12th International Conference on Sandwich Structures (ICSS-12) Lausanne - Switzerland, 19–22 August 2018
DOI: 10.5075/epfl-ICSS12-2018-102-104

TESTING AND FEM ANALYSIS OF A NOVEL FRP SANDWICH BRIDGE DECK

Maciej Kulpa[1] and Tomasz W. Siwowski[2]
[1]Rzeszow University of Technology, Poland. kulpa@prz.edu.pl
[2] Rzeszow University of Technology, Poland. siwowski@prz.edu.pl

1. INTRODUCTION

The feasibility and potential of FRP sandwich systems in civil engineering and construction have been successfully demonstrated by various applications including bridge decks and girders [1]. The growing need of durability enhancement of the Polish road bridges has recently caused the big impulse for research on new, durable, lightweight and easy to handle bridge decks, made of FRPs (fibre reinforced polymers). Therefore, within the framework of UE 7FP PANTURA project, three structural solutions of FRP sandwich deck fabricated by VARTM technique were elaborated, fabricated and tested under static load (Fig.1). On the basis of initial test results, the stiffness, load carrying capacity and dynamic behaviour of panels were estimated and the best solution for further research was chosen [2].

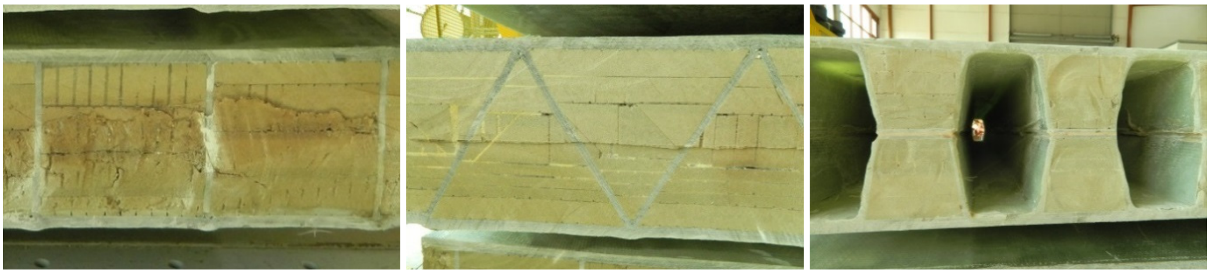

Fig. 1: Three structural solutions of sandwich FRP bridge deck elaborated within the framework of PANTURA project.

The best solution was tested on full-scale specimens to evaluate its behaviour under service, ultimate as well as dynamic load. The panel fulfilled the required criteria for ultimate capacity, serviceability and safety, therefore its application in prototype bridge construction is planned. The detailed FEM model of the panel was also elaborated and after validation against the test results, it is planned to be used in designing of the actual FRP bridge. The deck structure, some test results of the full-scale panel as well as FEM model validation procedure is presented in the paper.

2. SANDWICH DECK CONFIGURATION

According to comprehensive research works [3] and taking manufacturing aspects into account, the deck with trapezoidal ribs and internal openings turned out to be the best solution (Fig.1, right). Except for the best structural behavior, considering strength and stiffness, the following manufacturing aspects also heavily contributed to choosing this solution: simplifying the VARTM fabrication process and its control, improving panel faces quality and savings of core material. The panel is made of two symmetrical parts which are bonded together in the panel's mid-plane. Thus, the resultant sandwich deck is made of GFRP faces and foam core stiffened with the internal GFRP ribs.

The structural form and material selection for faces and core were determined by the initial plate analysis [3]. As a result, the sandwich bridge deck consists of two 17 mm thick GFRP faces and 222 mm thick PVC foam core, having the total thickness of 256 mm. The deck laminates are made of two types of glass fabrics: bi-directional braided fabric B-E 0/90 with the grammage g=800 g/m^2 and bi-directional knitted fabric X-E ±45 with the grammage g=800 g/m^2, and epoxy resin as the matrix. The fabric architecture as well as the overall dimensions of faces and ribs are shown in Fig.2. The 3 mm thick layer of standard epoxy adhesive was used to bond two parts of the deck.

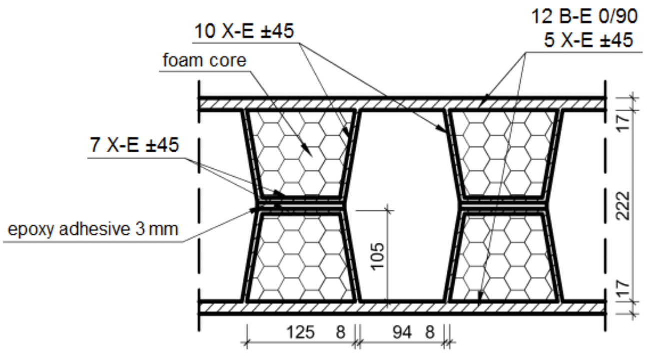

Fig. 2: FRP sandwich deck configuration.

3. FULL-SCALE DECK PANEL TESTING

Test Scheme

The static tests were carried out on the full-scale deck panel with the dimensions of 5.3 × 1.9 m. To simulate the actual bridge deck behavior, the two-span scheme of the panel 2 x 2.4 m was applied in the test. The LM-1 loading scheme according to Eurocode [4] was chosen to evaluate panel static behavior under bending and shear (Fig.3). Panel displacements and composite strains of the top and bottom faces were measured in several dozen points of the panel.

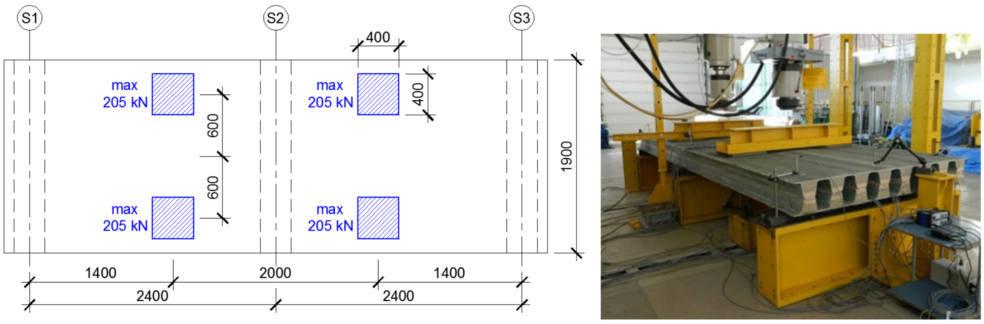

Fig. 3: LM-1 loading scheme and the full-scale panel on the test stand.

Panel Behavior

The behavior of the panel under static load was linear until failure. The exemplary "load-displacement" and "load-strain" plots in full range of static loading are shown in Fig.4. The initial plot curvature is due to support adjustment in the first loading phase. The mid-span deflection under service design load was only L/627, which is more than two times less than the allowable value assumed in design (L/300). Maximum mid-span strain in composite of the bottom face was about 1.25 ‰, which constitutes only 7% of the GFRP failure strain. No failure of the panel was observed under the full-range loading, i.e. the total load of 820 kN, which corresponds to the design load of LM-1 model according to Eurocode [4].

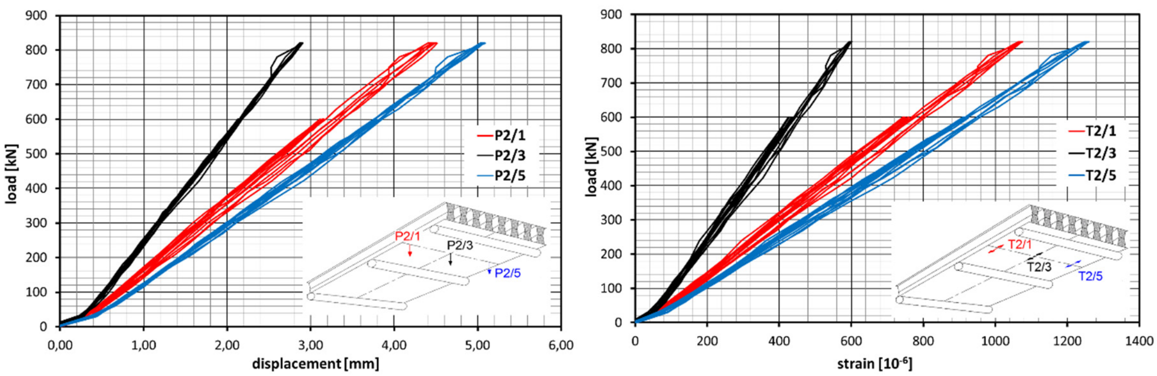

Fig. 4: Exemplary "load- displacement" (left) and "load- strain" (right) plots.

4. FEM ANALYSIS

ABAQUS code was used for FEM analysis of the panel. All panel laminates were modelled with 4-node shell elements based on Mindlin's plate theory. The 25 × 25 mm shell elements had the assumed thickness taking the actual number of laminas into consideration. Additionally, the 8-node solid elements were used to model the bondline as well as supporting steel plates and contact elements. To simplify the FEM calculations, foam core was not modelled but this is a safe side design assumption. The numerical model of the FRP sandwich panel consisted of total of 75 072 shell elements and 13 008 solid elements (Fig.5).

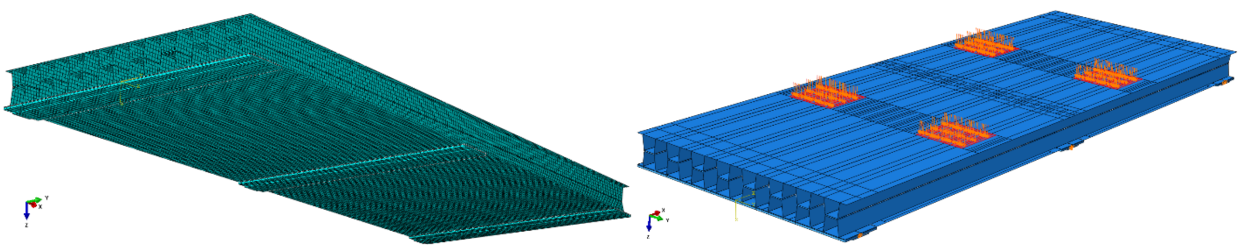

Fig. 5: FEM model of the panel and the loading.

The characteristic parameters for the both bi-directional glass-fiber laminas were determined by material testing and used to define the orthotropic layers in shell elements. The testing load was modelled by means of uniform pressure applied on four squares with the dimensions of 0.4 x 0.4 m according to Eurocode [4].

The FEM analysis results were obtained in the form of displacement or strain maps in each particular laminas of all panel composites (faces and core ribs). The exemplary displacement and strain maps are shown in Fig.6.

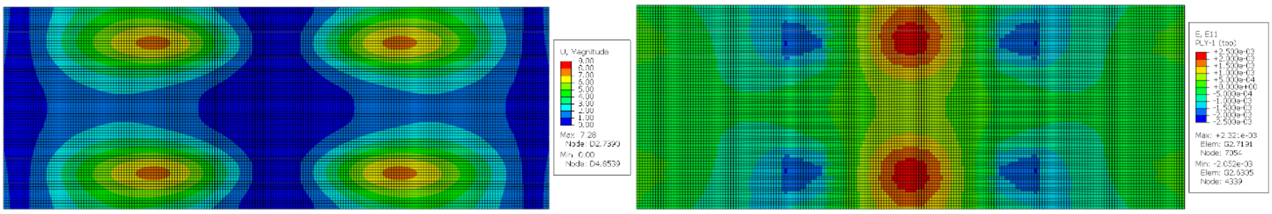

Fig. 6: Displacement map of the bottom face laminate (left) and longitudinal strain map of the top face laminate under the LM-1 characteristic loading.

5. FEM MODEL VALIDATION

The FEM model validation was performed by comparing the experimental and numerical results obtained in each discrete displacement and strain measurement point. Two plots of arbitrary displacements and strains in the mid-section of panel span under the characteristic LM-1 loading are shown in Fig.7, to compare experimental and numerical values.

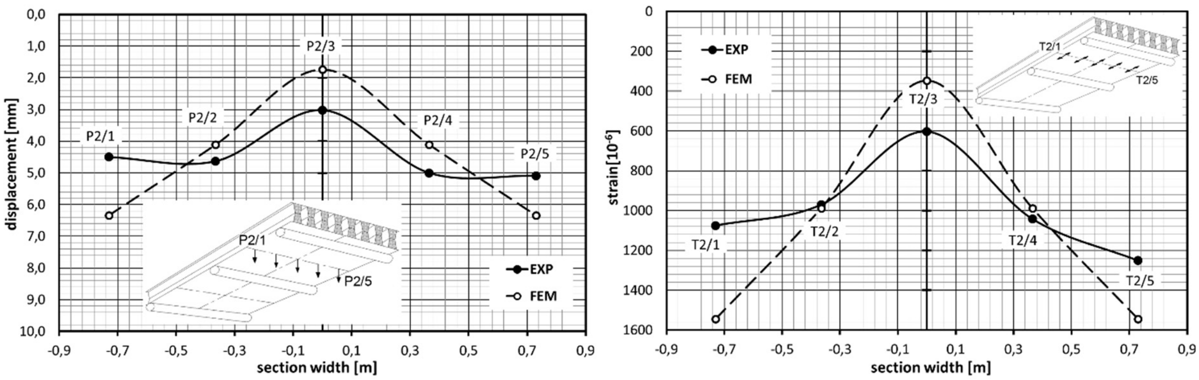

Fig. 7: Experimental versus numerical displacements (left) and strains (right) comparison.

The comparison shows that experimental displacements and strains close to panel edges are 25% lower than the numerical ones but the latter are as high as in the panel's longitudinal axis. The transverse stiffness of the actual panel was higher than the relevant stiffness obtained numerically. The basic reasons of the difference between experimental and numerical behavior of the FRP sandwich panel are: foam core was not modelled and the "pressure" patch load did not exactly simulated the actual bearing. Moreover, the material parameters applied in FEM analysis were determined for laminas not laminates. Finally, some internal flaws and geometrical nonconformities decreasing the overall quality of the actual panel also influenced these differences. Because the average differences for particular load cases were too big, it was decided to modify the numerical model. However, having in mind the application of the FEM model in practical design, the numerical model that does not increase the computation time has been created.

As the transverse stiffness of panel was the main reason of FEM/experiment discrepancy, the material parameter adjustment seemed to be the most efficient way to modify numerical model. Using *lamina* material model implemented in ABAQUS code, trial and error procedure was applied to find equivalent material properties in particular laminas and to obtain the displacement and strain compatibility in most load cases and points of the panel. The procedure was stopped if the average discrepancy between experimental and numerical values was below 15 %. In this way, the final design model was created without increasing the number of finite elements and computation time [5]. Moreover, the further research carried out on the novel FRP deck panel confirmed its satisfactory strength and stiffness to be implemented on site in bridge redecking.

REFERENCES

[1] A.Manalo et al., "State-of-the-art review on FRP sandwich systems for lightweight civil infrastructure", *J. Comp. Constr.*, 2017; 21: 16 p.

[2] M.Kulpa et al., "Structural shaping of FRP bridge decks", *J. Civ. Eng., Env. Arch.*, 2015; XXXII: 21 p. (in Polish).

[3] M.Kulpa et al., "Modelling and numerical analysis of GFRP composite panels of bridge deck", *3rd Polish Congress of Mechanics & 21st Computer Methods in Mechanics*: 2015, 2p.

[4] PN-EN 1991-2: *Eurocode 1: Actions on structures. Part 2: Traffic loads on bridges*, PKN, Warsaw: 2011.

[5] M.Kulpa, Strength and stiffness of FRP bridge decks, Dissertation, Faculty of Civil and Environmental Engineering and Architecture, Rzeszow University of Technology, Poland, 2017 (in Polish).