MODELOVANIE A ANALÝZA DODATOČNE PREDPÄTÝCH ĽAHKÝCH BETÓNOVÝCH NOSNÍKOV S VYUŽITÍM NELINEÁRNYCH MODELOV

MODELLING AND ANALYSIS OF POST-TENSIONED LIGHT WEIGHT AGGREGATE CONCRETE BEAMS USING NONLINEAR MODELS

Omar BACARREZA[1], Jan L.VÍTEK[2], Petr KONVALINKA[1]

Abstrakt

Článok sa zaoberá praktickými aplikáciami nelineárnych modelov betónu pre prípady, keď sa očakáva nelineárna odozva zaťaženého betónu. Skúma sa numerický model upravený pre komerčný program na metódu konečných prvkov – ANSYS. Výsledky testov pri kritickom zaťažení ľahkých dodatočne predpätých betónových nosníkov boli použité pre zistenie vhodnosti materiálového modelu betónu implementovaného do programu ANSYS pre predikciu kritickej odozvy týchto nosníkov.

Kľúčové slová: Betón; dodatočne predpätý; modelovanie konečnými prvkami.

Abstract

This paper deals with the practical application of nonlinear models of concrete when nonlinear response of concrete under loading is expected to occur. The numerical model adopted by the commercial finite element software ANSYS is discussed. Results of ultimate load tests on post-tensioned light weight aggregate concrete beams were used to assess the suitability of the concrete material model implemented in ANSYS in predicting the ultimate response of these beams.

Keywords: Concrete; post-tensioning; finite element modelling.

INTRODUCTION

Beams from Light Weight Aggregate Concrete (LWAC) were produced and prestressed after 7 days. They were loaded after 28 days with a loading block for a period of one year and finally loaded until failure; the results from these tests are compared to a computer model created in the general purpose finite element code ANSYS.

To obtain the computer model as close as possible to the experiment, all the material properties of the concrete (modulus of elasticity, ultimate strength, etc.) used in the computer model were obtained from the experimental data. The values obtained from the measurements were calculated according to the Czech national standards. The material properties of the prestressing steel were provided by the producer.

The deformations were measured using the vibrating wire strain gauges and the deflections with potentiometers. The computer model uses a non-linear geometric and non-linear elastic material mode.

[1] Ing. Omar BACARREZA, doc. Ing. Petr KONVALINKA, CSc., ČVUT v Praze, omar.bacarreza@fsv.cvut.cz, petr.konvalinka@fsv.cvut.cz

[2] prof. Ing. Jan L.VÍTEK, CSc., Metrostav a.s. a ČVUT v Praze, vitek@metrostav.cz

Lektoroval: doc. Ing. Jozef BOCKO, CSc., KAMaM, SjF TU v Košiciach, jozef.bocko@tuke.sk

EXPERIMENTAL SPECIMENS (MODELS)

The experimental models are three beams made from post-tensioned LWAC, with a nominal density ρ = 1900 kg/m^3. They are simply supported. The nominal dimensions of the beams are 6600 mm x 600 mm x 260 mm. The span of the beams is 6000 mm between supports.

The beams were prestressed using two monostrands placed 80 mm above the bottom surface and 150 mm from the sides.

In order to monitor the deflections, three potentiometers were placed on the beam (one in the middle of the span and the other two below the point loads); and four vibrating wire strain gauges were embedded in the beams to measure the strains in the middle of the span and in the support region (Fig.1).

The beams were loaded symmetrically and monotonically, in four-point bending mode, with point loads 0.7m either side of the mid-span location, until failure.

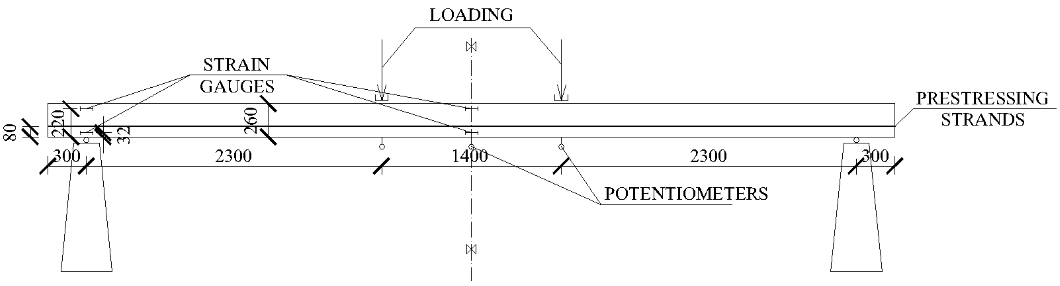

Fig.1 Geometry and loading of the samples

THE FINITE ELEMENT MODEL

The requirement to include the nonlinear response of reinforced concrete in capturing the ultimate response of the post-tensioned beams demands the use of the Solid65 element in ANSYS. The existence of a symmetry plane resulted in a possibility to model only half of the model.

In formulating the model for the post-tensioned beams the three dimensional spar elements (uniaxial tension-compression element with three degrees of freedom at each node) with plasticity, Link8, were employed for the post-tensioning cables. The post-tensioning was modelled via an initial strain in the tendon elements, corresponding to tendon tensile forces, in a preliminary load stage with the self weight of the structure. Subsequently the loading to failure was applied in the next stage in a manner consistent with the test data. The model was meshed using bricks.

MATERIAL MODELS

ANSYS provides the element Solid65, a three-dimensional eight noded solid isoparametric element, having three degrees of freedom at each node, to model the nonlinear response of brittle materials. This element uses the criterion for failure of concrete due to a multiaxial stress state after Williams and Warnke.

The element includes a smeared crack analogy for cracking in tension zones and a plasticity algorithm to account for the possibility of concrete crushing in compression zones. Each element has eight integration points at which cracking and crushing checks are performed. The element behaves in a linear elastic manner until either of the specified tensile or compressive strengths are exceeded. Cracking or crushing of an element is initiated once, if one of the element principal stresses exceeds the tensile or compressive strength of the concrete, respectively (in an element integration point). Cracked or crushed regions, as opposed to discrete cracks, are then formed perpendicular to the relevant principal stress direction with stresses being redistributed locally. The element is thus nonlinear and requires an iterative solver. In the numerical routines the formation of a crack is achieved by the modification of the stress-strain relationships of the element to introduce a plane of weakness in the requisite principal stress direction. The amount of shear transfer across a crack can be varied between full shear transfer and no shear transfer at a cracked section. The crushing algorithm is similar to a plasticity law in that once a section has crushed any further application of load in that direction develops increasing strains at constant stress. Subsequent to the formation of an initial crack, stresses tangential to the crack face may cause a second, or third, crack to develop at an integration point.

The link modelling allows the elastoplastic response of the reinforcement to be included in the simulation.

RESULTS AND DISCUSSION

The values obtained from the computer model are compared to the ones obtained from the experiments, the load-deflection response and the load-strains graphs are presented in the figures below.

The numerical model predicts an ultimate load of around 60.5 kN and captures relatively good the nonlinear behavior of the load deflection response. The ultimate load reached in the experiment is 62.6 kN. It can be seen in the numerical model that the response of the model is linear until the first crack has formed at approximately 25kN which is the same as in the experiment. After this point the load-deflection response of the numerical model and the experiment is different, with the load-deflection response of the numerical model being almost linear, and the experimental being curved.

The midspan deflection at maximum load recorded from test is about 107 mm, while the deflection obtained in the numerical model is 116 mm. The ultimate deflection from the test is 163 mm meanwhile Ansys cannot record the post failure behavior of the model.

It can be seen in the load-strain graph that the strains in the top fibers obtained from the experiments and numerical model are very close, while those in the bottom fibers behave in similar manner as the load-deflection response.

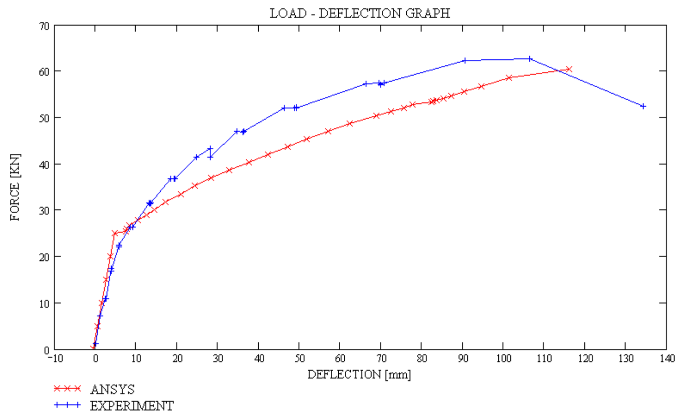

Fig.2 Load-deflection response

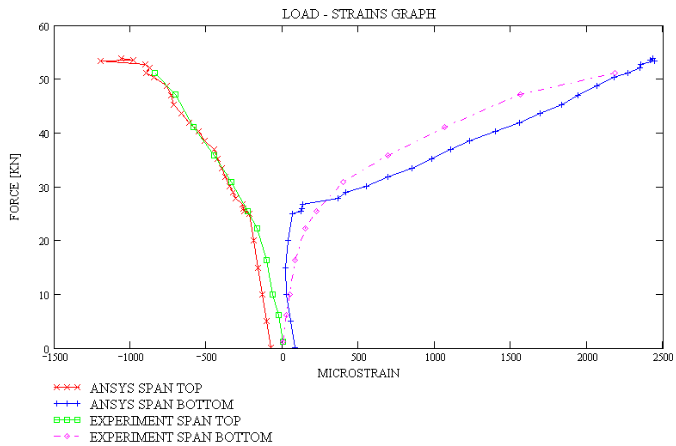

Fig.3 Load-strains graph

CONCLUSIONS

The numerical model for the post-tensioned beams was consistent with the test results. Clearly the correlation of test and numerical data depends on the assignment of accurate linear and non-linear material properties as appropriate which might be a rather complicated task.

This research has been supported by Grant Agency of the Czech Republic project No. 103/03/0952 and project No. 103/06/1474.

REFERENCES

[1] ANSYS, ANSYS Manual Set, 1998, ANSYS Inc., Southpoint, 275 Technology Drive, Canonsburg, PA 15317, USA.

[2] WILLIAM, K.J., WARNKE, E.D.: *Constitutive Model for the Triaxial Behaviour of Concrete*, Proceedings of the International Association for Bridge and Structural Engineering, 1975, 19, p.174, ISMES, Bergamo, Italy

[3] ČSN ISO 6784 *Stanovení statického modulu pružnosti v tlaku*.

[4] ČSN ISO 4012 *Stanovení pevnosti v tlaku zkušebních těles*.