PAPER • OPEN ACCESS

Analysis of an experimental tension structure

To cite this article: L Kapolka 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **566** 012014

IOP Conf. Series: Materials Science and Engineering **566** (2019) 012014 doi:10.1088/1757-899X/566/1/012014

Analysis of an experimental tension structure

L Kapolka[1],

[1] Department of Structural Engineering, Faculty of Civil Engineering, Technical University of Kosice, 042 00 Košice, Slovakia

Abstract. This article builds on the article written last year. The paper is the comparison between the results from the first numerical pre-analysis of the construction of an adaptive hyperbolic paraboloid, the results of which were used as input data for the design of the experimental device in the laboratory at the Faculty of Civil Engineering of the Technical University in Košice with the newest results gained from the numerical model created according to documentation from the manufacturer of this device. The structural numerical model was modeled in regard to geometry, materials and cross-sections characteristics, weight distribution, existing eccentricity and it was created in Dlubal software based on finite element method.

1. Introduction

Following the results of my previous post [1], the results of which were used as input data for the design of experimental frame of adaptive hyperbolic paraboloid structure in a laboratory, it was necessary to verify and compare the results of pre-analysis with the results of exact prototype, which was calculated in finite element method software according to specification after delivery by the external contractor, Inova Praha. Creating more precise calculation model consisted of studying the documentation from the manufacturer of device, followed by creating of an axis-spatial model in AutoCAD software according to the corresponded documentation and afterwards the geometry was imported into the DLUBAL RFEM 5.17 calculation software [2] and subsequently to individual members were assigned their cross-sections with their associated eccentricities. The joints were modeled at the joint sites, the dimensions of the anchor bars and the action members were adjusted to fit the specified geometric-physical model, and the shape and dimensions of the membrane surface were preserved in accordance with the initial requirements of technical textiles fabrication and also technical specifications of the fabric were included.

2. Specification of construction

The dimensions of the frame in the axes are 3.34 x 3.34 x 2.50 m, cross sections of beams were used according to the documentation as rectangular hollow sections (80/100/6; 140/80/6), square hollow sections (80/80/6 ; 100/100/8; 140/140/8; 160/160/8), sheet metal (PL6; PL30; PL45), anchored to the frame through the "U" profile (135/150/45/50) linked with rods full circular section (D46). The action members were modeled as full circular section (D145). There were used modulus of elasticity E_B = 210 GPa. Edge ropes were modeled as rope cross-section members (D8) with modulus of elasticity E_R = 130 GPa and initial curvature s = 0.20 m. The membrane surface was modeled with parameters of the Serge Ferrari Precontraint 502 technical fabric with a thickness of 0.56 mm, modulus of elasticity in the warp direction is E_x = 1057MPa and in weft direction is E_y = 612 MPa. The modulus of shear was established as G_{xy} = 240 MPa and Poisson’s ratio is υ_{xy} = 1.01. The floor plan dimensions of the technical fabric are

IOP Conf. Series: Materials Science and Engineering **566** (2019) 012014 doi:10.1088/1757-899X/566/1/012014

2.00 x 2.00 m, with the height of 0.50 m in the saddle of hyperbolic paraboloid and a total height of 1.00 m.

3. Finite element analysis

There were used 2D triangular membrane surface finite elements with a mesh size of 50 mm and 1D beam finite elements with 10 divisions per member for beams and also 1D cable finite elements with 10 divisions per member for cables. The form-finding of membrane surface was calculated with the value of axial force of surface $n_x = n_y = 4.0$ kN / m width. Due to the stiffness of the frame, had to be increased the robustness of the computational solution in order to find the initial state of equilibrium for n_x and $n_y = 4.0$ kN / m width. Subsequently, this model was loaded in a similar way as pre-analysis model and afterwards the combinations of these loads were created.

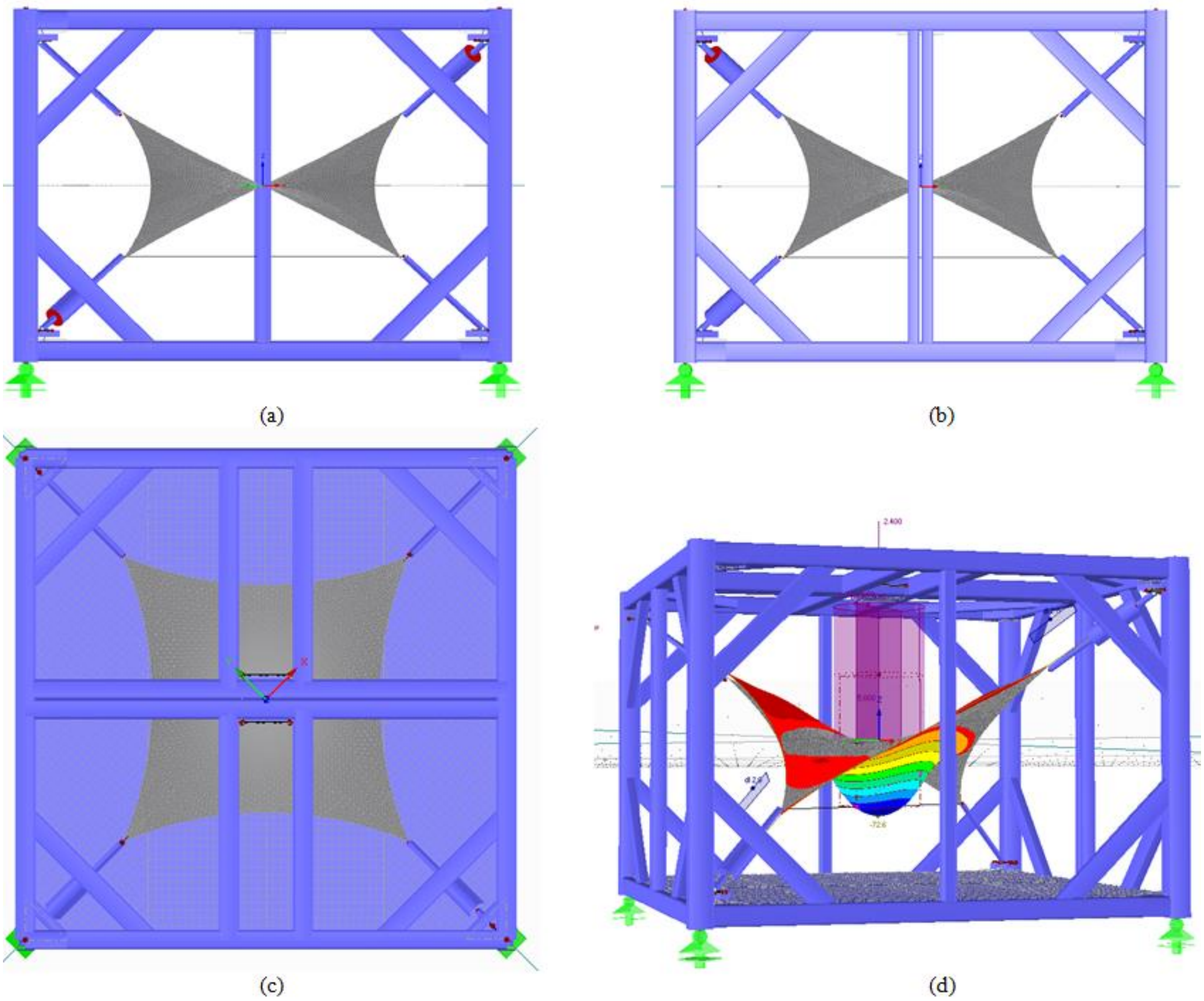

Figure 1. Detailed computational model.
(a) left view, (b) right view, (c) top view, (d) axonometric view of model

4. Analysis of the results

Due to the large number of combinations, only the most relevant data were included in the graphical comparison of the results, i.e. limit values in terms of the maximum and minimum shortening/extension of the action members and also the combination with no shortening/extension with the load.

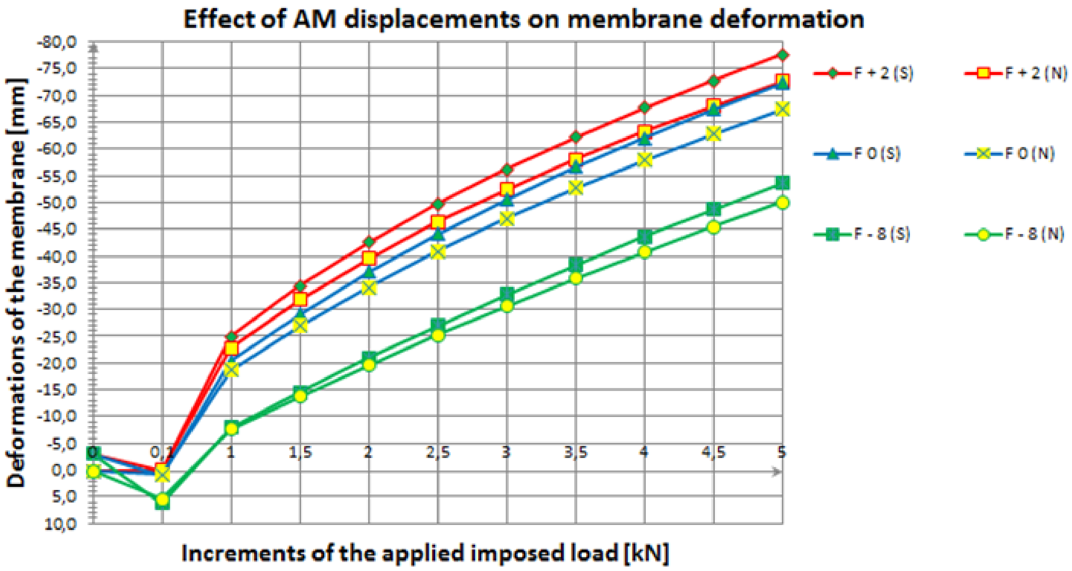

Figure 2. The influence of action members displacement to membrane deformation, the comparison of pre-analysis results (S) and new-analysis results (N).

As can be possible to see at figure 2, apart from the relative differences in the form-finding process and the combinations exclusively with its own weight, it can be stated that the maximum absolute difference in the deformation of the membrane surface is the 5.00 mm, from the combination (F + 2) which represents the incrementally loaded structure by the imposed load and the extension of the action members by 2 mm. The maximum relative difference of deformation is the 9.14% from the combination (F 0), the incrementally loaded structure by the imposed load without changing the length of the action members. At the figure 3, the maximum absolute difference in the axial force in the action members is the 2.00 kN from the combination of (F 0), the incrementally loaded structure by the imposed load without changing the length of the action members, which is also the maximum relative difference established at value 9.15%. At the figure 4, the maximum absolute difference in axial forces in the warp surface direction is the 0.28 kN/m width, for the minimal value of axial forces, combination (F 0), but the maximum relative difference is the 21.02% for the minimal value of the axial forces in the warp surface direction from the combination (F + 2). Similarly at the figure 5, the maximum absolute difference in axial forces in the weft surface direction is the 0.31 kN/m width, for the minimal value of axial forces, combination (F 0), but the maximum relative difference is the 30.13% for the minimal value of the axial forces in the weft surface direction from the combination (F + 2).

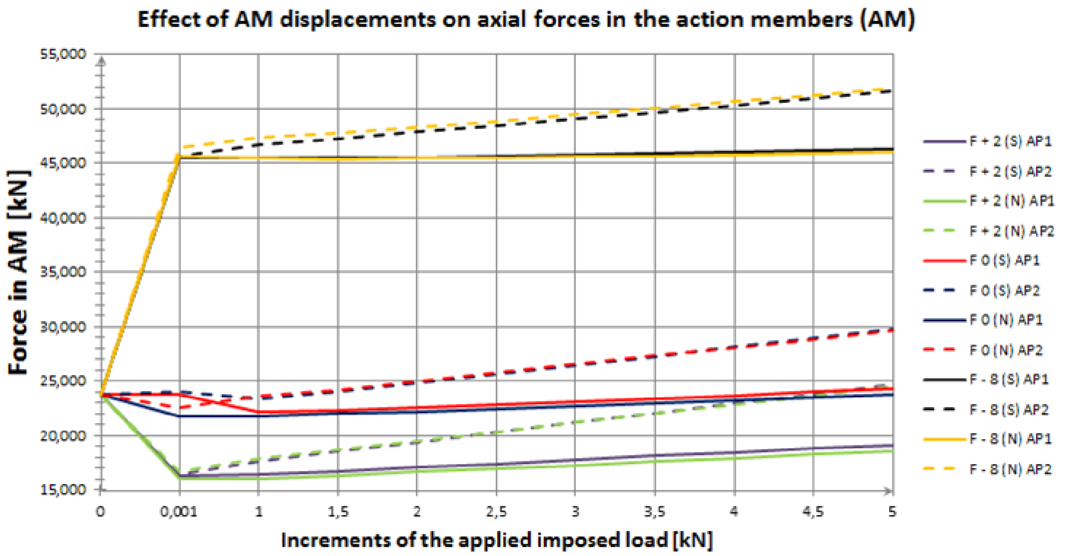

Figure 3. The influence of action members displacement to axial forces in action members, the comparison of pre-analysis results (S) and new-analysis results (N).

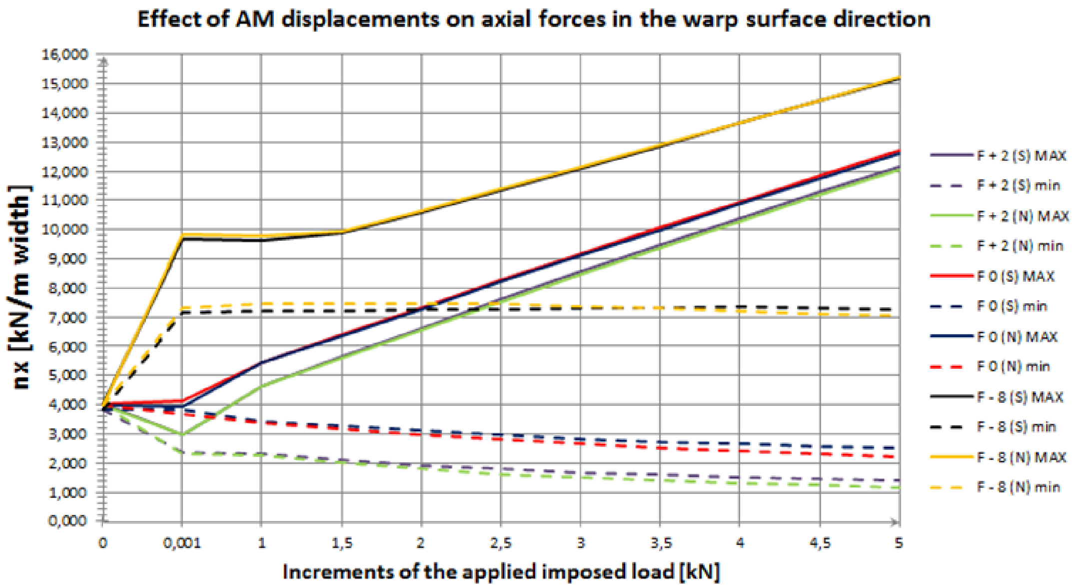

Figure 4. The influence of action members displacement to axial forces in the warp surface direction, the comparison of pre-analysis results (S) and new-analysis results (N).

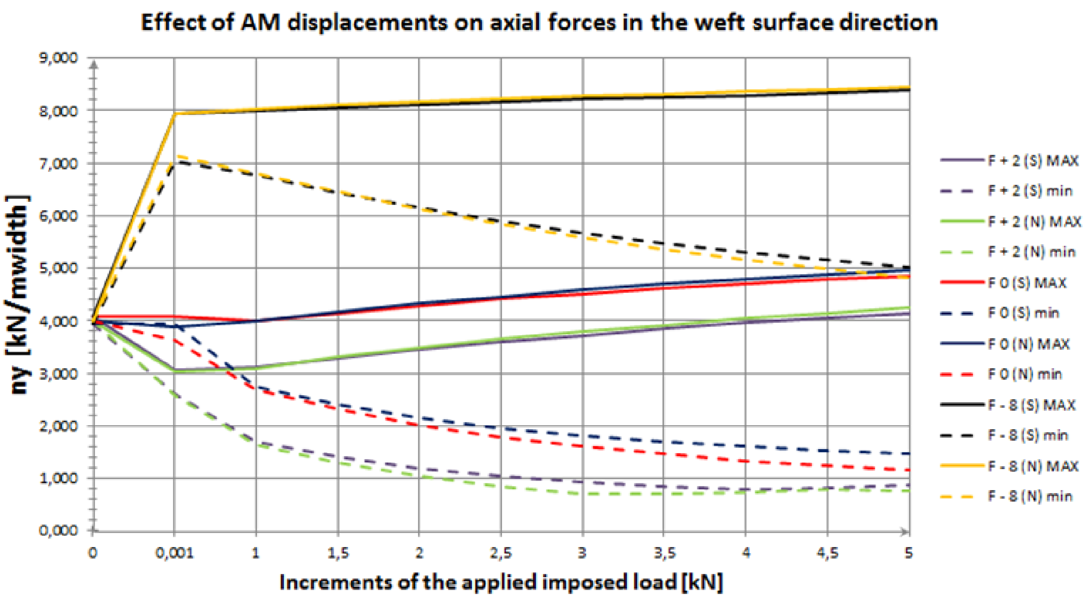

Figure 5. The influence of action members displacement to axial forces in the weft surface direction, the comparison of pre-analysis results (S) and new-analysis results (N).

5. Conclusion

From the results of the detailed computational model, it can be seen that there is no wrinkling at the membrane surface. The magnitude of the relative differences between the results of the pre-analysis and the results of that model are inversely proportional to the stiffness of the membrane surface (the smaller axial forces in the surface means the more significant relative difference). Due to the fact that the experimental device is limited by the force in the action members to the value 50 kN, the creation of the exact model is needed. These results will be compared in the near future with the results of the Ansys software and the results obtained from the experimental device.

6. References

[1] Kapolka L 2018 Static analysis of an adaptive model of membrane structure *Young Scientist 10th international scientific conference on civil and environmental engineering* (Košice: Technická univerzita v Košiciach) p 22
[2] Dlubal RFEM 5.17, student version of the finite elements method software

Acknowledgments

This work was supported by The Slovak Research and Development Agency under the contract No. APVV-15-0777 and also by The Slovak Scientific Grant Agency under the contract No. VEGA 1/0302/16.