



Comparing Stresses and Compressions for Three Elements by Numerical Solution and COMSOL Multiphysics Results

Mohamed .F. Khsheba,¹ Hesn .A. Alazrg,² Salah A. Ashareef,³ Tareq .M. Eissa⁴

HIGHER INSTITUTE OF SCIENCE OF AND TECHNOLOGY, QASR BIN GHASHIR

Alhosam.khsheba@gmail.com

Received: 29/6/2026 Revised: 20/7/2026 Accepted: 28/7/2026 Publication: 29/8/2026

الملخص

يقارن هذا البحث بين النتائج التي تم الحصول عليها باستخدام الحل العددي وبرنامج COMSOL Multiphysics في هذه الدراسة، يتم حساب الإجهادات والانضغاط في كل جزء من القضيب باستخدام طريقتين مختلفتين تعتمد الطريقة الأولى على الحل العددي، بينما تعتمد الطريقة الثانية على المحاكاة باستخدام برنامج COMSOL Multiphysics. بعد ذلك، تتم مقارنة النتائج الناتجة من الطريقتين للتحقق من مدى توافقهما. فإذا كانت النتائج متطابقة أو متقاربة، فإن ذلك يؤكد صحة النموذج المستخدم. أما في حال وجود اختلافات، فسيتم توضيح أسباب هذه الفروقات ومناقشتها في قسم النتائج والمناقشة.

الكلمات المفتاحية: العناصر الخطية، الإجهادات، الانضغاط، مصفوفة المعاملات، الشد، القضيب

Abstract

This paper compares the results between numerical solution and COMSOL multiphysics. In this problem we will calculate the stresses and compression in each section by two ways. At the first way we use numerical method, and the second way we use COMSOL multiphysics method. After that we will compare between them. To verify these results, if the results are equal it will be satisfied, or the results have difference after that we will describe the reasons in results and discussion section.

Keywords: Linear elements, Stresses, Compressions, Coefficient matrix, Tension, Rod.

1. INTRODUCTION

Solid mechanics is the study of the deformation and motion of solid materials under the action of forces. It is one of the fundamental applied engineering sciences, in the sense that it is used to describe, explain and predict many of the physical phenomena around us. It is also the branch of physics and mathematics that concerns the behavior of solid matter under external actions. It has been advanced by participating in major inventions throughout history, such as buildings, ships, automobiles, railways, petroleum refineries, engines, airplanes, nuclear reactors, composite materials, computers, and medical implants. In such connections, the discipline is also known as engineering mechanics, often practiced within civil engineering and mechanical engineering.

This paper includes an analytical study in terms of stresses and compressions and mechanical properties and its impact on the rod. Stress is a quantity derived from force, also known as the engineering science, the force acting per unit area. Also, in mechanics, compression is the application of balanced inward (pushing) forces to different points on a material or structure, that is, forces with no net sum or torque directed so as to reduce its size in one or more directions due to increase in pressure from both sides or from one side.

2. Procedure for Paper Submission

Figure (1) that has three linear elements and we want to illustrate this figure. Here in this problem we will find the stresses and compressions in each section.

The first section is from steel metal and it has length 16 (in) and its diameter is 3.19 (in), the second section is from aluminum metal and it has length 16 (in) and its diameter is 2.76 (in), and the third section is from copper metal and it has length 16 (in) and its diameter 2.25 (in).

In addition, after we find the results in each section we will compare these results with the result that we got from the COMSOL multiphysics.

Now we will start our steps to solve this problem with numerical solution and we will show several laws to find solution:

$$[K^e]\{u^e\} = \{f^e\} + \{Q^e\} \longrightarrow (1)$$

In the equation (1) the matrix is called the coefficient matrix, or stiffness matrix in structural mechanics applications. The column vector is the source vector, or force vector in structural mechanics problems. As in the equation illustrated above, $(u_1, u_2, \dots)$ and $(f_1, \dots)$ called primary and secondary element nodal degrees of freedom; hence, it cannot be solved without having an additional (n) conditions. Some of these are provided by the boundary conditions and the remainder by balance of the secondary variables at nodes common to several elements. And (u) is displacement in each section of the rod.

To find coefficient matrix in each section we use this following equation:

$$[K^e] = \frac{A.E}{h} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \longrightarrow (2)$$

Where:

A = area of section for the rod.

E = Young's modulus.

h = the length of section for the rod.

In this problem the assembly of three elements of equal length $h_1 = h_2 = h_3$ we use matrix shape:

$$[k^e] = \begin{pmatrix} k_{11}^1 & k_{12}^1 & 0 & 0 \\ k_{21}^1 & k_{22}^1 + k_{11}^2 & k_{12}^2 & 0 \\ 0 & k_{21}^2 & k_{22}^2 + k_{11}^3 & k_{12}^3 \\ 0 & 0 & k_{21}^3 & k_{22}^3 \end{pmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} p_1^1 \\ p_2^1 + p_1^2 \\ p_2^2 + p_1^3 \\ p_2^3 \end{Bmatrix}$$

The boundary condition in the matrix above is $U_1 = 0.0$, because there is no displacement at fixed point.

Next figure defines distribution of the forces in each member can be computed from the element equations:

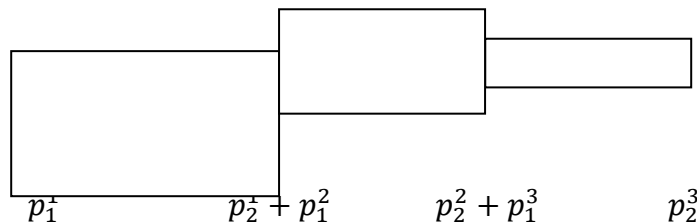


Figure 1: Distribution of The Forces In Each Member

When we mentioned in equation (3) above that the boundary condition $U_1 = 0.0$, in this case we should drop column and row after that the final matrix shape will be:

$$[K^e] = \begin{pmatrix} k_{22}^1 + k_{11}^2 & k_{12}^2 & 0 \\ k_{21}^2 & k_{22}^2 + k_{11}^3 & k_{12}^3 \\ 0 & k_{21}^3 & k_{22}^3 \end{pmatrix} \begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} p_2^1 + p_1^2 \\ p_2^2 + p_1^3 \\ p_2^3 \end{Bmatrix} \rightarrow \quad (4)$$

From this matrix above we can find displacement in each section of the rod (U_2, U_3, U_4) by calculation.

After that we can use this formula to find an elongation in each section:

$$\delta = \frac{P.L}{E.A} \longrightarrow (5)$$

Where:

δ = elongation in each section (in)

P = load in each section (lb)

L = length in each section (in)

A = area in each section (in²)

3. Fixed Constraint and Boundary Condition

First of all we will start with fixed constraint in this problem using following equation:

$$U = 0 \longrightarrow (6)$$

Here consider fixed constraint in the bottom part of the element that means there is no displacement in this case at point and also there is no deflection.

Secondly we will use following equation to find forces in each element:

$$\sigma \cdot n = F_A \longrightarrow (7)$$

$$F_A = \frac{F_{tot}}{A}$$

Here we will denote the boundary condition in each section of the rod. As known in the first section the boundary condition equal zero, the boundary condition in the second section is positive value because we have two forces effect in tension direction, in the third section equal zero because there is no forces effect in this section, in the fourth section equal negative value because there is force effect in compression direction.

4. Geometry and Computation Domain

Geometry in COMSOL multiphysics is considered an important part in the program and create figure with its elements and dimensions. In this step we create geometry as shown in the figure (2) and we will talk about how we draw each section with its dimensions. Now we are going to draw all our elements:

In the first section, steel rod the height is 16 (in) and the width is 1.59 (in), and (r, z) are (0, 0). In the second section, aluminum rod the height is 16 (in) and the width is 1.38 (in), and (r, z) are (0, 16). In the third section, copper rod the height is 16 (in) and the width is 1.125 (in), and (r, z) are (0, 32).

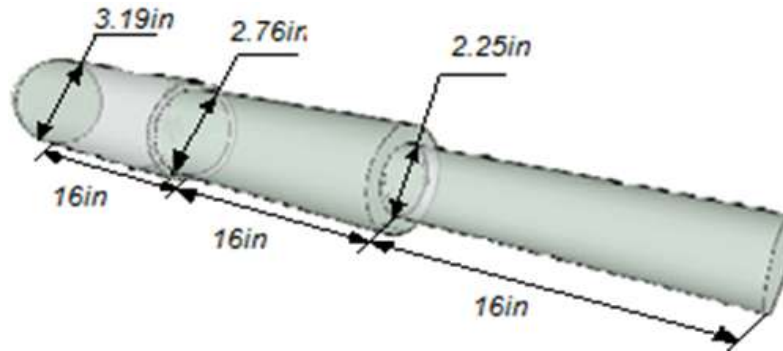


Figure (2): the computational domain (the unit of dimensions is in)

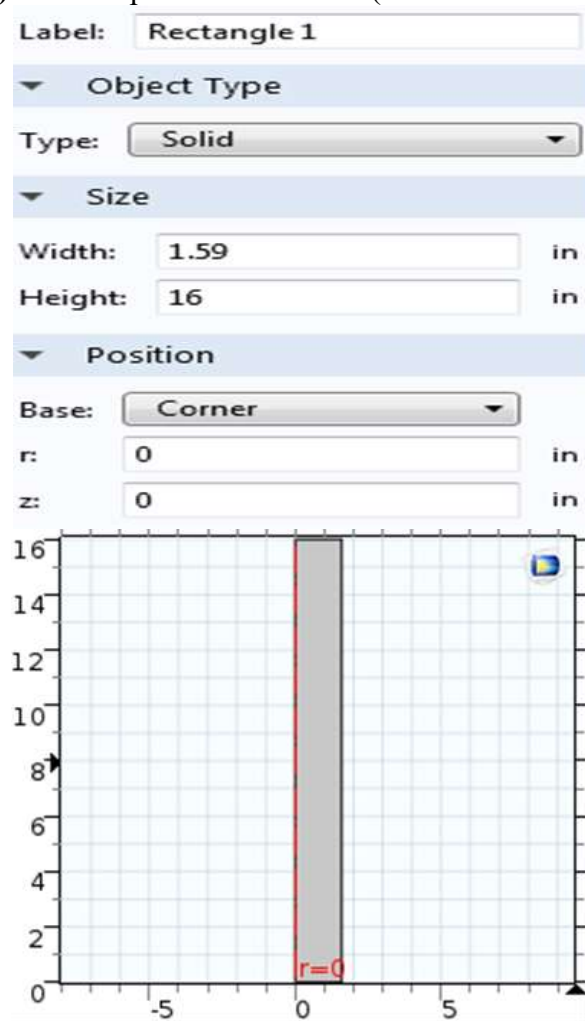


figure (3): illustrates the create element by COMSOL metaphysics

In this stage, we will apply two kinds of meshes because the results will change with the change in the meshes and we will take the closest result with the result that we got from numerical result. After that we will choose this result and procedure comparison to investigate from numerical stresses and compressions results.

First of all, we started with finer mesh Fig (4) and got these results for displacements and stresses respectively ($U_1 = 0$, $U_2 = 0.0002$ in, $U_3 = -0.000333$ in, $U_4 = -0.000867$ in) ($\sigma_{st} = -20.36$ pa, $\sigma_{al} = -2.30$ pa, $\sigma_{co} = -3.46$ pa)

Secondly, we will use another kind of mesh and it will be extra coarse mesh Fig (5) after we applied it we got these results for displacements and stresses respectively ($U_1 = 0$, $U_2 = 0.0002$ in, $U_3 = -0.000333$ in, $U_4 = -0.000867$ in) ($\sigma_{st} = -120.50\text{pa}$, $\sigma_{al} = -2.36\text{pa}$, $\sigma_{co} = -3.49\text{pa}$).

All previous values were satisfied in finer and extra coarse mesh except principle stress in steel section was not satisfied for two kinds of meshes. It is supposed to be a positive result because of that the steel section is exposed to tension stress. Therefore, we will choose the finer results because its results are the best.

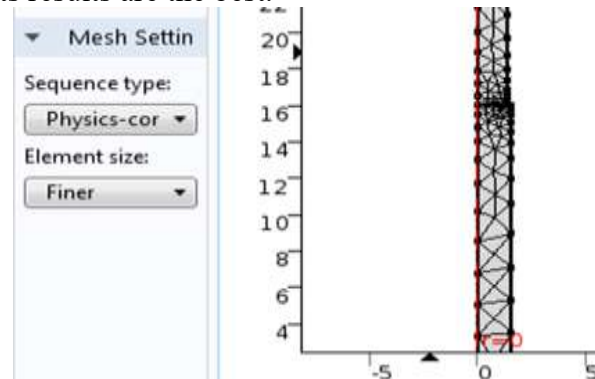


Figure (4): results for displacements and stresses for finer mesh

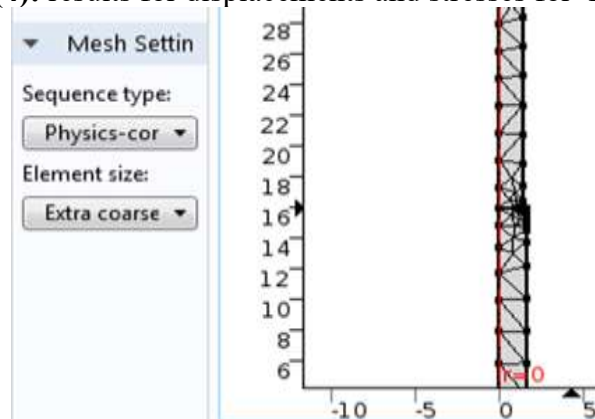


Figure (5): results for displacements and stresses for extra coarse mesh

5. Results and Discussion

The results were performed in a two dimensional (2D) domain and the results will include stresses and compressions. After procedure for stresses and compressions by COMSOL multiphysics for this reason, we got our results for displacement in each section. The displacement in steel element at length (16 in) approximately = 0.0002 in, the displacement in aluminum element at length (32 in) approximately = -0.000333 in, the displacement in copper element at length (48 in) approximately = -0.000867 in. As in figure (6) below:

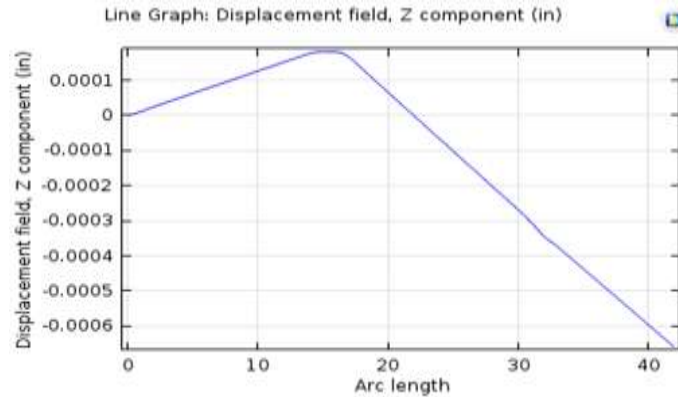


Figure (6): illustrates distribution displacement in each section in Z axis.

We noticed the displacement increase from zero until 16 in because there is tension stress here, while the displacement decrease from 16 in until 48 in because there is compression stress here.

Now we will mention numerical solution for displacement values:

($U_1 = 0$, $U_2 = 0.0002$ in, $U_3 = -0.000333$ in, $U_4 = -0.000867$ in)

Hence, we concluded that there is no difference between the numerical solution and the COMSOL multiphysics result both of displacement results were satisfied for two ways.

Now we will talk about the principal stress for both of ways (COMSOL multiphysics and numerical results). We had results of principal stresses for COMSOL metaphysics as shown in figure (7) below. While we clicked on each section of the rod we found following values as showing in the next table:

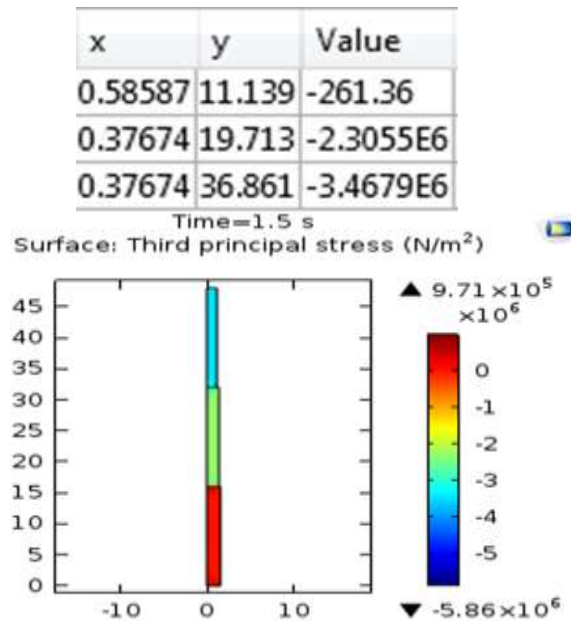


Figure (7): principal stresses for COMSOL metaphysics

In this case when we clicked on the steel element the value for principal stress was -261.36 Pa, and when we clicked on the aluminum element the value for principal stress was -2.3055 Pa and in the last element when we also clicked over it the value was -3.467 Pa. As showing in the figure (8) below:

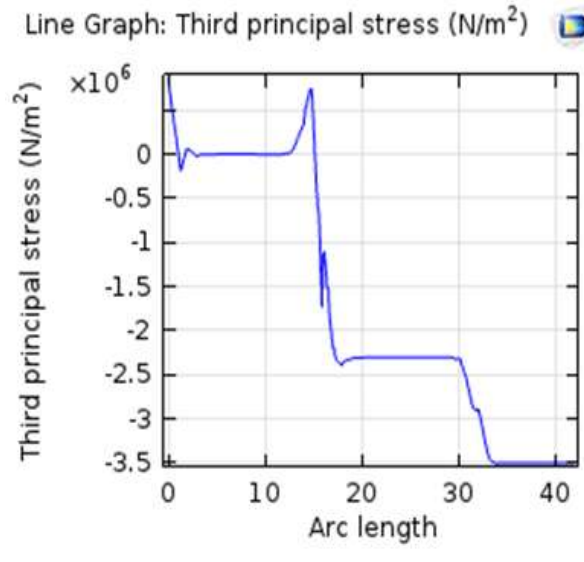


Figure (8):Third principal stress (N/m^2)

Now we will mention numerical solution for principal stresses after conversion of units from (psi) to (Pascal) with multiply in this number (6894.7) after that the values will become ($\sigma_{st}=2.58 \times 10^6 \text{ pa}$, $\sigma_{al} = -2.29 \times 10^6 \text{ pa}$, and $\sigma_{co} = -3.44 \times 10^6 \text{ pa}$).

Hence, we concluded that there is no difference between the numerical solution and the COMSOL multiphysics results at aluminum and copper elements, however, with steel element the result for COMSOL multiphysics was difference value with numerical result.

Finally, we are going to explain why the result of the COMSOL has difference with the result of numerical? Because it depends on the change of the mesh whenever, we change another type of mesh the values for three elements will diverge. Hence, we found that the best mesh is finer mesh for the result of COMSOL to be close from numerical result.

6. Conclusion

As we mentioned above our paper will solve a problem by two ways, numerical method and COMSOL multiphysics method, and we solved that problem with both ways, after that we made comparison between them.

7. References

- [1] J. N. Reddy, An Introduction to the Finite Element Method, Second Edition, Department of Mechanical Engineering, Texas A&M University, College Station, Texas, USA, pp. 86–93, 1993.
- [2] COMSOL AB, COMSOL Multiphysics® User's Guide, COMSOL AB, Burlington, MA, USA, pp. 31–35, 2023.
- [3] James M. Gere, Mechanics of Materials, Sixth Edition, Stanford University, Thomson Brooks/Cole, pp. 107–109, 2004.
- [4] Xu, J., Tang, L., Wang, H., Cao, Q., and Jiang, X., "Analytical Element for Compressive Bar Considering Flexural Displacement," Journal of Jiangsu University (Natural Science Edition), Vol. 40, No. 2, pp. 167–171, 2019.
- [5] Jafari, A., Broumand, P., Vahab, M., and Khalili, N., "An eXtended Finite Element Method Implementation in COMSOL Multiphysics: Solid Mechanics," Finite Elements in Analysis and Design, Vol. 202, Article 103707, 2022.
- [6] Stefani, F. A., Frascio, M., and Niccolini Marmont Du Haut Champ, C. A., "Choice of the Stress Integration Scheme for Accurate Large-Deformation Finite Element Analysis,"

Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, Vol. 237, No. 17, pp. 3977–3986, 2023.

[7] Yang, F., Bao, Y., Du, M., and Li, X., "Numerical Investigation of the Axial Compressive Behavior of a Novel L-Shaped Concrete-Filled Steel Tube Column," Materials, Vol. 18, No. 4, Article 897, 2025.