FEM analysis of steel-concrete composite bridges with encased perforated steel strips

Patrícia Vaňová[1,*], *Vincent* Kvočák[1], *Viktória* Kožlejová[1], *Daniel* Dubecký[1] and *Ruslan* Kanishchev[1]

[1]TUKE, Faculty of Civil Engineering, Vysokoškolská 4, 042 00 Košice, Slovakia

Abstract. One of the newest contributions to the science of bridges are steel-concrete composite bridges with encased perforated steel strips. Their main advantages are their low height, low usage of materials and therefore lower cost of the bridge. They are mainly used as short span railway bridges (under 20m of length). Main focus of today researchers and structural engineers in this area is to improve the geometry of the steel strip – and therefore find new types with lower usage of material and higher shear strength. At Technical University of Košice, Faculty of Civil Engineering, several new types have been developed and tested. In order to properly measure the shear strength, the push-out tests had to be carried. They track the dependence between load (typed by loading conditions) and slip. These experiments were performed at Laboratory of Excellent Research in Košice. In this article is closely described software modelling of a strip developed at Faculty of Civil Engineering, the experiments performed as well as the results found.

1 Introduction

Numerical study of performed experiments is important in order to fully watch and understand inside stresses and behaviour of the specimen during the testing as well as for the parametric study.

Several papers were published about this topic in the past. Oguejiofor and Hosain have described one of the first finite element analysis of push-out tests of specimens resisting shear which contained steel perfobond strip, as early as 1995. They created the numerical model in ANSYS [1]. Bouchair et al. did the analysis of push-out tests of specimens with shear studs in ATENA software [2]. Nguyen and Kim analysed the experiments using ABAQUS software [3].

In this article the FE modelling approach of the push-out tests performed onto the specimens with continuous shear connectors using ABAQUS/CAE software is closely described.

---

[*] Corresponding author: patricia.vanova@tuke.sk

2 Geometry

The specimens tested were made out of I-beam IPE 220 cutted longitudinally. The steel strips contained holes of 20mm in diameter within 100mm from each other, placed in the middle of the strip by height. 12mm reinforcement bars, marked 10 505 (R), were placed through each hole.

3 Push-out test

Standard push-out tests were performed onto three specimens. The load was measured by hydraulic press software and the slip by two inductive displacement sensors placed at the bottom of the middle steel metal.

a)   b)
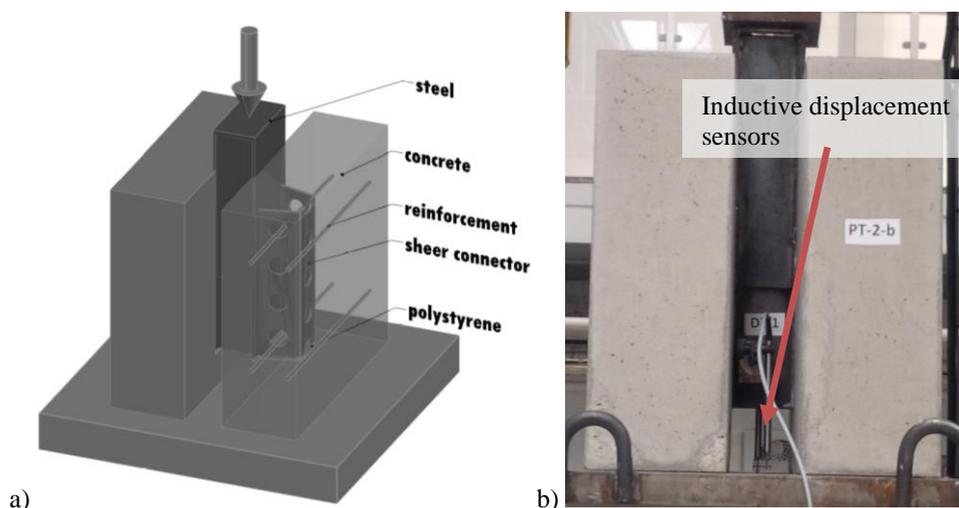
Fig. 1. Push out-tests a) Arrangement of the specimen, b) The real specimen in hydraulic press.

4 FEM model

The entire experiment was simulated in ABAQUS/CAE software [4]. The goal was to find a model of which results had lower deviation than 60%, suitable for usage in the further parametric study.

4.1 Material properties

4.1.1 Steel

Steel used for continuous shear connectors was tested by Faculty of Metallurgy at Technical University of Košice. Tensile tests were performed onto three samples. Yield point was set at the 315.3MPa (Fig. 2). For Young's Modulus and Mass density the normative values were used.
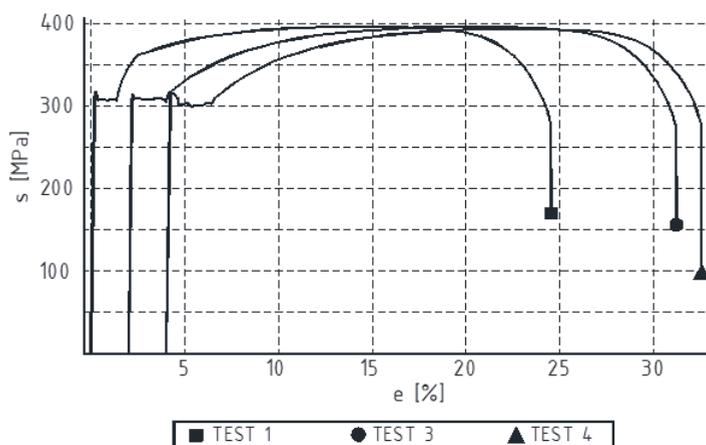
Fig. 2. Tensile tests of steel used for continuous sheer connectors.

4.1.2 Concrete

For two concrete blocks, the concrete marked C30/37 of European quality was used. Compressive tests and four-point flexural tests were provided by Laboratory of Excellent Research. The characteristics used for the FEM model are shown in Table 1.

Table 1. Material characteristics of concrete used in software model.

|				
DENSITY		Mass density	2.54E-09	tonne/mm3
ELASTIC		Young's Modulus	33 423.80	MPa
		Poisson's ratio	0.2	-
CONCRETE DAMAGED PLASTICITY	Plasticity	Dilatation Angle	30	°
		Eccentricity	0.1	mm
		fb0/fc0	1.16	-
		K	0.667	-
		Viscosity Parameter	0	-
	Tensile behaviour	Yield Stress	3.7	MPa
		Cracking Strain	0	-
	Compressive behaviour	Yield Stress	39.72	MPa
		Cracking Strain	0	-

4.2 Interaction

Option 'general contact' was chosen to perform the interaction between steel and concrete parts. 'Hard contact' was set as a global property assignment. One individual property assignment was present between the surface of the strip and the surface of concrete, which comes into contact – tangential behaviour was added, with friction coefficient equal 0.12.

4.3 Load

In the load module, firstly, the boundaries were fixed – as pinned at the bottom of both concrete blocks. Secondly, loading was done in 114 loading conditions, from which 50LCs created the cycle by dropping and increasing between 100kN and 350kN, while the specimen still acted elastically. Maximum load, after which the specimen broke, was 1200kN.

4.4 Mesh

The defined element for both steel and concrete parts was 4-node linear tetrahedron C3D4. 8-node reduced integration C3D8R element was selected for reinforcement.


a)   b)
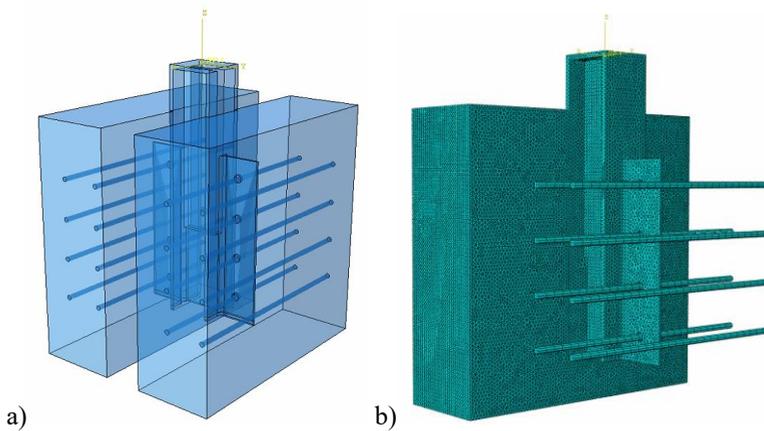
**Fig. 3.** ABAQUS model a) General model, b) Meshed model.

5 Results

The shear resistance of the specimen was determined at 750kN – at the end of the first, linear (elastic), section of the diagram (Fig. 4), during which the specimens were in good condition, no cracks could be seen. During the second section the slip started to grow and visible cracks were developed. After reaching 1050kN the slip started to rapidly rise – until the polystyrene between steel and concrete was compressed, the steel strip started to push into the concrete and the specimen was breached. This – third – section was accompanied by visible crack propagation as well as the loud cracking sound effects.

FEM model showed similar results (Fig. 5). The shear resistance was obtained at the same LC – 750kN, but presented with 0.5mm higher slip, which manifested the safe side of the FEM model. The most stressed place of the steel strip occurred at the interaction point between the border of hole and the reinforcement bar.
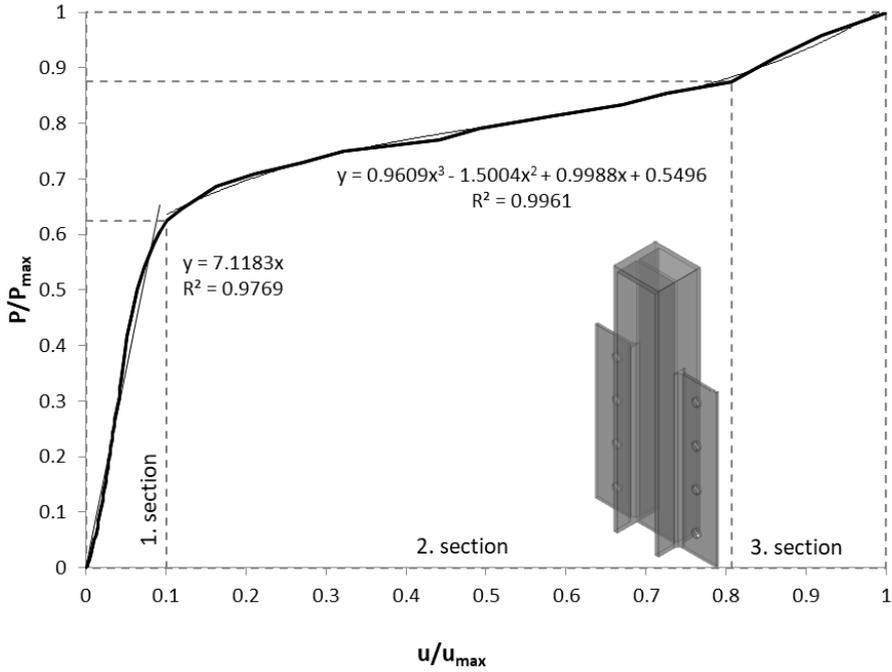
Fig. 4. Diagram showing results of push-out tests.
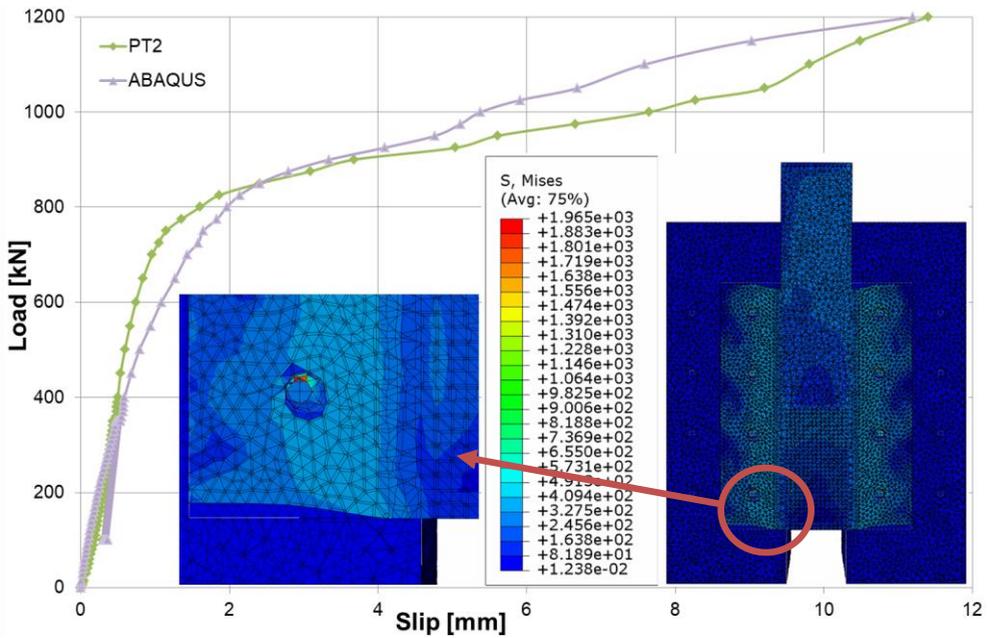
Fig. 5. Comparison between experiment results and the FEM software results.

6 Conclusion

The finite element model was created in ABAQUS software to simulate the real experiment of push-out tests. The deviation between slip of the experiment and FEM model was lower than 60% during whole process, except for the LCs #2-7 (5 – 50kN). This could be explained by minor shortcomings between the concrete blocks and steel pad of hydraulic press, which could not be put into the software model. In conclusion, the FEM model proved to be sufficiently accurate for use in further parametric study.

References

1. E. C. Oguejiofor, M. U. Hosain, *Computers & Structures* **62**, 4 (1995)
2. A. Bouchair, J. Bujnak, P. Duratna, A. Lachal, *Steel Structures and Bridges* **40**, 102 – 107 (2012)
3. H. T. Nguyen, S. E. Kim, *J. of Constr. Steel Research* **65**, 1909 – 1920 (2009)
4. ABAQUS/CAE: User's Manual. Available on the internet:

    <https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usi/default.htm?startat=pt03ch15s12hlb03.html>