Static Analysis of a Tensegrity Bridge using the Finite Element Method

M.H. Zgoul, A. Alzamer, M. R. Elayyan, M. A. Quran

Department of Mechanical Engineering, Faculty of Engineering
University of Jordan
Amman 11942, Jordan

* Corresponding author. Email: m.zgoul@ju.edu.jo

Abstract—Computer Aided Design (CAD) and Finite Element Analysis (FEA) of a tensegrity footbridge structure are investigated in this work. Challenges encountered throughout the modelling process and the analysis were highlighted and addressed. A static analysis of the finalised model was then carried out using the Finite Element Method. Throughout the work, modelling challenges and analysis shortcomings were resolved by constantly modifying the CAD model. Results showed little deformations in the tensegrity footbridge structure under the given loads.

Keywords: Tensegrity structure, static analysis, Finite Element Method, Computer Aided Design.

I. INTRODUCTION

Tensegrity structures are innovative spatial systems composed of struts and cables that are used in a wide range of modern engineering applications. Such structures are considered to be free-standing pin-jointed structures. Stability of tensegrity structures is provided by the self-stress state in tensioned and compressed elements. They, furthermore, hold the advantage of being light weight structures and also give the impression of transparency due to their assembly forms. Another advantage of tensegrity structures is the high strength to weight ratio, meaning that, for example, a tensegrity bridge would be very light compared to a traditional one, while supporting and holding the same load as the other.

A definition of tensegrity structures has been proposed by Motro [1]: “A Tensegrity is a system in stable self-equilibrated state comprising a discontinuous set of compressed components inside a continuum of tensioned components”. Under this definition, systems where compressed elements are interconnected are also considered as tensegrity structures. The concept of tensegrity structures has been the focus of multiple research works and has received significant interest among scientists and engineers throughout a diverse range of disciplines ranging from architecture, civil engineering, biology, robotics to aerospace engineering [2], and there are still more areas of research in that field that have not been explored yet, furthermore it has aesthetic architectural view.

Despite their appearance, the design of tensegrity structures is not simple due to the fact that these systems are reticulate, spatial and self-stressed [3-8]. In this research, the focus would be on the implementation of the Finite Element Method (FEM). The main advantage of this method is that it is computerized; as a result the analysis of this kind of structures will become faster. A footpath bridge is taken as case study to present the challenges in modelling and analyses of a typical tensegrity structure.

II. MODELLING

Modelling a tensegrity structure using various Computer Aided Design (CAD) techniques is essential for a successful analysis of the structure. While it might seem to be insignificant, it is indeed the base this research work. Failure to create a realistic CAD model will result in false analysis results and simulations of the structure.

A typical tensegrity module takes the form of a continuous ring with a single strut circuit. This module is called a pentagon module [9,10]. It contains 15 nodes describing 3 pentagonal layers as shown in Figure 1. The middle pentagonal layer nodes are rotated about the longitudinal axes with respect to outer pentagon by 36° in the counter-clockwise direction. The pentagon module comprises 15 struts held together in space by 30 cables forming a ring shaped tensegrity unit. Struts can be separated into diagonal and intermediate struts based on their topology. Diagonal struts connect outer and inner pentagon nodes while intermediate struts connect middle pentagon nodes to outer and inner pentagon nodes. Similarly, cables are separated into 10 layer cables and 20 x-cables. Layer cables connect nodes of the two outer pentagons while x-cables connect middle pentagon nodes to inner and outer pentagon nodes as shown in Figure 2.

The model selected for this work, tensegrity footbridge, is extracted from a previous research work [2] as shown in Figure 3. The tensegrity structure is made up from struts and cables. Initially, the main and secondary struts and cables were modeled according to the specifications reported in the literature [2] as shown in the table 1. A pentagon configuration was selected being the most efficient configuration [2]. Furthermore this configuration gave the tip of the lead to the
modeling; the key issue was to imagine a layer in the middle of the segment which was shifted about half of (360°/number of angles) of the shape of the layer.

The model contained 340 nodes and 370 elements, and was fully constrained at both sides. Nevertheless, results from this analysis did not yield a convergent solution and thus the CAD model was re-created.

Due to the 3D nature of the problem, and the choice of the pentagon module member the footbridge was remodeled as shown in Figure 5. It shows the initial 3D CAD model of the segment/module of the footbridge based on the specifications presented earlier. Although the initial CAD model looked acceptable, a problem with the intersections/connections between the different parts existed where those intersections/connections created a lot of modeling challenges when introduced to the finite element software. The CAD model had to be improved further to eliminate challenges related to the connections by simply connecting each element to the other using tangent connections with holes acting as a reference to the different connections as shown in Figure 6.

| TABLE 1: FEASIBLE CONFIGURATIONS OF THE TENSEGRITY BRIDGE [2]               |
|-----------------|--------|--------|----------|----------|----------|
|                 | D x t  | A      | I        | I        | A         |
|                 | mm x mm | mm²    | mm³      | mm      | mm²      |
| Square          | 114.3 x 3.6 | 1250 | 1.920 x 10⁸ | 39.20 | 300  | 100   |
| Pentagon        | 101.6 x 3.6 | 1110 | 1.330 x 10⁸ | 34.70 | 300  | 100   |
| Hexagon         | 95.0 x 3.2 | 923  | 0.973 x 10⁷ | 32.30 | 300  | 100   |

To begin with, the footbridge was modeled as one-dimensional problem using the finite element software as shown in Figure 5 with element to represent the trusses, cables and different joints carry both tensile and compressive loads.
Although modifications to the initial CAD model made it more acceptable and realistic, a new challenge emerged; limitation of the hardware. Computational time is an important factor and thus it should be reduced to allow running a relatively complex model on a standard personal computer thus reducing the cost of analysis. The modified CAD model had holes which led to the generation of a rather large number of elements in the finite element software and thus a further modification to the CAD model was required.

A second modification to the original CAD model saw the elimination of the holes. Also, the cables were replaced by springs subjected to pretension so that the model can be simpler and easier to handle in the FEA software.

Figure 7 shows the final CAD model, made up into a full footbridge with springs instead of cables, and a walkway assembled to represent a tensegrity bridge as shown earlier in Figure 3.

III. FINITE ELEMENT WORK

The purpose of the finite element analysis is to mechanically analyse the behaviour of the footbridge under specific loads and conditions. To begin with, the CAD model is imported in the finite element software and an initial mesh is generated (Figure 8).

Two loads were applied to the model: Standard earth gravity and a 4 kPa-pressure (distributed load) subjected to the walkway area as reported in the literature [2]. A static structural analysis was then carried out using the finite element software. Results were set to view the footbridge behaviour under the pre-mentioned loads and those results are (von Misses stress, directional deformation, total deformation and others). Figure 10 shows the directional deformation of the footbridge (x-direction) with a minimum value of $-4.281 \times 10^{-3}$ m and maximum value or $3.6056 \times 10^{-3}$ m at the locations shown in the figure.
IV. ANALYSIS AND RESULTS

During the process of simulating the bridge, a number of challenges were encountered and solved:

High mesh resolution: The initial problem encountered was that the automatic meshing process generated a large number of mesh elements which lead to an insufficient memory errors (Figure 11). This problem was solved by editing the geometry of the bridge so that complex details (small holes) in the struts, which were responsible for the fine mesh, are removed.

Defining cables: The second challenge was the cables. Initially, the cables were intended to be modelled as thin struts with different material properties to simulate cable behavior (Figure 12). But due to moment resistance of these thin struts the cables were removed and replaced by spring connections. These springs have a pretension so in a compression case the springs will behave as cables, meaning that they will not carry a compressive load/stress (Figure 13).

The preload for the springs can be calculated from the following equation:

\[ F = A \sigma \]  

Where:
- \( F \): is the preload (N).
- \( A \): cross sectional area for cables (m²).
- \( \sigma \): pre-stress in cables (Pa).

And the stiffness of the springs can be calculated from the following equation:

\[ k = \frac{EA}{l} \]  

Where:
- \( k \): cable stiffness (N/m).
- \( E \): Modulus of elasticity (Pa).
- \( l \): length of the cable (m).
- \( A \): Cross sectional area for cable (m²).

Joint definition: The third problem was setting the large number of joints and choosing the suitable type of joints that resembles true strut connections. At first the joints were defined automatically using the finite element software which resulted in the presence of redundant connections that in turn caused the simulation results to be unrealistic. Thus, the automatic joints were removed and then recreated manually. Second step was to shift the resulting joint reference into their correct position.

Applying suitable loads: At the beginning the walkway segment was not included in the CAD model which would not have resulted in appropriate simulation results. Thus, the walkway was included in the model and appropriate loads were applied on it as mentioned earlier.

V. CONCLUSIONS

Handling tensegrity structures using FEM software can be a little bit challenging due to a wide range of factors that are related to the complexity of modelling of such structures. However, FEM certainly facilitates the process of design; which will increase the usage of those types of structures and will help in spreading the use of such structures in different engineering applications.

CAD modelling of complex tensegrity structures poses no challenges or threats when appropriate modelling techniques are implemented. However, handling CAD models in finite element software might create a few challenges, particularly at interfaces between the different elements of a tensegrity structure. In the present work, the interference between struts and cables created a multitude of challenges when exported to the finite element software. Thus the main effort in modifying the CAD model for the tensegrity structure analysed in this work was to connect all cables and struts without any
interferences. Other challenges such as joints’ reference location were overcome by manually defining their positions.

Tensegrity footbridge structure has enough strength to sustain loads as a traditional footbridge with relatively small deflection. Unsymmetrical results in the model are probably due to the stress concentration that is produced by the spring connections.

REFERENCES