VERIFICATION OF STRESS AND DISPLACEMENT RESULTS OF AN OPTIMIZED TRUSS

Tarique Samir Bezerra Sena

Caio Rodrigues Laranjeira Vilar

tarique.samir@hotmail.com

caiorlvilar@gmail.com

Federal University of Parana

Avenida do Batel 1597 Apto 901, 80420-090, Parana, Brazil

Richard A. Regueiro

regueiro@colorado.edu

University of Colorado, Boulder

1111 Engineering Dr. 428 UCB, ECOT 441, 80309-0428, Colorado, United States

Abstract. *The paper presented here proposes to verify the mechanical behavior of an optimized simply supported truss with 5 nodes and 6 bars, obtained from a genetic algorithm optimization using population size 450. For the purpose of developing this paper, it was used as reference an optimization already performed in a previous article and its results. In order to perform this ascertainment, we compared the structure's stress and displacement results to those found through the finite element software Abaqus® and an analytical analysis. The first consists of modeling and analyzing the optimized truss using Abaqus® software. This analysis has the objective of confirming the reference paper results through a software that has been used widely by finite element specialists. The second is based on a Matlab® application to develop the stiffness matrix of the optimized structure. The applications ratified that the results obtained in the reference paper are consistent.*

Keywords: *Truss, Stress, Displacement, Finite Element Analysis, Stiffness Method.*

1 INTRODUCTION

Most processes with high benefit/cost ratio are developed by optimization methods. In engineering, especially in structural problems, it consists of developing lighter structural systems that are within the allowable stress and displacement requirements. The process related to solve optimal problems consists of minimizing the cross sectional area of members without exceeding the constraints of allowable stress and displacement. Nevertheless, most papers that approach optimal problems only perform stress and displacement analysis of the final optimized structure using a finite element (FE) algorithm developed by the authors themselves, as we notice in Camp et al. (1998) and Cazacu and Grama (2014) and Camp et al. (1998). Furthermore, no analytical solutions are presented to ratify that the optimized solution follows the constraints. Then, the paper presented here proposes to verify whether the constraints of an optimized truss performed by Deb and Gulati (2001) are met through a commercial finite element package and an analytical method.

2 METHODOLOGY

The truss (see Fig. 1) is composed by six members, five nodes and it has two overlapping members that are shown with a gap in the figure. All members are made of the same material (E = 68,950 MPa; $\rho = 2{,}768$ kg/m³), but each one has a different cross sectional area (see Table 1). The constraints for axial stress and displacement are 172MPa and 50.8mm respectively.

Table 1. Members cross sectional area

Member	1	2	3	4	5	6
Area (m²)	3.37×10^{-3}	13.10×10^{-3}	9.42×10^{-3}	5.01×10^{-3}	18.19×10^{-3}	13.32×10^{-3}

In order to perform this ascertainment, we compared the truss member axial stress and displacement results to those found through the FE software Abaqus® and stiffness method as analytical solution.

2.1 Stiffness Method Analysis

An efficient manner to analyze both statically determinate and indeterminate structures, the stiffness method generates forces and displacements directly. Furthermore, it is quite simple to program in a computer, resulting in calculations efficiently performed. The Matlab® code we use to verify the results obtained by Deb and Gulati (2001) was based on the work of Raza (2015). The code generates the truss member stiffness matrices and calculates not only the forces in each member and the displacements, but also the support reactions.

Some properties must be known, such as length, cross sectional area, and Young's modulus of each member. We also need to identify each member and node of the truss and establish global and member coordinates. Once we have all these data in the input file, we can run the code.

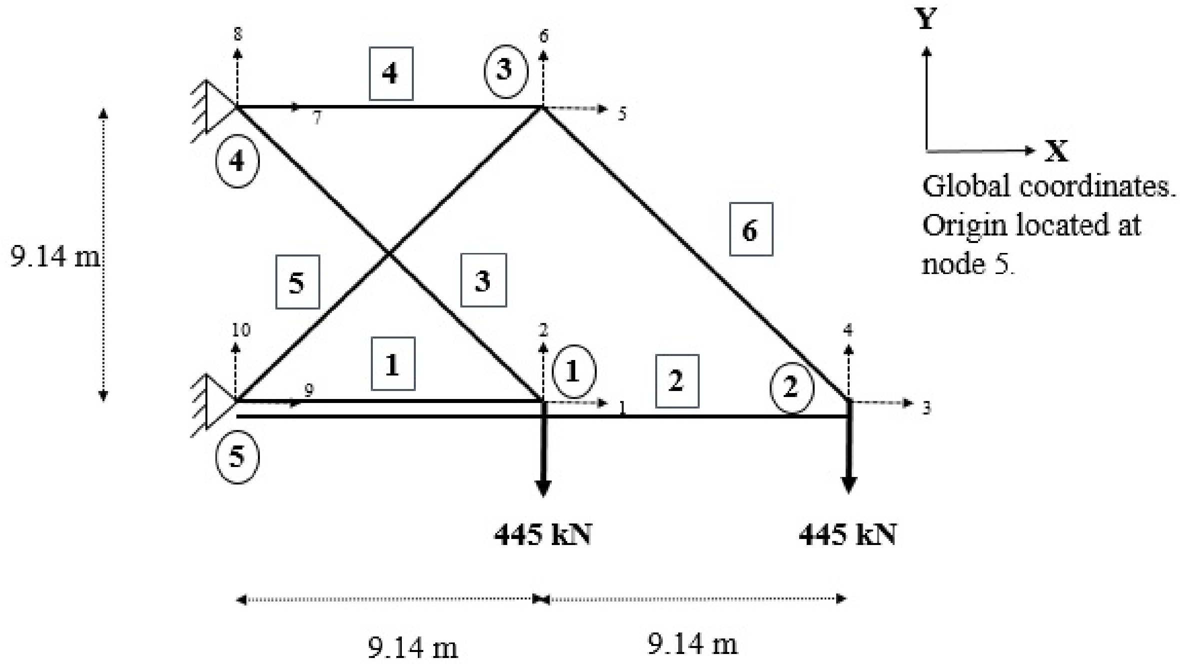

Figure 1. Truss elements

In order to apply the stiffness method for trusses, we have to divide the structure into a series of discrete finite elements (members) and nodes (joints). Relating the force displacement properties to the force equilibrium equations, we can determine the member stiffness matrix. Then, we can sum all these partial matrices to find the structure stiffness matrix K and apply boundary conditions. Once we know the structure stiffness matrix, we can find the displacements and then the forces in each member (Hibbeler, 2012). That is exactly what the Matlab® code does.

2.2 Finite Element Analysis

Fish and Belytschko (2007) states that FE analysis is a numerical method used to approach many types of engineering problems to obtain an approximate solution and it is often performed with support of software. The software we have chosen to use was Abaqus® from Simulia because it is a FE program which is widely used in the research community, which can make it easier to reproduce this methodology in other research project.

In order to solve the truss with two overlapping members we divided it into two parts as shown on Fig. 2. Furthermore, the truss analysis was done using all the parameters showed in methodology section.

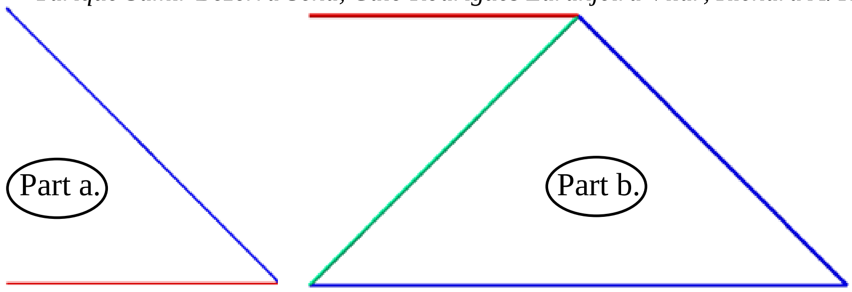

Figure 2. Truss divide into two parts

3 RESULTS AND DISCUSSION

After we performed stress and displacement analysis through stiffness method and FE analysis we obtained satisfactory results. As we can check, the values of axial stress are under 172MPa which is the limit stress for the optimized truss (see Table 2). Furthermore, the displacements in Table 3 show that values are under the imposed limit of 50.8mm.

Table 2. Comparison of the results for Stresses

Member	Stress of member (MPa.)		
	Deb and Gulati (2001)	Matlab®	Abaqus®
1	132	132	132
2	48.0	48.0	47.9
3	47.2	47.2	47.1
4	125	125	125
5	48.9	48.9	48.8
6	47.3	47.3	47.1

Table 3. Comparison of the results for Displacements

Node	Displacement of Node (mm.)			
	Axis	Deb and Gulati (2001)	Matlab®	Abaqus®
1	x	-17.5	-17.5	-17.5
	y	-50.8	-50.8	-50.7
2	x	-12.5	-12.5	-12.5
	y	-50.8	-50.8	-50.6
3	x	6.48	6.48	6.47
	y	-19.2	-19.2	-19.2

The calculations performed through the Matlab® code based on the work of Raza (2015) showed that the results presented in Deb and Gulati (2001) are consistent. Actually, the stresses and displacements presented in the base paper are the same found in the stiffness analysis. It is important to notice that "Automatic stabilization" was used to perform the FE analysis in both parts of truss. It was necessary because the truss was presenting large deformations which were not expected due to the material and loads. However, the results obtained from Abaqus® were still satisfactory.

4 CONCLUSION

The results obtained from this work show that Deb and Gulati (2001) presented precise values for the six-bar optimized truss. This comparison was possible because they have not only shown the final area for each optimized member, but the stress and displacement values as well. Then we conclude that the constraints for the optimized truss were met.

Hence, we can assert that this sort of verification is fundamental to ratify the results of previous papers. We also suggest that researchers publish the results for stress and displacement in papers which cover optimization problems, because we found only a few that we could see the results and perform a comparison.

Acknowledgements

We would like to thank the academic support from Professor Richard A. Regueiro and the financial support from "CAPES" (Coordenação de Aperfeiçoamento de Pessoal de Nível Superior) during our exchange program.

REFERENCES

Camp, C., Pezeshk, S., & Cao, G., 1998. Optimized design of two-dimensional structures using a genetic algorithm. *Journal of Structural Engineering*, vol. 124, n. 5, pp. 551-559.

Cazacu, R., & Grama, L., 2014. Steel truss optimization using genetic algorithms and fea. *Procedia Technology*, vol. 12, pp. 339-346.

Fish, J., & Belytschko, T., 2007. A first course in finite elements. Chichester, West Sussex: Jhon Wiley & Sons.

Deb, K. & Gulati, S., 2001. Design of truss-structures for minimum weight using genetic algorithms. Finite Elements In Analysis And Design, 37(5), pp. 447-465.

Hibbeler, R., 2012. *Structural analysis*. Boston: Prentice Hall.

Raza, S., 2012. *Matlab Code For Stiffness Method To Solve 2D Frames & Trusses*. *Structural Engineering Forum Of Pakistan*.