SUBMISSION AND LIMITATIONS OF CIVIL ENGINEERING TASKS USING ANSYS TOOL IN NATIONAL SUPERCOMPUTER CENTER IT4INNOVATIONS

Zdenka NEUWIRTHOVA, Radim CAJKA

Department of Structures, Faculty of Civil Engineering, VSB – Technical University of Ostrava,
Ludvíka Podéště 1875/17, 708 00 Ostrava-Poruba, Czech Republic

zdenka.neuwirthova@vsb.cz, radim.cajka@vsb.cz

DOI: 10.35181/tces-2020-0013

Abstract. *This article deals with a problem of soil-structure interaction on a simplified model using the commercial software Ansys on the Anselm supercomputer in the National Supercomputing Center IT4Innovations in the VSB-Technical University of Ostrava. For a purpose of large-scale modelling tasks by supercomputer knowing program limitations are required as well as the area and scope of the selected solution. Therefore, a numerical model was created with increasing size, and therefore degrees of freedom until a maximum size of the task was determined because any larger task can´t be computed. The maximum size of the task and solution times obtained by the supercomputer was compared to the maximum size and computation times received by a standard computer workstation.*

Keywords

Ansys, Finite Element Method, Supercomputer.

1. Introduction

Interaction between soil and a foundation is a complex task in which we need to take into account the influence of the physically-nonlinear behaviour of both the soil and the foundation as well as their interaction. Therefore, a suitable contact element must be selected to express the deformation and stress transmission through the foundation. A computational model of soil and the foundation is also very important to proper numerical modelling of the physically-nonlinear behaviour.

Although theoretical and numerical computational models of problematics are being constantly developed by many scientists all around the world [1-9] the optimal procedure to provide sufficiently accurate results that work under all circumstances has not been accomplished yet. For this reason, scientists perform experimental testing to deep the theoretical basis. Based on the experimental testing development of new numerical models are carried out for better simulation of the real behaviour of the soil-structure interaction [3]. Experimental testing and studies are performed worldwide to provide the deformations of the soil through the concrete slab, punching of slab [4],[5],[6], the tension inside of the soil [7].

Behaviour of the described problem can be described by the elastic half-space theory, by numerical contact models with major simplifications, or by advanced complex models that take into consideration all nonlinearities. The last model usually requires a fine finite element mesh which is related to high computing requirements. It may occur that the capacity of a standard workstation will be insufficient in such cases. Parallel computations and supercomputers are required for solving large tasks. The tasks with fine finite element meshes bring the great possibility to model individual reinforcements of concrete, like steel bars in reinforced concrete or fibers in the fibre-concrete. It is possible to use more advanced models of bonding between reinforcements and concrete. The supercomputer provides a wider range of options.

2. Methods

For a better understanding of the issue, special testing equipment was constructed at the Faculty of Civil Engineering at VŠB – Technical University of Ostrava [19], and experimental tests of loading slabs were performed there [3], [5], [16]. Numerical models are developed among the experiments but a capacity of a standard workstation is insufficient for the research at the moment. The work aims to explore the possibilities and limitations of the supercomputer and its application to the solved problem. A parametric numerical model was created and solved using Ansys Multiphysics HPC on the Anselm supercomputer and standard workstation using Ansys APDL 18.0. Computation times of each task were tracked and compared.

2.1. Experimental testing

The experimental testing was done using special testing equipment named Stand Fig. 1, which is placed outside the premises of the Faculty of Civil Engineering in Ostrava (Czech Republic). The testing equipment [9] consists of a steel construction anchored by micro piles to the subsoil ensuring the bearing capacity and a hydraulic press which relies on the steel construction to apply the vertical load to the tested slab as shown in the figure. The slab has been concreted in the centre of the testing area so that the results are not affected by an uneven ground settlement due to the eccentric placement between foundation strips under the steel construction. The maximum force induced by the testing equipment is 1 000 kN. Vertical deformation is monitored by sensors Fig. 2.

Slab no. D10-G01 was selected from a series of tests made in 2016 year. This slab consists of plain concrete only without any reinforcements. C 20/25 concrete type was used. Few samples were produced together with the slab and tested in the laboratory. Following characteristics were measured:

- average compressive strength of concrete (cube test) $f_{c,cube} = 25.11$ MPa
- average compressive strength of concrete (cylinder test) $f_{c,cyl} = 20.03$ MPa
- average tensile strength $f_t = 2.10$ MPa
- average elastic modulus $E = 17\,000$ MPa.

Fig. 1: Special testing equipment Stand.

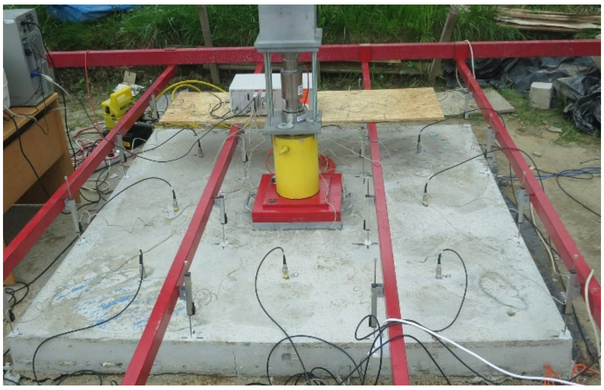

Fig. 2: Layout of sensors on tested slab.

2.2. Numerical modelling

Based on the measured experimental data and inputs the following numerical model was created. The model is based on Boussinesq´s theory for a stress distribution under a vertical concentrated load. So the elastic half-space was considered as a cubic section of the soil with parametric dimensions set as edge *h* - Fig.3.

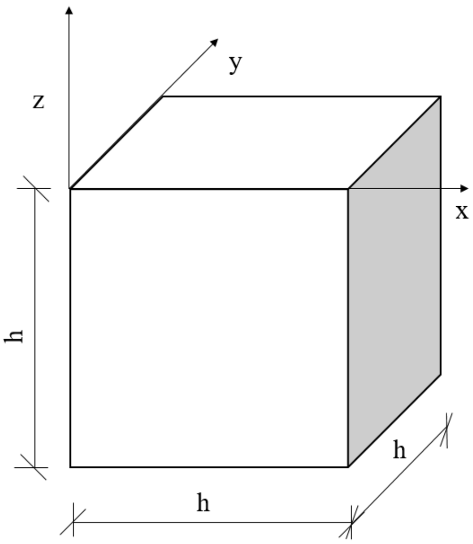

Fig. 3: Soil model cube dimensions.

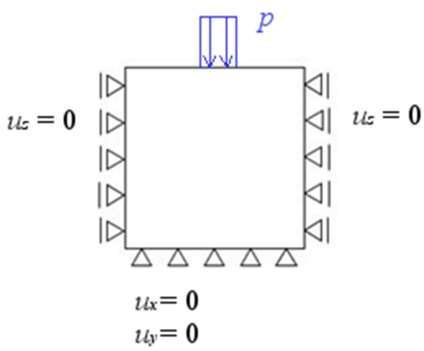

Fig. 4: Boundary conditions.

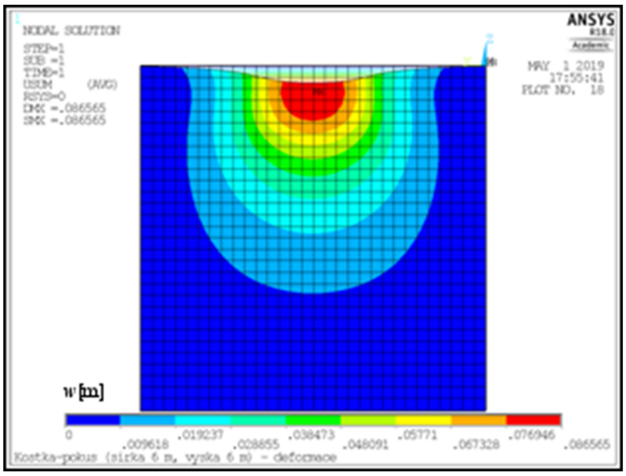

Fig. 5: Deformation in the middle cross-section of cube

Behaviour of the soil model was characterized by the modulus of elasticity E = 12.5 MPa and the Poisson coefficient μ = 0.25. Self - weight was neglected. The boundary conditions are shown in Fig.4. The pressure p = 87 500 Pa (which corresponds to the maximum load of experimental testing) was applied on 2×2 m area (which represents the dimensions of the concrete slab). The concrete slab itself was neglected.

This task was solved with increasing parameter h in steps by 2 m up to the limitations of the selected solving method. That means that any larger task cannot be solved because of insufficient computational capacity.

3. Results

Relative to the specified parameters the type of the task is linear elasticity. Any nonlinearities have not been considered at this moment.

3.1. Standard workstation

Firstly, the described task was computed on a standard workstation up to the computational capacity. Workstation with Intel Core i5-6500 CPU and 16 GB RAM was used. As we can see from the results (Tab.1), the biggest possible resolved calculation on this workstation has 1 536 000 degrees of freedom. The largest task cannot be completed due to the insufficient amount of available memory.

3.2. Supercomputer Anselm

Lately, the same calculations were performed on the Anselm supercomputer. The task was computed as distributed to the 512 cores. The computation time of tasks is recorded in Tab.2. The number of 512 cores was selected due to the limitation of HPC licenses available on supercomputers in the IT4Innovations. Due to this restriction is impossible to distribute the calculation to the bigger amount of cores.

Tab.1: Computation time depending on the number of degrees of freedom computed on a standard workstation

Dimension [m]	Model size	Computation time	
h	**Number of degrees of freedom**	**min**	**s**
4	24 000	0	02
6	81 000	0	10
8	192 000	0	37
10	375 000	1	48
12	648 000	5	10
14	1 029 000	13	46
16	1 536 000	27	56
18	The calculation cannot be completed		

Tab.2: Computation time depending on the number of degrees of freedom computed on the Anselm supercomputer

Dimension [m]	Model size	Computation time	
h	**Number of degrees of freedom**	**min**	**s**
4	24 000	0	04
6	81 000	0	04
8	192 000	0	05
10	375 000	0	07
12	648 000	0	12
14	1 029 000	0	20
16	1 536 000	0	31
18	2 187 000	0	50
20	3 000 000	1	20
22	3 993 000	1	59
24	5 184 000	2	52
26	6 591 000	4	09
28	8 232 000	9	09
30	10 125 000	11	07
32	12 288 000	21	17
34	14 739 000	26	00
36	17 496 000	37	33
38	The calculation cannot be completed		

3.3. Discussion

The computation performed on the supercomputer allows solving tasks of multiple sizes compared to a standard workstation as can be seen from Tab.2 and Tab.1. In the case of linear elasticity, the solved task is 11.3× larger. Both calculations – on the standard workstation and the supercomputer – were limited by the available memory. The calculation on the Anselm was also limited by the availability of the HPC licenses.

Based on the measured times the following graph was evaluated to compare the calculation times dependent on the size of the task on each device in a logarithmic scale on both axis, Fig.6.

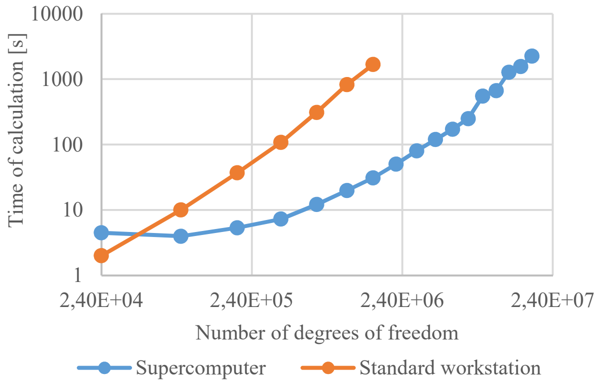

Fig. 6: Comparison of time of calculation on standard workstation and supercomputer

4. Conclusion

The parametric study was performed to find the limitations of the standard computer and the supercomputer Anselm for linear elastic calculations. For every task, the time of solution was measured and times were compared and evaluated into the graph (Fig.6). The biggest solved calculation by the standard workstation has 1.5 million degrees of freedom and calculation took 28 minutes. While solving the same task with 1.5 million degrees of freedom on the supercomputer took only 31 s (0,5 min). The biggest solved calculation by the supercomputer has 17.5 million degrees of freedom and took 37.5 minutes on 512 cores using 512 HPC licenses. The supercomputer allows solving tasks of larger sizes or calculations of the same size in a much reduced time. The solution in the Ansys on the supercomputer is limited by available HPC licenses. The use of an independent PERMON toolbox will be considered in upcoming projects to avoid this limitation. The largest task solved by the PERMON had 1.5 billion unknowns for the linear elasticity.

Acknowledgements

This paper was supported by The Ministry of Education, Youth and Sports from the Large Infrastructures for Research, Experimental Development and Innovations project „IT4Innovations National Supercomputing Center – LM2015070“, and by the Moravian-Silesian Region under the program “Support of Science and Research in the Moravia-Silesia Region 2017” (RRC/10/2017).

References

[1] TOMASOVICOVA D. and N. JENDZELOVSKY. Stiffness Analysis of the Subsoil under Industrial Floor, *3rd International Conference on Structural and Physical Aspects of Construction Engineering (SPACE)*, 2015 vol.190, 365-370. DOI: 10.1016/j.proeng.2017.05.350

[2] KRALIK J., ROSKO P. and J. KRALIK, Effectiveness of Probabilistic Methods to Analyse Probability of Structural Failure Using ANSYS Software. *International Conference of Numerical Analysis and Applied Mathematics (ICNAAM 2017)*, 2018, vol 1978. DOI: 10.1063/1.5043808

[3] CAJKA R., LABUDKOVA J. and P. MYNARCIK, Numerical solution of soil - foundation interaction and comparison of results with experimental measurements. *International Journal of GEOMATE*, 11(1), 2116-2122, 2016.

[4] HALVONIK J. and L. MAJTANOVA, Experimental Investigation of the Maximum Punching Resistance of Slab-Column Connections, *Slovak Journal of Civil Engineering*, 26(3), 22-28, 2018, DOI: 10.2478/sjce-2018-0017

[5] HEGGER J.,RICKER M.,ULKE B. and M. ZIEGLER, Investigations on the punching behaviour of reinforced concrete footings, *Engineering Structures*. 29, pp. 2233-2241, 2007, DOI:doi.org/10.1016/j.engstruct.2006.11.012

[6] KOTSOVOU M. G., and E. VOUGIOUKAS,, Assessment of design methods for punching through numerical experiments, *Computers and Concrete*. 17(3), 2016, DOI:10.12989/cac.2016.17.3.305

[7] HRUBESOVA E., MOHYLA M., LAHUTA H., BUI T.Q. and P.D. NGUYEN, Experimental analysis of stresses in subsoil below a rectangular fiber concrete slab, *Sustainability* (Switzerland), 10 (7), art. no. 2216, 2018, DOI: 10.3390/su10072216

[8] CAJKA R., and J. LABUDKOVA, Dependence of deformation of a plate on the subsoil in relation to the parameters of the 3D model, *International Journal of Mechanic*. pp. 208-215, 2014, ISSN: 19984448, 2014.

[9] CAJKA R., KRIVY V. and D. SEKANINA, Design and Development of a Testing Device for Experimental Measurements of Foundation Slabs on the Subsoil, *Transactions of the VSB - Technical University of Ostrava, Civil Engineering Series*. 11 (1), 1-5, 2011 DOI: 10.2478/v10160-011-0002-2, 2011.

[10] FEDA J., and Z.P. BAŽANT, Stress in subsoil and methods of final settlement calculation. *New York: distribution for the U.S.A. and Canada, Elsevier/North Holland*, 1978. ISBN 04-449-9800-4.

[11] KOHNKE P., ed. ANSYS Theory Reference. Release 15.0. U.S.A.: SAS IP, 2013.

About Authors

Zdeňka NEUWIRTHOVÁ was born in Bílovec, Czech Republic. She received her M.Sc. from VSB - Technical University of Ostrava, Faculty of Civil Engineering in 2017. Her research interests include numerical modelling of concrete structures and soil-structure interaction, Finite Element Method, supercomputer modelling.

Radim ČAJKA was born in Ostrava, Czech Republic. He received his professor degree from VSB - Technical University of Ostrava, Faculty of Civil Engineering in 2008. His research interests include Concrete Structures, Foundations of Structures Buildings on Undermined Areas, Soil - Structure Interaction, Finite Element Method, Temperature Loaded, Structures Fire Resistance of Structures.