Applicable FEM Models for Layered Beams

Jakub Javořík[1], Pavel Nekoksa[1], Jan Kledrowetz[1], Rohitha Keerthiwansa[1]
[1]Faculty of Technology, Tomas Bata University in Zlin. Vavrečkova 275, 762 72 Zlín. Czech Republic. E-mail: javorik@ft.utb.cz

We need to create appropriate and effective numerical (FEM) model to optimize properties of a composite product. The creation and evaluation of an efficient numerical model that could be used for an analysis of layered composite is the aim of this work. The model has to be able to take into account the properties and layout of the individual layers and must allow effective change of these parameters; thickness, material and number of layers especially. Various models of the same product are created and compared. The models differ in the type of used FEM elements. The results of models (deformation primarily) were compared with the result of analytical computation. Further, time and computational requirements of individual models are also evaluated. Element types used for investigated models are: 1D elements, 2D plane stress solid elements, 2D plane strain solid elements and shell elements. Models created form 1D and shell elements showed a close agreement with the analytical solution, and they provide the appropriate tools for the definition of layered structures and for the analysis of results.

Keywords: FEM model, composite, layered beam

1 Introduction

A need of the efficient simulation and prediction of the behavior of composite materials occurs more and more in the field of design and development of new products [1]. Numerical models, described in this work, will be used for the development of composite parabolic leaf springs.

Leaf springs are often used as damping elements, especially for vehicles. It is very suitable to use parabolic profile of the spring, especially for mono-leaf springs. In this case, a spring cross-section height increases parabolically from the ends of the spring towards the center, which means that the cross-section of the spring is largest in the most loaded point (middle of spring). Thanks to the parabolic profile there is constant stress in the entire length of the spring and so the all material of the spring is used equally (as opposed to a spring with constant cross-section). Therefore, this type of the spring allows maximal use of spring material with minimal spring weight, which is nowadays required increasingly in the area of transport vehicles.

Numerical methods based on the finite element method (FEM) are nowadays commonly used during the design and development of these components [2-5]. The aim of this work is to find such way of making a numerical model of the parabolic springs that allows effective modifications of input parameters of the spring, especially, of the spring longitudinal profile, i.e. the height of the cross-section along the length of the spring and material data (such as the definition of the layered composite, etc.). We need to create a numerical model that allows the analysis of mechanical behavior of springs with different geometry (or material structure).

2 Methods

There are several ways how to create the numerical model of composite leaf spring, especially, regarding the model geometry. The geometry will depend on the topology of chosen FEM element. It is possible to use 1D (LINE) elements, 2D planar elements (plane stress or plane strain) or shell elements (SHELL). Several models of one (same) parabolic spring will be created using these different elements and the model that enables efficient modeling of springs with variable parameters and that provides accurate results will be searched.

2.1 Model Geometry

Idealized geometry of mono-leaf parabolic spring/beam was chosen for the analysis. Basic dimensions of the model are shown in Fig. 1.

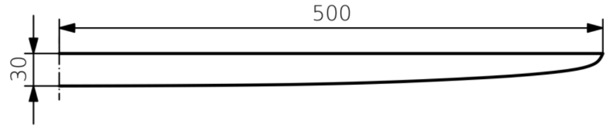

Fig. 1 *Geometry of the idealized parabolic beam [mm]*

Regarding the symmetry of the beam, only half model can be used. Assuming a total spring length of 1000 mm, we can create a model with a half-length of 500 mm, with an expected plane of symmetry, which is shown as the dashed line on the left side in the Fig. 1. The beam width will be constant with value of 50 mm. The thickness in the center is 30 mm and it decrease towards the ends parabolically to zero. Parabolic change of thickness h_x is given by equation

$$h_x = h_0\sqrt{\frac{x}{l}}, \tag{1}$$

with parameters shown in Fig. 2. From Fig. 1 follows that h_0 = 30 mm and l = 500 mm.

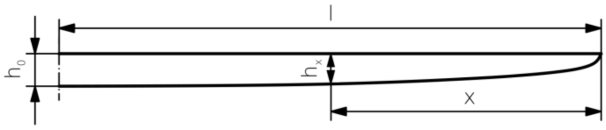

***Fig. 2** Geometric parameters of the model*

2.2 Material

Linear material model (Hooke's law) was chosen for the analysis using material parameters of common steel, i.e.: elasticity modulus E = 210 000 MPa and Poisson's ratio ν = 0.3. As different numerical models (varying in the type of the FEM element) will be compared, the material must be the same for all models. Therefore a simple isotropic material was chosen for this comparison. The models will be however created in such way that allows defining more complex materials (especially layered composites).

2.3 Analytical calculation of beam deformation

As a reference, the analytical solution of the beam is computed. Thanks to the idealized shape of parabolic spring profile, it is possible quite easily and precisely to describe and solve the deformation of the model analytically. Relation for the deflection in the middle of parabolic spring is used:

$$v = \frac{8Fl^3}{Ebh_0^{\,3}} \tag{2}$$

where E is the modulus of elasticity and b is the width of the spring. Other parameters and boundary conditions are shown in Fig. 3. The force F applied in the half model is 5 000 N.

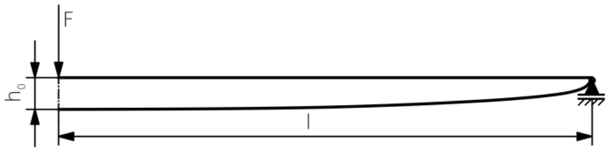

***Fig. 3** Parameters used for the analytical solution*

2.4 FEM models

Four different numerical models of the beam described above were created:

- model created from 1D beam elements
- model created from planar 2D elements – plane stress state
- model created from planar 2D elements – plane strain state
- model created from shell elements

The models differ only by the types of FEM elements. Material properties (see above), the load and boundary conditions are the same in all models.

a) Model from 1D elements

The geometry of the model is created by a simple straight line 500 mm long. The model is made of 100 elements of 5 mm length. The problem is defined as a planar (2D) and elements are able of the definition of the layered composite material. Load and boundary conditions defined in Fig. 3 are shown in detail in Fig. 4.

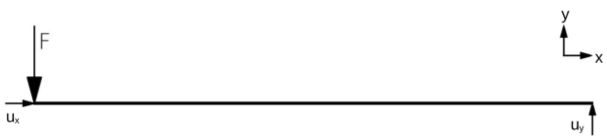

***Fig. 4** Loads and boundary conditions of model from 1D elements*

User subroutine of MSC Marc system (based on the FORTRAN programming language) is used to define the variable thickness of the elements. Using equation (1), the subroutine calculates and set appropriate thickness for each node of the FEM model. Therefore, application of user subroutines allows whenever easily and quickly change parabolic profile of spring, i.e. define any thickness of the spring.

b) Model from 2D elements – plane stress

Due to the geometry of the beam, it is obvious that it is not purely a plane stress problem (neither plane strain). Nevertheless, the possibility of using these models was also verified. The model consists of planar quadrilateral elements creating a planar surface of the half-profile of parabolic beam (Fig. 5).

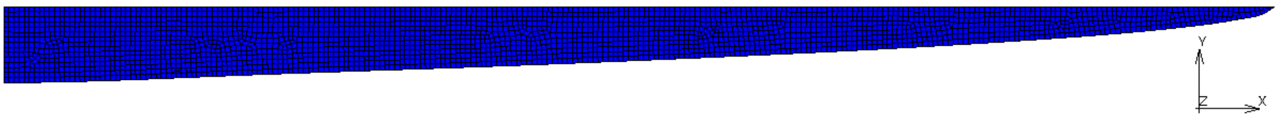

Fig. 5 *FEM model from 2D planar elements*

c) Model from 2D elements – plane strain

The geometry of the model itself is the same as in the previous case and corresponds to Fig. 5. The difference is in the type of used element. In this case plane strain element has been used, which (unlike the previous plane stress element) allows the definition of multiple layers through surface of the element. Scheme of the plane strain multilayered element is shown in Fig. 6.

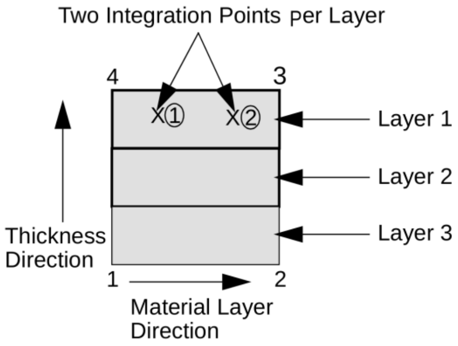

Fig. 6 *Scheme of the plane strain multilayered element*

d) Model from shell elements

Model geometry consists of the surface which represents one quarter of the beam and thus has dimensions 25x500 mm (Fig. 7). Surface is made of shell elements, for which, depending on their position (i.e. length = *x* direction), the variable thickness is defined to achieve the parabolic profile. The thickness of the elements is again defined by the user subroutine as in the model from 1D elements.

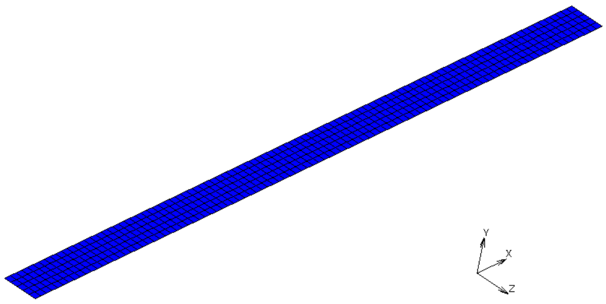

Fig. 7 *Quarter model from the shell elements*

Boundary conditions are similar to the 1D elements model. Regarding the third dimension of the problem and the quarter symmetry of the model, symmetry in the *xy* plane is also defined at one edge of the surface by removing the displacement in the *z* axis and rotations around the *x* and *y* axes. The same boundary conditions as in previous models then remain on two opposite edges of the surface (Fig. 8).

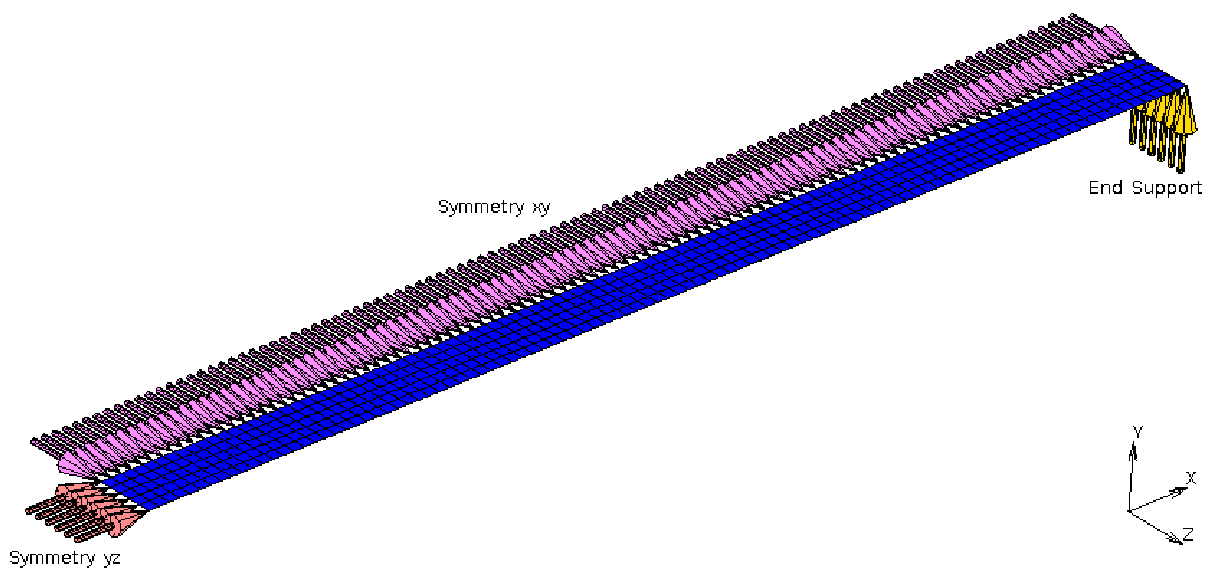

Fig. 8 *Boundary conditions of the shell model*

3 Results

The main result is the final deflection of the spring for the specified load. Furthermore, relation between the deflection and applied load is examined, which enables set up the spring characteristic during the whole range of loading.

3.1 Analytical calculation

Based on equation (1) beam deflection was calculated as

$$v = \frac{8 \cdot 5000 \cdot 500^3}{2{,}1 \cdot 10^5 \cdot 50 \cdot 30^3} = 17.64mm \quad (3)$$

Equation above follows to the linear dependence of deflection on the loading force during the analytic solutions.

3.2 FEM models

Deflection of the model from 1D elements is shown in Fig. 9 with a maximum of the value 17.59 mm in the middle of beam. Similar value is achieved even in the case of the model from 2D plane stress elements, where the deflection in the middle is 17.55 mm (Fig. 10). Deflection is only 15.96 mm in the model with the planar strain (Fig. 11), which confirms the inappropriateness of this model for the analysis of the beam with such geometry.

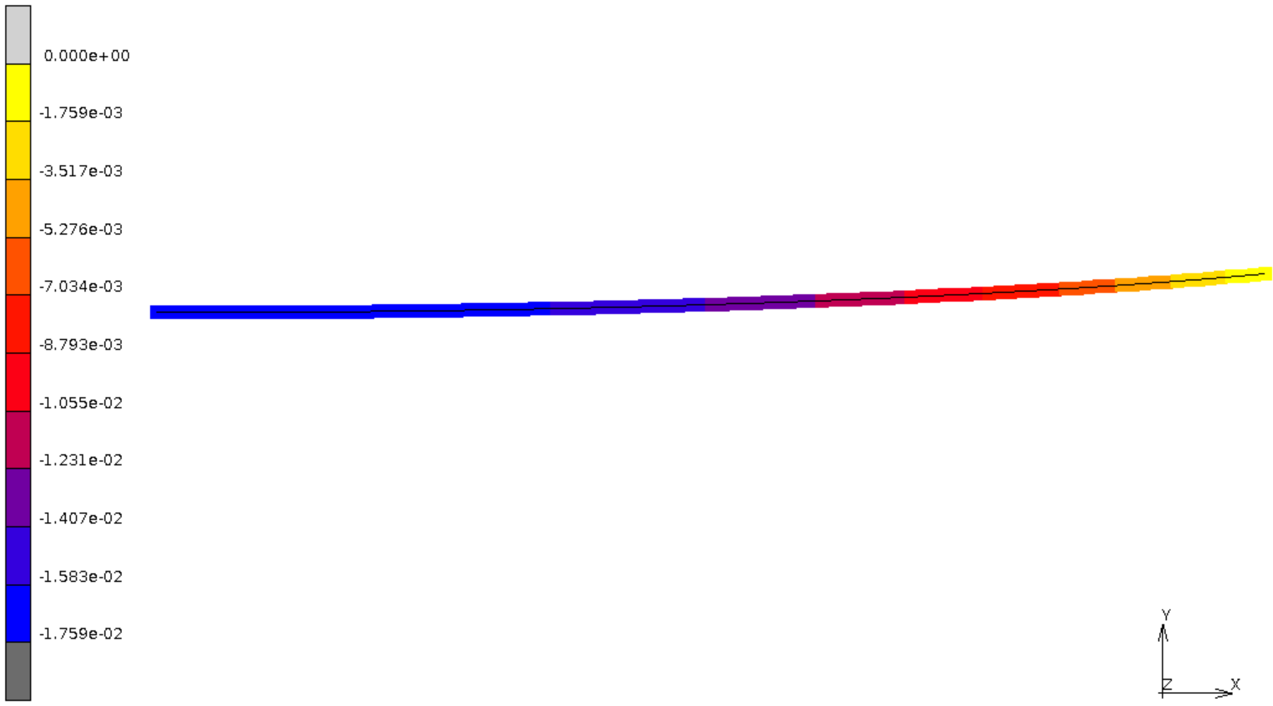

Fig. 9 *Deflection [m] of the model from 1D elements*

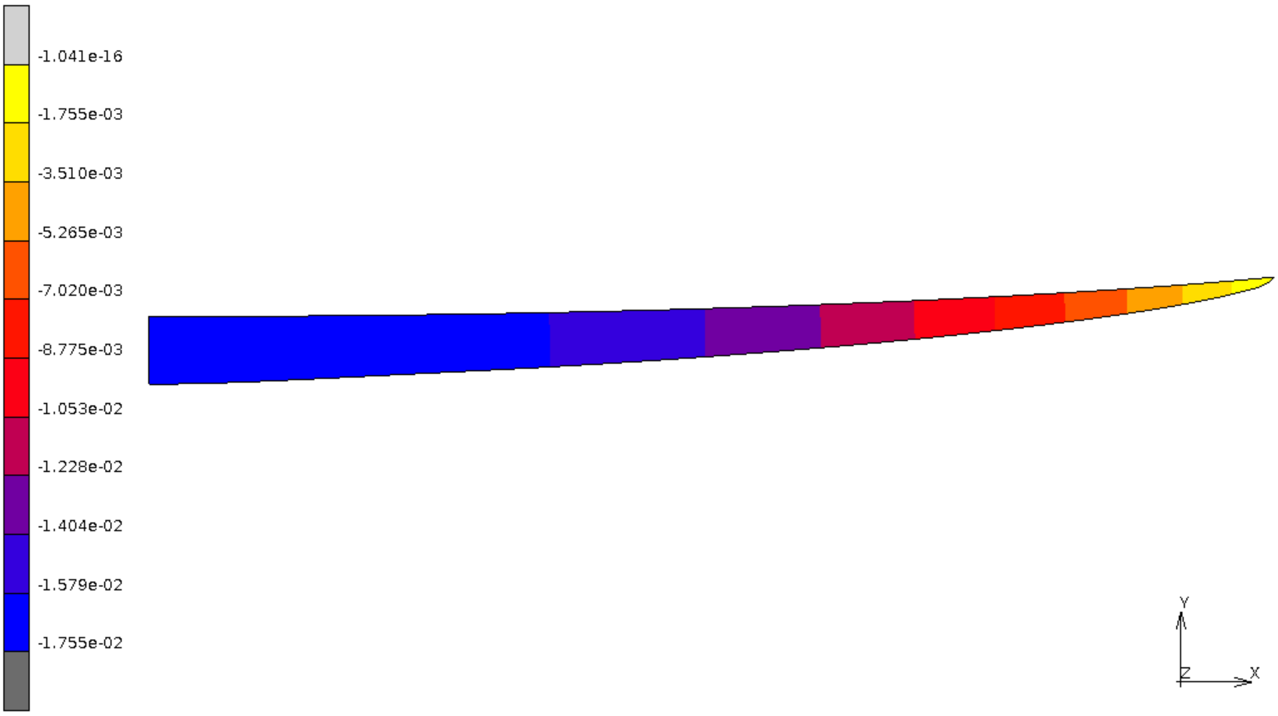

Fig. 10 *Deflection [m] of the model from 2D elements – plane stress*

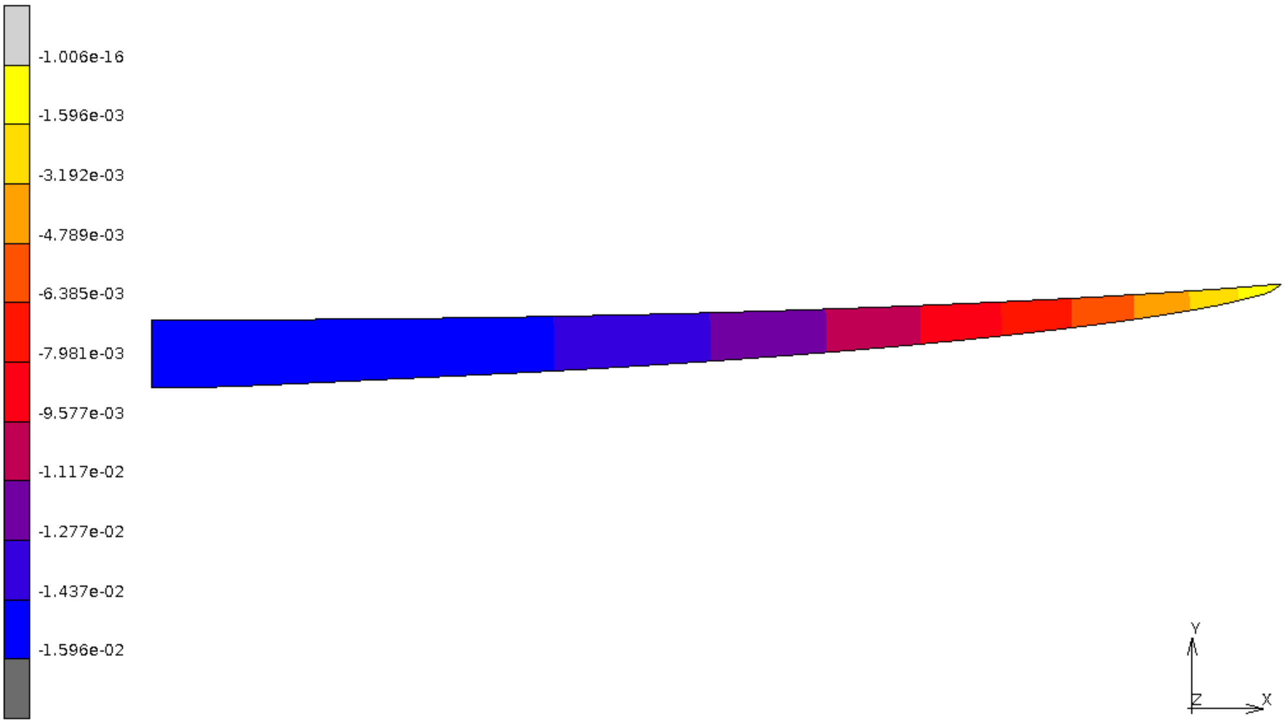

Fig. 11 *Deflection [m] of the model from 2D elements – plane strain*

Unfortunately, in the case of 2D planar models, layered structure of elements can be defined only for a plane strain elements. For this reason, it is not appropriate to use even the plane stress model.

Last from investigated models – shell elements model is closest to the analytical solution. Deflection of this model is 17.65 mm (Fig. 12). Finally, the dependence of the loading force on the deflection for the model from shell elements is presented (Fig. 13). The graph shows strong linear relation between these parameters, even though nonlinear iterative FEM analysis (including large displacements) was used to compute these results.

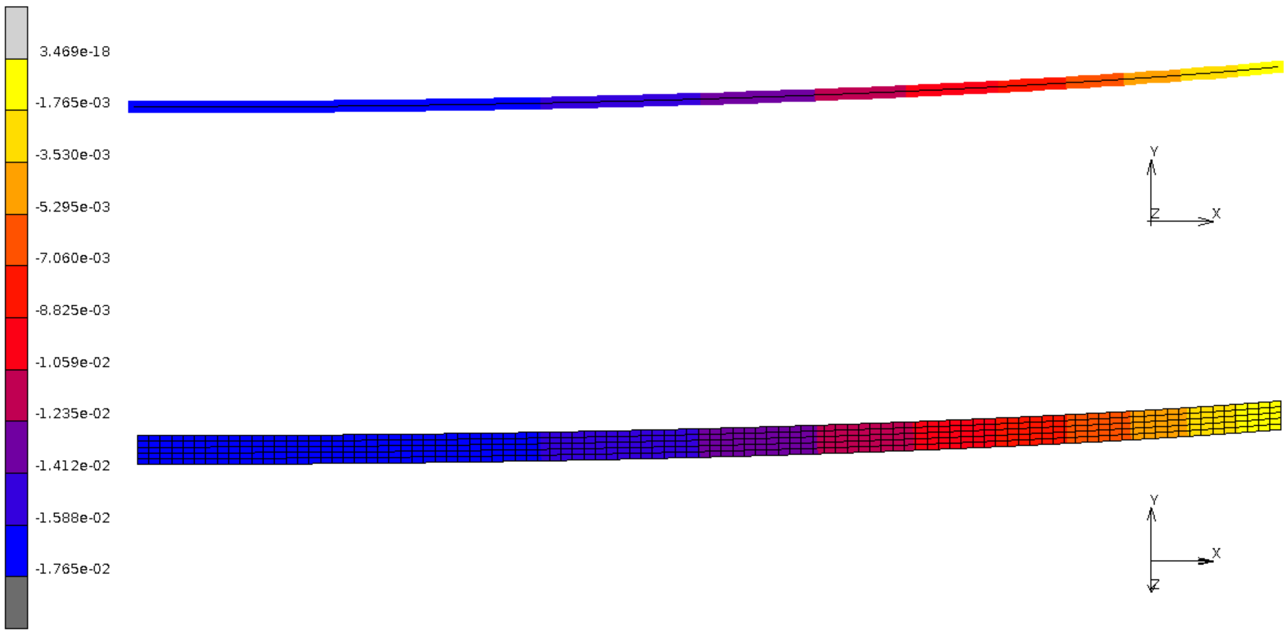

Fig. 12 *Deflection [m] of the model from shell elements*

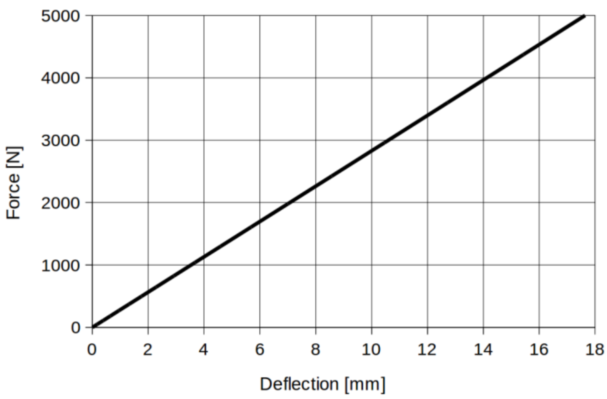

Fig. 13 *Dependence of the loading force on the deflection for the model from shell elements*

4 Conclusion

In accordance with the theory of elasticity it was confirmed that plain strain theory cannot be applied for this purpose. This fact eliminates the possibility of using the 2D planar elements and to solve the problem as two-dimensional because the 2D planar layered elements allow using only the plane strain theory (plane stress can be used only with non-layered 2D elements). Therefore, 2D plane stress elements, which provide significantly better results than the plane strain elements, can be used only for non-layered materials. Very accurate results are achieved in the model from 1D elements. These elements allow a very efficient and accurate solution of the model with minimal computing performance and time requirements. Certain limitations in amount and type of obtained results could be a disadvantage of this model. Thus, the model from shell elements is the most suitable for an analysis of layered beams with variable cross-section. Such model provides very accurate results and is very effective, especially in the process of geometry modification and analysis of results.

Acknowledgement

This work and the project is realized with the financial support of the internal grant of TBU in Zlin No. IGA/FT/2017/002 funded from the resources of specific university research.

References

[1] RUSNAKOVA, S., FOJTL, L., ZALUDEK, M., RUSNAK, V. (2014). Design of material composition and technology verification for composite front end cabs. In: *Manufacturing Technology*, Vol. 14, No. 4, pp. 607 – 611. UJEP. Czech Republic.

[2] ŽMINDÁK, M., MĚŠKO, J., PELAGIČ, Z., ZRAK, A. (2014). Finite element analysis of crack growth in pipelines. In: *Manufacturing Technology*, Vol. 14, No. 1, pp. 116 – 122. UJEP. Czech Republic.

[3] DELYOVÁ, I., HRONCOVÁ, D., FRANKOVSKÝ, P. (2014). Analysis of simple mechanism using MSC Adams. In: *Manufacturing Technology*, Vol. 14, No. 2, pp. 141 – 145. UJEP. Czech Republic.

[4] JAVORIK, J., BILEK, O. (2014). Numerical analysis of bushing of car stabilizer. In: *International Journal of Mechanics*, Vol. 8, No. 1, pp. 289 – 297. NAUN. USA.

[5] JAVORIK, J. (2016). Numerical Optimization of Large Shade Sail Support. In: *Manufacturing Technology*, Vol. 16, No. 4, pp. 707-712. UJEP. Czech Republic.