International Journal of GEOMATE, July, 2018 Vol.15, Issue 47, pp.103-110
Geotec., Const. Mat. & Env., DOI: https://doi.org/10.21660/2018.47.3576
ISSN: 2186-2982 (Print), 2186-2990 (Online), Japan

INTERACTION OF NONLINEAR NUMERICAL MODEL OF SFRC SLAB AND NONLINEAR NUMERICAL SUBSOIL MODEL

*Jana Vaskova and Radim Cajka

Faculty of Civil Engineering VŠB-Technical University of Ostrava, Czech Republic

*Corresponding Author, Received: 16 May 2017, Revised: 20 Dec. 2017, Accepted: 20 Jan. 2018

ABSTRACT: Experimental measurements of the concrete slab in interaction with subsoil are compared with numerical analysis based on Finite Element Method FEM. Experimental measurements are conducted with using an experimental device constructed at the Faculty of Civil Engineering VSB – Technical University of Ostrava. In the article, several types of nonlinearities enter into FEM analysis. Subsoil-structure interaction requires an iterative solution procedure - and therefore structural nonlinearity. Material nonlinearity was used in the numerical model of the slab. This model also allows the creation and development of cracks in concrete - similar as during the experimental load test. Material nonlinearity was also used in the numerical model subsoil - to ensure of apposite subsoil behavior. The purpose of this paper is to compare resulting deformation of the slab with values observed during experimental loading test. It was concluded that the good agreement between the experimental results and numerical simulation was observed.

Keywords: Foundation structure, Experimental measurement, Soil – structure interaction, Interaction models, FEM calculation

1. INTRODUCTION

With the beginning and subsequent development of computer technology, numerical methods have also been used to solve the interactions between base and subsoil. The most well-known numerical methods are Boundary Element Method (BEM) and Finite Element Methods (FEM). Currently (and also in the past) many experts have been dealt with a numerical model based on Finite Element Method (FEM) – eg. Zienkiewicz a Taylor [1], Králik, Jendželovský [2], Kolář, Němec [3]. The most widely used software, which solves interaction tasks, are Scia Engineer, Ansys, Trimas, MKPINTER, RFEM an RF-Soilin and Plaxis. Despite much commercial software and non-commercial software, a computational model was not still found to accurately capture the behavior in the interaction of the foundation structure and subsoil. The difficulty of designing an accurate static design of foundation structures lies in several aspects, for example, the influence of physical-nonlinear behavior of the structure or the uncertainty associated with the description of the properties and behavior of the foundation soil, because it is natural material and its properties cannot be determined unambiguously. Despite these uncertainties in input parameters, Finite Element Method (FEM) is an excellent computing method for a whole range of tasks, including interaction tasks. The Finite Element Method (FEM) allows the solution of a so-called complete interaction system including "subsoil - foundation structure - upper structure". Complete interaction system is more demanding to compute, but provides more accurate results than a simplified interaction system "subsoil - foundation structure". However, the question is, how accurate are input data and parameters entering the calculations. Both 2D and 3D finite elements can be used to solve of numerical analyzes of the complete interaction systems and also simplified interaction systems.

2. EXPERIMENTAL LOADING TEST

A combination of experimental tests, laboratory tests, field tests and numerical modeling is optimal to obtain reliable results from analyses of subsoil-structure interaction. Combination of all the foregoing approaches was also used in this paper. In 2016, experimental loading test of concrete slab was realized using the experimental facility built in the campus of Faculty of Civil Engineering, VŠB - the Technical University of Ostrava in the Czech Republic. For this experimental loading test, numerical analyses have been done in the program ANSYS, PLAXIS, SCIA ENGINEER, MKPINTER, all based on the finite element method (FEM).

Fig. 1 Experimental equipment, called Stand

Fig. 2 Experimental loading test

The concrete was a mixture of C 25/30 XC2. It was concrete with consistency S3, with the limit value of the minimum content of cement (CEM I 42.5 R VL) of 280 kg/m^3 and for the maximum grain size to 16 mm. A modulus of elasticity of concrete was obtained with laboratory tests, which were conducted in the day when the slab was loaded. The modulus of elasticity was E = 19.75 GPa. Poisson's ratio of concrete was $\mu = 0.2$. The compressive strength of concrete f_c = 20.03 MPa was also obtained by laboratory tests. The slab dimensions were 2.00 x 2.00 x 0.15 m. The slab model was loaded by the hydraulic press in the slab center. The load area was 400 x 400 mm. The subsoil had these properties – Poisson coefficient $\mu = 0.35$, modulus of deformability E_{def} = 12.5 MPa. Load step was 25 kN/30 minutes. Loading was carried out up to the load during which the experimental load test of the slab was failed (345 kN). A load of 345 kN applied to the loading area (400 x 400 mm) was also used in numerical analyses.

3. NONLINEAR NUMERICAL MODEL OF INHOMOGENEOUS HALF-SPACE

The soil is naturally and heterogeneous material. Because of it, its properties differ from the idealization of linear elastic, isotropic and homogeneous material. That's the reason why the calculated values of settlement differ from the actual settlement. This can be appropriately dealt with using the inhomogeneous half-space [4]. In inhomogeneous half-space, the modulus of deformability of the subsoil varies continuously with the increasing depth. In the inhomogeneous half-space, there is a different concentration of the vertical stress in the axis of the foundation than that in the homogeneous half-space. The formula (1) based on the minimum of deformation work was derived by Frölich [5].

$$E_{def} = E_0 z^m \qquad where \qquad m = \frac{1}{\mu} - 2 \qquad (1)$$

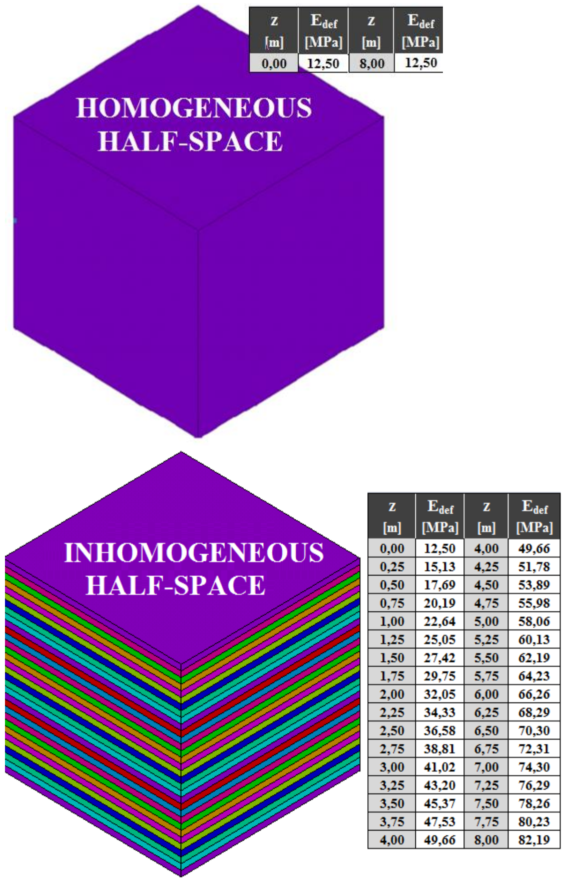
where

E_0 – modulus of deformability at the surface

z – z-coordinate (depth)

m – coefficient depending on Poisson's ratio μ

Value of modulus of deformability E_{def} continuously increases with increasing depth according to the formula (1) in an inhomogeneous half-space. When the subsoil model is created as an inhomogeneous nonlinear continuum, results are significantly less affected by the selected geometry subsoil model than subsoil model created as a linear homogeneous continuum. Inhomogeneity of the numerical model of the subsoil was created by dividing into separate layers (Fig. 3), in which the modulus of deformability grown in layers.



z [m]	E_{def} [MPa]	z [m]	E_{def} [MPa]
0,00	12,50	8,00	12,50

z [m]	E_{def} [MPa]	z [m]	E_{def} [MPa]
0,00	12,50	4,00	49,66
0,25	15,13	4,25	51,78
0,50	17,69	4,50	53,89
0,75	20,19	4,75	55,98
1,00	22,64	5,00	58,06
1,25	25,05	5,25	60,13
1,50	27,42	5,50	62,19
1,75	29,75	5,75	64,23
2,00	32,05	6,00	66,26
2,25	34,33	6,25	68,29
2,50	36,58	6,50	70,30
2,75	38,81	6,75	72,31
3,00	41,02	7,00	74,30
3,25	43,20	7,25	76,29
3,50	45,37	7,50	78,26
3,75	47,53	7,75	80,23
4,00	49,66	8,00	82,19

Fig. 3 Homogeneous half-space model and inhomogeneous half-space model, ANSYS

4. NUMERICAL ANALYSES IN ANSYS

The 3d numerical model was created in ANSYS. The input data were taken over the experimental loading test. Subsoil model was created as a three-dimensional model using 3D finite element SOLID 45. Subsoil model was created as homogeneous half-space and also as inhomogeneous half-space, see Fig. 3. Based on a parametric study [6], [7], [8], an area representing the subsoil was 8.0 x 8.0 x 8.0 m. Physical nonlinearity associated with material properties was used. The nonlinear material model was performed by Drucker-Prager model (Fig. 4).

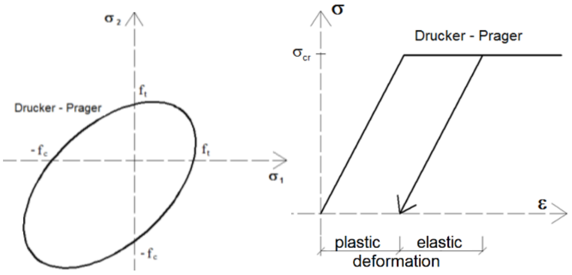


Fig. 4 Drucker-Prager, Nonlinear material model

SOLID 65 was used for spatial modeling of the experimentally loaded slab. It was necessary for the numerical model to take into account the influence of nonlinearities and cracks in concrete (occurring during the continuous loading of the concrete slab). These were taken into account by the use of the finite element SOLID 65. SOLID 65 enables non-linear calculation of concrete structures by Willam - Warnke criterion (Fig. 5). This model of the behavior of quasi-brittle material considers both tensile damage (forming cracks) and pressure damage (crushing the material). Fracture properties of concrete are also described eg. [9].

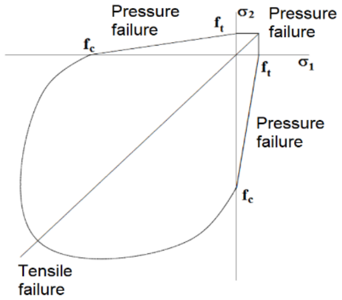
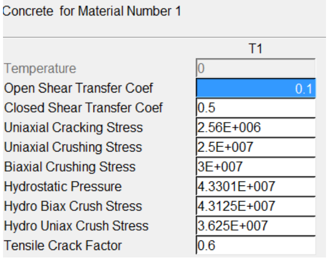


Fig. 5 Willam - Warnke criterion

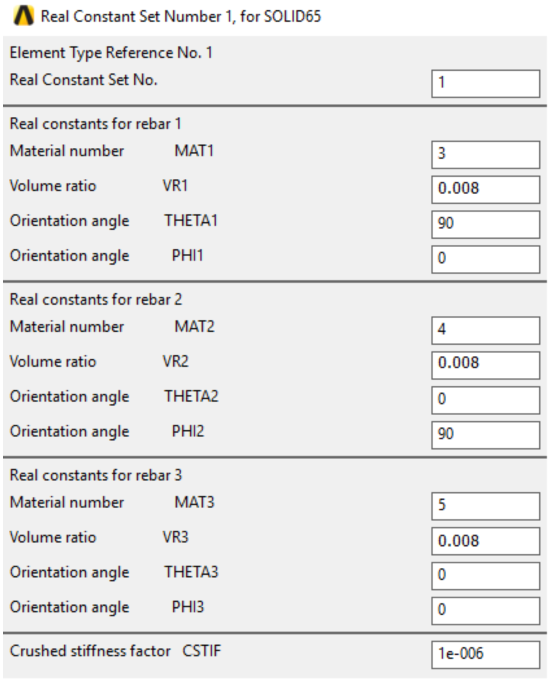
The parameters entered in the calculation are shown in Fig. 6 and Fig. 7. The parameters for the concrete are in Fig. 6.


Concrete for Material Number 1

	T1
Temperature	0
Open Shear Transfer Coef	0.1
Closed Shear Transfer Coef	0.5
Uniaxial Cracking Stress	2.56E+006
Uniaxial Crushing Stress	2.5E+007
Biaxial Crushing Stress	3E+007
Hydrostatic Pressure	4.3301E+007
Hydro Biax Crush Stress	4.3125E+007
Hydro Uniax Crush Stress	3.625E+007
Tensile Crack Factor	0.6

Fig. 6 Parameters for the concrete model

The parameters of the scattered reinforcement are in Fig. 7, and at the bottom, there is a diagram of the geometry of the finite element SOLID 65.


Real Constant Set Number 1, for SOLID65

Element Type Reference No. 1		
Real Constant Set No.		1
Real constants for rebar 1		
Material number	MAT1	3
Volume ratio	VR1	0.008
Orientation angle	THETA1	90
Orientation angle	PHI1	0
Real constants for rebar 2		
Material number	MAT2	4
Volume ratio	VR2	0.008
Orientation angle	THETA2	0
Orientation angle	PHI2	90
Real constants for rebar 3		
Material number	MAT3	5
Volume ratio	VR3	0.008
Orientation angle	THETA3	0
Orientation angle	PHI3	0
Crushed stiffness factor	CSTIF	1e-006

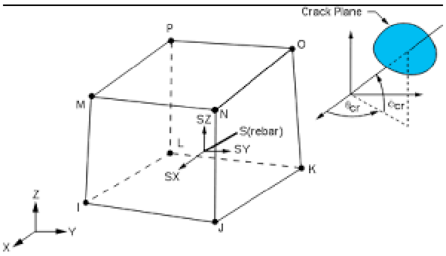


Fig. 7 Parameters for the SOLID 65

Fig. 8 shows a schematic layout of fibers in the model of the concrete slab using the 3D finite element SOLID 65, where the distribution of fibers is modeled evenly and in all three directions.

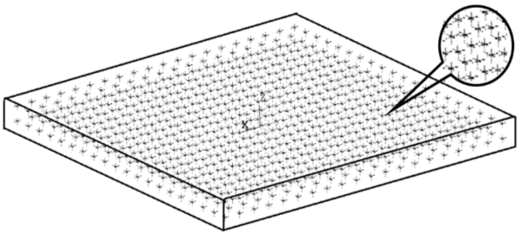


Fig. 8 Fibres in the model of the slab using the 3D element SOLID 65

The comparison of slab model without the influence of reinforcement and cracks with slab model with the influence of reinforcement and cracks is shown in following figures. The comparison is made for inhomogeneous subsoil model 8.0 x 8.0 x 8.0 m and a variant of boundary conditions B. The following figure (Fig. 9) shows the deformation of the model slab with the application of the 3D finite element SOLID 45 without consideration of the impact of fibers and cracks. Because of the way of loading the central part of the slab, there are also maximum vertical deformations in the central part of the slab (marked with the red area). The maximum deformation in the middle of the slab has a value of 8.271 mm.

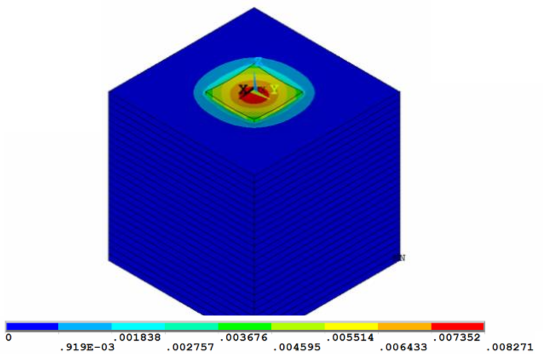


Fig. 9 Deformation of the slab model (modeled by SOLID 45) placed on the subsoil model

The following figure (Fig. 10) represents a magnified view of the deformation of the model slab. As a result of the use of the SOLID45 element, which does not take into account the influence of cracks, the model slab is not damaged by cracks (Fig. 10).

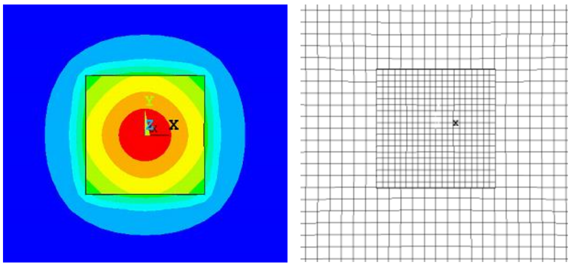


Fig. 10 Deformation of the model slab with no cracks

Fig. 11 shows the deformation of the model slab with the application of the 3D finite element SOLID 65 with consideration of the impact of fibers and cracks (placed the model subsoil for the above-mentioned boundary conditions). Because of the way of loading the central part of the slab, there are also maximum vertical deformations in the central part of the slab (red area). The maximum deformation in the middle of the slab has a value of 19.774 mm.

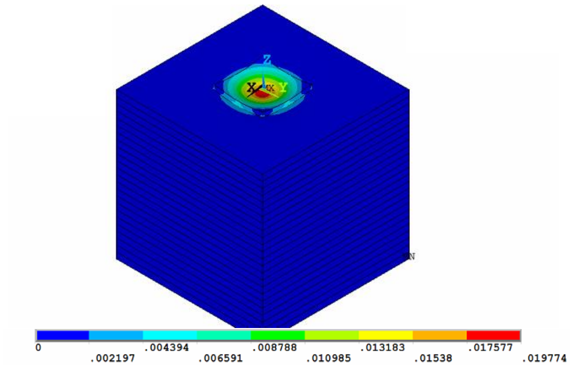


Fig. 11 Deformation of the slab model (modeled by SOLID 65) placed on the subsoil model

Fig. 12 represents a magnified view of the deformation of the model slab placed on the model subsoil. As a result of the use of the SOLID 65 element that allows thrust damage (crack formation) and pressure damage (crushing the material), it is also possible to draw the model of the slab damaged by cracks. In the figure (Fig. 12), the area damaged by indented cracks is also marked.

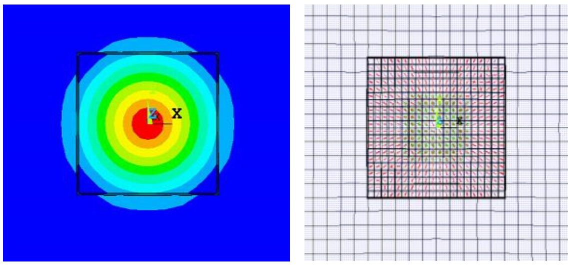


Fig. 12 Deformation of the model slab with cracks

If the spatial numerical model is created using 3D finite element, results are largely dependent on the chosen parameters entering into the calculation. It has been proven by parametric analysis in author´s articles, eg. [6], [8]. Important parameters influencing the results of the numerical analysis are, among other, the size of the modeled area representing the subsoil, the choice of boundary conditions or mesh size.

5. NUMERICAL ANALYSES IN PLAXIS 3D FOUNDATION

Model in the PLAXIS 3D FOUNDATION program, despite the title, it is not a full 3D model of the subsoil because the spatial model is created by stretching the cut to depth. In PLAXIS 3D FOUNDATION, therefore, a concrete slab was created as a 2D linear model, which was also computed for ANSYS for comparison purposes (above).

PLAXIS 3D FOUNDATION is primarily used to solve geotechnical problems. In this computing system is a specialized module designed to analyze subsoil-structure interaction. The 3D numerical model in PLAXIS 3D FOUNDATION was created with the same geometry of the subsoil model as in numerical analyses in ANSYS - in order to compare the results obtained by numerical analysis in ANSYS. The area representing the subsoil model was 8.0 x 8.0 x 8.0 m. The subsoil model is created as a homogeneous half-space and inhomogeneous half-space, as in ANSYS including the same values of the modulus of deformability. All input parameters are the same as input parameters in numerical analyzes in ANSYS and therefore according to experimental load test. The boundary conditions are automatically created in the PLAXIS 3D FOUNDATION and cannot be changed. The boundary conditions prevent by horizontal shifts of the nodes in the walls of the subsoil model and the vertical and horizontal shifts of the nodes in the lower base of the subsoil model. No boundary conditions prevented the nodes shifting in the upper base of the subsoil model because it represented the terrain. The boundary conditions in numerical analysis in ANSYS were the same. The finite-element mesh was created automatically by 15 nodal finite elements. The Mohr-Coulomb material model was used for the subsoil and consequently, the difference between tensile and compressive strength can be described.

When the subsoil model was created as a homogeneous half-space (Fig. 13), the maximum calculated vertical deformation at the center of the slab was 13.23 mm.

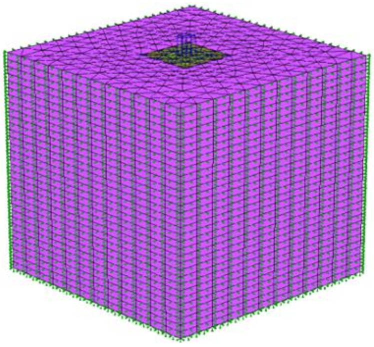


Fig. 13 Homogeneous half-space model, PLAXIS 3D FOUNDATION

The deformation measured during the experiment (using the sensor closest to the slab center) was 15.05 mm. In the inhomogeneous model of the subsoil (Fig. 14) the value of the modulus of deformability increased with the depth according to formula (1).

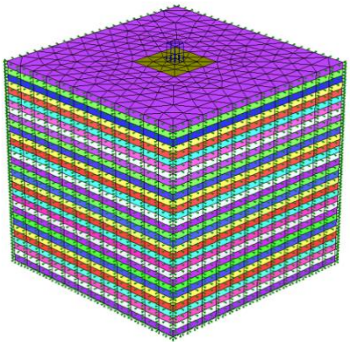


Fig. 14 Inhomogeneous half-space model, PLAXIS 3D FOUNDATION

In the PLAXIS 3D FOUNDATION the subsoil model was divided into 32 layers (Fig. 14). In these layers, the value of the modulus of deformability gradually increased with the depth (as in ANSYS). When the subsoil model was created as the inhomogeneous half-space, the maximum calculated vertical deformation at the center of the slab was 9.68 mm (Fig. 14).

6. NUMERICAL ANALYSES IN SCIA ENGINEER

Subsoil-structure interaction can also be solved using the SCIA ENGINEER, commercial software [10]. The results will be compared with the results obtained by the 3D numerical models created in the ANSYS, PLAXIS 3D FOUNDATION and with the experimental values.

In SCIA ENGINEER [10], the so-called standard model of subsoil according to ČSN 73 1001 [11] is applied, which is based on the theory of elastic half-space of the modified structural strength of soil. If the design in the program is modeled according to Eurocode 7 [12], the value of the structural strength coefficient is automatically $m = 0.2$ and is unchanged. It is a surface model of the subsoil, which is characterized by the parameters C_x, C_y, C_z. The properties of the modeling subsoil are inter alia entered through the parameters of the subsoil C_{1x}, C_{1y}, C_{1z}, and C_{2x}, C_{2y} [10]. C_{1z} is the compressibility parameter of the elastic subsoil in the z-axis direction, which represents the resistive elastic resistance against the vertical displacement w. C_{2x} and C_{2y} are shear deformation parameters that take into account the shear interaction of the subsoil. C_{1x} and C_{1y} are parameters of pliability of the subsoil in the x-axis direction, respectively y-axis direction, which represents resistance against horizontal displacement u, resp. v (shifts in the plane of the slab). The SCIA ENGINEER offers calculation module SOILIN to solve the interaction of foundations with the subsoil. SOILIN calculates the parameters of the subsoil C_{1z}, $C_{2x,}$ and C_{2y}. Parameters C_{1x}, C_{1y} are always specified by the user. The SOILIN module calculates the settlement based on the elastic half-space. Then subsoil parameters C are automatically calculated. Whereas the subsoil parameters C affect the contact stress, and this affects settlement, and all interdependencies are valid also conversely, it is an iterative calculation. The iteration cycle is finished when the calculated value of shift or calculated value of contact stress in two consecutive cycles is almost unchanged (use of quadratic norm). In the author's article [10] the dependence of the deformations calculated by the SOILIN module on the input values of the individual subsoil parameters were monitored and evaluated. Differences in deformations occur depending on the parameters C_{2x} and C_{2y}. If $C_{2x} = C_{2y} = 0$, these parameters do not enter the SOILIN iterative process and the influence of the surrounding soil environment is not taken into account in the calculations. If the parameters C_{2x} and C_{2y} are non-zero, they enter the iterative calculation and the influence of the surrounding soil environment is taken into account [10]. In the calculation of the interaction of the subsoil with the slab loaded during the experiment, the SOILIN module was used. The input values of the parameters C_{2x} and C_{2y} are non-zero, i.e. the influence of the surrounding soil environment has been taken into account.

The same values were entered in the calculation as in all previous programs in which this task was also solved (ANSYS and PLAXIS 3D FOUNDATION). Parameters of the geological profile were defined by the modulus of deformability $E_{def} = 12.5$ MPa and the Poisson coefficient $\mu = 0.35$. In addition, the soil mass density $\gamma = 20.0$ kN/m^3, the structural strength coefficient m = 0.2 (according to EC 7 [12]) and the layer thickness (defining the geological profile) h = 8.0 m were given. In this model, the maximum calculated vertical deformation in the center of the slab was 6.26 mm (Fig. 15).

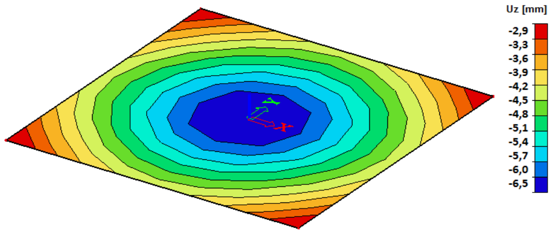


Fig. 15 Surface subsoil model, SCIA ENGINEER

7. NUMERICAL ANALYSES IN MKPINTER

To compare the results of the solution of the interaction of the subsoil with the concrete slab, this task (with the same input values) was also analyzed in the non-commercial program MKPINTER [13], [14]. Calculation of the deformations and internal forces of the slab is done by the finite element method (FEM) using isoparametric plate elements with shear effect. The course of contact stress affects the deformation of the slab and of the subsoil too. To cause the contact stress of the same deformation of the slab and subsoil in this nonlinear interaction, this solution is performed by an iterative method. The calculation of stress, settlement and contact stress is solved by the universal method of calculation using Jacobian transformation. In MKPINTER, subsoil-structure interaction is solved using numerical integration calculations of stress and settlement of the modified elastic half-space by structural strength. For more information about MKPINTER see [13-14].

8. COMPARISON OF RESULTS

This article compares the resulting vertical deformations calculated in the software based on FEM - ANSYS, PLAXIS 3D FOUNDATION, SCIA ENGINEER, MKPINTER FEM, and also vertical deformations calculated according to recommended normative procedures.

In the graph (Fig. 16), the vertical deformations calculated by the various computational programs and procedures are compared. The vertical deformations calculated by the solution of the interaction of subsoil with a concrete slab in ANSYS, PLAXIS 3D FOUNDATION, SCIA ENGINEER and MKPINTER are compared. The settlement calculated according to the normative procedures recommended in ČSN 73 1001 [11] and Eurocode 7 [12] are also included in the chart.

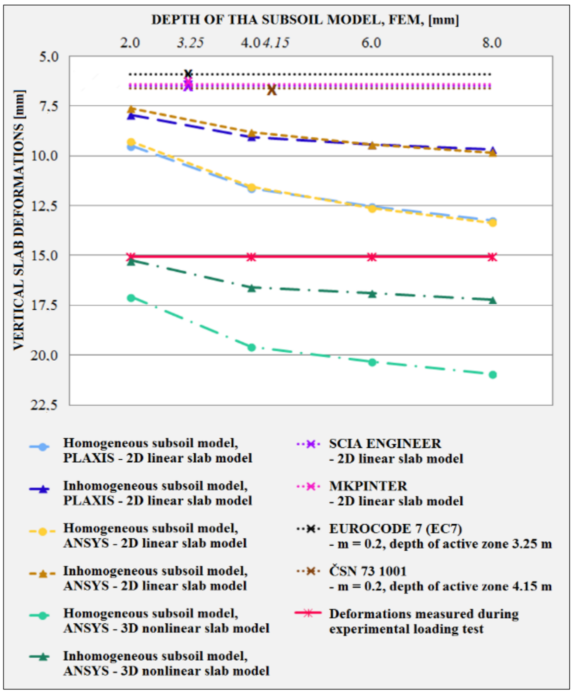


Fig. 16 Comparison of results – deformations

The red full line indicates the slab deformation measured at the last loading test step, i.e. at a time when the slab was significantly damaged by cracks.

The graph shows vertical deformation dependence on the depth of the 3D subsoil models created in ANSYS (yellow and green) and PLAXIS 3D FOUNDATION (blue). The light lines indicate vertical deformations calculated on the homogeneous model of the subsoil. The dark lines indicate the vertical deformations calculated on the inhomogeneous model of the subsoil in which the value of modulus of deformability increases with the depth. In tasks where the slab was modeled as a 2D linear model (without the impact of cracks), there was a very good accordance in the results from ANSYS (dark and light yellow dashed line) and PLAXIS 3D FOUNDATION (dark and light blue dashed lines). Due to the impact of the cracks in the concrete slab model is not taken into account, the vertical deformations calculated by the above-mentioned models are less than the measured deformation of the already extensively damaged concrete slab. This is also valid for vertical deformation obtained in SCIA ENGINEER (purple) and MKPINTER (pink), in which the concrete slab was also created as a 2D linear model without the impact of cracks. There are the subsoil surface model based on the elastic half-space of the modified structural strength of soil (according to EC 7 [12], m = 0.2) in programs SCIA ENGINEER and MKPINTER, in comparison with ANSYS and PLAXIS 3D FOUNDATION. It is a surface model of the subsoil and its results do not depend on the chosen depth of the area as it is with 3D models in ANSYS and PLAXIS 3D FOUNDATION. Using the procedure according to ČSN 73 1001 [11], the depth of the deformation zone was 4.15 m and the settlement was 6.55 mm for m = 0.1 (brown dotted line with a cross). The value of the correction coefficient of overload m = 0.1 was determined according to the table in ČSN 73 1001 [11]. According to Eurocode 7 [12], it is always m = 0.2, the depth of the deformation zone is 3.25 m and the settlement was 5.89 mm (black dotted line with a cross). For calculations of the depth of active zone according to the both above-mentioned recommended standards, it is clear from the graph that different models of subsoil and slab models (which are not uniquely specified in the standards) can obtain a large range of calculated deformation values.

The calculations whose resulting vertical deformations were greater than the measured deformations of the damaged slab are marked by green lines. Specifically, this is the deformation calculated in the ANSYS, in which the concrete slab was created as a 3D non-linear model with the influence of cracks. The highest values of the calculated deformations indicate that in the numerical analyzes performed in this way the influence of the cracks was taken into account, leading to a reduction of stiffness of the concrete slab model and its greater deformations. Of course, these deformations are also dependent on the subsoil model created as a homogeneous continuum (light green) and as an inhomogeneous continuum (dark green).

The average difference was of about 66% - between the results of the 2D linear slab model without the influence of cracks and the 3D nonlinear slab model with influence of cracks (in the comparison the deformations obtained from the ANSYS). The average difference between the calculated deformations in the homogeneous and inhomogeneous subsoil models was about 29%. The 3D nonlinear model of the subsoil has always been retained when comparing.

9. CONCLUSION

This article compares the resulting vertical deformations calculated in the software based on FEM - ANSYS, PLAXIS 3D FOUNDATION, SCIA ENGINEER, MKPINTER FEM, and also vertical deformations calculated according to recommended normative procedures.

A good accordance was in the results from the ANSYS and PLAXIS 3D FOUNDATION - programs in which 3D finite elements were used, and from the SCIA ENGINEER and MKPINTER - programs, which in which 2D finite elements were used. A significant difference in the calculated deformations occurred when comparing the results from the ANSYS and PLAXIS 3D FOUNDATION with results from the SCIA ENGINEER and MKPINTER in which the subsoil-structure interaction is solved based on the theory of elastic half-space modified by structural soil strength.

For calculations of the depth of active zone according to the recommended standards, it is clear that different models of subsoil and slab models (which are not uniquely specified in the standards) can obtain a large range of calculated deformation values.

Although it has been shown that the resulting deformation of the interaction of foundation structure and subsoil created as an inhomogeneous continuum model provide deformations to better delineate the real deformations, prudence is in place when creating a numerical model of subsoil created by spatial finite elements. By choosing the geometry of the solved area without previous experience with the aforementioned parameters, a numerical model can be created that would adversely affect the calculated deformations and lead to dangerous and unreliable results.

10. ACKNOWLEDGEMENTS

This outcome has been achieved with the financial support of the project GACR No. 16-08937S „State of stress and strain of fiber-reinforced composites in interaction with the soil environment“.

11. REFERENCES

[1] Zienkiewicz, O. C., Taylor, R. L., The Finite Element Method, Volume 2: Solid Mechanics, Oxford, Butterworth-Heinemann, 479 p, 2000.

[2] Kralik, J., Jendzelovsky, N., Contact problem of the reinforced concrete girder and non-linear Winkler foundations, Geomechanics 93, Ostrava, Pages 233-236, 1994.

[3] Kolar, V., Nemec, I.: Modelling of Soil Structure Interactions, Prague, 336 p, 1989.

[4] Feda, J., Tenseness of subsoil and calculation methods for calculation of final subsidence, ACADEMIA, 152 pages, 1974. (In Czech)

[5] Frölich, O. K.: Druckverteilung im Baugrunde, Mit Besonderer Berücksichtigung der Plastischen Erscheinungen, Vienna, 1934.

[6] Labudkova, J. Cajka, R., Numerical analyses of the interaction of steel-fiber reinforced concrete slab model with subsoil, Frattura ed Integrita Strutturale, Volume 11, Issue 39, January 2017, Pages 47-55, 2017.

[7] Cajka, R., Labudkova, J., Mynarcik, P., Numerical solution of soil-foundation interaction and comparison of results with experimental measurements, International Journal of GEOMATE, Volume 11, Issue 1, 2016, pp. 2116-2122.

[8] Labudkova, J., Cajka, R., Application of an Inhomogeneous Halfspace for the Solution of Foundation-Subsoil Interaction based on the Finite Element Method, in Proceedings of the Fifteenth International Conference on Civil, Structural and Environmental Engineering Computing, Civil-Comp Press, Stirlingshire, UK, Paper 9, 2015. doi:10.4203/ccp.108.9

[9] Sucharda O., Pajak, M., Ponikiewski T., Konecny P., Identification of mechanical and fracture properties of self-compacting concrete beams with different types of steel fibers using inverse analysis, Construction and Building Materials, 138, pp. 263-275, 2017, doi:10.1016/j.conbuildmat.2017.01.077.

[10] Labudkova, J., Cajka, R., Numerical analysis of the interaction of steel-fiber concrete slab with subsoil and comparison with values measured during experimental loading test. Proceedings of the International Conference FIBRE CONCRETE, Volume 2015-January, 2015, Pages 292-301, 8th International Conference Fibre Concrete 2015: Technology, Design, Application, FC 2015; 10 – 11 September 2015, Prague; Czech Republic;

[11] Eurocode 7 - Geotechnical Design, Part 1: General Rules, European Committee for Standardization. EN 1997-1: 2004, Brussels

[12] ČSN 73 1001. Foundation engineering. Foundation soils under flat foundations, Czech Standards Institute, Prague (in Czech).

[13] Cajka, R., Analysis of Stress in Half-space using Jacobian of Transformation and Gauss Numerical Integration, Advanced Materials Research, Switzerland, p.178-186, 2013.

[14] Cajka, R., Accuracy of Stress Analysis Using Numerical Integration of Elastic Half-Space, Applied Mechanics and Materials, Trans Tech Publications, ISSN: 16609336, Switzerland, p.1127-1135, 2013.