Petru DUMITRACHE

PARAMETRIC MODELLING AND FINITE ELEMENT ANALYSIS OF A TENSEGRITY STRUCTURE

Abstract: *In the introductory part of the paper, general aspects related to tensegrity structures are briefly presented. Next is presented the principle model of the adopted tensegrity structure and the parameters used in its CAD modelling. The calculations included in the paper allow the determination of the axial forces in the wires of the structure and the conditions in which it retains its tensegrity structure character. Using a virtual model of the structure, corresponding to a set of modelling parameters, in the part dedicated to finite element analysis are determined both the axial force in the wires of the structure and the stresses in its other members. The end part of the paper contains observations and conclusions.*

Key words: *Tensegrity, CAD modelling, parametric modelling, virtual model, finite element analysis.*

1. INTRODUCTION

The term of tensegrity was introduced by the American architect Richard Buckminster Fuller in order to describe his vision of an unconventional architecture, which seemed to be built by nature and not by humans.

In accordance with [1], the tensegrity structures are a system of construction which utilizes the tensile properties of structural materials to the fullest advantage.

Extending the definition contained in [2], I consider that a tensegrity structure is a structure that maintains its integrity and stability only through structural members required only by axial forces. In other words, a tensegrity structure is a structure that can take on other loads in addition to its own weight and maintain its integrity and stability through members subjected exclusively to stretching.

Because the tensegrity structure is designed to maintain its integrity and stability by means of members subjected only to stretching, it is reasonable for them to be made as structural elements that can only take axial forces, such as wires and cables.

Tensegrity structures have a number of advantages over other structures. Among the advantages of tensegrity structures, the most important are the following:

- Considering, for example, the case of constructions, compared to their classical strength structures whose integrity and stability are based in particular on the ability of structural members to withstand compression, the tensegrity structures have the major advantage of high load-bearing capacity, because the tensile strength of a longitudinal member is larger than its buckling (compressive) strength.
- Because members of the tensegrity structure that ensure its integrity and stability are loaded only with axial forces, their sizing leads to a reduction in material consumption, compared to case if these members would be subjected, simultaneously, to other types of loading (e.g., bending).
- Because structural members with the role of maintaining the integrity and stability of the structure can only take on axial forces, a tensegrity structure can have two states: a functional state and a "folded", non-functional state. In the "folded" state, the structure occupies a substantially smaller volume than that corresponding to the functional state. As a result, tensegrity structures have the advantage of the reduced volume of storage and transport.

There are other advantages of tensegrity structures, but the purpose of this paper is not to present all the advantages of tensegrity structures.

Before addressing other issues related to tensegrity structures, it is important to point out that they have important practical applications, in this context mentioning only the Kurilpa Bridge in Brisbane, Australia, currently the largest structure of its kind in the world.

2. PARAMETRIC MODELLING

The CAD Autodesk Inventor platform was used for modelling.

The virtual model of the structure consists of two solids whose dimensions are controlled by parameters.

The modelling parameters are defined as *User Parameters* inside one of the two solids and are referred to in the model of the other solid and in the virtual model of the structure by the *Link* option in the *Parameters* window.

It is specified that the virtual model of the structure was generated for the purpose of finite element analysis. In this context, all the components of the structure that are made of the same material and, at the same time, are bonded together have been embedded in a solid. Thus, on the finite element analysis platform, only the mesh corresponding to one solid is generated and not for several.

In principle, the modelled structure, considered to be made of homogeneous materials, corresponds to the schematic model presented in Figure 1.

Figure 1 shows that the considered structure admits a plane of symmetry, resulting in the equality of the axial forces that occur in the inner and outer wires if the following conditions for loading the structure are also met:

- the structure is loaded only by its own weight;

- loads, other than the own weight of the structure, are applied only to the *Top* solid and are symmetrical in relation to the plane of symmetry of the structure.

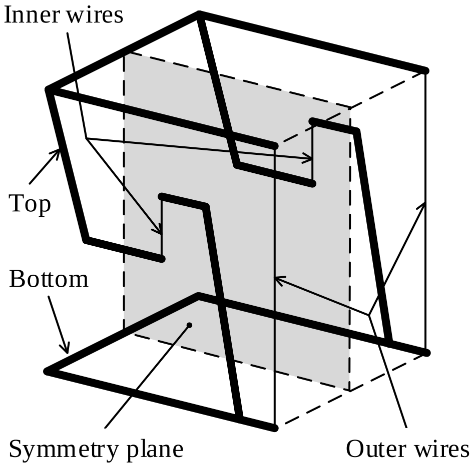


Figure 1 Schematic model.

Figure 2 is a screenshot showing the *Top* solid whose parameter-controlled dimensions have been highlighted. As previously mentioned, some of these parameters are also used to control the dimensions of the *Bottom* solid and the virtual model of the entire structure.

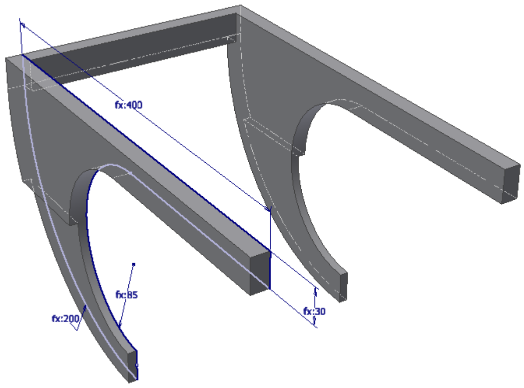
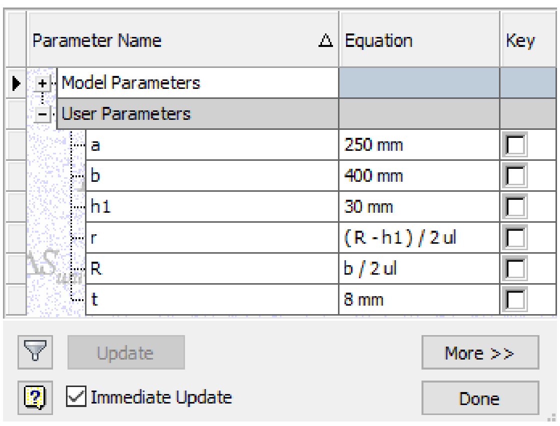


Figure 2 Dimensions of the model, controlled by parameters.

The values associated with the parameters used in the modeling of the structure can be seen in Figure 3.



Parameter Name	Equation	Key
+ Model Parameters		
− User Parameters		
a	250 mm	☐
b	400 mm	☐
h1	30 mm	☐
r	(R - h1) / 2 ul	☐
R	b / 2 ul	☐
t	8 mm	☐

Figure 3 Names associated with parameters.

Regarding Figure 3, in correlation with Figure 2, the following observations are made:

- The values adopted for parameters *b*, *r*, *R* and *h1* (see Figure 3) are also visible in figure 2.
- The values of the parameters *a* and *t* are not visible in Figure 2, the values of these parameters representing distances used for performing extrusions in modelling.

Figure 4 shows the virtual model of the structure, composed of the two solids (*Top* and *Bottom*) to which a rectangular plate was added, placed above the solid *Top*.

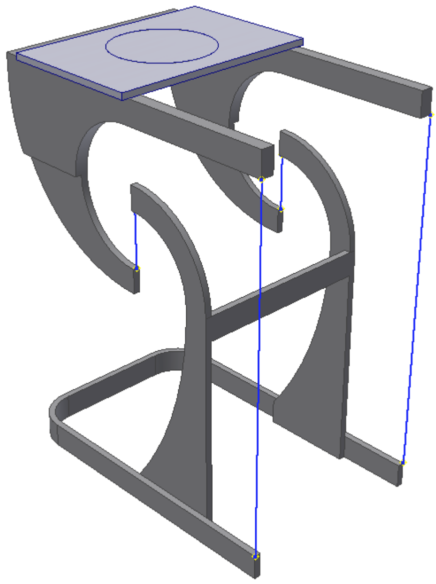


Figure 4 The virtual model of the structure.

The plateau is in contact with the *Top* solid and is placed so as to admit the same plane of symmetry as the structure (see Figure 1). The use of the plateau modelling is motivated by its use as a means of transmitting to the structure of the various vertical loads, respecting the symmetry of the system.

The plateau is designed as a set of two solids, as seen in Figure 4, the motivation of this approach being the possibility of applying different ways of loading the structure in the finite element analysis environment.

3. CALCULATIONS

This paragraph is dedicated to deducing only the calculation relations of the axial forces from the wires of the structure and establishing the loading conditions so that the structure is a tensegrity structure. Here, by wires are meant one-dimensional bodies, which connect the solid components of the structure.

It should be noted that the determination of the stresses that develop in the solid 3D components of the structure when it is loaded with its own weight and possibly with other loads, is possible only numerically, the most convenient method being the finite element method.

In order to deduce the mentioned calculation relations, the following hypotheses were considered satisfied:

- The structure is loaded only by vertical forces, acting on the solid *Top*.
- The forces that load the structure form a system of forces that admits as a plane of symmetry the plane of symmetry of the structure (see figure 1). In other words, the loading of the structure complies with the conditions set out in paragraph 1 of the paper.
- The structure is considered loaded with both its own weight (G) and a vertical force (F), satisfying the symmetry conditions expressed above. In addition, the force F has the same direction as the weight of the structure.
- The weight of the wires that connect the 3D components of the structure is neglected.

The deduction of the calculation relations for the axial forces that occurs in the wires of the structure is made using the calculation schemes given in Figure 5 and Figure 6.

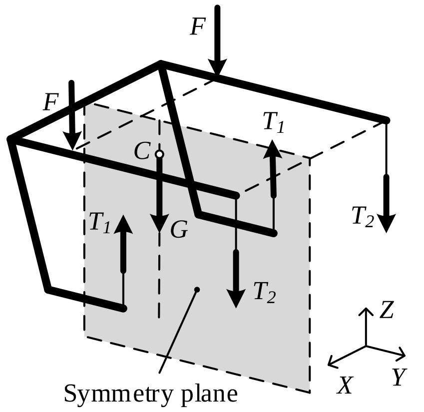


Figure 5 Calculation scheme (3D view).

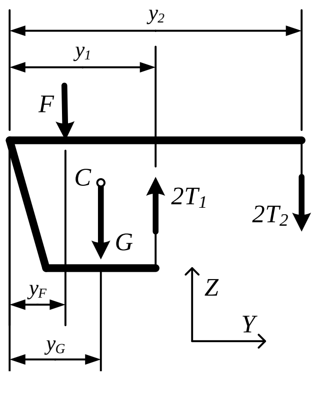


Figure 6 Calculation scheme (2D view).

Based on the calculation scheme in Figure 6, imposing the static equilibrium conditions, it results:

$$\begin{cases} 2T_1 - 2T_2 = F + G \\ 2T_1\left(y_1 - y_G\right) - 2T_2\left(y_2 - y_G\right) = F\left(y_F - y_G\right) \end{cases} \quad (1)$$

Solving the system (1), it is found:

$$T_1 = \frac{F\left(y_2 - y_F\right) + G\left(y_2 - y_G\right)}{2\left(y_2 - y_1\right)} \quad (2)$$

$$T_2 = \frac{F\left(y_1 - y_F\right) + G\left(y_1 - y_G\right)}{2\left(y_2 - y_1\right)} \quad (3)$$

Considering that the structure is loaded only with its own weight (G), it results:

$$T_{1(F=0)} = \frac{G\left(y_2 - y_G\right)}{2\left(y_2 - y_1\right)};\ \ T_{2(F=0)} = \frac{G\left(y_1 - y_G\right)}{2\left(y_2 - y_1\right)} \quad (4)$$

From (4) it is found that if the axial forces in the wires of the structure are caused only by the weight of the upper part (F = 0), both wires are stretched, i.e. the structure is a tensegrity structure.

From (2), it is easy to deduce that if $F > 0$, the inner wires are stretched. The outer wires can be stretched or compressed depending on the position of the force *F*. Obviously, the structure is a tensegrity structure only if both $T_1 > 0$ and $T_2 > 0$. It remains to establish the limitation of the position of the force *F*, so that $T_2 > 0$.

Imposing $T_2 > 0$ in (3), it results:

$$y_F \in \left[0;\ y_1 + \frac{G}{F}\left(y_1 - y_G\right)\right] \quad (5)$$

4 FINITE ELEMENT ANALYSIS

The finite element analysis of the structure studied in this paper aims to validate the formulas for the axial forces T_1 and T_2 as well the condition (5) that limits the position of the force F.

Finite element analysis input data:

- Analysis type: *Static Stress with Linear Material Models*
- All parts of the structure are made of aluminium with the following physical and mechanical properties:
  - Mass density: $\rho = 2700\ kg/m^3$;
  - Modulus of Elasticity: $E = 68900\ MPa$;
  - Poisson's Ratio: $\mu = 0.33$.

Considering the structure fixed at its bottom, under the conditions specified in the previous paragraphs, the structure has material, load and constraints symmetry in relation to the plane of symmetry shown in Figure 2.

Therefore, in the finite element modelling of the structure, it is rational to use this symmetry. The use of symmetry in the finite element model imposes appropriate restrictions on the nodes of the model which are in the plane of symmetry, so that these nodes are maintained in this plane when the structure deforms due to the applied loads.

Two cases were analysed:

- The structure is loaded with its own weight (*G*) and the weight of the plate (*F*), positioned so that the vertical of its centre of mass coincides with the vertical of the centre of mass of the solid *Top*.
- The structure is loaded with its own weight (*G*) and the weight of the plate (*F*), positioned at the limit of maintaining the character of the structure (tensegrity structure).

In both cases the symmetry of the structure was used.

Correct modelling of the structure wires requires that they be modelled as truss-type finite elements, which is not possible in Autodesk Inventor. As a result, an external FEA environment was used.

For the solid *Top*, with the modelling parameters shown in Figure 3, using *iProperties* from *Autodesk Inventor*, we find $y_1 \cong 120.107\ mm$, $y_G = 120.107\ mm$ and $m_{Top} = 2.653\ kg$.

In the same way F is calculated. As a result, we obtain:

$$G = G_{Top} = m_{Top} g \cong 26.026\ N \qquad (6)$$

$$F = G_{Plateau} = m_{Plateau} g \cong 10.144\ N \qquad (7)$$

Noting that in the first case of analysis, $y_F = y_G$, by the appropriate customization of the calculation formulas for the axial forces T_1 and T_2, deduced in the previous paragraph, we obtain:

$$T_{1\_formula} \cong 25.309\ N\ ;\ T_{2\_formula} \cong 7.224\ N \qquad (8)$$

After solving the numerical calculation model associated with the finite element model, we obtained:

$$T_{1\_FEA} \cong 25.318\ N\ ;\ T_{2\_FEA} \cong 7.205\ N \qquad (9)$$

From the analysis of the relative errors between the values (8) and (9), results the correctness of the finite element modelling of the problem, and at the same time validates the formulas deduced in the previous paragraph. Figure 7 shows a screenshot of the FEA environment showing the values of the axial forces T_1 and T_2.

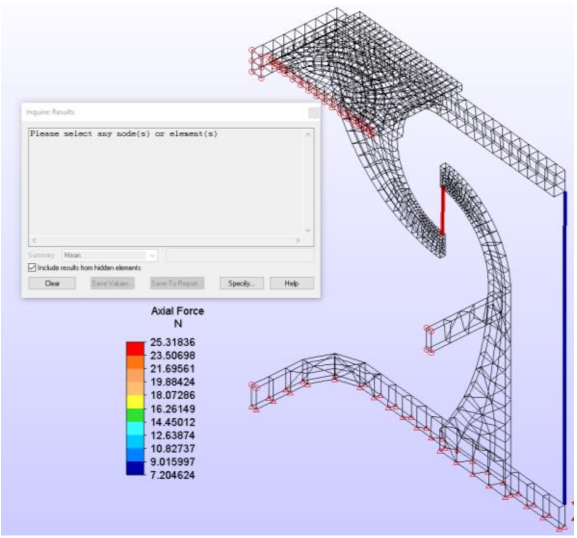


Figure 7 Axial forces in the structure wires (Case 1).

In the second case, considering G and F given by (6) and (7) respectively and using condition (5), we obtain $y_F \in [0;\ 404.978]\ mm$. After positioning (in Inventor) the plateau at the distance $y_{F\max} = 404.978\ mm$ and finite element analysis of the corresponding model, it is obtained that $T_2 \cong 0\ N$ (see Figure 8), which validates the condition (5).

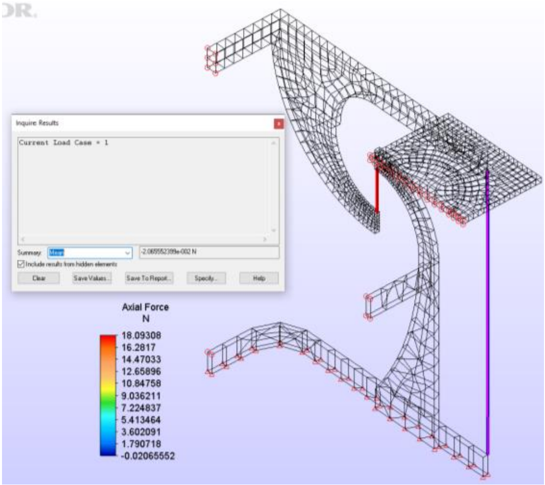


Figure 8 Axial forces (T_2) in second case.

5 CONCLUSIONS

The elaboration of this paper allows the formulation of the following conclusions:

- Where possible, it is recommended that the finite element analysis of a system compatible with this approach, to be carried out after a prior study of the loading conditions that ensure that the system maintains the required characteristics.
- The modelling with adequate finite elements of the components of the mechanical structures is a requirement of great importance, the observance of which contributes to obtaining interpretable, relevant, plausible and truthful results.
- The beauty of tensegrity structures demonstrates that they happily combine beauty with usefulness.

REFERENCES

[1] Fuller R. B. (1962) *Tensile-Integrity Structures*, US Patent no. 3063521, Nov. 13, 1962.

[2] Gómez-Jáuregui V. (2010). *Tensegrity Structures and their Application to Architecture*. Santander: Servicio de Publicaciones de la Universidad de Cantabria. ISBN 978-84-8102-575-0.

[3] *** Waguespack C. (2014) *Mastering Autodesk Inventor 2015 and Autodesk Inventor 2015 LT* ISBN: 978-1-118-86213-1

[4] *** ALGOR v.23.00 (2009) *User Manual*

Author:

Assoc. Prof. Dr. Eng. Petru DUMITRACHE, 'DUNAREA DE JOS" University of Galati, „Engineering Sciences and Management" Department, E-mail: pdumitrache@ugal.ro