Application of the ADINA Software for Tasks of Educational and Technical Practice

Veronika Valašková, Jozef Vlček,
University of Žilina
Univerzitná 8215/1, 010 26 Žilina
Slovak Republic

Abstract—Development of the designing process in civil engineering brings the possibility for more complex analysis of the structures. Computational software can be helpful for regular design but also for identification and solving of more specified problems. Finite Element Method (FEM) became one of the most used numerical methods. Numerous commercial programs are based on this method. However, these programs are more complex and require skilled designer to fully utilize their potential. This article brings the lead to apply the FEM method implemented in commercial software ADINA to solve the problem of structural mechanics. It is necessary to connect the theoretical approach of FEM modeling and experience from real case problems. Skilled designer can calibrate the numerical model to obtain relevant outputs. However, certain cases still require a detailed and individual approach during designing that is verified by the in-situ monitoring and observation. Lead of the student to the flexible treatment of the design is one of the important tasks of the educational process. This involves the basic knowledge about appropriate analysis of the problem for effective utilization of the computational software for particular applications even for more complex solutions.

Keywords—ADINA, Bathe's method, FEM, Pasternak.

I. Introduction

The University of Žilina has an accredited field of study Civil Engineering at the Faculty of Civil Engineering. Within this field of study, students are taught at the bachelor's and master's level of education. As part of the engineering study, it is necessary to complete the compulsory subject: Finite methods of mechanics. The syllabus of this course is to learn to independently create numerical models in the Adina software and the subsequent analysis of the results obtained from these computational models.

ADINA is a commercial engineering simulation software program that is developed and distributed worldwide. The company was founded in 1986 by Dr. Klaus-Jürgen Bathe and is headquartered in Watertown, Massachusetts, United States. ADINA is numerical software based on Finite element method and is used in industry as well as in academia to solve structural problems, fluid problems, heat transfer, electromagnetic problems, fluid-structure interactions and thermomechanical problems [1].

Division according to numerical models in terms of their relation to time:

· stationary analyzes (static analysis)

· non - stationary analyzes (dynamic analysis).

A number of systems behave in such a way that when the ambient conditions change, a transition process takes place, during which the state parameters of the system change over time. After a certain time, the state stabilizes when the time change of the system parameters is zero. Stationary analysis solves precisely this final state, without which the transition process would be solved (taken into account). The solution is time independent.

In contrast, the goal of non-stationary analysis is to obtain the state parameters of the system as a function of time. The choice of the type of analysis is related to the goal of the solution and is decided by the problem solver. The terms static and dynamic analysis are introduced in the field of mechanics. A distinction is made between cases where inertial forces are and are not considered in the calculation. Sometimes the terms static and stationery are confused. Similarly, so stationary with the term dynamic. In most cases, this is true, but not always.

In the static analyzed of the building structure, the interaction of all its components must be taken into account: the superstructure, the foundation structure and the subsoil. The interaction of the foundation and the subsoil is often referred to as a contact task or interaction. In design practice, it cannot be required that the static calculation of the structure

be supplemented by an equally detailed analysis of the state of stresses and deformations in the subsoil. The real subsoil is idealized. Various computational models of the subsoil are created, which are to serve for a reliable design or assessment of the superstructure and foundation structure [2].

However, the finite element method is not the only method that can effectively solve the problems of structural mechanics. The boundary element method is one of the methods used in numerical software. For efficient and fast calculation, however, we leaned towards numerical software ADINA, which is based on the principle of FEM. As a first step in modeling, it is necessary to choose the right discretization of the numerical model. In terms of dimension, the model can be planar or spatial. Both of them have their advantages and disadvantages. Therefore, it's necessary to have a thorough idea of how the model should work at the very beginning of numerical analysis and what results we are interested in engineering practice. After mastering these problems, however, numerical modeling is a great benefit and we can use it to obtain adequate results for solving individual assignments.

II. INTRODUCTION OF NUMERICAL MODEL

They are based on the assumption that the subsoil behaves as an isotropic elastic body, referred to as an elastic half-space, characterized by a modulus of elasticity E and a Poisson's number v. The stresses and strains of such a body are investigated by the methods of mathematical theory of elasticity.

When solving the problem of the structure-subsoil interaction, at first glance, it seems most advantageous to use, regardless of the model of the superstructure used, the spatial model of the subsoil. From the point of view of technical possibilities of computer technology currently used in our country, but also from the point of view of considerable discrepancy between theoretical possibilities of the model and real possibilities of geotechnical survey of physical properties of subsoil, this approach is quite unrealistic. Therefore, certain hypotheses and simplifications are introduced to solve this problem in order to reduce the three-dimensional problem by one dimension. Such a two-dimensional model is called surface models, because all the deformation characteristics they work with relate to the surface of the subsoil, more precisely to the contact area between the subsoil and the structure [3,4].

A. Model of Pasternak

This subsoil model was described by Pasternak [5,6]. Model respects shear spread in the soil and the creation of the sinkhole in the vicinity of the building, Fig. 1.
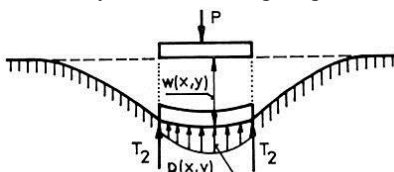

Fig. 1 Model of Pasternak

Model of Pasternak uses 2 constants C_1 and C_2. Local session constant C_1:

$$p_1(x,y) = C_1 \cdot w(x,y) \qquad (1)$$

Constant C_2 [N/m] of the local session:

$$t_{xz} = C_2 \cdot \gamma_{xz} = C_2 \frac{\partial w(x,y)}{\partial x}; \quad t_{yz} = C_2 \cdot \gamma_{yz} = C_2 \frac{\partial w(x,y)}{\partial y} \quad (2)$$

between the intensity of the transverse shear force t [N/m] and the bevel $\gamma = \partial_w / \partial_s$ of the subsoil surface, characterizes the shear distribution in the soil, Fig. 2.
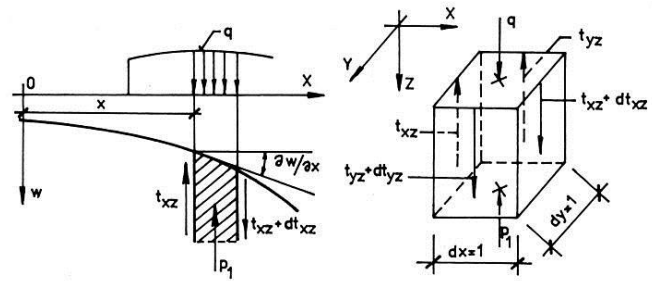

Fig. 2 The course of shear distribution in the soil

For the resulting substrate reaction, $p(x,y)$ is:

$$p(x,y) = p_1(x,y) + p_2(x,y)$$
$$= p_1(x,y) - dt_{xz}(x,y) - dt_{yz}(x,y) =$$
$$= C_1 \cdot w(x,y) - C_2 \cdot \frac{\partial^2 w(x,y)}{\partial x^2} - C_2 \cdot \frac{\partial^2 w(x,y)}{\partial y^2} \quad (3)$$

III. INTRODUCTION TO NUMERICAL SOLVER IMPLEMENTED IN ADINA

The finite element method (FEM) is one of the best known and most common used numerical methods in engineering practice, especially in construction tasks mechanics, currents, electromagnetism, thermodynamics, etc. Calculation method FEM belongs to the group of weight residues based on Galerkin's idea variational problem. This method is based on the replacement of infinite dimensional space v in the finite dimensional space v_n. Then we look for the v_n solution in the form functions with a combination of base functions to which the following applies:

$$f = \sum_{i=1}^{n} \alpha_i \phi_i, \qquad (4)$$
$$\int_\Omega L(u_n) \phi_i d\Omega = \int_\Omega p \phi_i d\Omega, \ i = 1, \dots n, \qquad (5)$$

where L is a linear operator and Ω is a boundary region. Then we obtain a set of linear equations for n unknown coefficients α_i

$$A_{ij}\alpha_{ij} = F_i, \qquad (6)$$

where A_{ij} is the stiffness matrix and F_i is the load vector [1].

One of the biggest problems of the FEM is the creation of a network (mash). In planar problems, triangular or quadrangular elements are used. In solving 3D spatial problems, spatial elements such as pyramids, blocks, tetrahedra are used.

A. Bathe's calculation method

Bathe's method (composite integration method) combines an explicit (Backward Euler) and an implicit (Newmark) computational method. As two independent calculation methods are combined, numerical stability of the calculation is ensured. The implicit method only is used to solve nonlinear problems, as the stability of the calculation is sufficient. In this case, the convergence of the calculation is sufficient, and it's necessary to obtain satisfactory numerical results. The essence of Bathe's calculation method is based on the division of the calculation into two-time sub-steps. In the first step the problem is solved using the Newmark method and in the second step by the Euler method. In Bathe's calculation method, the time step is divided into the exact half. In the modified Bathe method, it is possible to divide the time step variably. Due to problems with the stability of the calculation, it is possible to add most of the integration to the Newmark method and use the Euler method only as a numerical attenuation. The size of the substep is entered using the parameter γ, which is entered with rules greater than or equal to 0.5. Bathe's method is used to use the value $\gamma = 0.5$. For Newmark's method in calculating differential equations:

$$\{\dot{u}_{t_n} + \gamma.\Delta t\} = \{\dot{u}_{t_n}\} + [(1-\delta).\{\ddot{u}_{t_n}\} + \delta.\{\ddot{u}_{t_n} + \gamma.\Delta t\}].\gamma.\Delta t, \quad (7)$$

$$\{u_{t_n} + \gamma.\Delta t\} = \{u_{t_n}\} + \{\dot{u}_t.\gamma.\Delta t\}$$
$$+[(1-2\alpha).\{\ddot{u}_{t_n}\} + \{\ddot{u}_{t_n} + \gamma.\Delta t\}].\frac{(\gamma.\Delta t)^2}{2}, \quad (8)$$

where α and δ are Newmark parameters.

For Euler's method - the second substep applies:

$$\{\dot{u}_n\} = c_1.u_n + c_2.u_{n+\gamma h} + c_3.u_{n+1}, \quad (9)$$
$$\{\ddot{u}_{n+1}\} = c_1.\dot{u}_n + c_2.\dot{u}_{n+\gamma h} + c_3.\dot{u}_{n+1}, \quad (10)$$

where c_1, c_2 and c_3 apply:

$$c_1 = \frac{1-\gamma}{\gamma.\Delta t}, \quad (11)$$

$$c_2 = \frac{-1}{(1-\gamma).\gamma.\Delta t}, \quad (12)$$

$$c_3 = \frac{2-\gamma}{(1-\gamma).\Delta t}. \quad [1]. \quad (13)$$

IV. PROCEDURE FOR CREATING AND SOLVING A NUMERICAL MODEL

The solution to the task is a sequence of activities that may consist with the following steps:
· assignment of the task (investor, supervisor, teacher)
· analysis of the assignment and the meaning of the task
· selection of the theory to be used in solving the problem (bars, beam elements, shells, planar stress, etc.)
· choice of solution method (analytical, numerical)
· Compilation of a complete set of input data
· compilation of a computational model (simplification to reality)
· custom solution
· processing of results and their interpretation
· decision on next steps (end or continuation).
Therefore, in order for the task to be solved correctly, the solver must have the necessary knowledge, acquired both at school and through practice or self-study. To solve your own tasks, it is necessary to have a computing tool, which is currently the most common computer with the appropriate software. Probably the most important qualities in the creation of computational models are, first of all, creative abilities, experience, feeling and intuition [7].

V. COMMON MISTAKES WHEN CREATING A NUMERICAL MODEL

Everyone makes mistakes. Mistake is a reality that must be taken into account. The most serious mistake is the so-called qualitative error, when the given model does not have all the important features that are essential in terms of solving the problem. Such an error is most serious because it is difficult to detect. The model behaves correctly, but does not solve the problem we are looking for [8]. The opposite of the previous error is the complexity error, where the model contains, in addition to essential properties, also insignificant properties. In the best case, the time to solve the problem is extended, in the worst case, the solution of the problem may not be obtained at all. Another type of error is a quantitative error, where the model contains all the essential properties in terms of solving the problem, but their quantitative expression is insufficient to solve the problem at a given level of resolution. A common error that occurs during the implementation of the solution is a concretization error. In practice, this means that the results obtained in numerical analysis are in many cases not correctly and specifically incorporated into the tasks of engineering practice. The model contains all the essential properties, but in its own solution an error occurs, e.g., incorrect input of input data, error in inappropriate choice of used program. This type of error is easily detectable by checking the obtained results [9].

VI. CALCULATION OF THE NUMERICAL MODEL OF THE SUBSOIL IN ADINA

For the needs of numerical simulation, a computational model of a concrete block placed on a flexible Pasternak model was created. It is a simplified idealized planar model. This idealization is sufficient to obtain correct results with suitable accuracy. For more detailed results or a stress-response analysis of the subsoil or the concrete block itself, it would be necessary to create a spatial numerical model.

The advantages of the planar model are:
· simplicity of the numerical model satisfactory for static analysis
· geometry created through specific points in a plane
· time-saving creation of a numerical model
· simple and concise determination of boundary conditions
· possibility to choose simpler finite elements
· simplicity of presentation of obtained results

In this case, the concrete block simulates the load on the superstructure. Flexible subsoil and springs represent a real earth environment. Material parameters, stiffness and strength characteristics of specific materials were obtained from previous works of the authors. The superstructure represented

by the concrete block is meshed from the finite element 2D solid. The material parameters are as follows: $E = 4.2 \times 10^9$ Pa, $v = 0.33$ and $\rho = 2200$ kg/m^3. The subsoil consists of two elements, a beam element and a spring element. The stiffness of the springs is recalculated and represents a value $K = 39.01137 \times 10^5$ and damping $c = 1250$. The model is loaded by its own weight of the concrete block and it is a stationary analysis. We can state that this is a contact task. Figure 3 shows a computational model of the situation. The length of the subsoil is 10 meters, the concrete block is 4 meters length and 1 meter high. The springs are 0.2 m apart, and the finite element from which the concrete block is meshed has the same dimension.
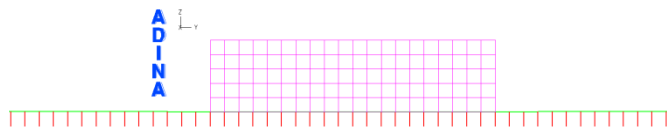

Fig. 3 Computational model in ADINA software

It is a simple static analysis in which it was used as a numerical solver based on Bathe's calculation method.

A. Script creation procedure - model creation

1. Input the basic coordinate 0; 0; 0
2. An extension of the line that defines the subgrade surface
3. Division of the line network into finite elements
4. Creating a group of elements (elementgroups)
5. Definition of materials
6. Creating a line element network
7. Copy the line
8. Assignment of boundary conditions to the line
9. Creation of spring elements between two lines
10. Creating the geometry of a concrete block
11. Creating a concrete block line
12. Division of a network of lines into finite elements
13. Creating a concrete block network
14. Entering boundary conditions for a concrete block
15. Defining the effect of self-weight of a concrete block
16. Start the calculation

The whole program script can be found in the appendix of this article.

VII. Discussion

The calculation of the numerical model took place in 2 seconds and the results are completed and in order.

A. Course of deformations in the z direction

From the given calculation model, the most interesting for us are the deformations that occur on the floor. Figure 4 shows the course of deformations and the bending line on a scale of 1: 100. The maximum deformation in z direction is located directly below the concrete block and has a value of 9.29275×10^{-3} m. The area that is not loaded but is affected by the concrete block is in this case 1.74 m on each side.
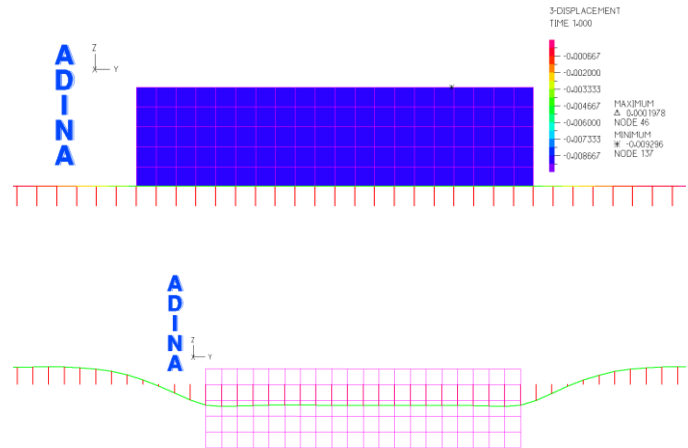
Fig. 4 Course of deformations

B. Course of internal stresses

Since this is a symmetrical problem, the stresses can be mirrored. Figure 5 shows the course of the internal stresses in the xx direction. It is clear from this figure, which the stress concentration occurs in the corners of the concrete block. The maximum value is 0.433 MPa. This value is safety and is out of the question affects the limit states of the concrete.
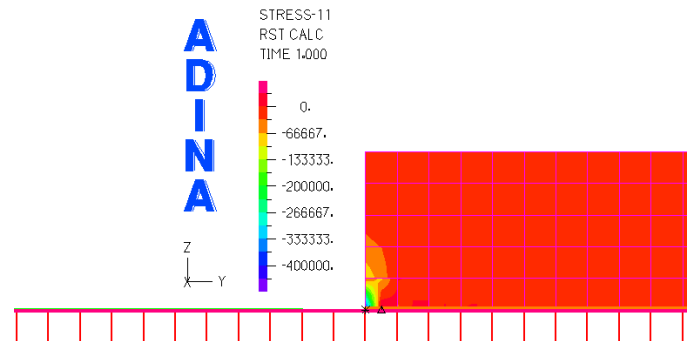
Fig. 5 Course internal stresses in xx direction

Figure 6 shows the course of the internal stresses in the yz direction. It is clear from this figure, which the stress concentration occurs in the corners of the concrete block. The maximum value is 0.400 MPa. This value is safety and is out of the question affects the limit states of the concrete.
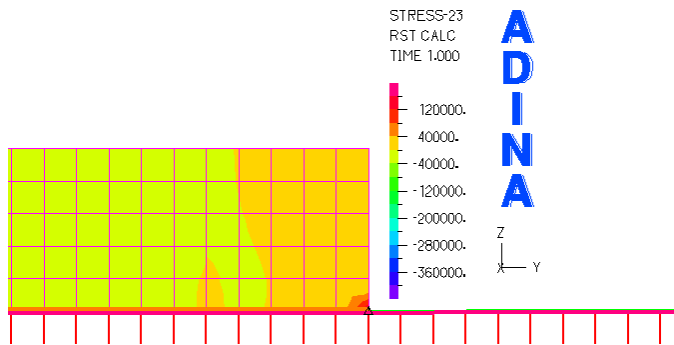
Fig. 6 Course internal stresses in yz direction

C. Course of internal forces

From the point of view of internal forces, the most interesting are the bending moment and contact forces on the ground arising on the subsoil. Figure 7 shows the bending moment in the y direction. The maximal bending moment occur directly under the outer corner of the concrete block and have a value 12.974 kNm. This value is also correct and in no case exceeds any limit state.
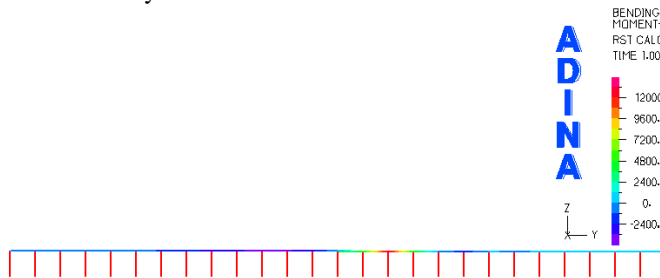
Fig. 7 Course of bending moments in y direction

Figure 8 shows the contact forces in the z direction. The maximal contact forces occur directly under the outer corner of the concrete block and have a value 98.161 kN. This value is also correct and in no case exceeds any limit state.

Fig. 8 Course of contact forces in z direction

VIII. Conclusion

The main aim of this article is to create the issue of numerical modeling. As finite methods of mechanics, but also the finite element method itself very actively contributes to solving the problem of applied mechanics. In principle, it is necessary to perform and follow all the steps for a correct and functional numerical model, which is possible to want to evaluate the correct results. Therefore, it is necessary to know how to decide the model as clearly as possible. ADINA

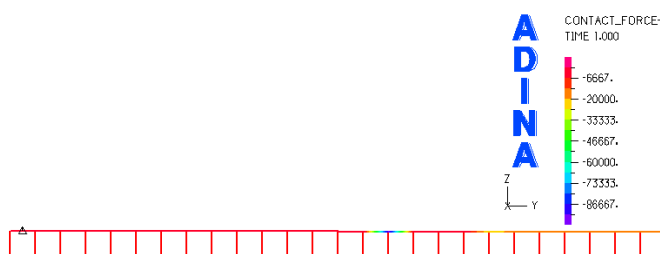enables the effective implementation of numerical calculations, replaces manual work and, within programming, forces the student to delve deeper into the essential problem and analyze the solution process in detail. Therefore, in other subjects, it is necessary to provide students with the widest possible space to take advantage of these opportunities.

Another benefit of this article is a simple demonstration of how to create a planar numerical model for the needs of static analysis of subsoil. This numerical model has to have well-defined boundary conditions so that the obtained results can be properly analyzed. As a further step and continuation of this numerical analysis, it would be appropriate to create a spatial numerical model, and compare the obtained results with each other. A large number of faculties and universities in Slovakia but also at the border, deal with the issue of numerical modeling with application to the field of study. Whereupon, it's very easy to find a large number of numerical models that are, in principle, very similar, but have different dimensions, geometry and numerical inputs. Other numerical software may be also used. This article also serves as a simple and brief explanation of the application of numerical modeling. Each student and reader can build their own numerical model according to the above procedure, verify the results and calibrate to their own proportions.

It is necessary to connect the theoretical approach of FEM modeling and experience from real case problems. Skilled designer can calibrate the numerical model to obtain relevant outputs. However, certain cases still require a detailed and individual approach during designing that is verified by the in-situ monitoring and observation. Lead of the student to the flexible treatment of the design is one of the important tasks of the educational process. This involves the basic knowledge about appropriate analysis of the problem for effective utilization of the computational software for particular applications even for more complex solutions.

Appendix

```
*
COORDINATES POINT SYSTEM=0
@CLEAR
1 0.00000000000000 0.00000000000000 0.00000000000000 0
@
*
LINE EXTRUDED NAME=1 POINT=1 DX=0.00000000000000
DY=10.0000000000000,
 DZ=0.00000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
SUBDIVIDE LINE NAME=1 MODE=LENGTH SIZE=0.200000000000000
*
MATERIAL ELASTIC NAME=1 E=4.200000000000000E+09
NU=0.330000000000000,
 DENSITY=2200.0000000000+00 ALPHA=0.00000000000000
MDESCRIP='Beam'
*
CROSS-SECTIO RECTANGULAR NAME=1
WIDTH=0.140000000000000,
 HEIGHT=1.00000000000000 SC=0.00000000000000
TC=0.00000000000000,
 TORFAC=1.00000000000000 SSHEARF=0.00000000000000,
 TSHEARF=0.00000000000000 ISHEAR=NO SQUARE=NO
*
```

```
EGROUP BEAM NAME=1 SUBTYPE=THREE-D DISPLACE=DEFAULT
MATERIAL=1 RINT=5,
 SINT=DEFAULT TINT=DEFAULT RESULTS=STRESSES
INITIALS=NONE,
 CMASS=DEFAULT RIGIDEND=NONE MOMENT-C=NO RIGIDITY=1,
 MULTIPLY=1000000.00000000 RUPTURE=ADINA OPTION=NONE,
 BOLT-TOL=0.00000000000000 DESCRIPT='NONE' SECTION=1,
 PRINT=DEFAULT SAVE=DEFAULT TBIRTH=0.00000000000000,
 TDEATH=0.00000000000000 SPOINT=2 BOLTFORC=0.00000000000000,
 BOLTNCUR=0 TMC-MATE=1 BOLT-NUM=0 BOLT-
LOA=0.00000000000000,
 WARP=NO ENDRELEA=ACCURATE
*
GLINE NODES=2 AUXPOINT=0 NCOINCID=ENDS NCENDS=12,
 NCTOLERA=0.00000000000000 SUBSTRUC=0 GROUP=1
MIDNODES=CURVED,
 XO=0.00000000000000 YO=0.00000000000000 ZO=1.00000000000000,
 XYZOSYST=SKEW
@CLEAR
1
@
*
LINE EXTRUDED NAME=2 POINT=1 DX=0.00000000000000
DY=0.00000000000000,
 DZ=-0.200000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
LINE EXTRUDED NAME=2 POINT=3 DX=0.00000000000000
DY=10.0000000000000,
 DZ=0.00000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
SUBDIVIDE LINE NAME=1 MODE=LENGTH SIZE=0.200000000000000
@CLEAR
2
@
*
PROPERTYSET NAME=1 K=3901137.00000000 M=0.00000000000000,
 C=1250.00000000000 S=0.00000000000000 NONLINEA=NO
*
EGROUP SPRING NAME=2 PROPERTY=1 RESULTS=FORCES
NONLINEA=NO,
 SKEWSYST=NO OPTION=NONE DESCRIPT='NONE'
PRINT=DEFAULT,
 SAVE=DEFAULT TBIRTH=0.00000000000000
TDEATH=0.00000000000000,
 6DOF-SPR=NO
*
SET EGROUP NAME=2
*
TRANSFORMATI TRANSLATION NAME=1 MODE=SYSTEM
SYSTEM=0,
 DX=0.00000000000000 DY=0.00000000000000 DZ=0.200000000000000
*
SPRING LINES DUPLICAT=NO NCOINCID=ALL
NCTOLERA=0.00000000000000,
 GROUP=2
@CLEAR
1 1 3 2 3 1 'DEFAULT' 'DEFAULT' 0.00000000000000 0.00000000000000 ,
'SAME'
@
*
FIXBOUNDARY TWO-D FIXITY=ALL
@CLEAR
2 0 'ALL'
@
*
FIXITY NAME=BEAM
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Y-ROTATION'
 'Z-ROTATION'
 'OVALIZATION'
@
```

```
*
FIXBOUNDARY TWO-D FIXITY=ALL
@CLEAR
2 0 'ALL'
1 0 'BEAM'
@
*
COORDINATES POINT SYSTEM=0
@CLEAR
1 0.00000000000000 0.00000000000000 0.00000000000000 0
2 0.00000000000000 10.0000000000000 0.00000000000000 0
3 0.00000000000000 0.00000000000000 -0.200000000000000 0
4 0.00000000000000 10.0000000000000 -0.200000000000000 0
5 0.00000000000000 3.00000000000000 0.00500000000000000 0
@
*
LINE EXTRUDED NAME=3 POINT=5 DX=0.00000000000000
DY=4.00000000000000,
 DZ=0.00000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
LINE EXTRUDED NAME=3 POINT=5 DX=0.00000000000000
DY=0.00000000000000,
 DZ=1.00000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
LINE EXTRUDED NAME=3 POINT=6 DX=0.00000000000000
DY=0.00000000000000,
 DZ=1.00000000000000 SYSTEM=0 PCOINCID=YES,
 PTOLERAN=0.00000000000000 DISTANCE=0.00000000000000
*
LINE POLYLINE NAME=3 TYPE=SEGMENTED
@CLEAR
5 0.00000000000000 0.00000000000000 0.00000000000000
6 0.00000000000000 0.00000000000000 0.00000000000000
8 0.00000000000000 0.00000000000000 0.00000000000000
7 0.00000000000000 0.00000000000000 0.00000000000000
5 0.00000000000000 0.00000000000000 0.00000000000000
@
*
EGROUP TWODSOLID NAME=3 SUBTYPE=AXISYMMETRIC
DISPLACE=DEFAULT,
 STRAINS=DEFAULT MATERIAL=1 INT=DEFAULT
RESULTS=STRESSES,
 DEGEN=DEFAUL FORMULAT=0 STRESSRE=GLOBAL
INITIALS=NONE FRACTUR=NO,
 CMASS=DEFAULT STRAIN-F=0 UL-FORMU=DEFAULT PNTGPS=0
NODGPS=0,
 LVUS1=0 LVUS2=0 SED=NO RUPTURE=ADINA
INCOMPAT=DEFAULT,
 TIME-OFF=0.00000000000000 POROUS=NO WTMC=1.00000000000000,
 OPTION=NONE DESCRIPT='NONE' THICKNES=1.00000000000000,
 PRINT=DEFAULT SAVE=DEFAULT TBIRTH=0.00000000000000,
 TDEATH=0.00000000000000 TMC-MATE=1 RUPTURE-=0 EM=NO
JOULE=NO
*
BODY SHEET NAME=1 LINE=3 DELETE-L=YES OPTION=LINE
SYSTEM=0,
 MERGE-IM=NO
*
SUBDIVIDE FACE NAME=1 BODY=1 MODE=LENGTH
SIZE=0.200000000000000,
 MAX-SIZE=0.00000000000000
*
GFACE NODES=9 NCOINCID=NO SUBSTRUC=0 GROUP=3
PREFSHAP=QUAD-DIRECT,
 BODY=1 COLLAPSE=NO SIZE-FUN=0 MIDNODES=PROJECT
METHOD=DELAUNAY,
 NLAYER=1 NLTABL=0 GEO-ERRO=0.00000000000000 SAMPLING=20,
 MIN-SIZE=0.00000000000000 AUTO-GRA=NO SIMULATE=NO,
 DENSITY=1.80000000000000 EVEN=SUM FLIP=NO REFINE=ALONG-
NORMAL,
 BLTABL=0 MESHING=MIXED PATTERN=AUTOMATIC 2DBLTAB=0,
 BLSHAPE=2.00000000000000
@CLEAR
```

```
1 1
@
*
LOAD MASS-PROPORTIONAL NAME=1
MAGNITUD=9.81000000000000,
 AX=0.00000000000000 AY=0.00000000000000 AZ=-1.00000000000000,
 INTERPRE=BODY-FORCE DESCRIPT='NONE'
*
APPLY-LOAD BODY=1
@CLEAR
1 'MASS-PROPORTIONAL' 1 'GROUP' 3 0 1 0.00000000000000 0 -1 0 0 0 ,
'NO' 0.00000000000000 0.00000000000000 1 0 'MID'
@
*
MASTER ANALYSIS=STATIC MODEX=EXECUTE
TSTART=0.00000000000000,
 IDOF=110011 OVALIZAT=NONE FLUIDPOT=AUTOMATIC
CYCLICPA=1,
 IPOSIT=STOP REACTION=YES INITIALS=NO FSINTERA=NO
IRINT=DEFAULT,
 CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=ATS
SOLVER=SPARSE,
 CONTACT-=CONSTRAINT-FUNCTION TRELEASE=0.00000000000000,
 RESTART-=NO FRACTURE=NO LOAD-CAS=NO LOAD-PEN=NO
SINGULAR=YES,
 STIFFNES=0.000100000000000000 MAP-OUTP=NONE MAP-
FORM=NO,
 NODAL-DE=" POROUS-C=NO ADAPTIVE=0 ZOOM-LAB=1 AXIS-
CYC=0,
 PERIODIC=NO VECTOR-S=GEOMETRY EPSI-FIR=NO STABILIZ=NO,
 STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE
FEFCORR=NO,
 BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO DEGEN=YES TMC-
MODE=NO,
 IRSTEPS=1 INITIALT=NO TEMP-INT=NO ESINTERA=NO
OP2GEOM=NO,
 INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z
OP2STR=DEFAULT,
 IRTIMES=0 OP2SAV=END
*
CGROUP CONTACT2 NAME=1 SUBTYPE=DEFAULT FORCES=YES
TRACTION=YES,
 NODETONO=NO FRICTION=0.00000000000000
EPSN=1.00000000000000E-12,
 EPST=0.00000000000000 DIRECTIO=NORMAL CONTINUO=YES,
 INITIAL-=ALLOWED PENETRAT=ONE DEPTH=0.00000000000000,
 OFFSET=0.00000000000000 OFFSET-T=CONSTANT CORNER-C=NO,
 TBIRTH=0.00000000000000 TDEATH=0.00000000000000 TIED=NO,
 TIED-OFF=0.00000000000000 HHATTMC=0.00000000000000,
 FCTMC=0.500000000000000 FTTMC=0.500000000000000 RIGID-
TA=NO,
 NORMAL-S=1.00000000000000E+11 TANGENTI=0.00000000000000,
 PTOLERAN=1.00000000000000E-08 RESIDUAL=0.00100000000000000,
 LIMIT-FO=1.00000000000000 ITERATIO=2 TIME-
PEN=0.00000000000000,
 CONSISTE=DEFAULT USER-FRI=NO DESCRIPT='NONE',
 CFACTOR1=0.00000000000000 CS-EXTEN=0.00100000000000000,
 ALGORITH=DEFAULT RTP-CHEC=NO RTP-
MAX=0.00100000000000000,
 XDAMP=NO XNDAMP=0.100000000000000 DISPLACE=DEFAULT,
 FRIC-DEL=DEFAULT GAP-VALU=0.00000000000000,
 EKTMC=0.00000000000000 GAP-COND=0 GLUEMESH=NO
*
CONTACTSURFA NAME=1 PRINT=DEFAULT SAVE=DEFAULT
SOLID=MULT BODY=0,
 ORIENTAT=AUTOMATIC MARQUEEB=0 DESCRIPT='NONE'
@CLEAR
3 1 1
@
*
CONTACTSURFA NAME=2 PRINT=DEFAULT SAVE=DEFAULT
SOLID=MULT BODY=0,
 ORIENTAT=INPUT SENSE=1 MARQUEEB=0 DESCRIPT='NONE'
@CLEAR
1 -1 0
```

```
@
*
CONTACTPAIR NAME=1 TARGET=2 CONTACTO=1
FRICTION=0.00000000000000,
 TBIRTH=0.00000000000000 TDEATH=0.00000000000000,
 HHATTMC=0.00000000000000 FCTMC=0.00000000000000,
 FTTMC=0.00000000000000 NX=0 NY=0 NZ=0 OFFSETCO=BOTH,
 EKTMC=0.00000000000000 GAP-COND=0
*
```

REFERENCES

[1] ADINA R & D "Theory and Modeling Guide", vol. I, ADINA, ADINA R & D, Inc., Watertown, MA, USA, 2012.

[2] L. Frýba, Vibration of Solids and Structures Under Moving Loads" ACADEMIA, Praha, Nordhoff International Publishing, Groningen, 1972.

[3] M. Reza Eslami, "Finite Elements Method in Mechanics", Springer, vol. 216, 2014, pp. 119-156.

[4] B. Zhu, "The Finite Element Method: Fundamentals and Applications in Civil, Hydraulic, Mechanical and Aeronautical Engineering", Wiley, 2018

[5] P.L Pasternak, "Osnovy novogo metoda rozčota fundamentalov na uprugnom osnovanii pri pomočši dvuch kueficientov posteli" GIT, Moskva, 1954.

[6] M.R. Madhav, H.B. Poorooshasb, "Modified pasternak model for reinforced soil", Mathematical and Computer Modelling, Elsevier, vol 12, iss. 12, 1989, pp. 1505-1509.

[7] S.L. Barbour and J. Krahn. "Numerical modelling–prediction or process", Geotechnical News, vol. 22, iss. 4, 2004, pp. 44-52.

[8] P. Mauro and M. Andres, "Common mistakes about numerical representations", Behavioral and Brain Sciences, vol. 32, iss. 3-4, 2009, pp. 346.

[9] M.N. Sadiku, S.M. Musa, K. Kirby, "Common mistakes made by students", IEEE Potentials, vol. 31, iss.4, 2012, pp. 9-16.

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

Jozef Vlček elaborated the theoretical part.
Veronika Valašková is the author of the programs.

Sources of Funding for Research Presented in a Scientific Article or Scientific Article Itself

Creative Commons Attribution License 4.0 (Attribution 4.0 International, CC BY 4.0)