Method for Calculating the Tunnel Lining of a Metro Station

Mirzakhid Miralimov1, а) and Abdullaziz Karshiboev1, b)

1*Tashkent State Transport University, 1 Temiryulchilar St., Tashkent 100167, Uzbekistan*

*a) Corresponding author:* [*mirzakhid\_miralimov@yahoo.com*](mailto:%20mirzakhid_miralimov@yahoo.com) *b) Laziz\_22\_92@mail.ru*

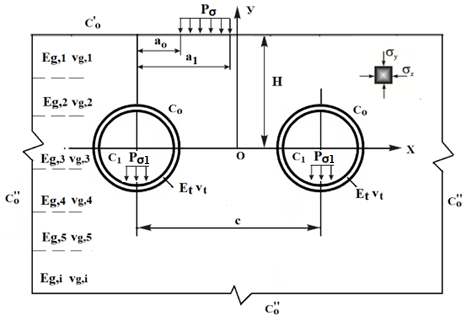
**Abstract.** This paper presents a practical method for analyzing the structural behavior of metro tunnel linings placed in different soil types. The method is based on the finite element approach and uses a two-dimensional elasticity model to study the interaction between the tunnel structure and surrounding soil. A combination of rod and isoparametric finite elements is applied to accurately simulate stress and deformation in the system. The calculation process is automated using Excel-based software integrated with TurboPascal programs. The proposed model allows for detailed evaluation of tunnel lining performance under static loads, including the effects of soil pressure and self-weight. Verification against analytical solutions confirms the method's accuracy and usefulness for engineering design and safety assessment of urban metro tunnels.

**Keywords:** Tunnel lining, metro station, finite element method, stress-strain analysis.

# INTRODUCTION

Today, the development of metropolitan infrastructure construction is considered the most effective means of solving transportation problems in modern megacities. In particular, developing new design solutions for subway tunnel linings and improving methods for calculating their structural integrity take leading positions in this field. In developed countries, including the USA, Switzerland, South Korea, Russia, China, and the Republic of Uzbekistan, the durability of transportation structures is ensured through the application of advanced modern design technologies and technical tools. In this direction, special attention is given to developing alternative design solutions for shallow metro tunnel linings, as well as modeling their interaction with various types of soil.

Let us consider a body in the XOY plane, represented by a two-dimensional region Ω and bounded by boundaries and . This region consists of two cavities enclosed by contours and . Within these contours, circular rings (reinforced concrete structures) are located, which possess physical properties different from the rest of the planar area. In turn, this remaining part may also be composed of media (various types of soil) with different physical characteristics.



**FIGURE 1**. Calculation schemes for the “ground-tunnel lining” system on flat terrain: *c* - distance between tunnels, *H* - depth, - external forces, and - boundaries of the flat area, С0 - outer boundary of the tunnel lining, С1 - inner boundary of the tunnellining, *Et, vt* - modulus of deformation and Poisson's ratio of the tunnel lining, *Eg,i, vg,i* - moduli of deformation and Poisson's ratios corresponding to the various (non-homogeneous) soil types surrounding the tunnel

As shown in Figure 1, the problem is reduced to a planar case in space where the domain is fixed and subjected to body forces, while forces and are applied at boundaries and , respectively. Let us assume that the deformations arising from these forces are small, for which the following basic equations are considered valid [1]:

1. Equilibrium Equations (Static Equations):

(1)

Here, *X̄* and *Ȳ* are considered volumetric forces. Alternatively, this can be expressed in matrix form.

Here (2)

2. Relationships between deformations and displacements (Geometric equations)

(3)

or here  (4)

3. Physical Equations (Hooke's Law)

 (5)

or

here

Thus, the equation of elasticity theory with boundary (limit) conditions can be written in the following form [3]:

Kinematic boundary conditions along the contour

Static boundary conditions on the contours and

Here, represents the displacement vector given on the contour, and are the stress vectors given on the contours, while and are the matrices of direction cosines.

We express from the second and third equations of system

Here (6)

Here, *D* - is the Hooke's law matrix that accounts for the physical properties of materials in various media.

When we substitute the value of into the first equation (2), we obtain the following differential equation of elasticity theory in matrix form for displacements:

or ,

(7)

Now,(5-6) in formulating the variational problem, we will use the differential equation (7) and its corresponding boundary conditions. Then, according to Lagrange's variational principle [4], the functional of the total potential energy *I* should attain a minimum value. We assume that the plane has unit thickness, and hypothesize that the displacement vector function *U(x,y)=[u(x,y),v(x,y)]* in the loaded state minimizes the functional of the total energy of the system in the following form:

(8)

In a limited way

here .

- potential energy of external forces, *П* - potential energy of internal forces, - thickness of a single layer of a flat region. The stationarity condition of the functional, together with the given boundary conditions, must be equivalent to the direct formulation of the problem. Then, based on the principles in [5], for a two-dimensional problem, the quadratic functional *F* can be written as follows:

(9)

Due to the unimodality of the problem of finding the functional minimum, by substituting equation (9) into the Euler-Poisson equation

(10)

Based on this, it is possible to obtain a direct formulation of the problem.

Maintaining such equivalence in the variational and direct formulation of problems allows for the application of a variational approach to solving problems using the finite element method. In this case, small changes in the displacement U led to a small change in the functional, denoted by δU. To obtain a useful expression for δU, it is necessary to integrate this expression by dividing it into parts. According to the necessary conditions for the functional minimum, the variation of the energy functional in actual displacements must be equal to zero. Since the total potential energy of a body consists of the sum of its deformation potential energy and the work done by external forces, relation (8) can be written as follows:

(11)

Now we apply the variational operation to the expression for the potential energy of deformation of a body:

Here, represents the specific potential energy of the body. It should be noted that in this case, the potential energy is a function of deformations, which in turn depend on the displacement field. Therefore, it is necessary to apply the rule of variation for a complex function with multiple variables. In this context, displacements are considered as independently variable functions, while deformations are treated as variable functions dependent on displacements.

It should also be emphasized that, from a formal mathematical perspective, the process of calculating the variation of a function is equivalent to calculating the differential of a function, meaning it is performed according to the same rules. As a result, we can obtain the following expression for *δW*:

(12)

Here, - represents the variation of the deformation vector. The product will be scalar, therefore

The following equation (12) can be expressed as:

(13)

Thus, the change in the potential energy of a body's deformation is expressed as follows:

The change in the work of external forces, taking into account that the displacement field is a variable independent function, is found through simple variation of the ratio:

(14)

Here, **-** represents the variation of the displacement vector.

Now let's substitute expressions (12) and (13) into formula (11) for the change in potential energy. We will then observe that the actual changes in the displacement vector provide a stationary value to the system's total energy functional:

(15)

To obtain a discrete model in the domain , we introduce a system of piecewise continuous basis functions and nodal displacements for the entire domain [5]. These basis functions are selected in such a way that they inherently satisfy the kinematic boundary conditions of the problem on the boundary :

(16)

Here, t=1......M represents the total number of nodes of the basis functions, which is equivalent to the total number of nodes in the system being modeled. During the modeling:

1. The given domain is divided into a finite number of elements using imaginary lines or points.
2. Finite elements are considered to be interconnected at a limited number of nodal points.
3. For each finite element, a function is given that precisely defines the sought-after characteristic (for example, displacements) within the element through the properties of its nodes.

According to this division, the volume and surface integrals in the expressions for potential energy and the work of external forces are equal to the sum of integrals over finite elements. Consequently, the integral relation (18) is expressed in the following form:

(17)

Here, - represents the total number of finite elements, while - denotes the number of finite elements that extend to the domain boundary. For basis functions within an element, the displacement field is determined using the following interpolation relationships:

(18)

The basic functions of an element's shape indicate the number of element nodes.

After substituting expression (17) into (16), performing some transformations, and calculating the overall sum, we obtain a system of equations in the following form:

(19)

From this, we obtain the following:

(20)

We obtain the solution of the system of finite element equations with respect to the general vector of nodal displacements using the following method:

Here, K represents the stiffness matrix of the system. Stress state analysis using the finite element method satisfies static equilibrium conditions and allows for the evaluation of stress changes arising from elastic properties, inhomogeneity, and alterations in geometric shapes. By determining the force at each nodal point based on the domain's self-weight and external forces at its boundaries, it is possible to solve a system of coupled equations based on the overall stiffness matrix with respect to the displacement of each nodal point. Once displacements are determined, it becomes possible to calculate stresses for each element.

*Computational methods, software, and algorithms.* Tunnel lining structures deform under the weight of their own mass, the pressure from overlying soil layers, as well as the horizontal and vertical external pressures exerted by the soil mass. [4].

According to the finite element method principle, the tunnel structure is divided into numerous small elements that are interconnected at nodal points. This methodology is called physical discretization of the structure. The calculation model should take into account, to the necessary degree, the design of the tunnel lining and the actual properties of the heterogeneous soils in the surrounding environment.

By applying formula (5) to each element and interconnecting them across all elements, we obtain a system of first-order equations that describe the state of the tunnel structure and the soil environment. Using this system, we determine the values of deformations and stresses within the elements based on the displacements at the nodal points.

We present the following analysis procedure for examining the stress-strain state of the "soil-tunnel lining" system using the finite element method [5]:

1) Creating a physical model for calculating a structure by dividing it into finite elements.

2) Construction of local stiffness matrices and external force vector matrices for elements.

3) Construction of the system's global stiffness matrix and external force vectors.

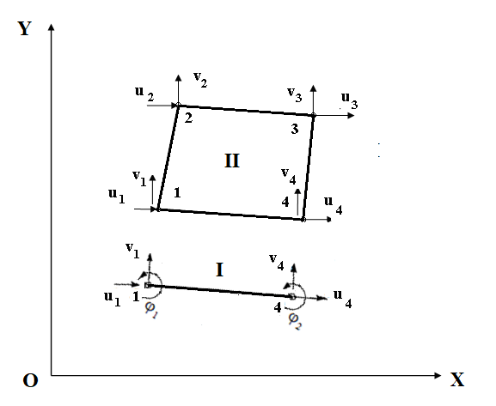
4) Solving a system of first-order equations that describe the state of tunnel lining structures and the soil environment.

5) Determine the stresses and deformations in elements using formulas (5).

In the computational method being developed for modeling the “ground-tunnel lining” system, we use two-node rod elements for the tunnel lining and four-node isoparametric finite elements for the ground mass (Figure 2) [5]. Following the general concept of constructing arbitrary isoparametric finite elements, we consider a natural local coordinate system that corresponds to the geometry of such elements and determine their stiffness matrices using the known expressions provided in [3]. For instance, the values of the stiffness matrix for a rod element can be presented as follows:

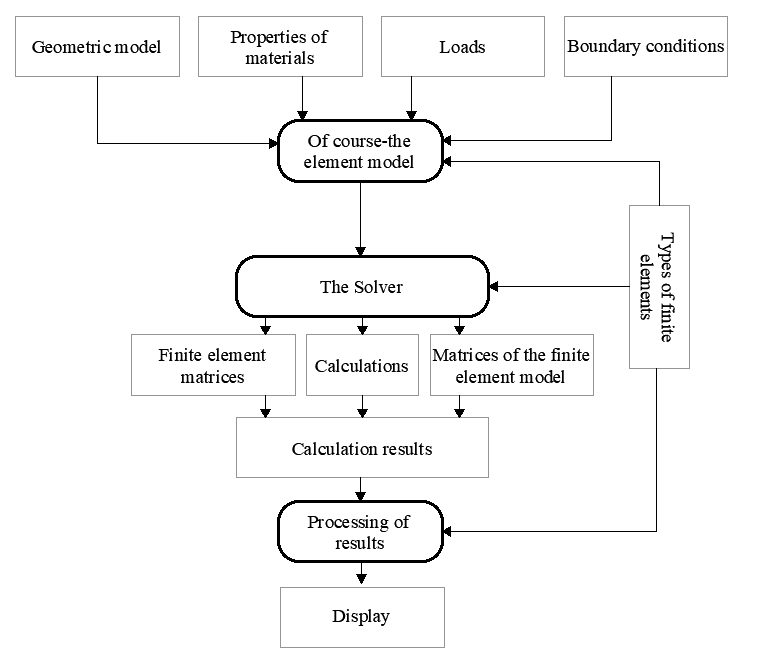
(21)

Here, E represents the elastic modulus of the rod material, I is the moment of inertia of the rod's cross-section, F is the cross-sectional area of the rod, and l denotes the length of the rod.

****

**FIGURE 2.** Finite elements used in modeling: I - rod element, II - isoparametric element

The algorithm for the method that takes into account the interaction between developed tunnel structures and the surrounding soil under static influences in real environmental conditions has been created in the “Program – Application” software. All processes are executed through computational modules accessed from the “Menu” table.



**FIGURE 3.** Algorithm for solving a problem in the programming package

In the developed programming package, problems are addressed through the algorithm-scheme presented in   
Figure 3. This scheme performs specifically defined tasks and corresponds to the expressions mentioned above. Once the system's differential equations are transformed into a system of algebraic equations, computational processes are initiated for each individual problem based on the developed algorithms.

In the problems under consideration, the boundary conditions for stress include the loaded or stress-free nodes of the field. Kinematic conditions encompass the constrained nodes. When performing calculations using the developed package, it is necessary to compile general information to input the finite element model of the structure.

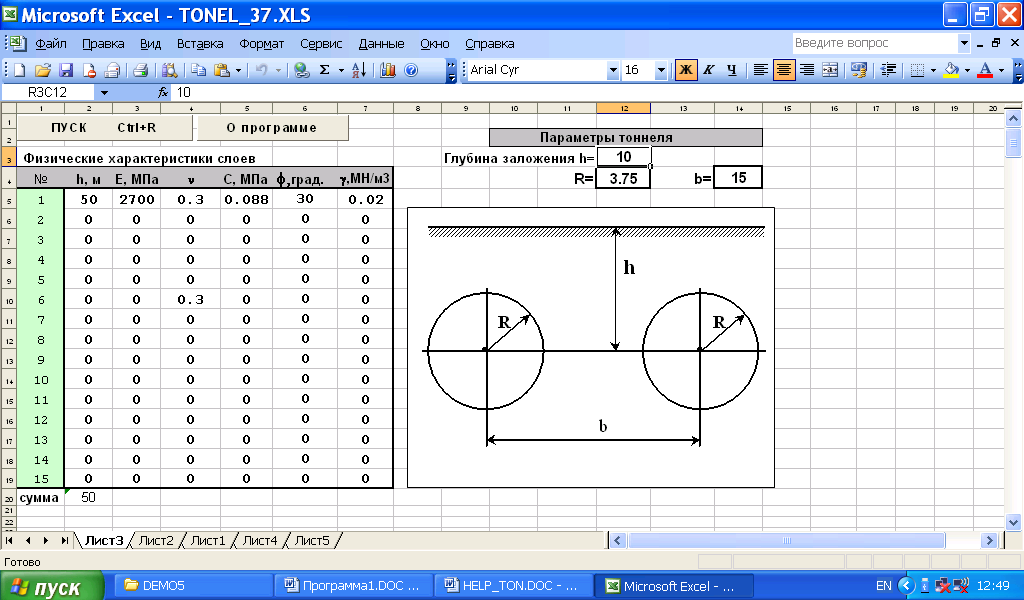
In this case, the node component codes can take two different values for boundary conditions: 0 and 1. If the condition value is 0, then these nodes are free from any constraints. However, if the value is 1, then boundary conditions are imposed on the specified degree of freedom for this node, and kinematic displacement conditions equal to zero are applied here.

The program is implemented as an “Excel-based application”, meaning that all calculation modules are managed from the Excel spreadsheet. The choice of such a system is explained by the following reasons:

1. Programs for finite element discretization, constructing the stiffness matrix, and solving systems of linear equations have been developed as separate executable files in TurboPascal-5.5 within the Microsoft Office environment. Excel has a built-in Visual Basic for Applications (VBA), which includes commands for launching files from the disk. As a result, all stages of managing the problem-solving process were implemented in Basic subroutines. In this process:
2. Excel includes functions for drawing and inputting tabular data, which are convenient for engineers;
3. All graphical and numerical results are entered directly into the table.
4. The printing of results is accomplished using built-in functions in the Excel program.
5. Upon completion of the calculations, the table contains all the initial data and information about the calculation results, thus serving as a ready-made report on the computation.

The calculation of tunnels consists of the following items:

1. When selecting the “Geological Column” menu item, a section displaying geological data appears in the table (Figure 4). Using this figure, it is possible to assess the accuracy of the entered information.



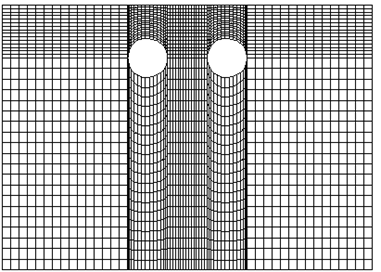
**FIGURE 4.** General view of the calculation program for the “soil-tunnel lining” system

2. The menu item “Grid Application” “Нанесение сетки” is designed for creating a finite element mesh. When this button is clicked, the “Tonl.exe” program launches from the disk - this program performs finite element division based on the given topology. This program utilizes computational geometry methods, particularly the method of dividing a curve into a specified number of segments, which is widely employed. The essence of this approach is as follows: two lines (broken or curved) are given, and they are divided into an equal number of segments. Then, straight lines connecting the corresponding points on these lines are drawn, and these lines are also divided into a specified number of segments [2].

The numbering of nodes and elements, consideration of boundary conditions, determination of physical properties for each finite element, and specification of loads arising from self-weight and surface forces are carried out using the “Tonl\_t.exe” program. Figure 5 illustrates the division of the structure into finite elements. Elements with different physical properties can be colored in various colors.

“Tonl\_sorcrd.exe” is a program that renumbers coordinates to ensure the minimum width of the ribbon. This process is performed as follows: the nodes are sequentially numbered along the long side of the structure, from one edge to the other.

“Tonl\_sort2.exe” sorts finite elements in the global coordinate system. This process is necessary because the correct method of constructing the stiffness matrix and solving the system of equations using the Gaussian method must be performed simultaneously. Therefore, finite elements should be arranged as follows: the node with the lowest number should always be used as the first node of each element, and the elements should be ordered so that the numbers of their first nodes are arranged in sequential order.



**FIGURE 5.** Division of the structure into finite elements

Communication between programs is carried out through data files.

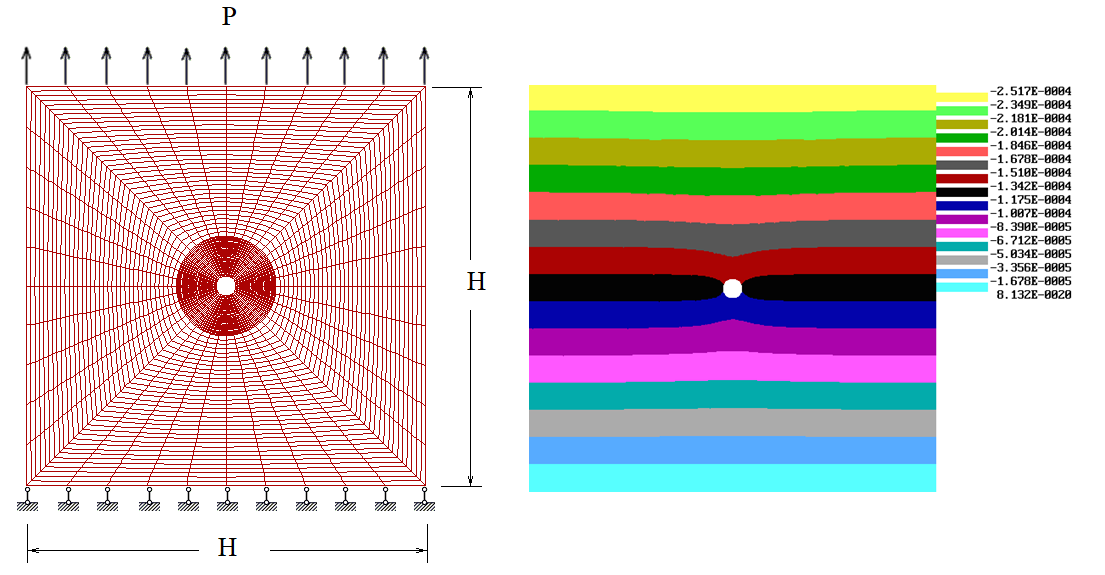
3. During the execution of the “Problem Solution” section, a stiffness matrix is constructed, a system of linear equations is solved, and the stresses in the soil are determined.

4. The section “Voltage isolines” is executed when it is necessary to construct isolines for σx ,σy ,τxy stresses.

5. The section “Force Diagrams in the Lining” is designed for constructing bending moment, transverse force, and normal force diagrams. A table of calculation results is compiled.

Solving the problem of stress concentration: We examine the tensile problem of a circular perforated structure located in a plane domain under a load of P=1 MPa (Figure 6, a). Its physical properties are as follows: E=300,000 MPa, v=0.35, L=40 m, d=2 m. The material consists of a metal plate. To model the loading process, isoparametric finite elements were used. σx and σy stresses were determined, and a vector field of principal stresses was constructed. The obtained results were found to approach the solutions presented in work [58] and the exact analytical solution [1] with a satisfactory degree of accuracy.

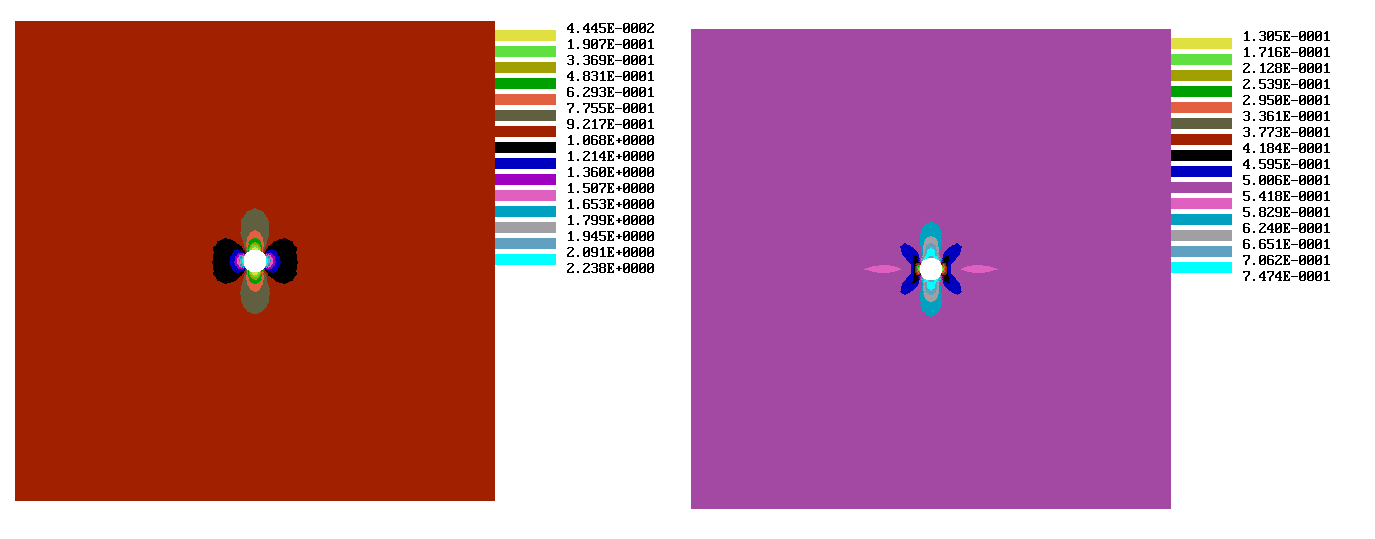
*а) b)*



**FIGURE 6.** Circular hole area (a) and results of vertical displacement field calculations (b)

Figures 6-7 show the vector field of the principal stresses and the isochromes of the maximum σ1 and σ3 stress values calculated at the centers of finite elements. Table 1 presents the values of the maximum stresses obtained at the edge of the plate hole in comparison with other solutions.

*а) b)*



**FIGURE 7.** Isochromes of maximum stresses, MPa: a - σ stress, b - τ stress

If the paper does not have the margins shown in Table 1, it will not upload properly.

**TABLE 1.** Margins and print area specifications

|  |  |  |  |
| --- | --- | --- | --- |
| Stresses around the hole's perimeter | Precise Solutions [1] | Regarding the solution according to | Solution obtained by the author according to |
| (МПа) | 1,000 | 0,996 | 0,996 |
| (МПа) | 3,000 | 2,899 | 2,899 |

A comparative analysis of the obtained results demonstrates the accuracy of the developed calculation methodology and allows for the conduct of further research.

**CONCLUSION**

The article presents a calculation method developed using advanced modern techniques of construction mechanics. This method takes into account the interactions between the structures of shallow circular metro tunnel linings and various types of surrounding soil.

Practical methods for modeling the deformation of structures and internal stresses arising under the influence of soil loads are presented, along with standards for determining their numerical indicators. In this approach, the soil is described based on the mechanics of continuous media - elastic laws, while the tunnel structure is considered as a planar system deforming together with the continuous media. This model takes into account the properties of each soil layer and the characteristics of the tunnel's permanent lining. For a possible variant of calculating the "soil-lining" system using finite elements, a calculation scheme for a planar area is provided. According to this scheme, the tunnel structure was represented by a system of planar elements with arbitrary shapes, having a thickness of 1.0 meter and geometric strength classification I.

The following analysis procedure is provided for calculating the forces in structural cross-sections under the influence of permanent and temporary loads. The following information about the algorithm can be specified:

To assess the reliability of the calculation method, model problems were solved as experimental objects and compared with exact solutions. In the calculation process, the plastic properties of the field material were conditionally disregarded, and stresses for this field were determined. Vector isochromes of principal stresses were constructed (at the centers of finite elements). The results showed that the obtained values satisfactorily approximated those obtained in practical work. This once again demonstrated the accuracy of the developed method.

A comparative analysis of the results demonstrates the applicability of the developed calculation method for studying the stress-strain state of shallow circular subway tunnel linings in various soil structures.

References

* 1. Z. Z. Wang and Z. Zhang, “Seismic damage classification and risk assessment of mountain tunnels with a validation,” Soil Dyn. Earthq. Eng. **45**, 45–55 (2013). <https://doi.org/10.1016/j.soildyn.2012.11.002>
  2. Chaotic Dynamics and Control of Systems and Processes in Mechanics, Proc. IUTAM Symp., Rome, June 2003, p. 459.
  3. M. Miralimov, Sh. Sh. Shojalilov, A. Karshiboev, and D. Usmanov, “Calculation method of reinforced concrete structures with account of nonlinear deformation of the material,” AIP Conf. Proc. **3045**, 030078 (2024). <https://doi.org/10.1063/5.0197797>
  4. M. Kh. Miralimov, S. Normurodov, M. Akhmadjonov, and A. I. Karshiboev, “Numerical approach for structural analysis of Metro tunnel station,” E3S Web Conf. (2021). [Scopus Author ID: 58578202800]
  5. P. P. Oreste, “A numerical approach to the hyperstatic reaction method for the dimensioning of tunnel supports,” Tunn. Undergr. Sp. Technol. **22**(2), 185–205 (2007). https://doi.org/10.1016/j.tust.2006.05.007