COMPARISON BETWEEN NUMERICAL ANALYSIS AND ACTUAL RESULTS FOR A PULL-OUT TEST

J. Gontarz[1], J. Podgórski[1], Józef Jonak[2], Marek Kalita[3], Michał Siegmund[3]

[1]*Faculty of Civil Engineering and Architecture, Lublin University of Technology, Nadbystrzycka 40, Lublin, Poland*

[2]*Faculty of Mechanical Engineering, Lublin University of Technology, Nadbystrzycka 36, Lublin, Poland*

[3]*KOMAG Institute of Mining Technology, Pszczynska 37, Gliwice, Poland*

e-mail: j.gontarz@pollub.pl, j.podgorski@pollub.pl, j.jonak@pollub.pl

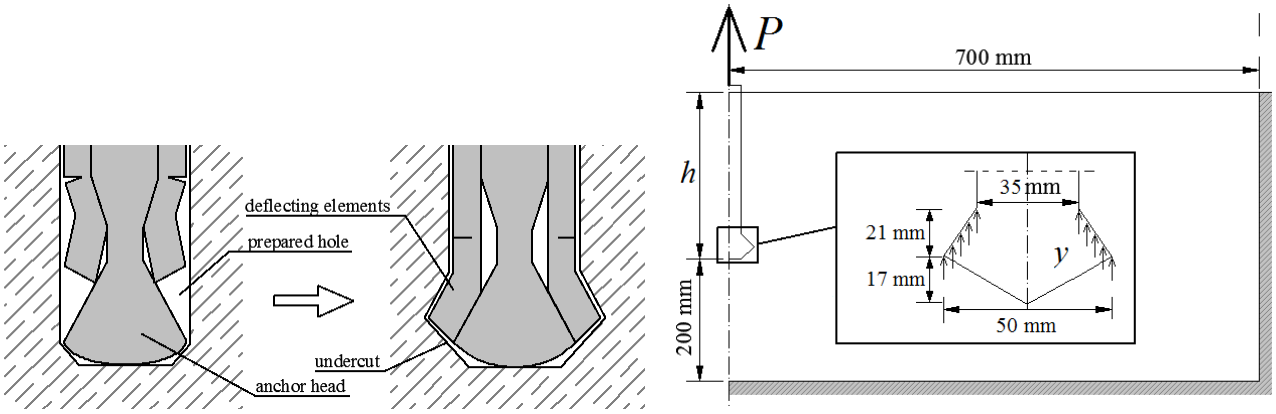
1. Introduction

The authors of this paper attended to analyze the pull-out test in which a pre-set self-undercut anchor is pulled out of a stone surface. The research consisted in finding the parameters of the selected material - sandstone from the Braciszów quarry. Next, a pull-out test was modeled in the Abaqus program and computer analysis was performed using X-FEM elements, which are elements simulating the crack independent of the finite element mesh [1]. The results obtained in the calculations were compared with the pull-out tests performed on actual rock. The aim of the described research is to find a way to calculate the force of pulling out the anchor for any material and for any length of anchoring.

The HILTI HDA-P M20x250/100 anchor was adopted for pull-out tests. To mount this anchor, it is placed in a prepared hole in a rock surface, and then the anchor is pressed. Then while drilling it undercuts itself with deflecting elements. Scheme of mounting the anchor is shown in Figure 1a.

2. Description of the computer model

The pull-out test was modeled in 2D stress state as an axially symmetrical task. The computational models is presented in Figure 1b, where *h* is the length of the anchor. The load was simulated by the *y*-direction displacement. The boundary conditions were modeled on the right and bottom edge. The size of the model was assumed to be large enough so that the boundary conditions did not affect the result.



a) b)

Figure 1: a) Scheme of mounting the anchor, b) Scheme of the task.

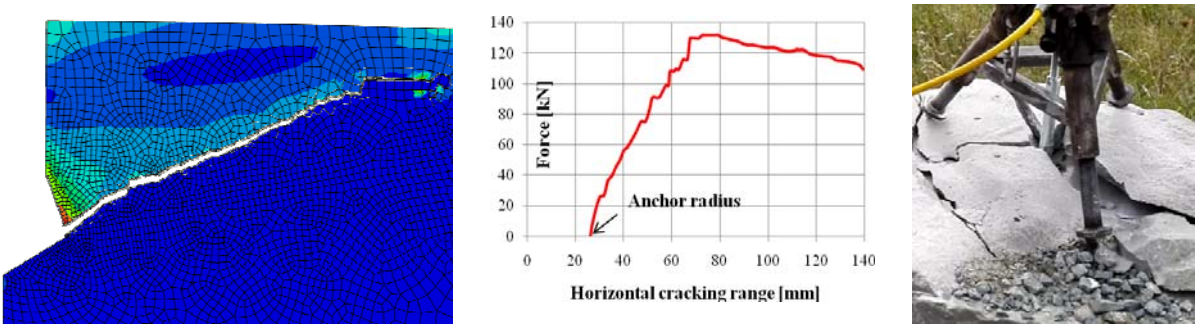
3. Material parameters

Several laboratory tests on the described sandstone were performed. Compressive strength tests were carried out with uniaxial compression. The Young modulus and the Poisson ratio were obtained with the help of reading the vertical and horizontal strains in the samples. Tensile strength was also obtained using a modified Brazilian test. The exact tensile strength was obtained based on the analysis of the stress field in the Abaqus

code and on the author's previous work [2]. Critical strain energy release rate in mode I was also obtained from the test of three-point bending of beams with notches. This value was calculated with the Bower equation [3]. The results of the experiments gave the following values: compressive strength f_c = 202.7 MPa, tensile strength f_t = 7.945 MPa, Young modulus E = 15.984 GPa, Poisson ratio ν = 0.275, Critical strain energy release rate G_{Ic} = 0.257 N/mm.

4. Computer simulation and actual pull-out test

The above material parameters were used to model the test in Abaqus for an 8 cm anchoring. The view of the damaged model is shown in Figure 2a. As it can be seen, the crack starts to propagate horizontally near the upper edge. For various program settings and different mesh, it was not possible to cause the crack to go through to the end, probably that there is a stress state with which the Abaqus program can't cope. The pulling-out force at the beginning grows and then decrease after reaching the radius of the pulling-out fragment of about 8 cm (Figure 2b), where the expected radius of the broken cone is 18.5 cm, basing on the crack angle. The maximum force was about 132 kN. Tests in the quarry were also made on the same stone and for the same depth of the anchor (Figure 2c). For three successful tests, the average pulling-out force is 162 kN. Inspection of the damaged stones allowed to state that the shape of the broken fragment is very similar to that in the computer simulation.



a) b) c)

Figure 2: Pull-out test. a) crack path obtained by computer simulation with Abaqus code, b) dependence of force on the crack's range, c) view of the detached fragment of the actual rock.

5. Summary

As it can be seen, the above results are similar. However, the results depend on many factors. The tested material is very heterogeneous because the test results were very different. For example, the tensile strength results fluctuated from 5.3 MPa to 10.6 MPa. Similarly, the pull-out test on actual rocks gave a result between 112 kN and 212 kN. Therefore, it can't be concluded that the X-FEM method in Abaqus, and in particular the given parameters, allow to obtain the correct result. It is therefore necessary to perform more pull-out tests on the actual rock. It is possible that it will be necessary to re-analyze all for a more homogeneous material.

Acknowledgments This research was sponsored by the Lublin University of Technology Statutory Funds S16/2017 and co-financed by the Polish National Science Center, project RODEST nr 2015/19/B/ST10/02817.

References

[1] S. Mohammadi, Extended Finite Element Method: Extended Finite Element Method: for Fracture Analysis of Structures, Wiley-Blackwell, 2007.
[2] J. Podgórski and J. Gontarz, Explanation of the mechanism of destruction of the cylindrical sample in the Brazilian test. *Advances in Mechanics: Theoretical, Computational and Interdisciplinary Issues*., CRC Press, 2016.
[3] A. F. Bower, Applied mechanics of solids, CRC Press, 2009.