IOP Conf. Series: Materials Science and Engineering 456 (2018) 012095 doi:10.1088/1757-899X/456/1/012095

Features of finite element analysis of steel-reinforced concrete slabs from hollow core slabs

A R Tusnin and A A Kolyago

National research Moscow State University of Civil Engineering Russia, Yaroslavskoe shosse 26, 2129337, Moscow, Russia

alex777002@gmail.com

Abstract. The present article is devoted to the calculation and finite element analysis of steel-concrete composite slabs from precast hollow-core slabs in the software package ABAQUS with the use of ABAQUS/Explicit and ABAQUS/Standard solvers taking into account the installation stage. The behaviour of the fragment imitating the steel-reinforced concrete work section on the basis of a beam with a zone of equal deformations is considered. The load was applied to two beam nodes at a distance of 0.5 m from the support, thus forming a zone of pure bending at the site of the slab's inclusion in the beam work. The inclusion in the work of precast hollow-core slabs is provided by the installation of stud anchor on the slab's ends welded to the upper flange of the beam, while the length of the support plates increasing. Based on finite element analysis, carried out engineering and theoretical studies, the authors obtained dependences describing the behaviour of steel-concrete composite slabs. The analysis of the results showed good convergence of the finite element analysis results and theoretical solutions. The discrepancy between the results of finite element analysis and theoretical methods was less than 5%.

1. Introduction

The interest in finite element analysis of steel-concrete structures of buildings is very high because of the problem importance, which solution allows to build in a shorter time, more rigid and durable, as a consequence, lighter, that also allows you to save on the foundations of the building and erect them where it was impossible.

The complexity of steel-concrete composite slabs simulation lies in the nonlinearity of construction and the necessity of considering installation stage. For these reasons, software used for the modeling of steel-concrete structures, that allow not only to model the design with the physical and geometric nonlinearity, but also to maintain the initial stress-strain state after loading installation loads (own weight of plates, weight of the concrete, etc.) and transfer it to the next steps, which takes into account the composite work of steel-concrete structures materials. The model is usually created by3D-elements (which are better suited for such problems, more accurate), since the difference of stresses in the structure is large, it leads to an earlier manifestation of the material nonlinear properties in separate zones. As well as there is a need to take into account the contact interactions between structural elements.

Abaqus was chosen to calculate the steel-concrete slab from hollow core slabs with stud anchors which has a lot of numerical models of materials, such as Concrete Damage Plasticity, which allow to obtain a solution of such problems with sufficient accuracy [1, 2]. The problems of numerical calculation of steel-reinforced concrete structures are considered in [4 - 6]. The numerical calculation takes into account the nonlinear properties of materials, the deformation of the bonds between a plate and a beam, the sliding of anchors relative to the concrete. To solve a nonlinear problem, software ABAQUS [4] and ANSYS [8] were successfully used. The results of numerical calculations are compared with experimental data [4, 7 - 9]. The analysis of the scientific literature has shown the successful application of the combination of volume, plane and rod elements for nonlinear calculations. Experimental verification of numerical calculations confirms the reliability of the computational method.

2. Problem setup

To solve the problems of modeling behavior and determining the stress-strain state, the implicit (ABAQUS/Standart) and explicit (ABAQUS/Explicit) solvers were used to obtain initial stresses in the structure from the installation stage loads and the subsequent quasi-static application of the load in the stage, taking into account the composite interaction features of the steel beam and hollow core slabs.

The model has dimensions 3x3 meters and consists of 3 main elements – beams, precast concrete slabs and interface (Figure 1).

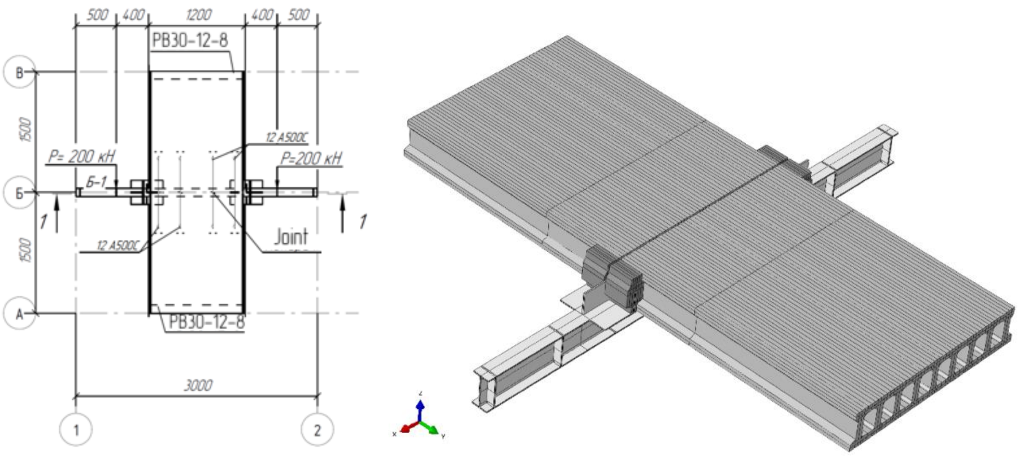

Figure 1. General view.

The I-beam 20B1 3 m long (Figure 2) with welded stud anchor (Figure 3) is divided into finite elements C3D8R with a size of approximately 40 mm.

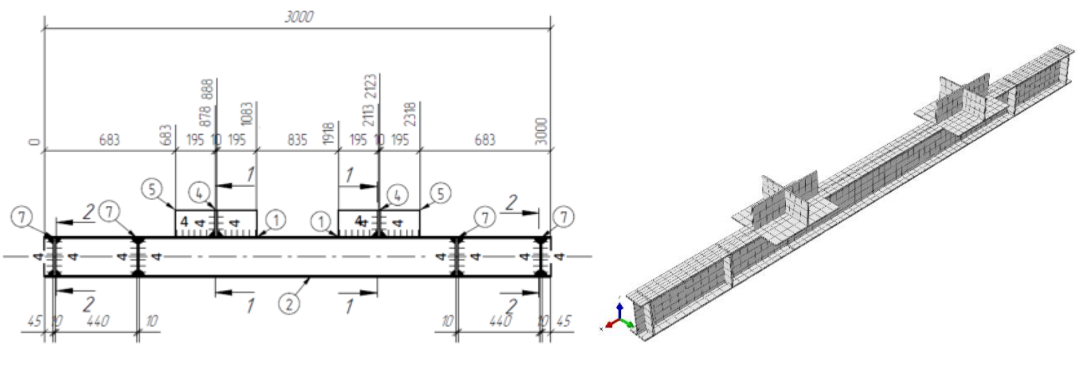

Figure 2. General view of beam.

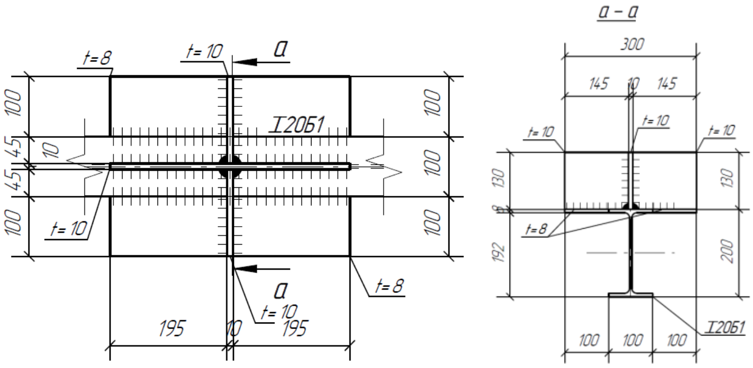

Figure 3. Stud anchor.

3. Methods

Precast slab PB-30-12-8 1.5 m long is divided into finite elements C3D8R with a size of approximately 40 mm (Figure 4). The concrete was modeled using the built-in model CONCRETE DAMAGE PLASTICITY module PROPERTY [ABAQUS/CAE User's Guide. 12.9.2 Defining plasticity] with parameters corresponding to concrete B25 (Table 1). Reinforcing modeled by T3D2 elements using the STRINGERS function of the PROPERTY module [ABAQUS/CAE User's Guide] module. 36. Skin and stringer reinforcements]. For down strands of reinforcing ropes the function MULTIPLE STRINGS were used (Figure 4), which allows assigning multiple wires in one split location to simplify the grid. Reinforcing bars were set by a linear material with Young`s modulus 147000 MPa for BP-I wire in the upper part of the plate and 150000 MPa for K-7 ropes in the lower part of the plate. Poisson's ratio was taken to be 0.33.

Table 1. Property parameter for Concrete Damage Plasticity.

Young`s modulus, MПa	Poisson`s Ratio	Dilation Angle	Eccentricity	fb_0/fc_0	K	Viscosity Parameter
300000	0.2	31	0.05	1.16	0.667	0

Table 2. B25 concrete parameters for Concrete Damage Plasticity model.

Compressive Behavior		Tensile Behavior	
Yield Stress	Inelastic Strain	Yield Stress	Cracking Strain
14.5	0.00	1.1	
18.5	0.002	1.5	0.0001
25	0.0035	1.8	0.0015
0.25	0.00525	0.15	0.0025

The function DEACTIVATED IN THIS STEP of the INTERACTION module were used [ABAQUS/CAE User's Guide was used. 15.13.13 Defining a model change interaction], to obtain the initial stresses in the construction, allowing to turn off the calculation items, modeling interface between stud anchors and precast hollow-core slabs. To get shaped form of interface due to the initial curvature of the steel beam, it was necessary to use the RESTART function of the JOB module [ABAQUS/CAE User's Guide. 19.6 Restarting an analysis] and then return the elements of interface, removed from the model using the function REACTIVATED IN THIS STEP of the INTERACTION module. The REACTIVATE ELEMENTS WITH STRAIN function is disabled so there are no deformations and stresses is formed in the interface elements, as after solidification of the concrete mixture.

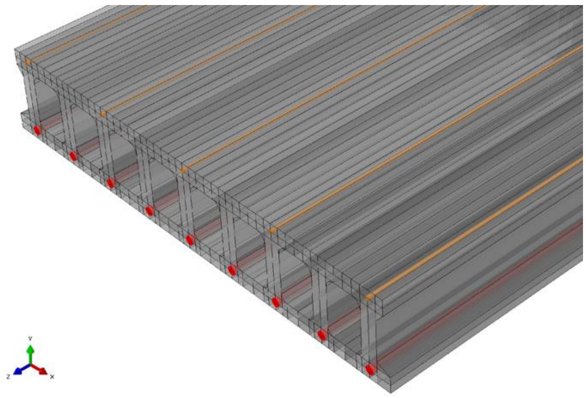

Figure 4. Precast hollow-core slab reinforcement.

4. Results

The model was calculated at the stage of installation in ABAQUS/Standart taking into account geometric and physical nonlinearity. A pressure load of 3.3 kN/m^2 was applied to the plates, corresponding to the weight of the plates. Initial stresses in the beam were obtained in structural elements (Figure 5) and other elements of the structure. As can be seen from the distribution of stresses in the beam and the location of the neutral axis of beam corresponds to the simple beam on 2 supports without including the plates.

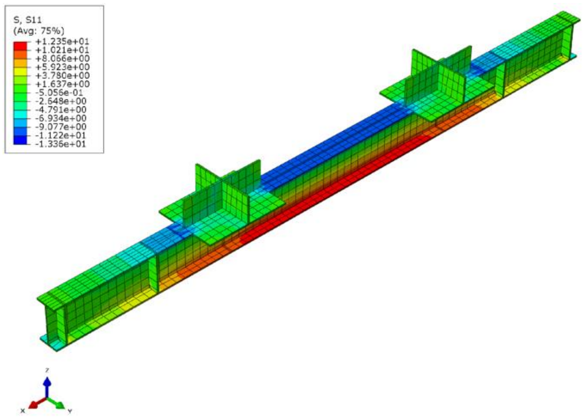

Figure 5. Initial normal stresses in the beam, MPa.

Using the IMPORT function [ABAQUS/CAE User's Guide] function. 10.8 Importing a model] was able to obtain structural elements and continue the calculation using ABAQUS/Explicit solver. In order to be in the initial stress-strain state before the application of the actual load for the structural elements the INITIAL STATE function of the LOAD module [ABAQUS/CAE User's Guide was used. 16.8.3 Creating predefined fields], elements modeling interface was specified material, appropriate for concrete B30.

Further, the calculation was made for an additional concentrated load applied to the beam nodes at a distance of 0.5 m from the supports, supported by ribs, thus simulating a zone of pure bending in the beam section, where composite work with the plate is provided and a steel-concrete section is formed. The load at each point was 100 kN (Figure 6).

The stress-strain state of the structure obtained in the calculation corresponds to the concepts of the steel-concrete slab-normal work. Tensile stresses in the beam prevail as a result of the neutral axis shift of the cross section after interfaces are frozen (Figure 7). In the plates appeared a compressed zone along the axis of the beam, which also indicates the inclusion of plates in the work (Figure 8).

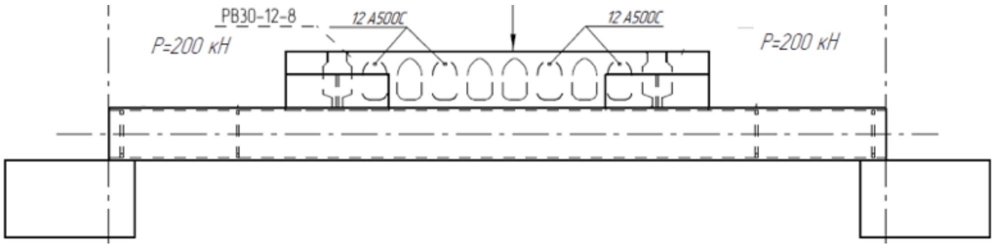

Figure 6. Loading scheme of structure.

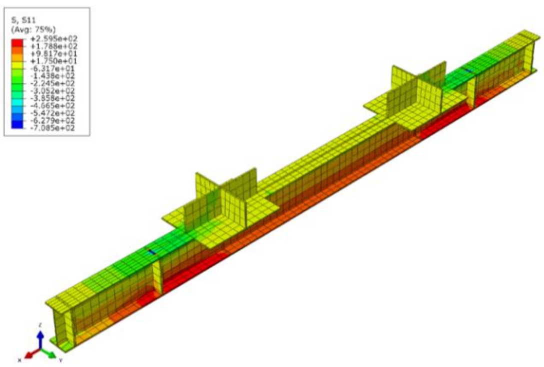

Figure 7. Normal stresses in the beam, MPa.

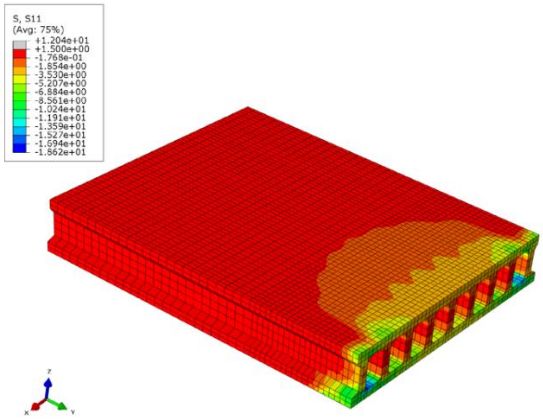

Figure 8. Normal stresses in the plate along the axis of the beam, MPa.

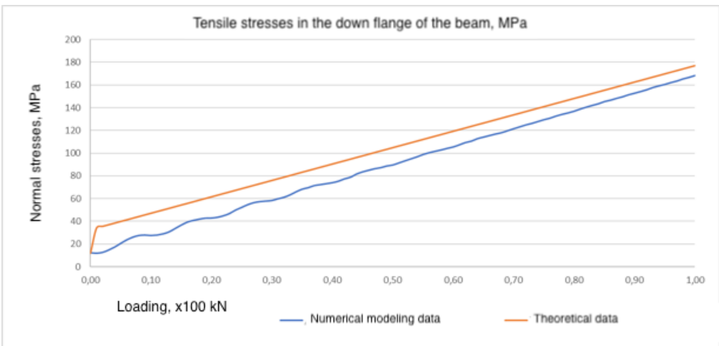

Figure 9. Tensile stresses in the down flange of the beam, MPa.

Theoretical studies were carried out on the basis of the reduced steel-concrete section, which characteristics were taken on the basis of claim 8.1.11SP63.13330.2012 Concrete and reinforced concrete structures, where it is said that the width of the concrete element in the cross section, entered into account, take the condition that the overhang shelf width in each direction from the edge should be no more than 1/6 span of the element. Therefore, the cross-section of steel-concrete composite beams has been introduced 0.25 m plate on each side on the basis of the slabs 1.5 m. span.

In order to determine the geometric characteristics of the reduced section, a composite section was compiled, where the shelves of the floor slabs were replaced as follows:

Modulus of concrete in compression was adopted with cal. 0.8 from the initial, thus amounted to 2. 4E6 t/m^2. The modulus of steel elasticity was adopted in the calculations of 2.1E7 t/m^2. The coefficient of the reduced module = 0.114. Thus, the thickness of the plates brought to the shelves was 3.47 mm. The geometric characteristics of the section are given in Table 3.

Table 3. Geometric characteristics of steel-concrete section.

Geometric characteristics		
Parametre	Value	Units
I_y	9758.857	cm^4
I_z	8661.835	cm^4
I_t	5092.262	cm^4
i_y	12.961	cm
i_z	12.211	cm
W_{u+}	513.157	cm^3
W_{v+}	346.473	cm^3

This modeling method of steel-concrete composite slabs from hollow core slabs with stud anchors showed good convergence of the numerical and theoretical solutions results (Figure 9). The discrepancy between the simulation results and the theoretical method was no more than 5%, which gives grounds for planning a full-scale experiment for verification of the numerical model.

References

[1] Silantiev A S 2012 *Ind. Civil Eng.* **2** 71–4
[2] Belostotskii A M and Vershinin V 2013 *Development of a verification report on the use of the Abaqus software package for solving construction profile problems* (Moscow: Moscow State University of Civil Engineering) pp 81–92
[3] ABAQUS/CAE User'sGuide
[4] Thevendran V, Chen S, Shanmugam N E and Liew J Y R 1999 *Finite Elements in Analysis and Design* **32** 125–39
[5] El-Lobody E and Lam D 2003 *Adv. Struct. Eng.* **6** 267–81
[6] Lam D and El-Lobody E 2005 *J. Struct. Eng.* **131** 96–107
[7] Quan Liang Q, Uy B, Bradford A and Ronagh H R 2004 *J. Const. Steel Res.* **60** 1109–28
[8] Mahmoud A M 2016 *Ain Shams Eng. J.* **7** 73–88
[9] Duan S J, Wang J W, Zhou Q D and Wang H L 2010 An experimental study on double steel–concrete composite beam specimens *Challenges, Opportunities and Solutions in Structural Engineering and Construction* (Boca Raton, USA: CRC Press) pp 209–14