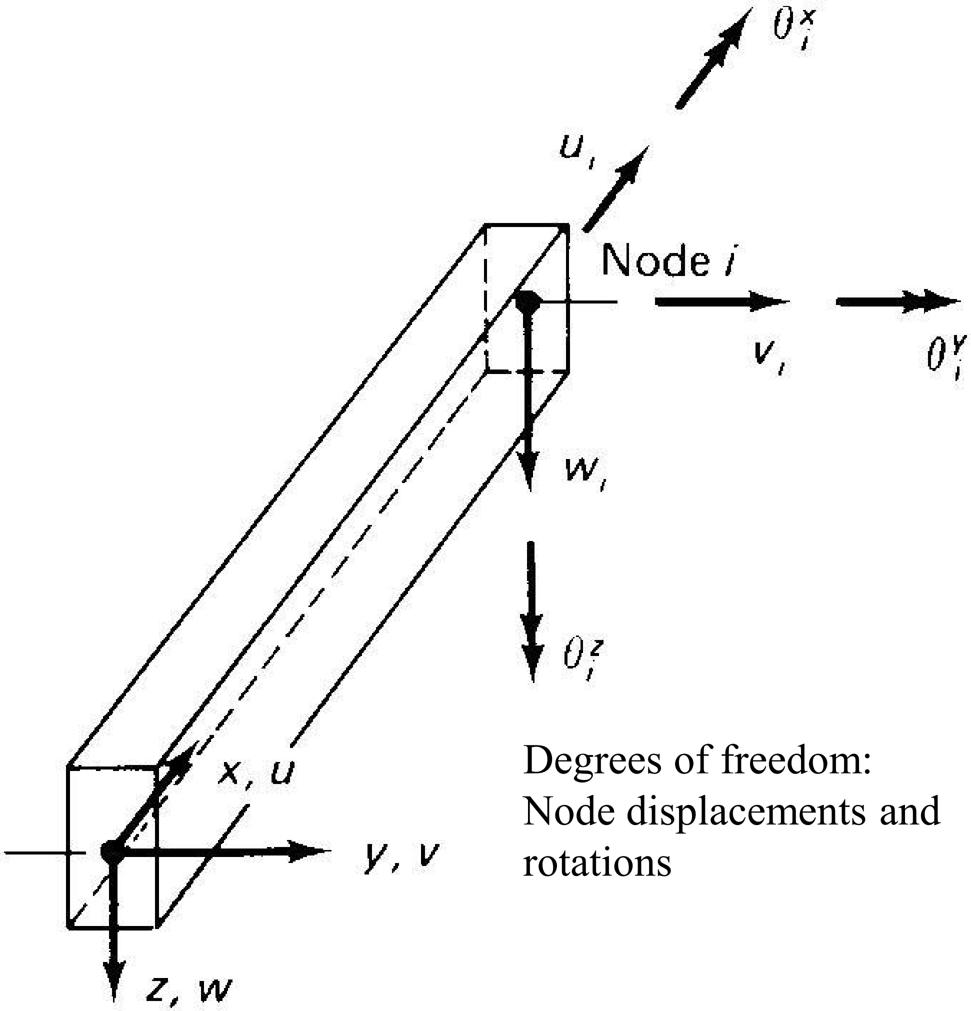
Types of Finite Elements

Structural (finite) elements



Degrees of freedom: Node displacements

(a) Truss element

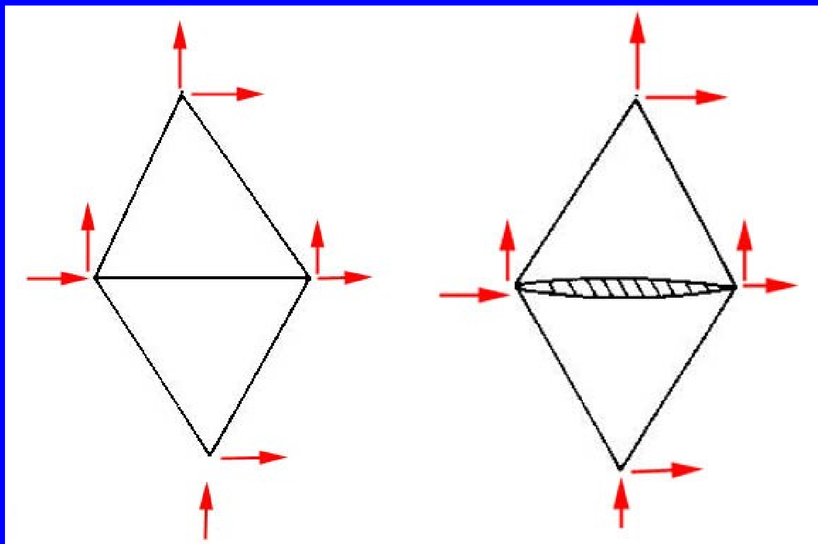


Degrees of freedom:
Node displacements and
rotations

(b) Beam element

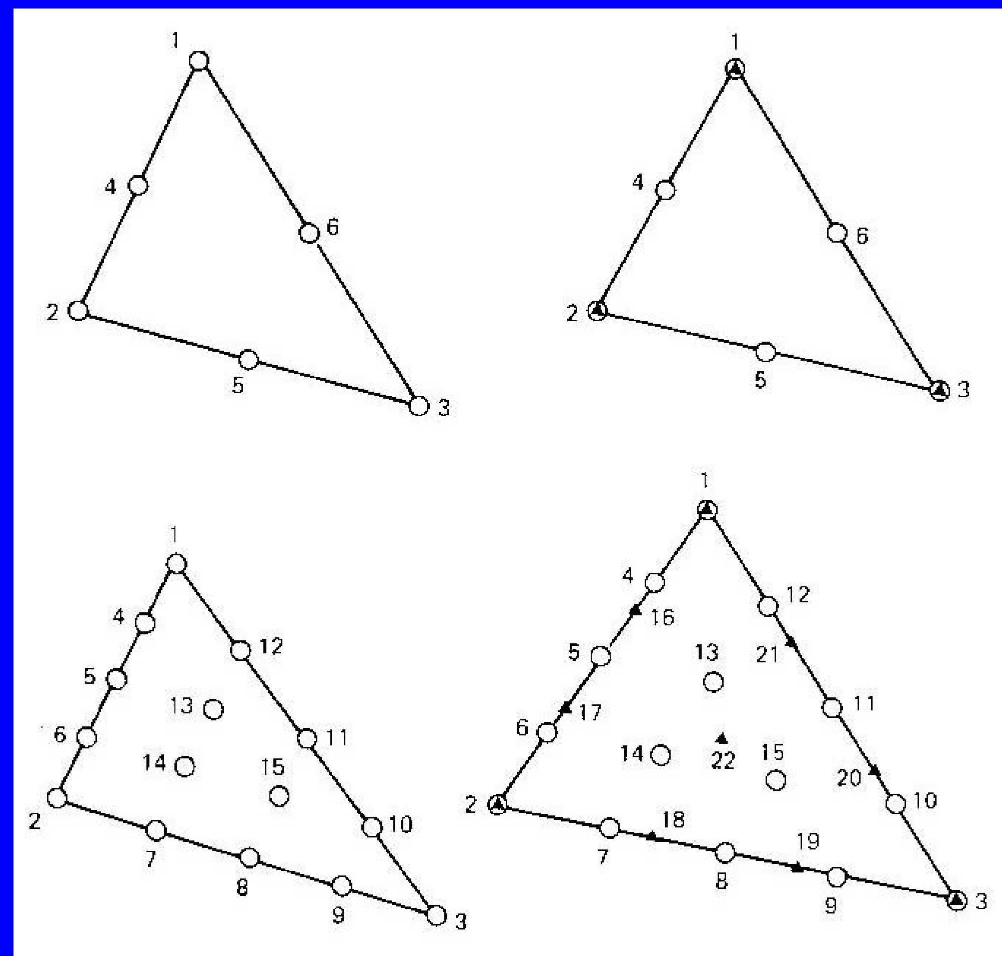
Types of Finite Elements

Triangular solid FE in 2D problems (e.g. plane strain)

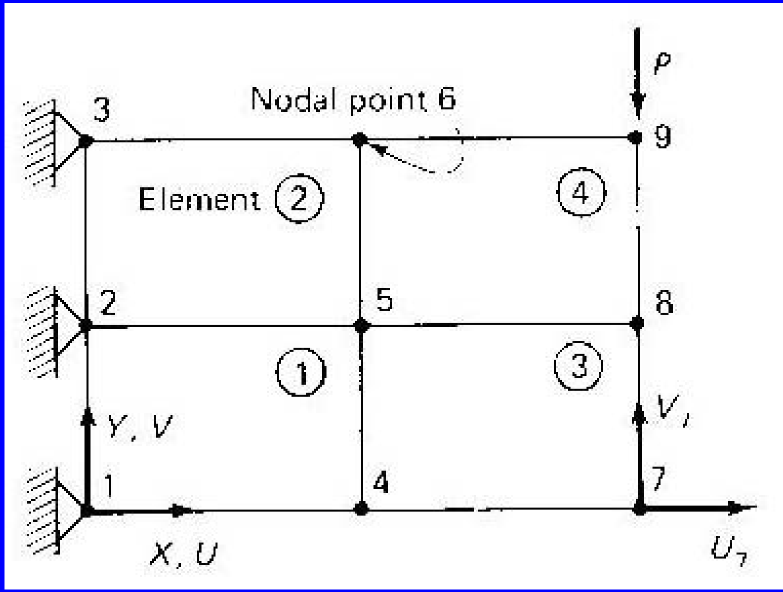


Node displacements (U_x , U_y) are the unknown variables to be calculated

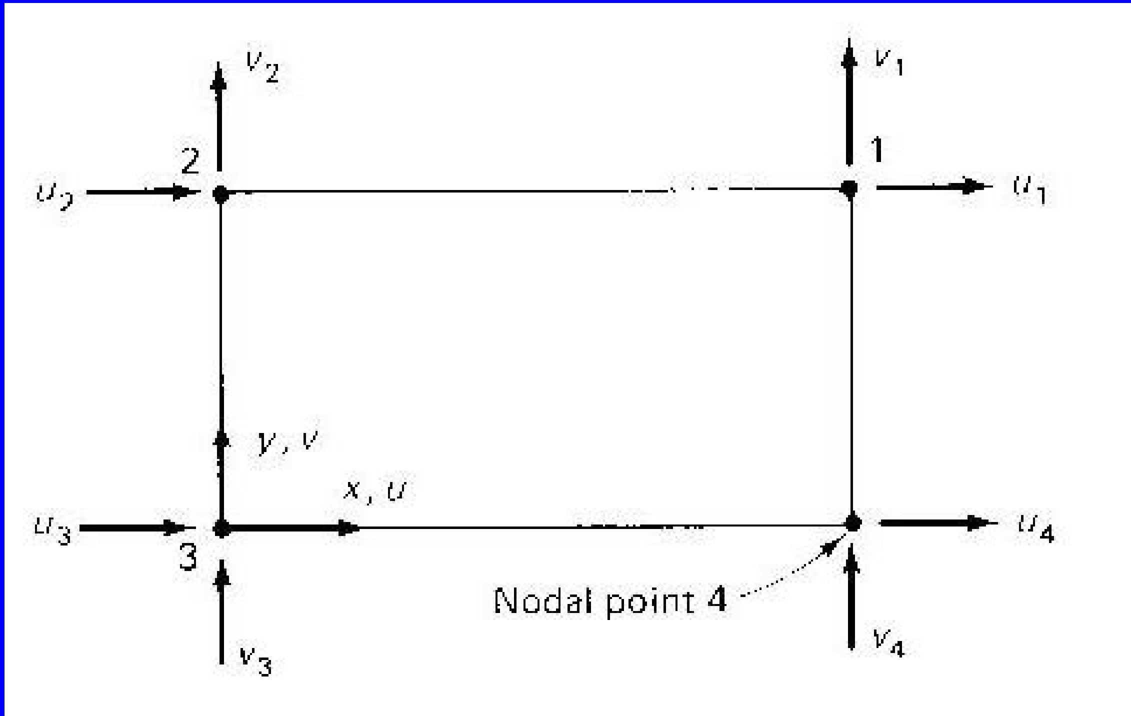
Triangular elements can have more than 3 nodes → higher order displacement distribution in each element → fewer elements are required



Types of Finite Elements



Quadrilateral solid FE in 2D problems (e.g. plane strain)

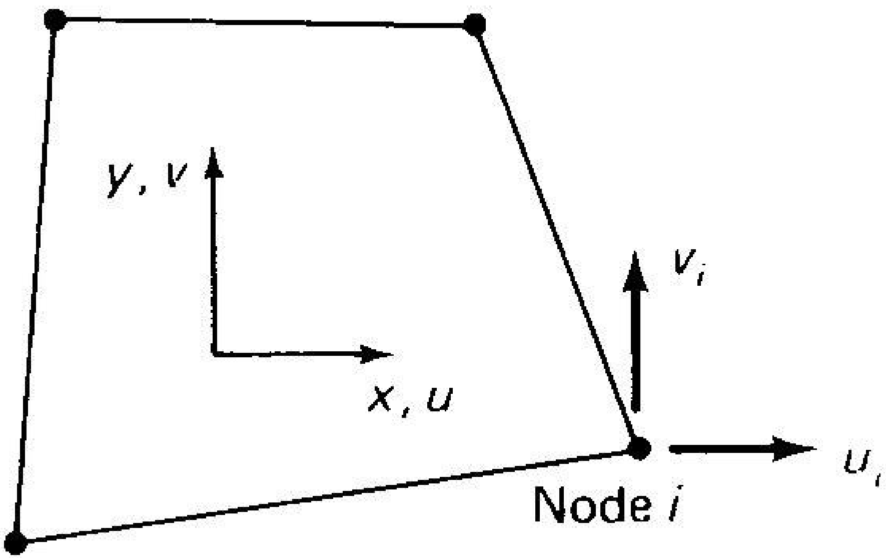


Node displacements (U_x , U_y) are the unknown variables to be calculated

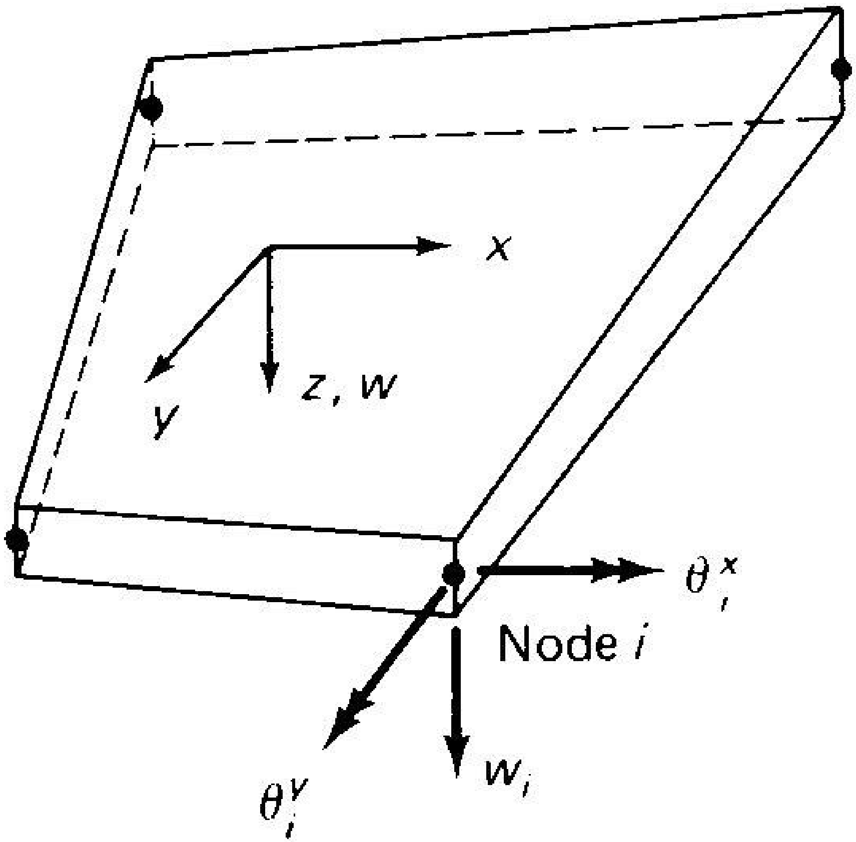
Types of Finite Elements

Quadrilateral solid FE in 2D problems

Left : Plane strain Right : Plates (slabs) under bending



(c) Axisymmetric, plane strain and plane stress element

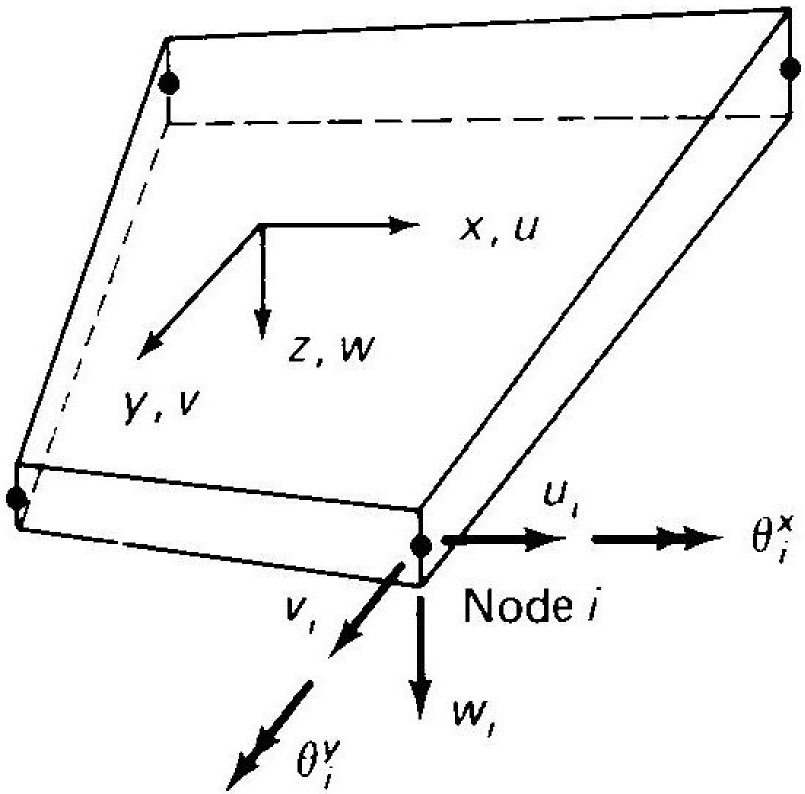


(d) Plate bending element

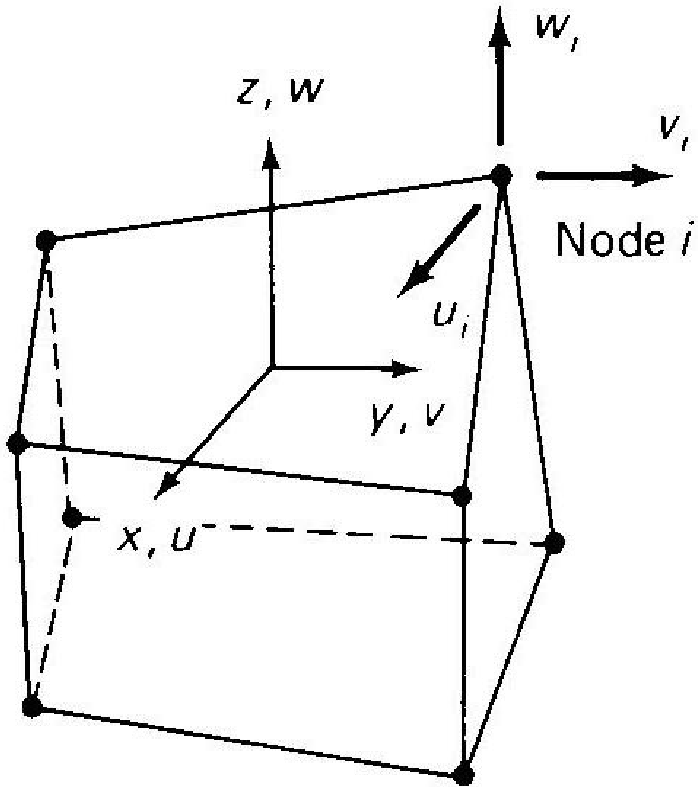
Types of Finite Elements

Quadrilateral solid FE

Left : Shells Right : 3D solids



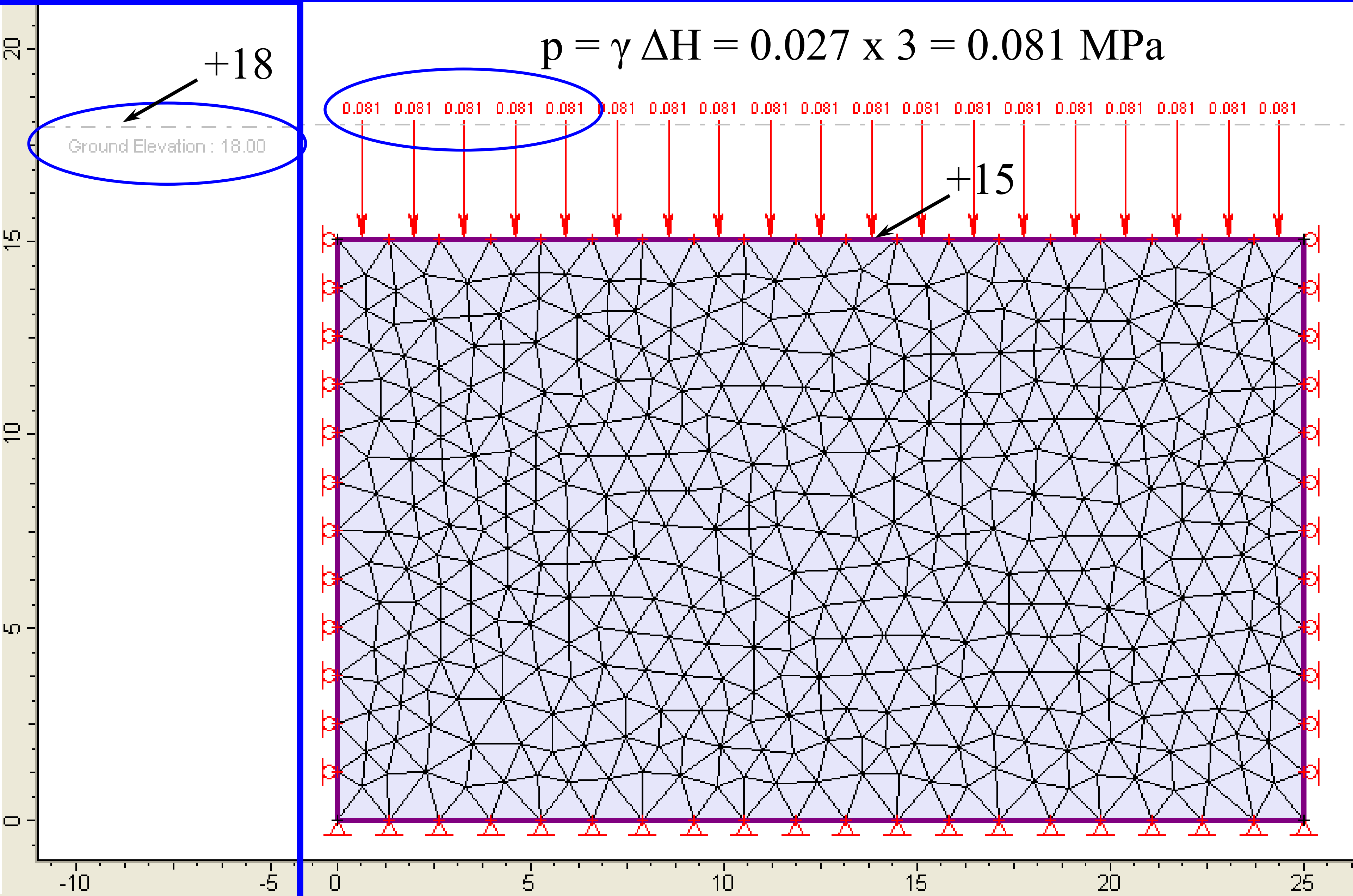
(e) Thin shell element (obtained by superimposing plate bending and plane stress)



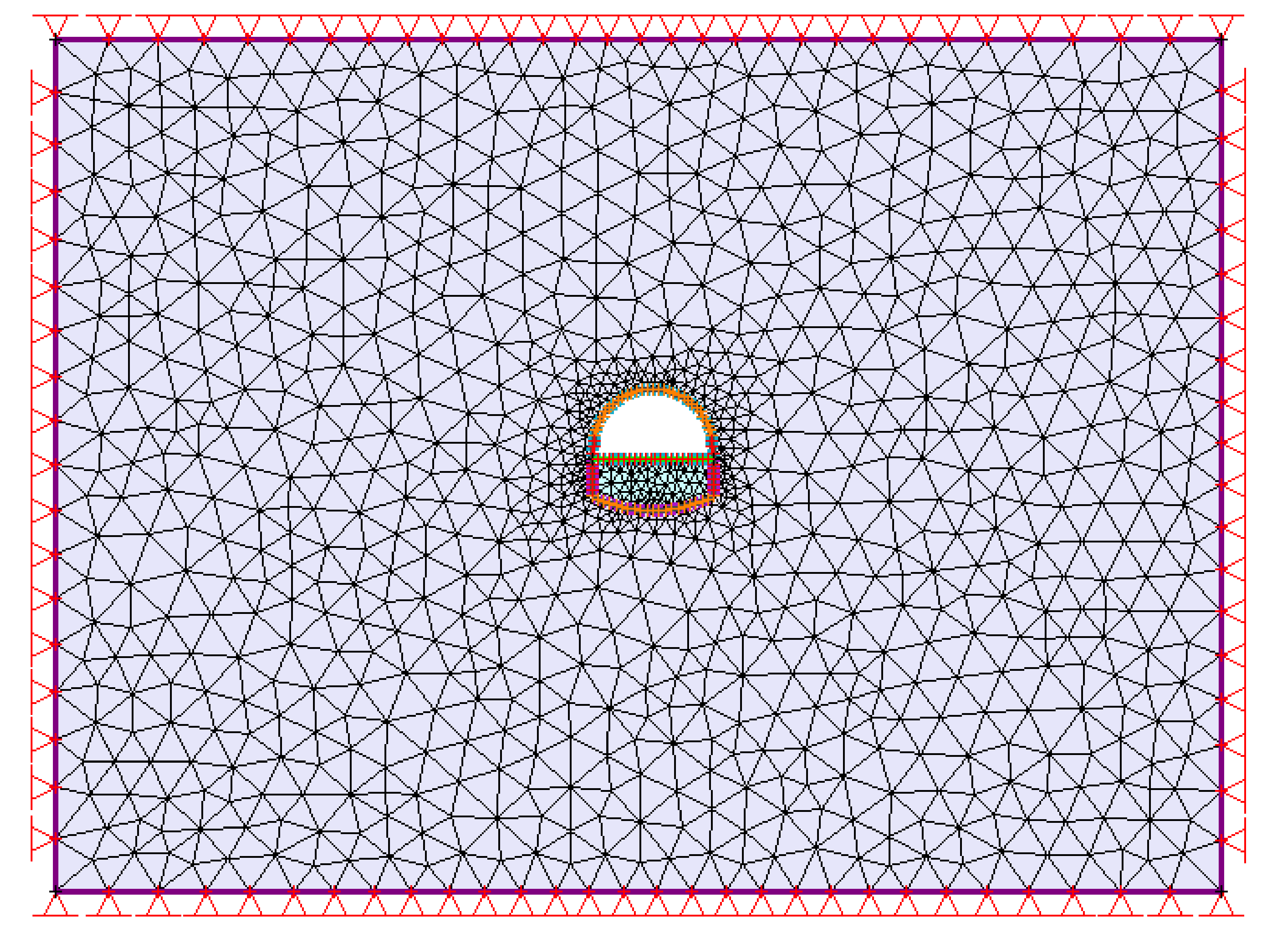
(f) Three-dimensional element

Initial condition (Field Stress = Gravity)

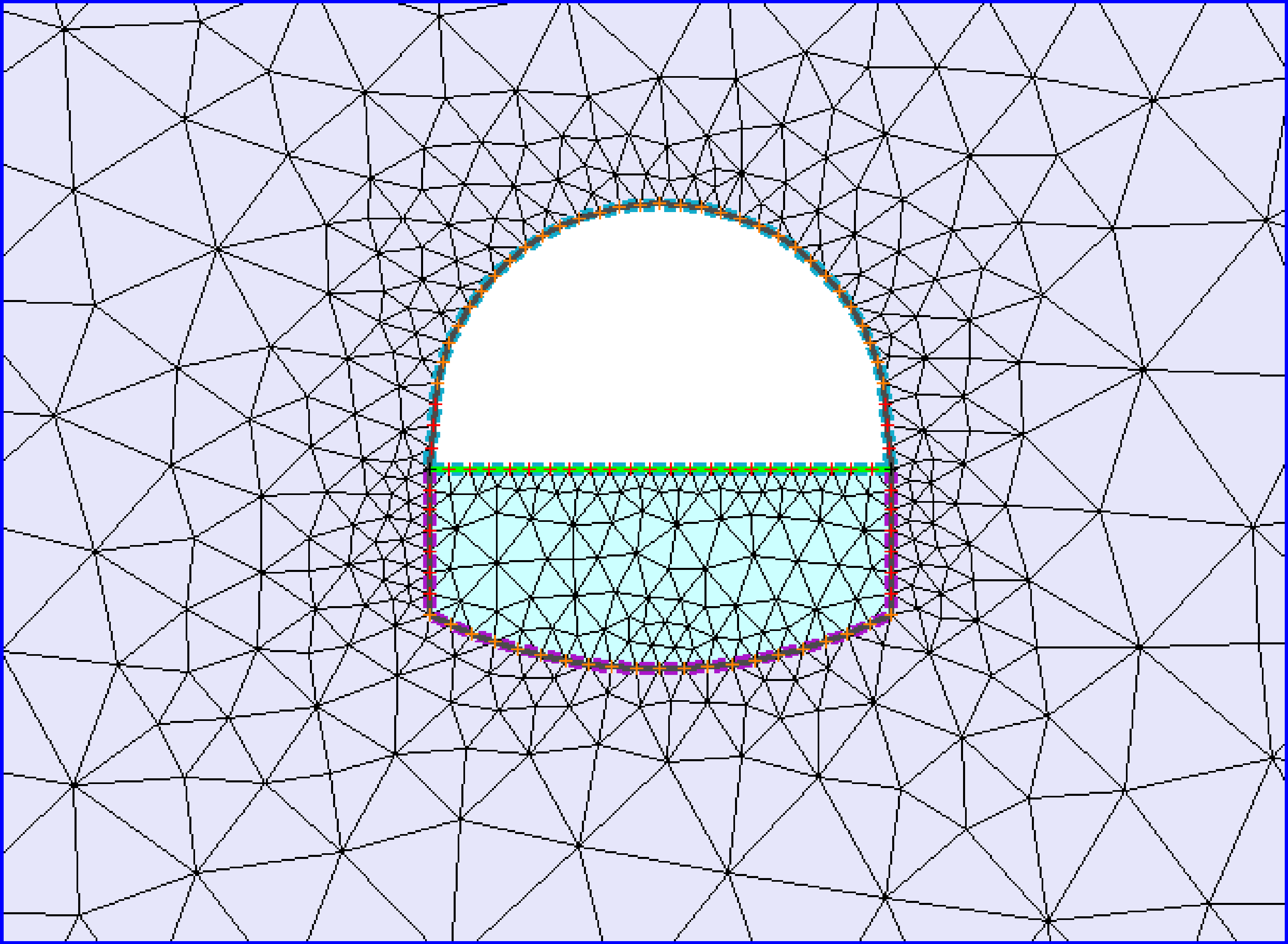
$$p = \gamma\,\Delta H = 0.027 \times 3 = 0.081 \text{ MPa}$$



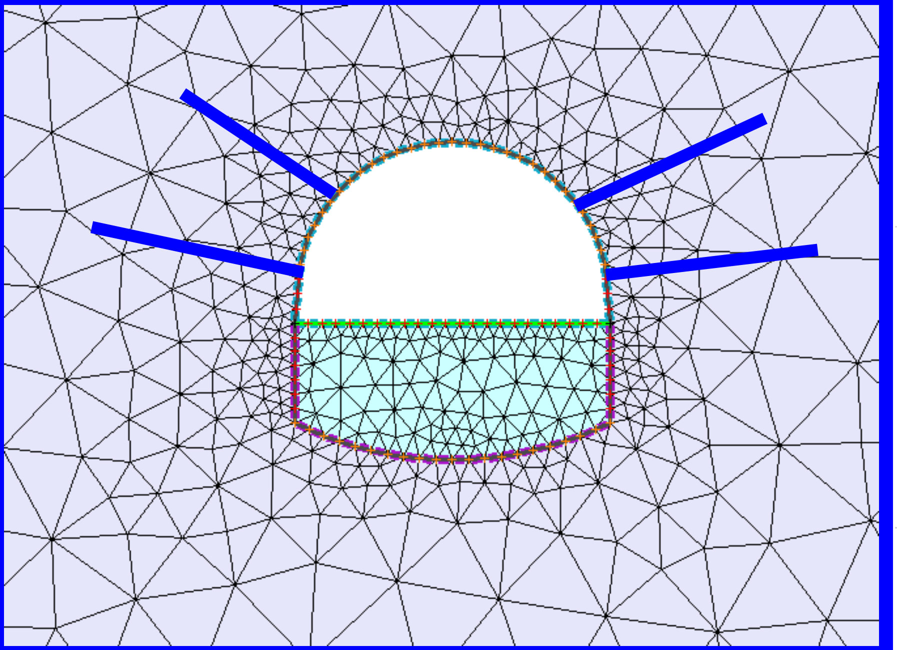
Finite Element discretization

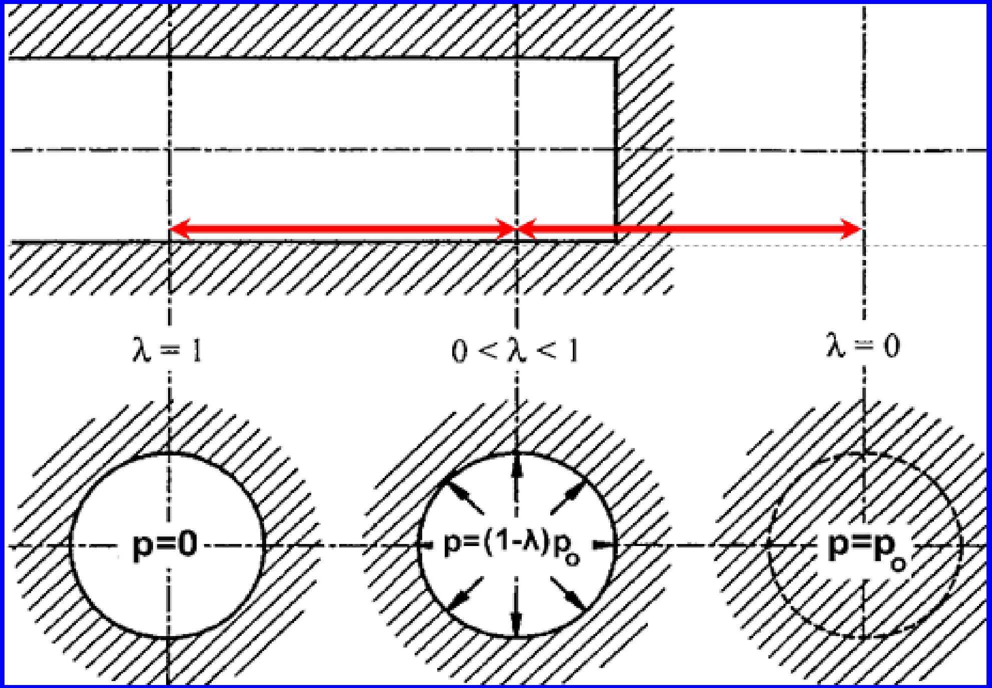


Finite Element discretization



Finite Element solution of tunnel excavation and support – Analysis steps





Each excavation and support phase includes at least two steps:

(1) Deconfinement by applying a λ-coefficient (λ < 1) or modulus reduction ($E_o \rightarrow E < E_o$) in the area to be excavated – this models excavation up to the installation of the support

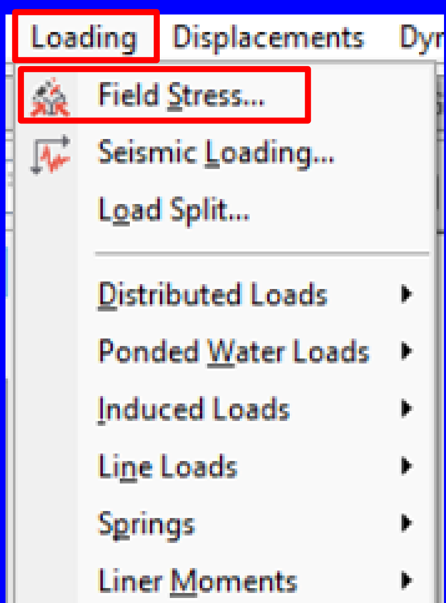
(2) Installation of the support and completion of the excavation (λ or E → 0)

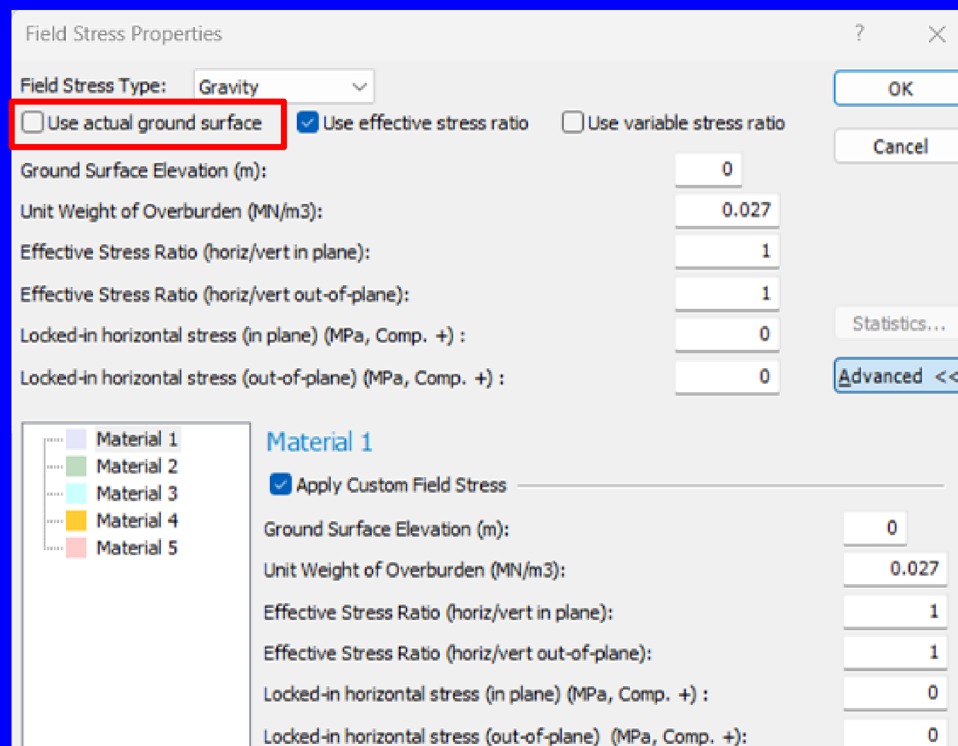
Note: If support is applied in several steps, No 2 is split in multiple sub-steps, each with some support and additional deconfinement.

Initial condition (stress field) of the model in the RS2 software

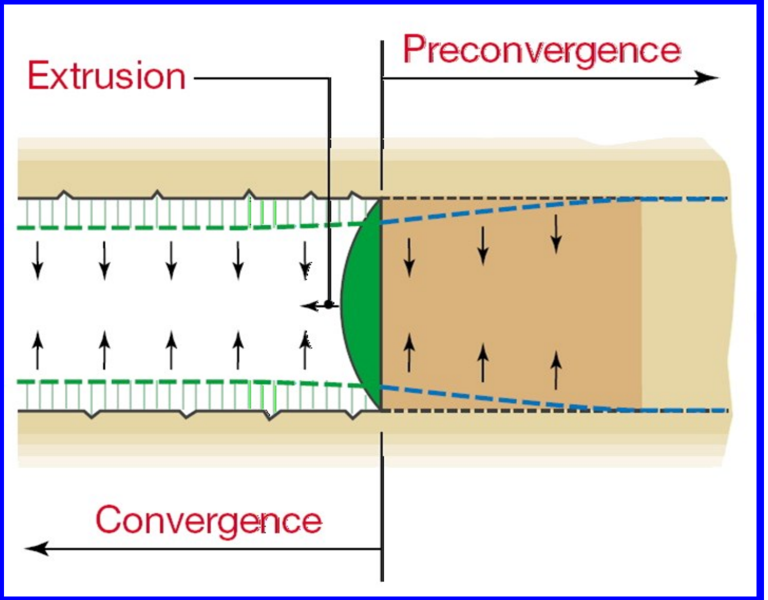
- Surface Tractions are computed using the surface forces and/or pressures
- Ground Body Forces (weight) are computed using the unit weight of the various ground layers.
- The initial (geostatic) stresses in the ground are computed using the “Field Stress” menu (options: “Constant” or “Gravity”) in the “Loading” tab

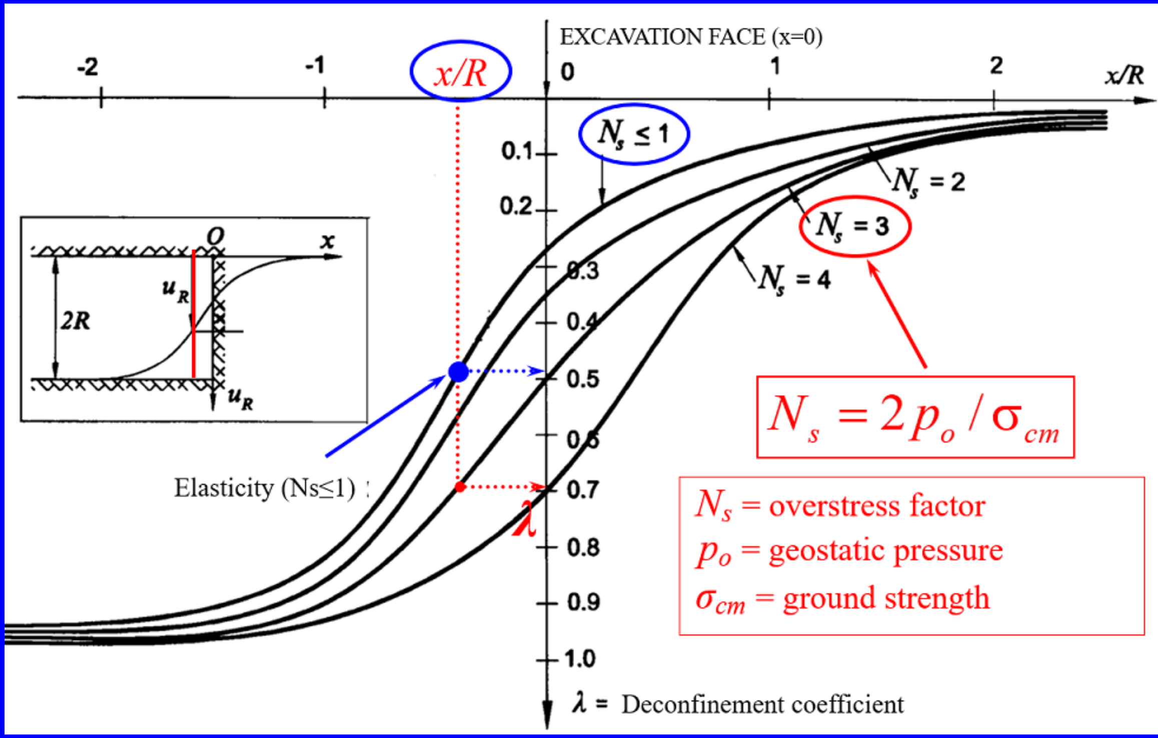
  If the “Ground surface elevation” is not equal to the actual ground surface of the model, the unit weight of the uppermost ground zone needs to be applied to the ground above the surface of the model (use the “Advanced” option for this material)
- The horizontal stress ratio (K_o) is defined in the “Field Stress properties”





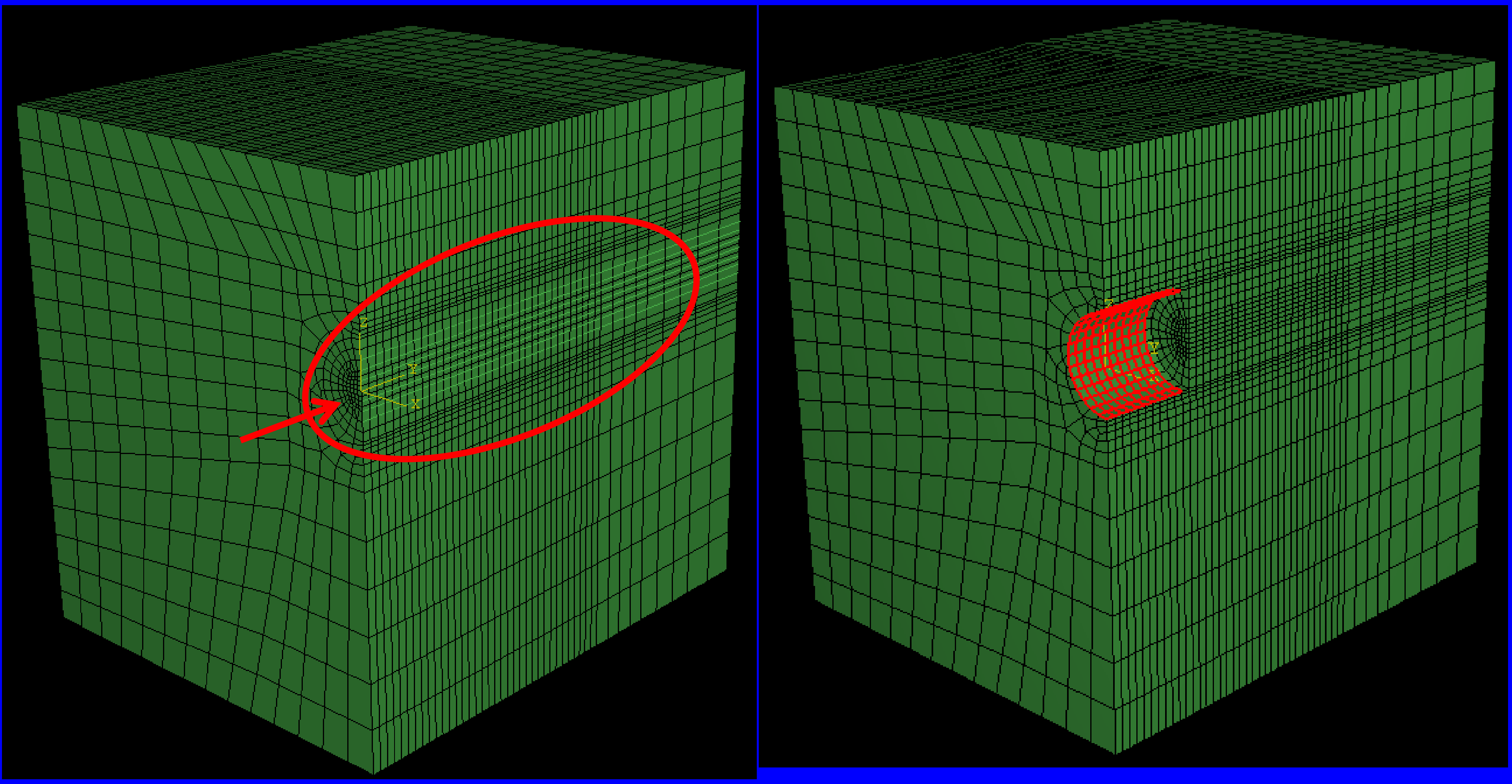
3D Finite Element analyses



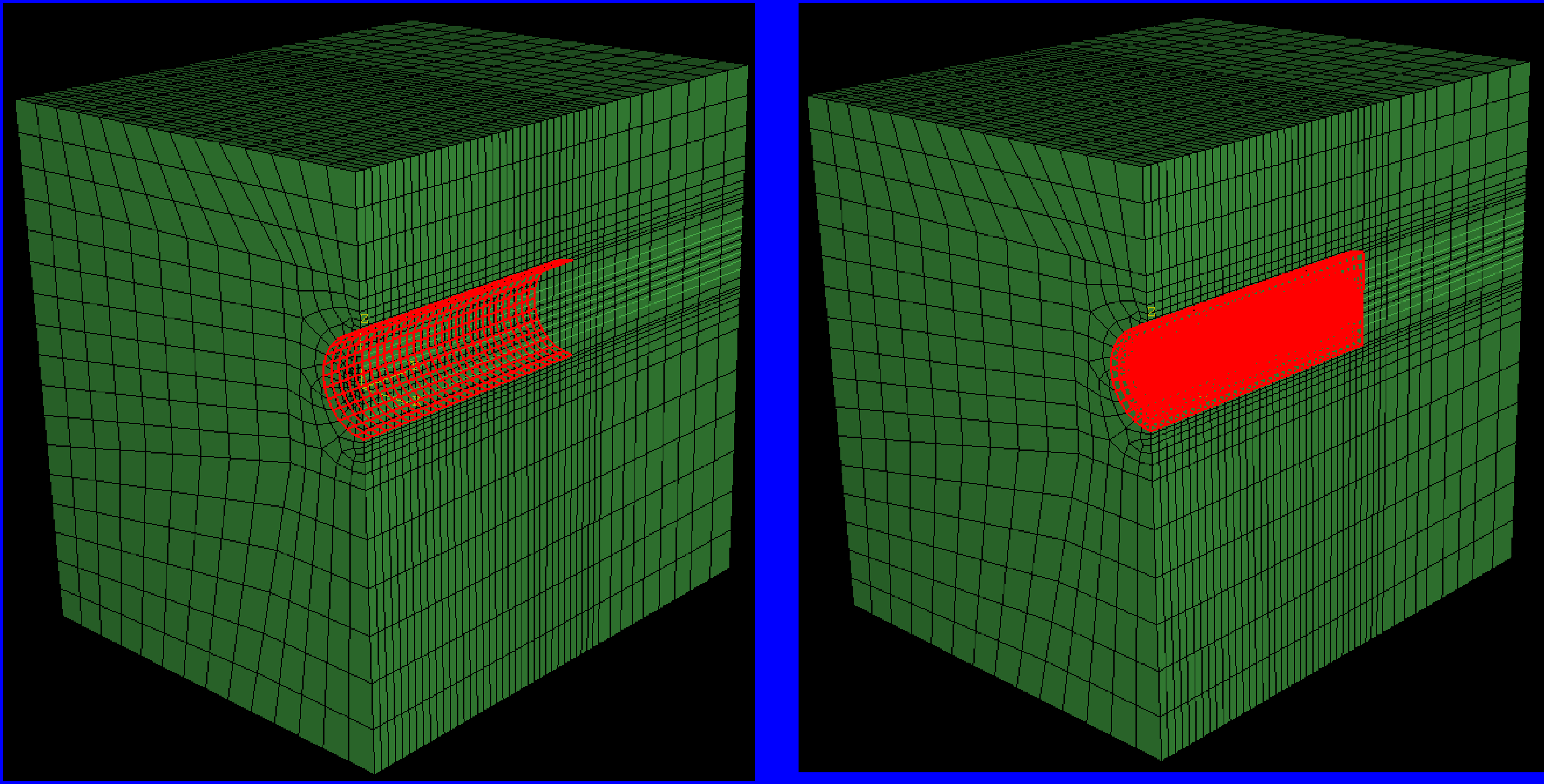


- 2D Finite Element analyses cannot model the extrusion of the excavation face. Thus, an "assumption" is required → the deconfinement coefficient.
- 3D Finite Element analyses do not require such an assumption, since face extrusion is included in the model

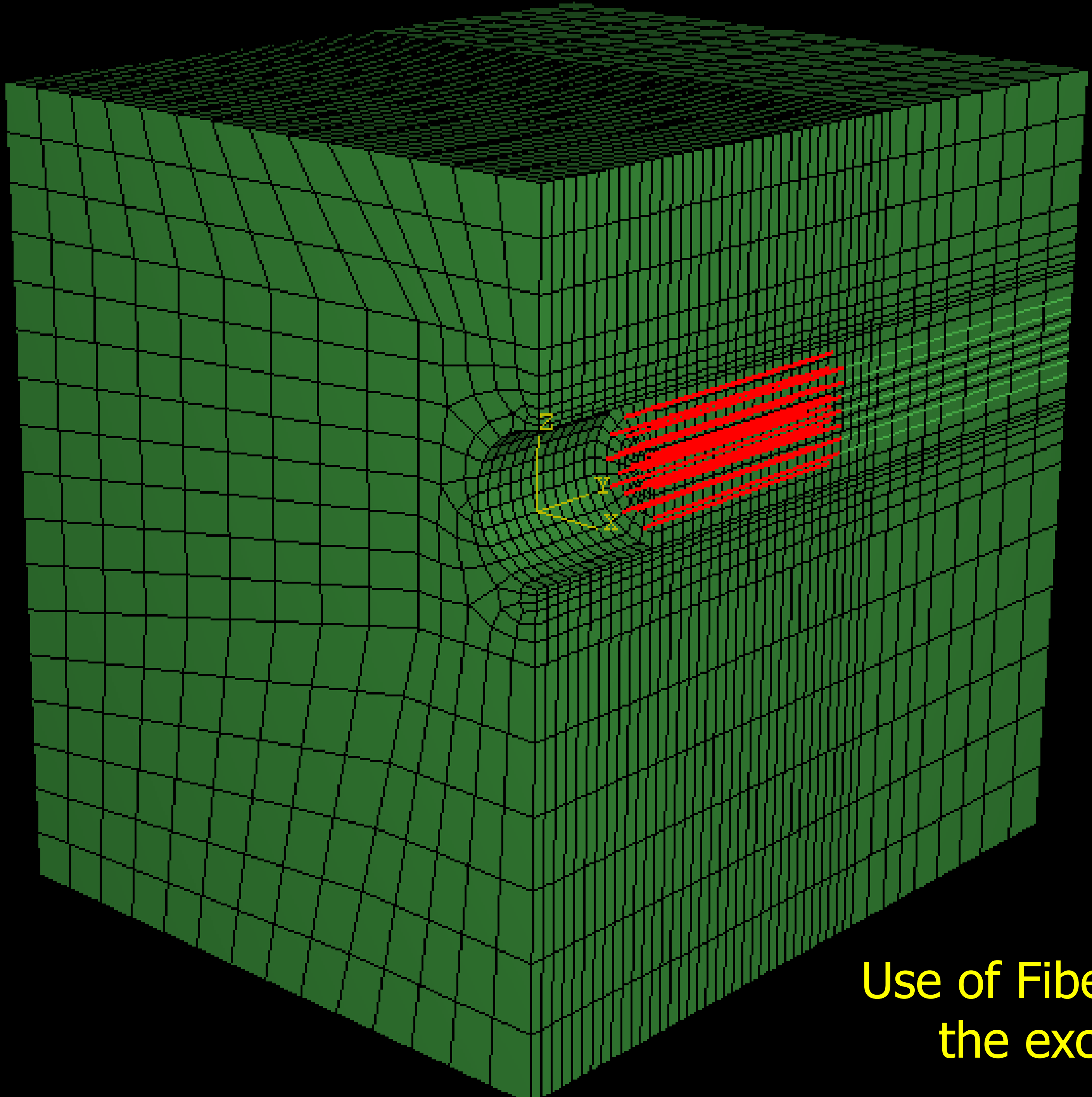
NOTE: The deconfinement coefficient is calculated from 3D FE analyses (e.g. via the Chern curves) and is an input parameter in 2D analyses.



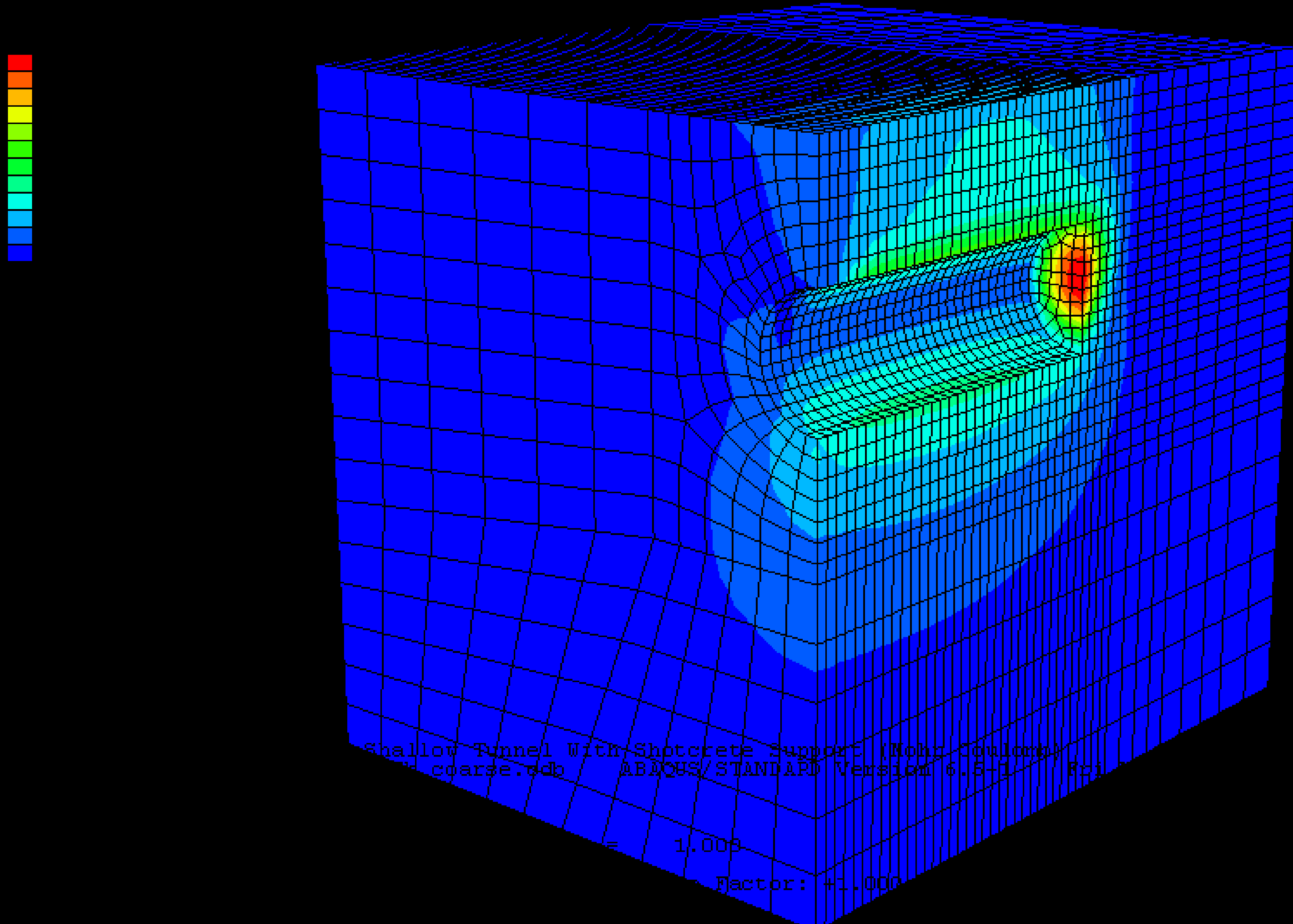
3D Finite Element model



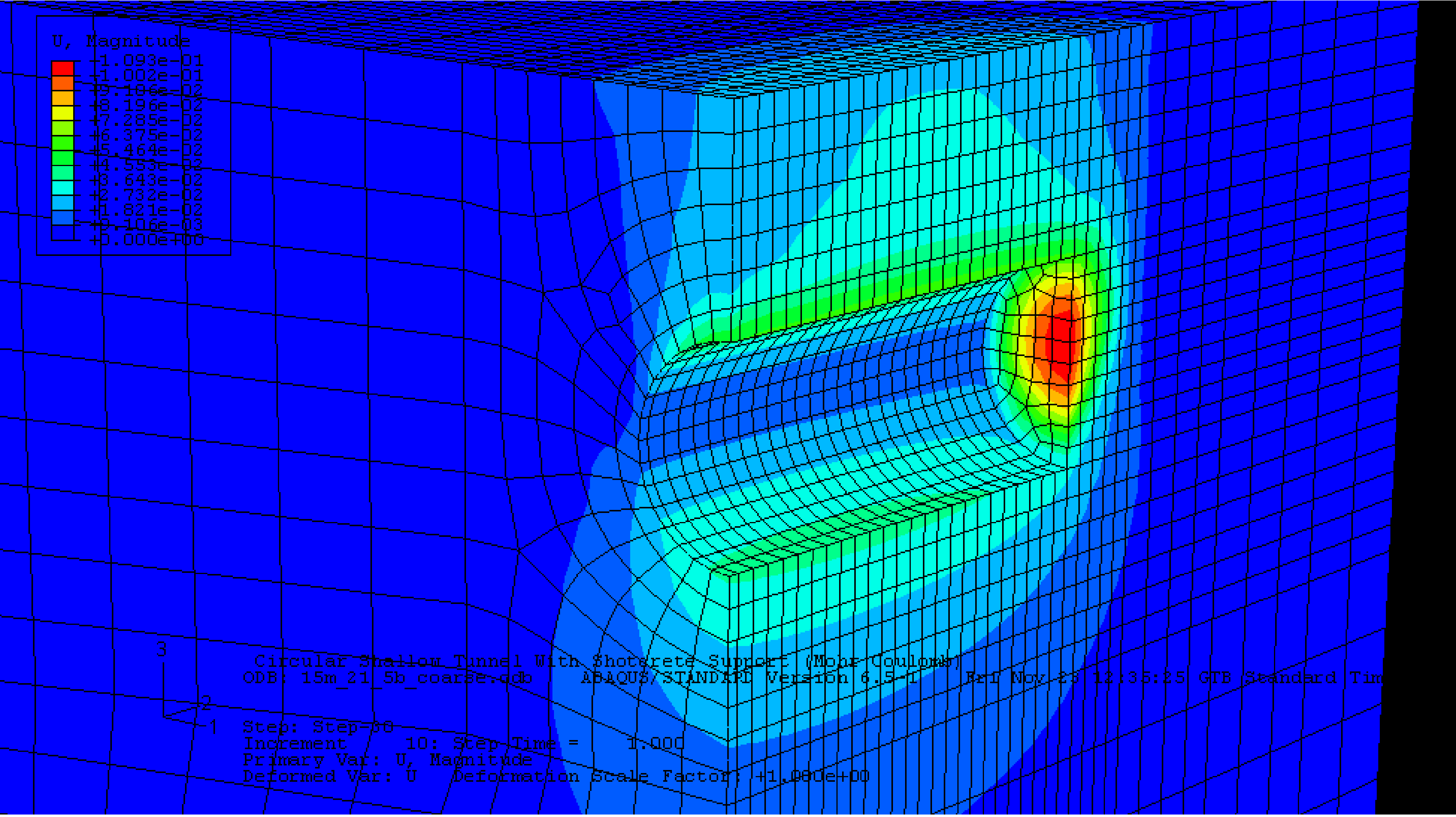
3D Finite Element model



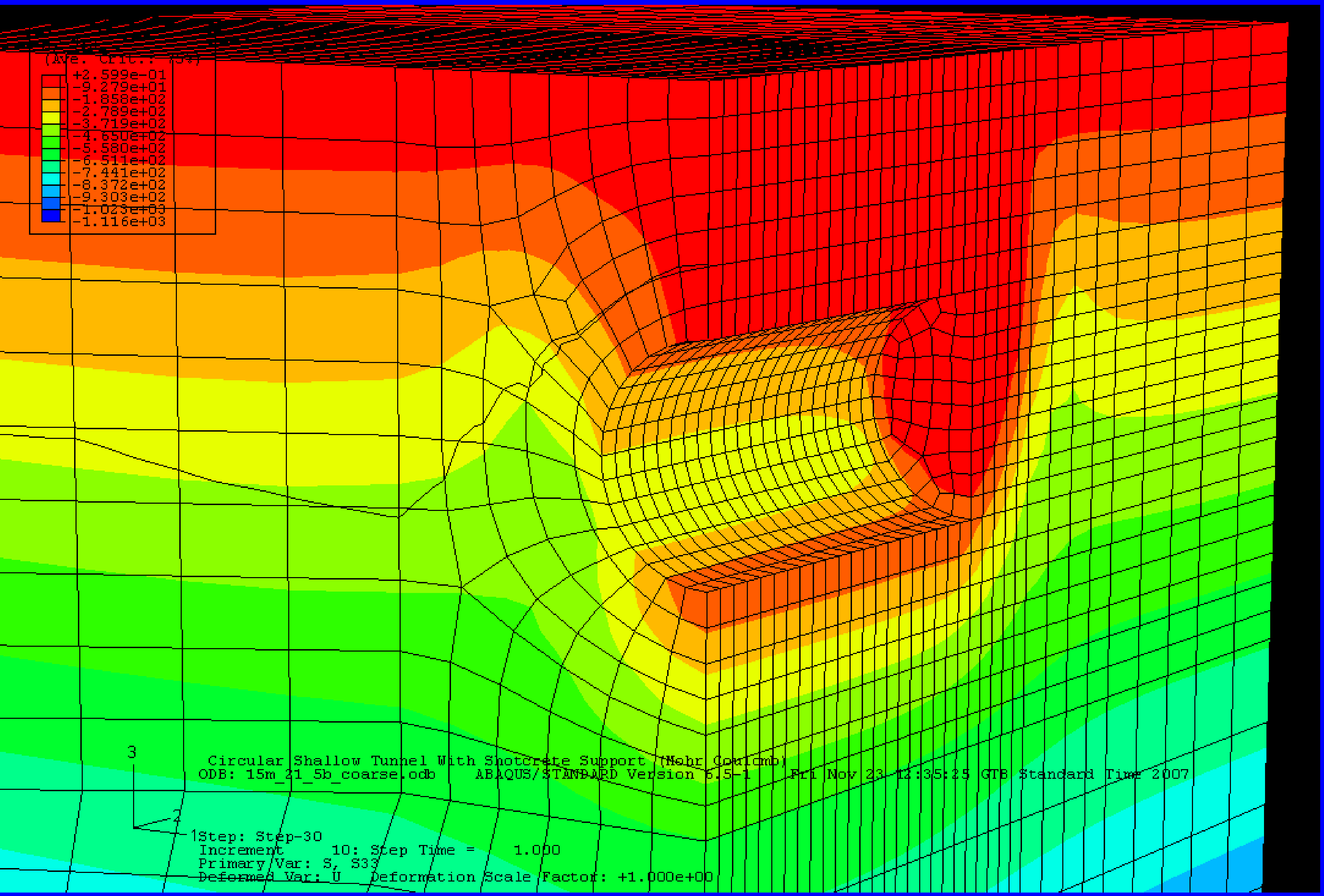
Use of Fiber Glass nails on the excavation face



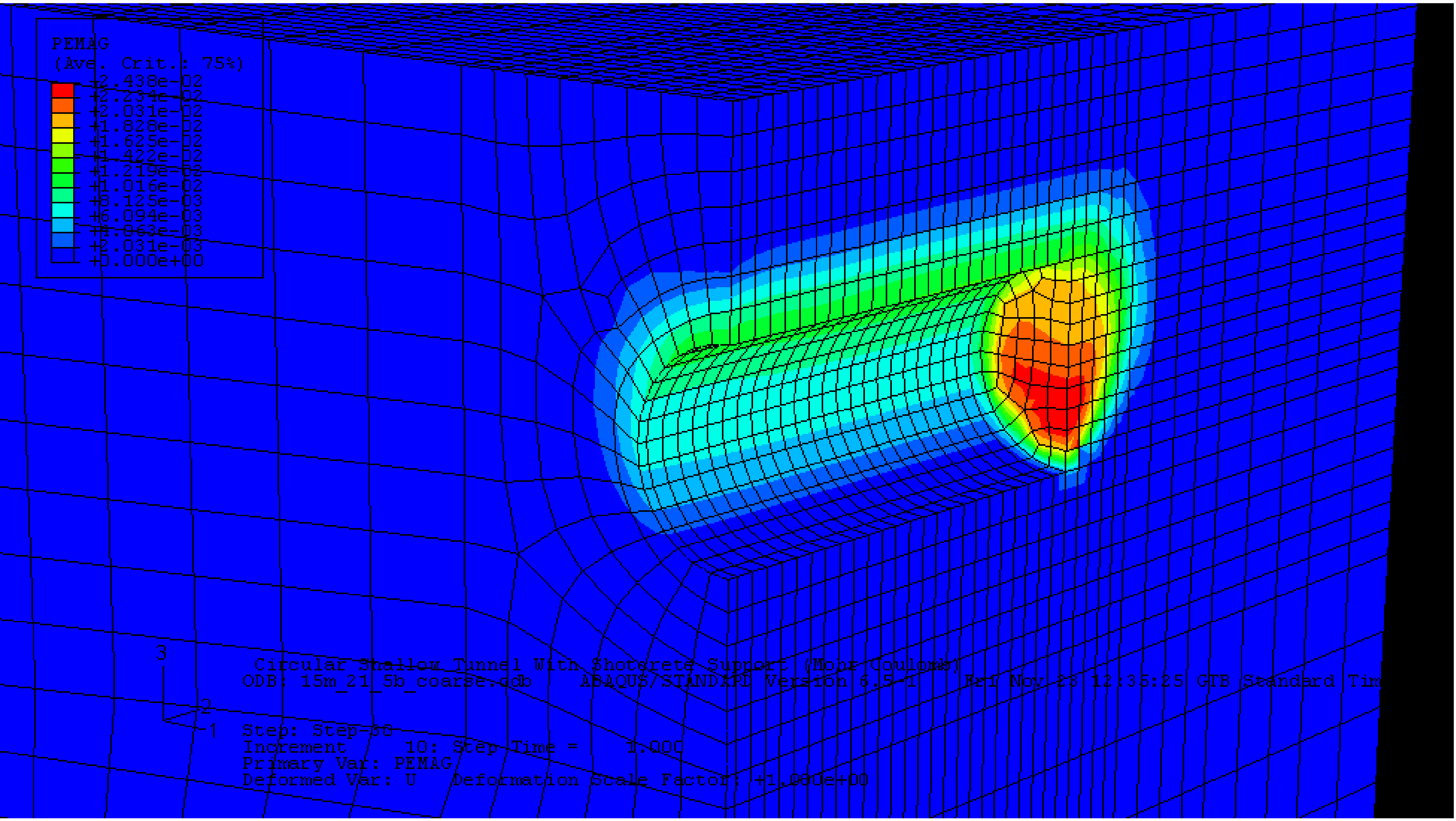
Typical results of the analysis: Total displacements



Typical results of the analysis: Total displacements



Typical results of the analysis: Mises shear stress



Typical results of the analysis: Plastic shear strain