18th International Conference
ENGINEERING MECHANICS 2012
Svratka, Czech Republic, May 14 – 17, 2012

pp. 186–187
Paper #163

MODELLING OF THE CEILING SLAB FROM THE HOLLOW CORE PANELS IN ANSYS PROGRAM

J. Kršík, J. Křiváková *

Absract: *The aim of this paper is to describe modeling of the Ceiling slab from the hollow core panels in ANSYS program. The real load tests are used from NO. VTT-S-07311-06 from VTT Technical Centre of Finland. Several simplifications are accepted. The calculated deformations are compared with deformations from the load test at the end.*

Keywords: ***Ansys, Prestressed Concrete, Hollow Core Panel, Deformations***

1. Annotation

The aim of this paper is to create the model of the ceiling slab from the hollow core panels using volume finite elements in ANSYS program. This slab will be used for verification of the shear stress test. Ceiling slab used is the same as that used in loading tests no. VTT-S-07331-06 from VTT Technical Centre of Finland.

Modeled panel has a height of 0,4 m, length 9 m and structural width of 1,2 m. Panel is lightweight by four cavities. Material properties of concrete are approximately equivalent to strength class C40/50. Multilinear strength-stress diagram is closed to parabolic. Steel cables are used for prestress St 1630/1860. Bilinear strength-stress diagram is applied. Element distribution of the forehead was applied with to the desired results (see Fig. 1). Division along the length is from the same reason chosen on 50 mm.

All volume parts are modeled from finite elements SOLID 185. All beam parts (prestress cables, reinforcement) are modeled from finite elements BEAM 188.

The slab is created from four panels, which are connected by in-situ concrete with strength class C30/37. The same concrete is also used for embedding of holes to a depth of 50 mm from the panel front.

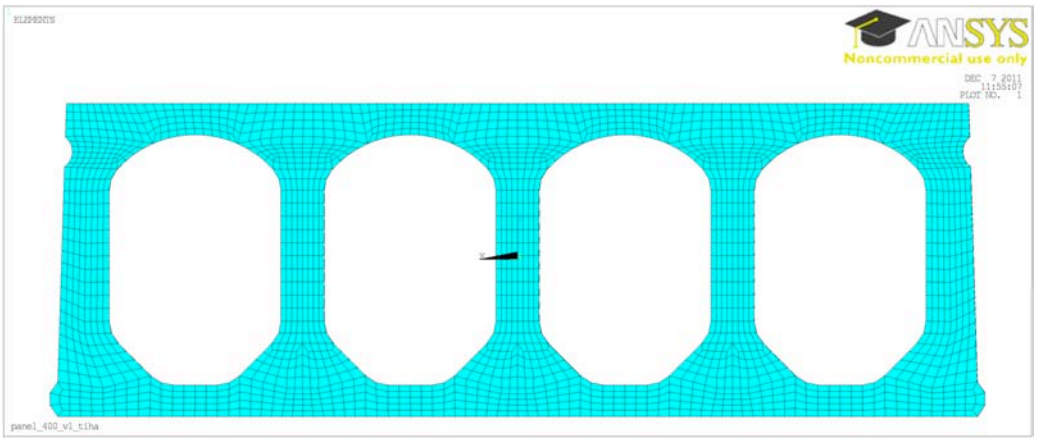

Fig. 1 The finite element net of forehead for 1 panel

* Ing. Jakub Kršík, Ing. Jarmila Křiváková, CSc.: Institute of Structural Mechanics, Brno University of Technology, Veveří 331/95; 602 00, Brno; CZ, e-mail: krsik.j@fce.vutbr.cz, krivakova.j@fce.vutbr.cz

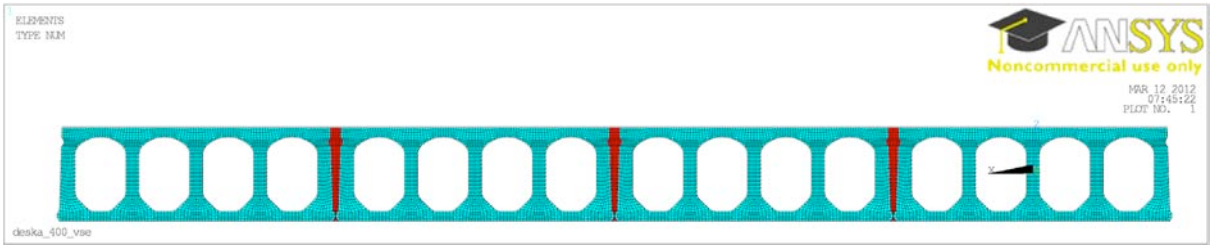

Fig. 2 Front view on the slab including in-situ concrete

The slab is in the loading test loaded trough complicated load system by resultant force 295,1 kN at the outside panels and by resultant force 292,2 kN at the central panels. Slab in the load test was destroyed by shear at these forces. This load is modeled as pressure on simple steel plates with dimensions 0,1 x 1,155 m and thickness 1 mm. These plates are located 1,2 m from the panel forehead.

The supports are modeled as elastic from finite elements LINK 11 with defined stiffness. The model stiffness is the same as stiffness of the real supports from the load test. Stiffness is calculated through an iteration procedure.

The calculation is performed in 10 load steps using Newton-Raphson numerical method. The results from every step are saved in order to obtain the curve of dependence deformation on increasing load value.

The results are evaluated in the same places as they were measured in the loading test. The values of deformations are compared.

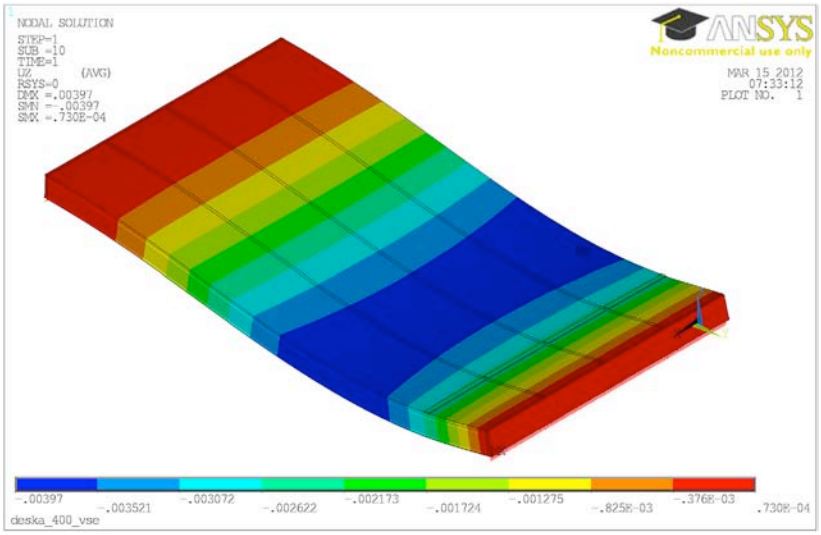

Fig. 3: Deformations of the slab under full load.

2. Conclusions

Only results after first iteration of support stiffness are presented. Shear stresses are allocated correctly in panel ribs.

It is necessary to made more iterations of support stiffness to achieve better agreement with measured deformations.