Acta Technica Napocensis: Civil Engineering & Architecture Vol. 55 No. 1 (2012)
Journal homepage: http://constructii.utcluj.ro/ActaCivilEng

Finite element modelling for tunneling excavation

Alexandra Raluca Moldovan [*1], Augustin Popa [2]

[1,2] *Technical University of Cluj-Napoca, Faculty of Civil Engineering. 15 C Daicoviciu Str., 400020, Cluj-Napoca, Romania*

Received 4 March 2012; Accepted 16 September 2012

Abstract

Nowadays the increase demand for tunneling construction in urban areas has evolve complex calculations and analyses in order to satisfy the correct simulation for this type of construction. At the present the tunnels are analyzed on the basis of Finite Element Method both in a 2D and 3D approach. The aim of this paper is to describe the modelling of a tunnel construction in a 2D approach using different methods. Previously a brief introduction in the finite element method principle and the most common used finite elements in a tunneling construction simulation will be discussed.

Rezumat

Datorită creșterii acerbe a construcției tunelurilor în zonele urbane, au fost dezvoltate calcule și analize complexe pentru a satisface cerințele acestui tip de construcție. În momentul de față, tunelurile sunt analizate cu ajutorul elementului finit, această problemă fiind modelată atât în 2D cât și în 3D. Scopul acestui articol este de a descrie diferitele metode de simulare a construirii unui tunel cu ajutorul elementelui finit în 2D. Deasemenea o introducere în metoda elementului finit, a principiului și a elementelor finite cele mai des întâlnite în simularea construcției unui tunel for fi discutate.

Keywords: Tunneling, Finite Element Method, 2D modelling, gap parameter method, convergent method, volume loss method, progressive softening method.

1. Introduction

The increased demand of public transportation in metropolitan areas and the scarcity of horizontal space led to an increased requirement for underground transportation. Nowadays the tunnels are analyzed on the basis of Finite Element Method.

The conventional analysis although are simple to use, and give relatively good results, still remain limited since the different patterns are considered separately: loads are determined using usually an elastic solution, whereas movements are calculated using empirical techniques. This is the reason why the finite element method is used lately to solve complex problems such as: simulating the construction sequences, complex ground conditions, realistic soil behavior modelling, complex hydraulic conditions, account ground treatments, account for adjacent structures, short and long

[*] Corresponding author: Tel./ Fax.: 0744142419, 0264523319
E-mail address: Alexandra.MOLDOVAN@cif.utcluj.ro

term conditions, multiple tunnels [3].

Available for almost six decades, firstly developed in the aerospace industry (1950), the finite element method is used nowadays by millions of engineers and scientists worldwide. While widely spread among engineering practice, is relatively recent in the use of geotechnical problems, because of the complex issues encountered in the geotechnical field.

In geotechnical engineering is usual to encounter situations when the total stress is split into effective and pore fluid pressure. In addition, geotechnical engineering involves the interaction between structures and soil, therefore in the finite element method both structures are necessary to be modeled. When a saturated soil is used means that there is no volume change, and this can be modeled for a isotropic elastic material as setting the Poisson radio 0.49, since the 0.5 value results in severe numerical problems as all the terms in the matrix that contains the effective constitutive behavior, becomes infinite [2]. The idea of finite element method is to break up the continuum into discrete number of smaller elements. Physical phenomena in engineering can be described using partial differential equations. The finite element method represents a numerical approach which can solve these partial differential equations easily, whereas using classical analytical methods this process will be impossible. It is helpful to use in engineering areas, where problems such as stress analyses, heat transfer, fluid flow and electromagnetic can be solved, allowing to predict the behavior of structural, mechanical, thermal, electrical and chemical systems in both design and performance analysis.

Although is considered the most exact calculation method, due the errors than can emerge, the FEM is often considered a tool to investigate the mechanism of behavior rather than obtaining precise predictions about the tunnel performance. It is to bear in mind that the finite element method can make a good engineer to become a better one, and a bad engineer to become a dangerous one. In order to asses successful analyses with "sound" results, the engineer firstly needs to have an understanding in the soil mechanics and the finite element theory, moreover an appreciation of the constitutive models limitation. The input date dictates the resulting output, and can be "wrong" even if the algorithm works properly. According to Wood and Clayton (1993) the modelling is subject to six sources of errors that might lead to poor predictions: modelling the geometry of the problem, modelling of construction method and its effects, constitutive modelling and parameter selection, theoretical basis of the solution method, interpretation of results, human error [2].

The paper will include a brief description of the finite element method, followed by the methods which are used to model a tunnel excavation in a 2D approach.

2. Finite element methods

2.1. Principle of Finite Element Method

A finite element analyzing program includes a pre-processing and post-processing phase. The former includes various modules for creating a model, defining material proprieties, specifying boundary conditions and external loads and meshing the assembly of the model. The results generated from the analysis can be enormous so it requires additional processing which represents the latter phase. The procedure of computational modelling using the FEM broadly consists of main steps: modelling of the geometry, meshing (discretization), specification of material property, specification of boundary, initial and loading conditions. In the following several steps that need to be accomplished for a Finite Element Method are described [6].

2.1.1. Idealization

This first step is important since is the assumption that needs to be done to idealize the "real problem". The problems that are analyzed with the finite element method are generally 3D problems in real life, but can be reduced to 1D, 2D or 3D models. This reduction in dimensions can be done by simplifying the model. This initial step can dramatically change the results; therefore a correct assumption should be done before starting to model [6].
Reducing the 3D to 2D can be done by input the third dimension or just analyzing the most significant part of the 3D problem as a parameter. The 2D analyses can be modeled as : plain stress, plain strain and axisymmetric approaches.
The plain stress is define as being a state of stress in which the normal stress s_z and the shear stresses s_{xz} and s_{yz} directed perpendicular to the x-y plane (out-of –plain) are assumed to be equal to zero. The plain stress approach is used to describe a problem if the stress is zero in the direction not being modeled. In this case is considered that the solid has a uniform thickness and the thickness is much less than the other two characteristic dimensions [6].
The plain strain is defined to be a state of strain in which the strain normal to the x-y plane (out-of – plain), ε_z, and the shear strains g_{xz} and g_{yz} are assumed to be equal to 0. The plain strain approach describes the problem where the strain is zero in the direction that is not being modeled. This case is used to model deep solids that cannot deform in the third plan where the dimension is very large in comparison to the other directions of the structure. This approach is used in the analysis of tunnels, dams and other geotechnical works [6].

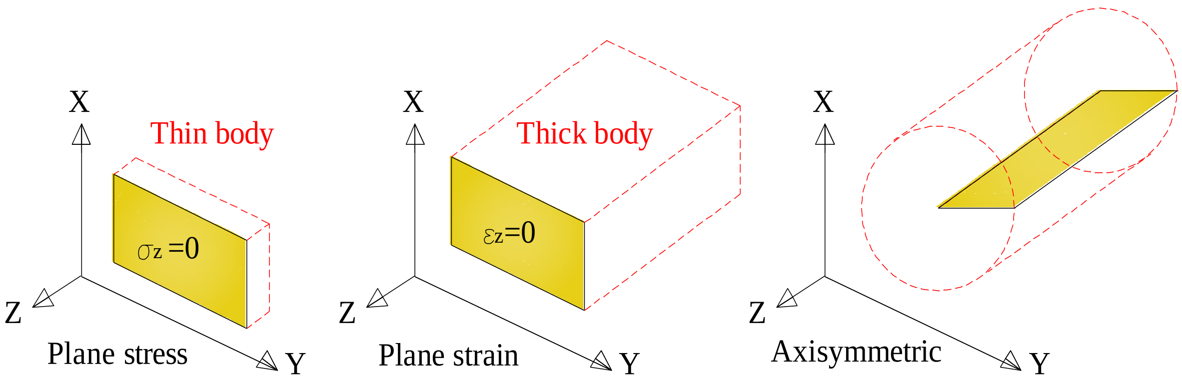

Figure 1. 2D approaches

The last approach the axisymmetric one allows the analyze of a 3-dimensional model which is rotationally symmetric about an axis. Although the input is 2D, the analysis results apply to the 3D problem [6].

2.1.2. Mesh discretization

The whole domain is discretized into a set of simple shapes or elements. The basic idea behind the FE method is to divide a region or a body into finite numbers (finite elements) which are connected between them by nodes (vertices). The elements can be in one, two or three dimensional space. For linear problems, a system of linear equations with the number of unknowns (nodal displacements, temperatures, etc.) equal to the number of nods needs to be solved [2].
Discretization plays an important factor which has a great impact on the results. Finest the mesh, the more accurate results, but more time consuming.
For 2D problems the finite elements are usually triangular or quadrilateral, having straight lines, and the nodes are located at the corners of the elements, or can have 6 respectively 8 nodes, the additional nodes being located at the midpoint of each side (fig.2).

The elements and nodes are numbered systematically, from left to right and from bottom to top. In a two dimensional problem there are two degree of freedom for each node: the u and v displacements (fig.3). In 3D problems for elements such as shell and beams have also rotational degree of freedom [5].

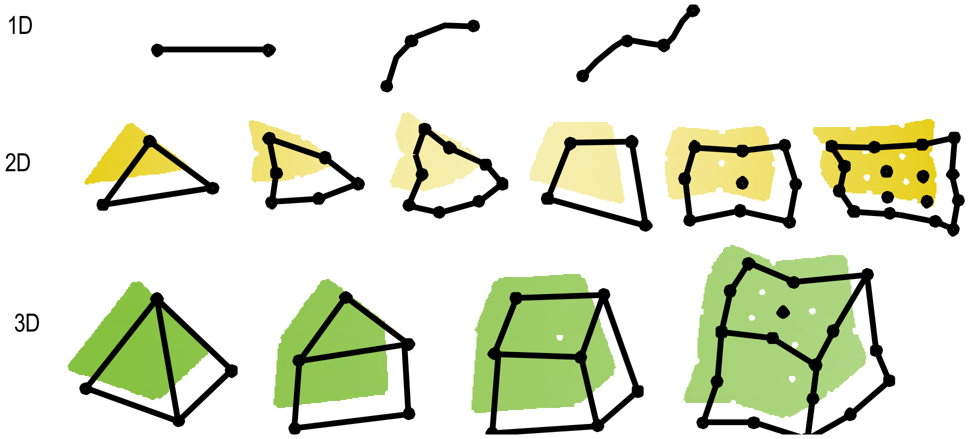

Figure 2. Typical finite element geometries [6]

Rules should be established to show how the variables vary over a established finite element. The displacements of nodes are solved by solving the global equations, followed by the secondary quantities such as stresses and strains. An appropriate variation principle needs to be used in order to derive the element equations [2]:

$\{K^e\}\{U^e\}=\{F^e\}$, where $\{K^e\}$ represents the element stiffness matrix.

For a certain element having the nodes noted with i,j,k the vector of displacement and nodal forces is (fig. 3):

$$\{U^e\}= \{U_{x,i}\ \ U_{y,i}\ \ U_{x,j}\ \ U_{y,j}\ \ U_{x,k}\ \ U_{y,k}\ \}^T$$
$$\{F^e\}= \{F^e_{x,i}\ \ F^e_{y,i}\ \ F^e_{x,j}\ \ F^e_{y,j}\ \ F^e_{x,k}\ \ F^e_{y,k}\ \}^T$$

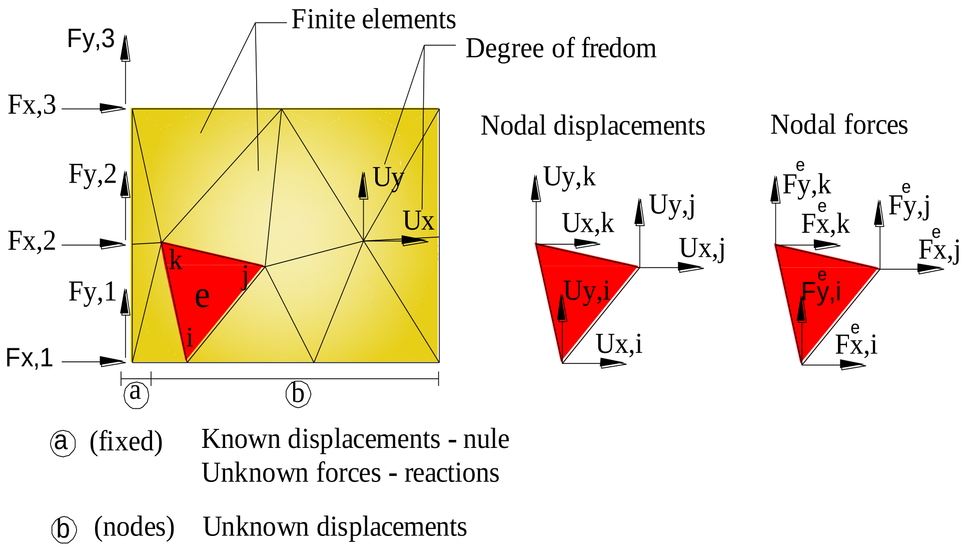

Figure 3. Discretization

The equations for each element are assembled into a set of global equations that will model the proprieties of the whole system. The global equation has the following expression:

$\{K\}\{U\}=\{F\}$, where$\{K\}$ is the global stiffness matrix

Where the following vectors represent the vector containing the unknown degree of freedom (nodal displacements) and the vector of all exterior nodal forces:

$$\{U\}= \{U_{x,1}\ \ U_{y,1}\ \ U_{x,2}\ \ U_{y,2}\ .....\ U_{x,NN}\ \ U_{y,NN}\ \}^T$$

$$\{F\}= \{F_{x,1}\ \ F_{y,1}\ \ F_{x,2}\ \ F_{y,2}\ .....\ F_{x,NN}\ \ F_{y,NN}\ \}^T$$ [5].

The displacements and the degree of freedom are calculated at each node, and at any point in the element the displacement are determined by interpolation the nodal displacements [5].

2.1.3. Application of boundary conditions

If boundary conditions are not available, then solutions cannot be obtained. The boundary conditions reflect the known values for certain primary unknowns [6].

In order to model the tunnel construction a variety of boundary conditions are required: boundary displacement conditions, surface traction, excavation of solid soil elements, construction of structural shell elements, hydraulic conditions, soil strata interfaces if any, tunnel lining. For the drained granular material condition an attention should be required for the hydraulic boundary conditions both during and after the excavation [3].

When speaking about boundary conditions, the term covers all possible additional conditions that can arise to fully describe a specific problem. According to the influence that the boundary conditions have on the global system of equation, they can be classified in: conditions that affect only the right hand side of the system equation, and the other group affects only the left hand side of the system equation. The former includes the line loads, surcharge pressure, loading conditions, forces from excavated and constructed elements. Where pressure boundary conditions are defined, firstly they must be expressed as equivalent nodal forces, and after being added to the right hand side of the equation. The latter includes displacement boundary conditions. Sufficient displacement conditions must be prescribed in order to retain the rigid body, and for 2D plane strain problem at least two nodes must have prescribed displacements in the x direction and one node in the y direction, or vice versa [2].

2.1.4. <u>Solution of global equation</u>

After the stiffness matrix was established on the boundary conditions were added, a mathematical simulation of equations is formed, which needs to be solved in order to find the values for the nodal displacements. In order to solve this enormous amount of equations, the matrix, which represents the system of equations defined by the model, is solved by different methods. The most used technique in the finite element method is based on Gaussian elimination. It is to be taken into account that for solving large problems, other methods can be used. The Gauss elimination is composed by two parts: the first one reduces the given system to a triangular form or echelon form (using elementary row operations), whereas the second uses back substation to find the solution of the above system [2].

The stiffness matrix is assembled for the whole mesh allowing to relate the displacements to the stresses. As a following step, the matrix is solved using the so-called “implicit” solution technique.

Where possible the results from the finite element analyses should be compared to an analytical solution [2].

2.1.5. Choice of element type

The behavior of each element is determined by the following aspects: family, degree of freedom (which is related to the element family), number of nodes, formulation and integration. The following figure (fig.4) shows the most common elements which are used in stress analysis. A major distinction between different elements families is the geometry [5].

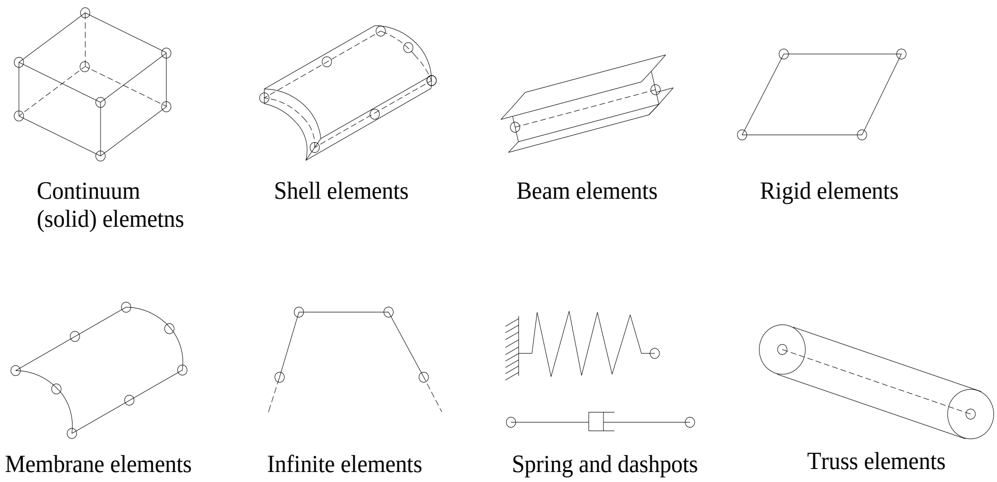

Figure 4. Different elements [5]

The degrees of freedom are calculated during the analyses and are fundamental variables. In the case of stress/displacement simulation the degrees of freedom are the translation and the rotation at each node. The latter one, rotational degree of freedom, applies in the case of shell, pipe and beam elements. Where heat transfer simulations are conduced, the degrees of freedom are temperatures at each node [5]. The element's formulation represents the mathematical theory used to define its behavior. In the Lagrangian, or material, description of behavior the element deforms with the material. In the alternative Eulerian, or spatial, description elements are fixed in space as the material flows through them. Eulerian methods are used commonly in fluid mechanics simulations [5]. When analyzing linear or complex nonlinear mechanical analyses, plasticity and/or large deformations, stress/displacement elements are to be used [5].

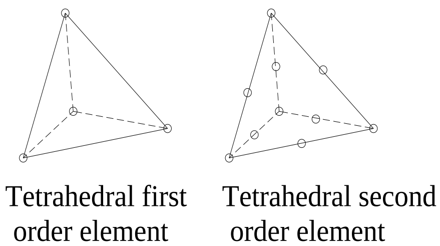

Figure 5. First and second order elements [5]

In the following the most common stress/displacements continuum and structural elements are presented. Several elements will be described which present an interest in modelling a tunnel excavation.

a.) Continuum elements

The continuum elements are the volume elements and can be used to model any shape of a model, and can be subject to any loading, boundary conditions and material proprieties. All three possible stresses: three normal and three shear should be taken into account. The continuum elements do not include structural elements such as beams, shells, membranes, and trusses; special-purpose elements such as gap elements; or connector elements such as connectors, springs, and dashpots. It is normally composed of a single homogeneous material. The continuum elements can be both stress/displacement and coupled temperature-displacement [5]. Although 3D solid elements can cover the other type of elements, this will not be a simplification of the problem, because the 3D models are the hardest to prepare and the most tedious to check for errors and demand for the most computer resources [5]. It is recommended when possible to use 1D (trusses, beams and frames) or 2D (2D solids and plates) structure instead of 3D solid ones [10]. When solid elements are used to represent a lining, a wide variety of constitutive models can be used. But one of the limitations of doing that is the lining thickness which is very thin related to the tunnel diameter and boundary distance. This means that in order to maintain the acceptable ratio between length and width a large number of elements is required [7].

b.) Structural elements

Since the geotechnical problems involve the interaction between soil-structure, in the finite element method analyses the structural components should be included as well. For the structural component using the 2D continuum elements is not the best option since the dimensions of structural components are much smaller compared to the overall geometry. As a result this solution will induce large number of elements and unacceptable aspect ratios. Moreover, this option cannot reveal the distribution of bending moments, axial and shear forces, unless additional calculations are conduced [2].

<u>Beam elements</u>

A beam is a structural member whose primary function is to resist transverse loads mainly through bending action. One of the dimensions is considerably larger than the other two, which is called longitudinal dimension or beam axis. The beam model can be based on Bernoulli-Euler theory and Timoshenko beam theory. The former is called also the classical beam theory, and assumes that the internal energy of a beam member is entirely due to bending strain and stresses. The model neglects transverse shear deformation and the cross sections (the intersection of the planes normal to the longitudinal dimensions with the beam) remain plane during the deformation and perpendicular to the longitudinal axis. On the other hand, the Timoshenko beam model incorporate a first order correction for transverse shear deformation effects and the cross-sections do not remain perpendicular to the longitudinal axis during deformation [6].

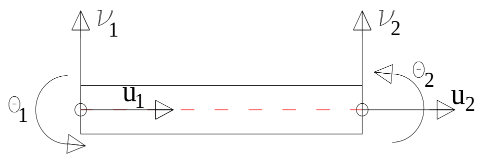

Figure 6. Degree of freedom for a 2D beam element [6]

The beam elements can be expressed in 1D, 2D and 3D. The one dimensional line elements also called truss elements, have only axial stiffness. They operate in terms of axial force, bending and torque. The beam element has the same number of nodes as the bar, but differs from this in the degree of freedom. The beam element has 3 degree of freedom at each node: the axial displacement u_i, the transverse displacement v_i and the rotation Θ_i. Two adjacent beam elements having a common node will have the same nodal unknown, meaning that they have the same displacement and rotations as they were be welded. A hinge in a beam allows the activation of releasing one or more degrees of freedom [6]. The beam element in the 3D space differs from the 2D version in terms of degree of freedom, each node having 6 degrees of freedom: 3 displacements and 3 rotations. An additional point is required to define the orientation of the principal axes of inertia (fig.7).

The element stiffness matrix is formed in the local 2D coordinate system first and then transformed into the global 3D coordinate system to be assembled [11].

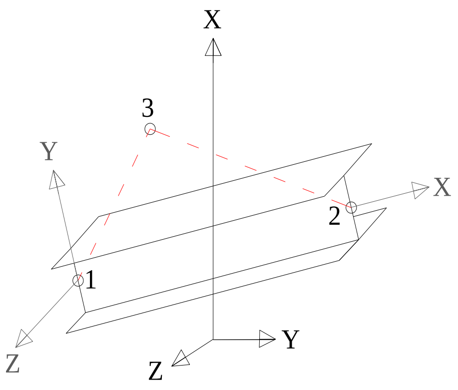

Figure 7. Degree of freedom for a 3D beam element [6]

The Timoshenko beam element will be more suitable to be used in significant shear deformations are expected, and generally in the cases where the height of the beam is greater than about 1/5th of the beam's span [6]. An advantage of the beam elements is that they have simple geometry and few degrees of freedom.

<u>Membrane elements:</u>

Membrane elements are useful elements for analyses which deal with soil-structure interaction problems and in plane stress, plain strain, axisymmetric and 3D analyses [6]. Are surface elements that transmit in-plane forces only, this means no moments, and have no bending stiffness [5].

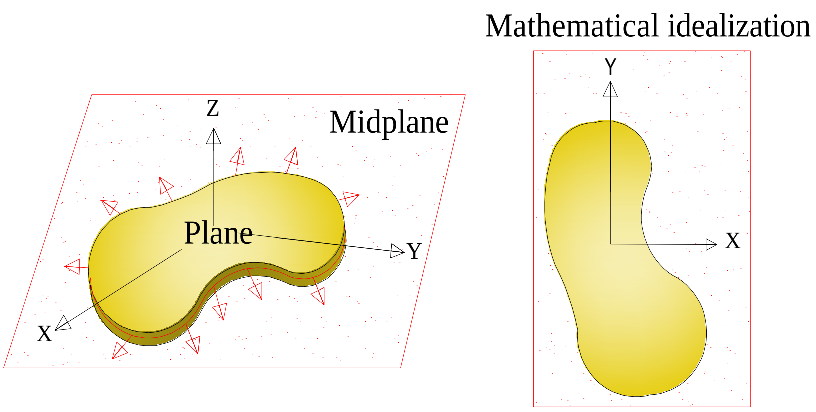

Figure 8. Membrane

For 2 D dimension, flat thin sheet of material are called membranes, plates and shells. The distance between the plate faces is called thickness and it should be 10% or even less than the shortest in-plane dimension. The midplane is situated halfway between the faces and the transverse direction and is normal to the midplane. The global axis z is orientated along the transverse direction and the x and y are placed in the midplane (fig.8). This is to say that the midplane equation is z=0. [6]
In 2D analyses: plain strain and axisymmetric ones, the membrane elements represent pin-ended membrane elements, which are capable to transmit forces tangential to the surface only (membrane forces). Differs from a spring because is curved and is treated in the same way as the other elements

in the analysis. Instead of using springs to express the same principle; using the membrane elements allow to express different behavior through constitutive laws and elasto-plastic formulation. In addition spring elements do not account for the effect of hoop forces in an axisymmetric approach [2]. The membranes which are plane elements follow several principals such as: all the loads applied to the element act in the midplane direction, and are symmetric with respect to the midplane, all supports are symmetric with respect to the midplane, the in plane displacements, stresses and strains can be taken to be uniform through the entire thickness [6].

Shell elements

Shell elements may be curved in space and can be considered as 2 and a half dimension –surface elements in 3D. They are appropriate to be used where the structure is in presence of the membrane stress combined with bending stresses, meaning that the forces in shells are membrane forces plus bending moments. The shell elements have five to six degree of freedom. Normally three displacements (one transverse and two in plane) and two rotations, and some shell elements have another rotation about the normal of the shell. Flat shell elements are used in plain stress or 3D modelling while the curved shells are used only in the 3D approach [6]. Zero thickness interface elements can suffer numerical instabilities when they have different stiffness from the adjacent continuum or from the structural beam elements. The shell elements are used to model structures which have one direction, namely the thickness, significant smaller than the other dimensions. The shells can be conventional or continuum ones. The former discretizes the body by defining the geometry at a reference surface, and have displacement and rotational degree of freedoms. The later, continuum shell elements, look-like three dimensional continuum models, but have similar behavior with the conventional shell elements in terms of kinematic and constitutive behavior (fig.9) [5]. Moreover these elements which have zero thickness present problems during mesh generation due to the adjacent nodes on each side of the element having the same coordinate [2].

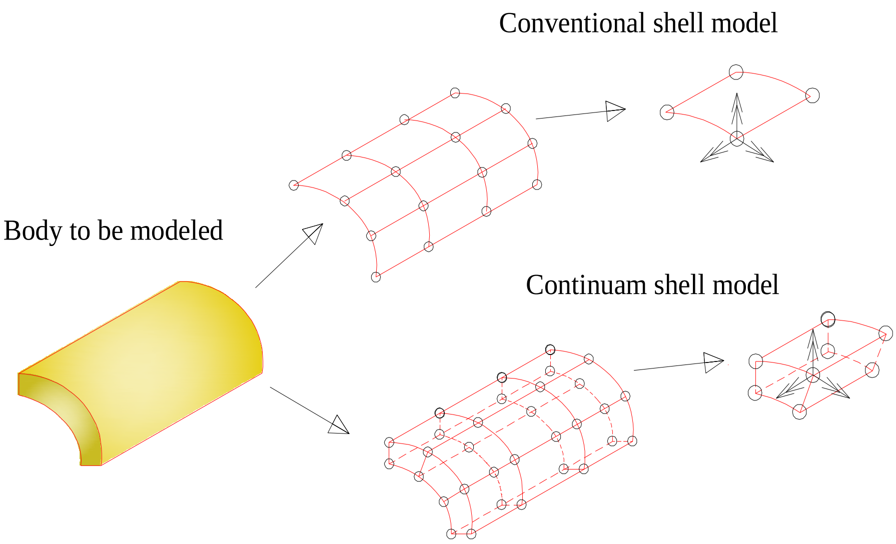

Figure 9. Conventional versus continuum shell elements [5]

The geometry of the shell is defined by its thickens, and load is carried by a combination of membrane action and bending moments. In addition no shell is completely free of bending. The bending appears near a point of loads, line loads, reinforcements, junctions, changes of curvature and supports. Plane elements are always plane and carry bending and twisting actions but no membrane actions. On the other hand the shell elements carry also the membrane actions and may be plane or curved in space [6].

c.) Connector elements

Springs

Spring elements are used to model actual physical springs as well as idealizations of axial or torsional components, can couple a force with relative displacement. They can also model restraints to prevent rigid body motion, and can be linear or nonlinear [5].

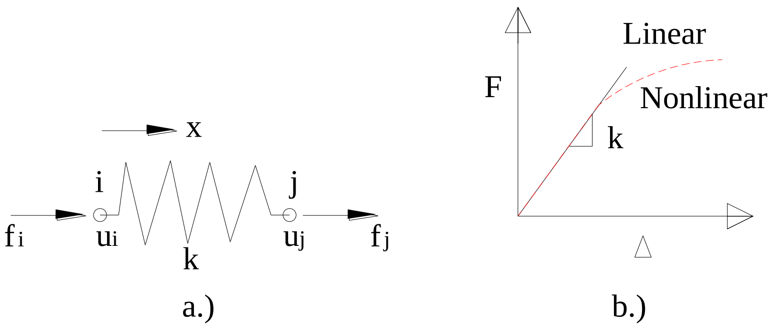

Figure 10. Spring a.) Spring element b.) Spring force-displacement relationship [11]

The spring element will have two nodes i and j, and the nodal displacements and nodal forces u_i, u_j and respectively f_i, f_j . The spring stiffness will be constant and is noted with the k value. The relation between the force and displacement is the following: $F=k\Delta$, where $\Delta = u_i - u_j$ [11].
They are assumed to be linear having a constant stiffness. They can be placed between two nodes in the mesh, to a single node, or being applied as a continuous spring along a part of the boundary of the mesh [2].

d.) Pore pressure elements

The finite element formulation can analyze the full drained problems and the fully undrained behavior. The former prescribes no change in the pore fluid, this implies that changes in effective and total stress are the same.
For clays with low permeability, the time (short term) used for installing the lining the soils parameters cam be considered as undrained. In the long term conditions, if the excess pore pressure is allowed to dissipate due to consolidation, will lead to changes in the effective stress inducing additional loads on the lining. Where a ground with high permeability is encountered, the hydrostatic pressure will not change in the case where the tunnel lining is watertight, or seepage forces will act on the lining if the tunnel acts as a drain [2].
Pore pressure elements are used to model fully or partially saturated fluid flow through deforming porous medium. These elements are used for soil and geostatic analysis. In addition to displacement degrees of freedom, the pore pressure elements have also pore pressure degrees of freedom. In the second-order elements, the pore pressure degrees of freedom are active only at the corner nodes. This type of elements are available only in the solid (continuum) element family [5].

e.) Coupled temperature-displacement elements

These types of elements are used in the problems where the stress analysis depends on the temperature solution and the thermal analysis depends on the displacement solution. This type of elements will have both displacement and temperature degrees of freedom. In modified triangle and tetrahedron elements the temperature degree of freedom are active at every node. In second-order

elements the temperature degree of freedom are active at the corner nodes [5].
These types of elements use linear or parabolic interpolation for the geometry and displacement. The temperature will use a liner interpolation. The elements used for coupled-displacement elements are from the following element families: solid, truss, shell, gap and slide elements [5].

3. Modelling tunnel excavation

3.1. General

The construction of a tunnel is a 3D problem, but since this is a time consumer, and needs powerful computers the 2D approach still dominates the modelling process nowadays. In a 2D analyze there can be used plain strain and axisymmetrical approach depending on the desirable results. The plain strain approach can be used when analyzing shallow tunnels and the effects of the surface settlements or to study the effects of tunnel construction on existing structures. In addition the plain strain analyzes can be used when transverse sections of multiple tunnels are modeled. The axisymmetric approach is widely used for deep tunnels, when the surface settlements are not of prime interest or when the face advance is analyzed. The two approaches can be used together in order to define the behavior of a 3D model, this process will be presented within this report [3].
When analyzing the tunnel effect, the initial conditions of the ground are of a major importance. The greenfield conditions should be inputted in the analysis. The distribution of vertical and horizontal effective stress of the ground is specified in terms of material unit weight, pore water pressure, and the coefficient of earth pressure at rest K_0. If more complex constitutive models are used, then in this step the initial void ratio and the hardening parameters may be required [3].
Another important step that needs to be taken into consideration is the simulation of previous construction activities that occurred in the surroundings of the site such as: excavation and construction of other structures. This step might involve several increments in the analyses, but can be neglected if the previous activities had a minimal disturbance in the soil conditions [3].
The following step, after defining the initial conditions, is the tunnel excavation. Although the modelling of the construction sequences considering the stress conditions is a 3D problem, it is reasonable to presume that is a plain strain problem. The zone of stress and strain distribution for a tunnel constructed in a homogeneous ground corresponds in a transverse direction to distances greater than 2-3 diameters from the point at which the tunnel ring is closed. At the advancing face the stress distribution is a 3D problem, but such models are complex to construct and difficult to analyze, in addition a lot of time is consumed using the 3D approach [3]. The effect of three-dimensional stress redistribution can be simulated using a 2D approach. For such 2D analyses the following techniques can be used: gap parameter method, convergent method, volume loss and progressive softening [3]. All of the above mentioned techniques allow a certain amount of deformation in the ground. The deformation is strongly dependent on the assumed degree of ground stress relief at the time of lining installation. The tunnels linings are composed by materials which are more uniform than the ground, and with a better understood behavior. In the case of spayed concrete, the proprieties of the material change according to the construction period. The “green” shotcrete (sprayed concrete) which is applied for immediate support, changes the strength and the stiffness during the construction period.
A specific attention should be paid to creep and shrinkage phenomenon which may significantly influence the stress and strains in the lining. The HME (Hypothetical Modulus of Elasticity) is often used to account these effects, as well as the stress distribution ahead of the face in the 2D models. For reinforced linings, the bars can be accounted in the model by modifying the bulk properties of the lining.

3.2. The gap model

The “gap” method was introduced in 1983 by Rowe et al. and assumes a predefined void which is introduced in the finite element mesh, representing the total ground loss expected. The ground parameter is a function of 3D deformation at the tunnel face, workmanship, lining geometry, soil remolding, tunnel machine shield, miss-alignment of the shield. When using different construction methods, the size of the void will vary. Is a useful method to predict the ground deformation using 2D finite element method or empirical correlations [7].

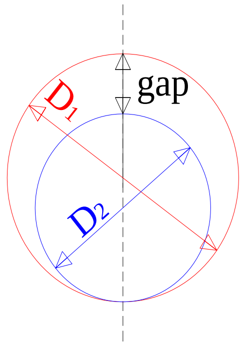

Figure 11. The gap method for modelling tunnel excavation [3]

The gap parameter represents the vertical difference between the initial tunnel position (D1) (before tunneling) and the final tunnel position (D2) at the crown, and is always fixed to zero at the invert. Through the advancing of the tunneling machine, the heading will move both radial and axially towards the face, but because of the weight of the lining the gap parameter will represent the distance at the crown and at the invert will have a zero value [7]. The void is placed around the final tunnel position, locating the soil boundary prior to excavation. The lining and the soil are treated as separate bodies, the analysis proceeds by removing boundary tractions at the perimeter of the opening, and monitoring the resulting nodal displacements. The soil-lining interaction will be activated at the node when the displacement of a node will indicate that the void had been closed, and the soil is in contact with the predefined lining position [3].
Ongoing research has analyzed the surface settlements for only undrained conditions, meaning that only immediate settlements have been used.
The gap parameter will be equal to:

$$GAP = G_p + u_{3D}^{*} + w$$

Where:
$G_p = 2\Delta + \delta$ represents the physical gap and is equal with the thickness of the tailpiece and the erection of the lining.
$u_{3D}^{*} = \frac{\delta_x}{2}$ represents the 3D effect of the of the face , and is the the magnitude of maximum axial intrusion at the tunnel face.
$w = 0.6\, G_p$ represents the workmenship parameter which is applied due to the occurrence of over-excavation or remolding of the soil adjacent to the tunnel [7].
The method of excavation consists in mechanical excavation followed by immediate erection of a tunnel lining within the tailpiece of a protective shield. The construction process can be simulated in the finite element analyses by deducing the tractions that would be acting around the surface of the tunnel prior to excavation and then followed by the removal of those tractions. The tail void left behind the tail skin can be represented by a diameter of the circular opening in the finite element method to be less large than the diameter of the lining [7].

3.3. The convergence method

This approach is a plain strain method which simulates a 3D effect. Since ground displacement occur prior to tunnel excavation, and the support can be installed with a time delay behind the excavation face, this approach can take into account this features. It can be spitted into: stiffness reduction method and load reduction method [3].

a.) Stiffness reduction method

This method implies the gradually reduction of material stiffness inside the periphery of the tunnel lining. In this manner the excavation process will be modeled accordingly. This procedure allows us to determine both the settlement ahead the face of the tunnel (prior to lining installation) and the settlements parallel to the tunnel. There is a certain amount of deformation before the lining is install therefore is important to determine the wall deformation prior to support installation [3].

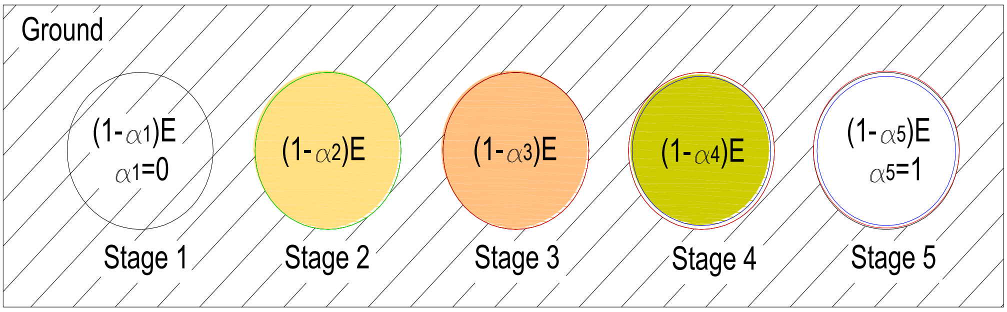

Figure 12. Stiffness reduction [4]

The material stiffness inside the periphery of the tunnel lining is going to be reduced by an α-factor resulting in a reduced Young modulus (1- αi)E. This is to say that for the material in the future opening, using a factor of α=0 represents the same stiffness as the one encountered outside the opening zone. On the other hand, when α=1 there is no stiffness for the material in the opening. Whether the material outside the tunnel has an initial state accounting the gravity and field stress, the material inside the future excavation zone has zero internal stress [4]. In order to calibrate the softening factor, an axisymmetric model is needed. The axisymmetric model provides the percentage of stiffness reduction that is to be used for each step of excavation. Since a tunnel posses rotational symmetry the tunnel can be easily modeled by using an axisymmetrical approach which reveals the deformation prior to lining installation. Before the installation of the shotcrete with a reduced Young's modulus (young shotcrete) the unloading factor changes from a value α_1=0 to α_2 value. Since the internal stiffness reduces, this will allow the ground to deform radially. When the α factor will change again, the shotcret's Young modulus will be changed for a value equivalent with the hardened concrete. In the stage 4, see fig.12, the inner lining is installed after the value of α_3 is changed to α_4. The stiffness is reduced the third time, in this situation the system composed by ground and shotcrete shell, moves radially inward.

b.) Load reduction method

This method follows the same steps as the method presented before. The difference here is that instead of reducing the stiffness, the pressure is reduced. In the initial state the internal pressure p_0 in the openings is equal with the external earth pressure [4]. The internal pressure is reduced with β-factor which can range between 0 and 1, where 0 is equal with the full internal pressure, and 1 corresponds to no internal pressure (fig.13).

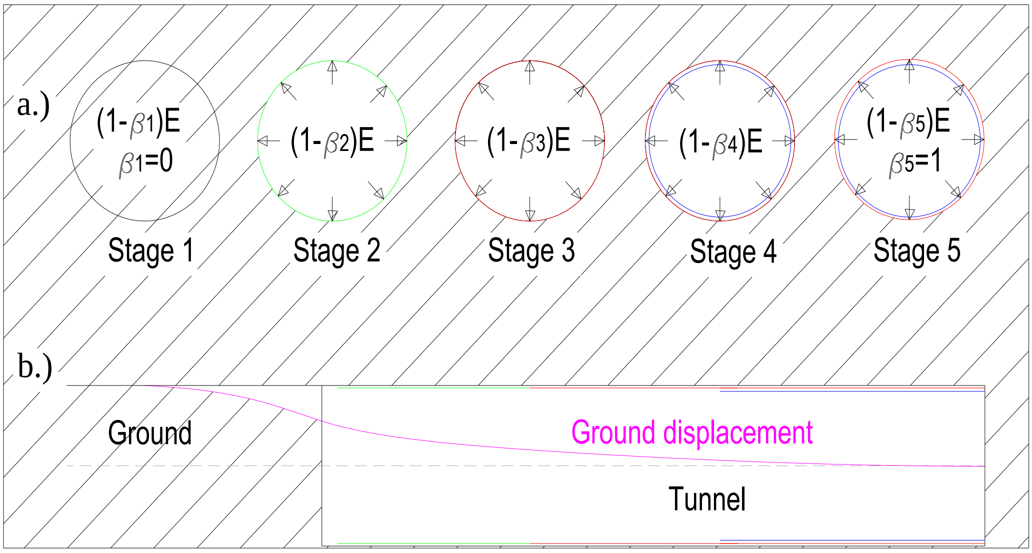

Figure 13. Load reduction method a.) Cross section b.) Longitudinal section [4]

The longitudinal section in the above figure presents the schematic ground displacement curve into the future opening before the tunnel is excavated and the displacement of tunnel perimeter before and after the initial support is installed.

3.4. The "volume loss control method"

This method is similar with the convergence method, but here the volume loss that will result on the completion of excavation will be prescribed, and for the case of earth pressure boring machine the out of plane component of the ground loss can be reduced [3].
This method is useful in back analyses of tunneling operations. The method assumes that the support pressure at the tunnel boundary is reduced in increments, and the volume loss which is generated can be monitored.

An alternative to the convergence method is for the program to calculate the equivalent nodal forces $\{F_0\}$ which represent the pressure exerted. The forces will be represented on what is to be the tunnel boundary by the soil to be excavated. This is linearly to the number of increments, over which the excavation takes place $\{\Delta F\} = \{F_0\} / n$ [7]. The $\{-\Delta F\}$ is equal but opposite to $\{\Delta F\}$, and is applies at the excavation boundary for each n increments of excavation. At each increment the volume loss can be monitored [7]. When the prescribed value is achieved the lining is installed. After lining is applied, the loading boundary condition $\{-\Delta F\}$ is still applied, introducing in the lining an initial stress. In some situations, depending on the lining stiffness, additional deformations and hence volume loss can occur, for this reason the lining should be installed before the prescribed volume loss is achieved in order to allow for additional value [8].

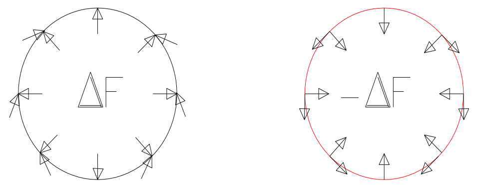

Figure 14. Modelling excavation of solid elements [3]

The analyze in soft clays may require the introduction of support pressure within the tunnel during excavation , because when lining is constructed during excavation, the subsequent unloading within the lining can result in a desire of the complete tunnel to move upwards, which may significantly reduce the surface settlements [7].

3.5. The "progressive softening" method

The method called the progressive softening is used normally for tunnels using a sequential excavation such as NATM (Sprayed Concrete Lining). In this case the tunnel can be composed by top heading, bench invert, and even side walls. The procedure of modelling is the same as discussed before for the convergence method; the only difference is that to each part of tunnel: top, bench, and invert individually softening percentage will be applied. [3]

4. Conclusions

This paper presented the basic parts in the finite element method, underlining the principle and the most used finite elements that can be encountered in a tunneling construction. Albeit, the 3D elements used in finite element method were described, the second part of the paper has presented the modelling approaches used for tunneling excavation only from a 2D point of view.
It is important to understand the behavior of each finite element and their limitation, in order to choose the right solution when modelling a structure, moreover in the geotechnical engineering which involves the interaction between structures and soil. Therefore, in the finite element method both the soil and the structure are necessary to be modeled when a tunneling construction is analyzed.

Several methods used for tunneling excavation such as gap parameter method, convergent method, volume loss and progressive softening were presented. The described methods reflect the conditions of construction and soil characteristics that can evaluate the stresses/strains induced in the lining and for any location of ground around tunnel and the surface settlements as well. Moreover, the results can lead for assessments of adjacent structures damage due to tunnel-induced differential settlements. The methods can be used for back-analyses of tunneling operations.
Is to be taken into account that although the numerical methods, generally speaking, have proved a good correlation with the monitoring data, there is some errors from the modelling point of view that might lead to poor predictions. Therefore, where possible the results from the finite element analyses should be compared to an analytical solution. In addition the behavior induced by tunnelling construction is influences by the standards of workmanship.

Acknowledgements

This paper was supported by the project "Improvement of the doctoral studies quality in engineering science for development of the knowledge based society-QDOC" contract no. POSDRU/107/1.5/S/78534, project co-funded by the European Social Fund through the Sectorial Operational Program Human Resources 2007-2013.

5. References

[1] Tunnel lining design guide. *The British Tunnelling Society and The Institution of Civil Engineers*, 2004.

[2] David M. Potts and Lidija Zdravkovic. *Finite element analysis in geotechnical engineering. Theory*. Imperial College of Science, Technology and Medicine ,1999.

[3] David M. Potts and Lidija Zdravkovic. *Finite element analysis in geotechnical engineering. Application*. Imperial College of Science, Technology and Medicine. Published by Thomas Telford Publishing, 2001.

[4] Heiko Mödlhamme. *Numerical Methods for Tunneling using ABAQUS and Investigation of Long-Time-Effects of the Shotcrete Shell and its Impact on the Combined Support System*. Master thesis, Montanuniversität University, 2010.

[5] ABAQUS/CAE User's Manual, Hibbitt, Karlsson & Sorensen, Inc. 2000.

[6] Dr. Ir. P. BOERAEVE. *Introduction To The Finite Element Method (FEM).* Institut Gramme – LIEGE, January 2010.

[7] K.M. Lee, R.Kerry Rowe, K.Y. Lo. *Subsidence owning to tunneling. I. Estimating the gap parameter*. Geotechnical Research Centre, Faculty of Engineering Science, The University of Western Ontaio,London, Canada, 1991.

[8] Par David Chapman,Nicole Metje,Alfred Stärk. *Introduction to Tunnel Construction*, 2010.

[9] *Ştefan Sorohan*. *Elemente finite* în *ingineria* mecanică . Catedra de rezistenta materialelor, Universitatea Politehnica din Bucuresti.

[10] G. R. Liu, S. S. Quek . *A Practical Course The Finite Element Method*. Department of Mechanical Engineering, National University of Singapore ,2003.

[11] Yijun Liu. *Lecture Notes: Introduction to the Finite Element Method*. CAE Research Laboratory.Mechanical Engineering Department. University of Cincinnati, USA, 2003.