



NUMERICAL MODELING USING THE ANSYS PROGRAM AND THE FINITE ELEMENT METHOD

R.A. Iskanderov, O.Y. Afandiyev

Department of Mechanics, Azerbaijan University of Architecture and Construction, Baku, Azerbaijan, r.iskanderov@gmail.com, o.afandi@yandex.com

Abstract: The issues discussed in this article were solved using both analytical and numerical methods. At the same time, the article discusses the stability of a flat rod made of a porous material, the ends of which are rigidly connected, the ends are connected by a hinge, one end is connected by a hinge, the other end is rigidly connected, and one end is rigidly connected, and the other is free, which was performed using the ANSYS program, which includes the finite element method (FEM).

The finite element method (FEM) for numerical modeling taking into account the mechanical properties of engineering systems with complex geometric structures is considered an approach capable of giving fairly honest results. For this reason, the application of the finite element method to the ANSYS program is used in all areas of deformable solid mechanics (Structural mechanics, mechanical engineering, shipbuilding, aerospace, etc.). In the process of implementing the ANSYS program in the study of structures with various mechanical, physical parameters and complex geometric configurations, the program offers a wide range of relevant elements. This includes beams, shells, thin-walled cylindrical and spherical covers, structures with a shaft system, etc. The choice of the type of listed elements should correspond to the geometric configuration, the possible nature of the deformation and the intention of the analysis.

To implement the proposed method and program, a computational model must be built. The elements that will be used in building the model can be imported from a computer-aided design (CAD) system. After defining the model, the mechanical parameters of the elements selected for the model are taken into account and the reaction forces are specified. Finally, at the end, a finite element network is built.

In the presented article, using the ANSYS program, depending on the shape of the connection of the ends of the straight-axis rod, the stability problem was solved (a critical force was found) and in all cases a stress-strain spectrum was performed.

The following conditions are accepted in the issues under consideration, geometric dimensions, and coordinate system:

- the points of the rod in the direction of the y axis do not accept an offset, and other values do not depend on y ,
- the thickness of the rod in the direction of its y axis is equal to one,
- the origin is above the median plane of the undeformed rod, the length of the rod is $2L$, and $-l \leq x \leq l$.
- the width of the rod in the direction of the z -axis is $2h$ and $-h \leq z \leq h$.

a) The case when both ends of the rod are rigidly pinched

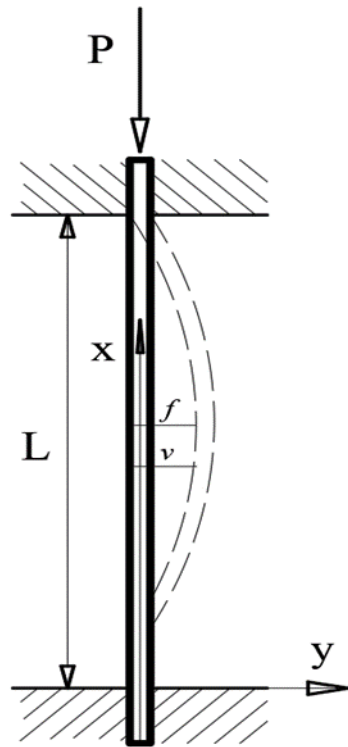


Figure 1.

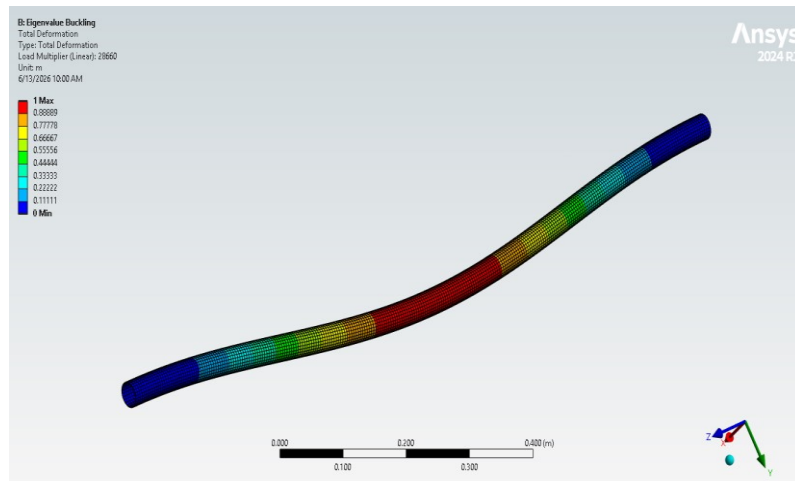


Figure 1. a) Complete deformation.

b) The case when both ends of the rod are pivotally connected

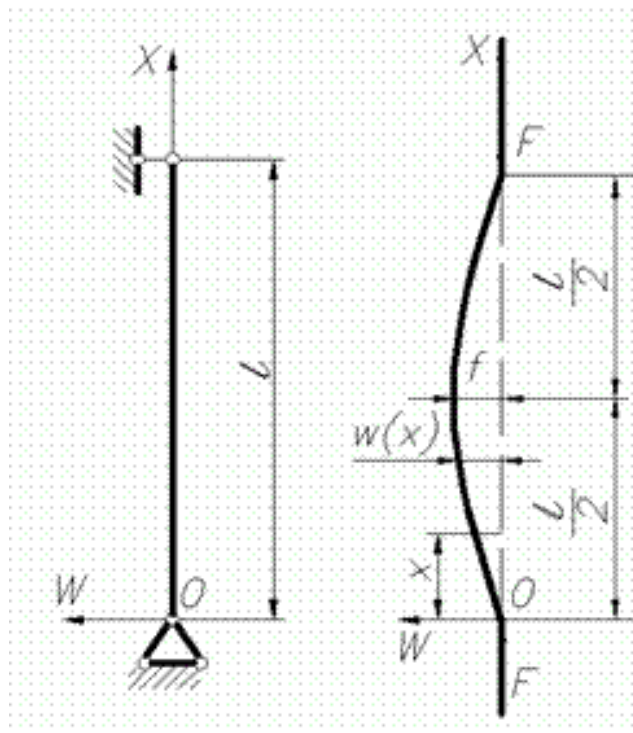


Figure 2.

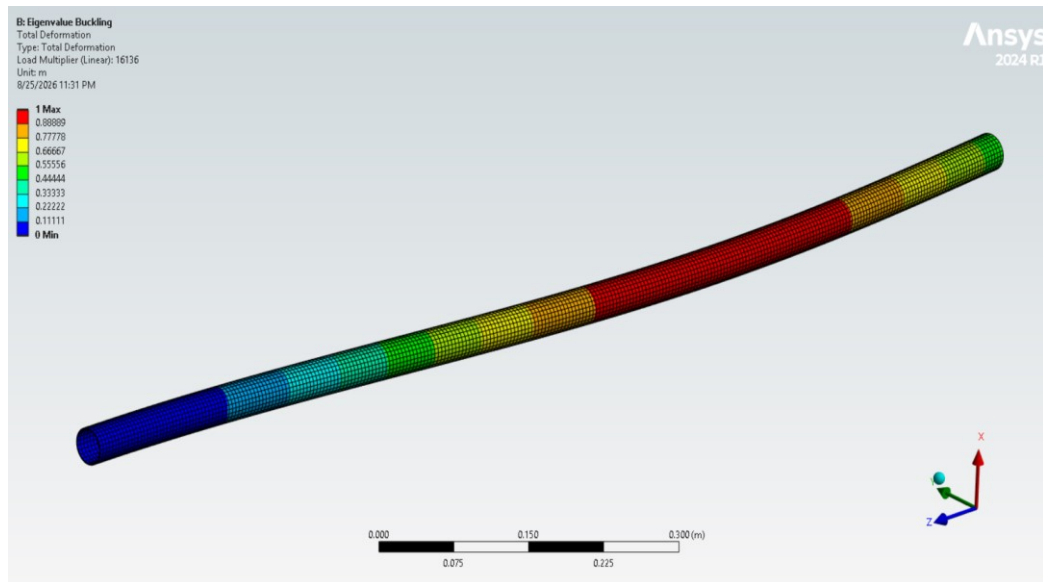


Figure 2. a). Intrinsic value of the strength loss analysis result, strength coefficient = 2.8659×10^5

Critical force acting on the structure - $P_{cr} = 1 \times 16136 = 16136 \text{ N}$

c) The case where one end is rigid and the other end is connected by a hinge.

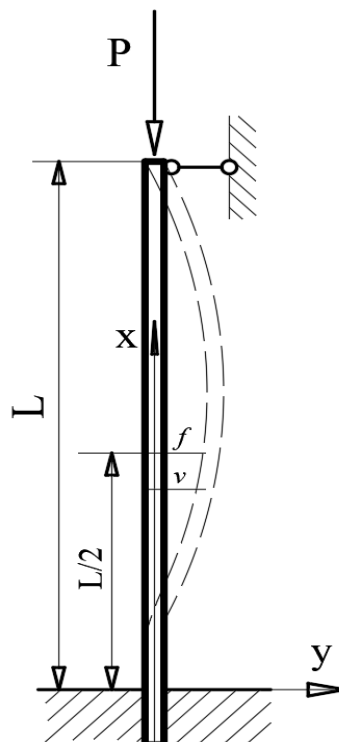


Figure 3.

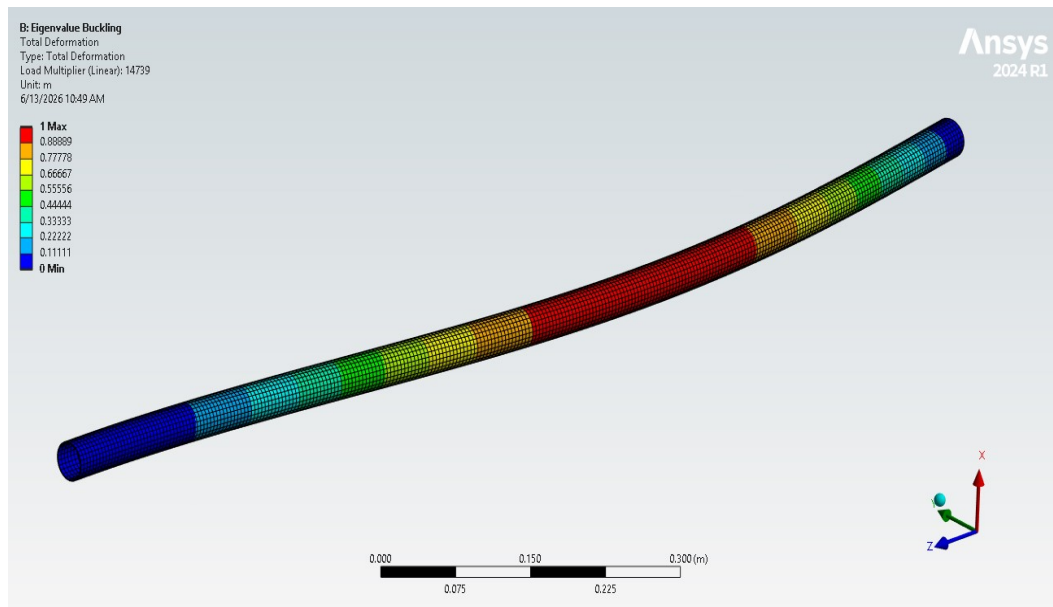


Figure 3. a)

Intrinsic value of the strength loss analysis result, strength coefficient = 2.8659×10^5

Critical force acting on the structure - $P_{cr} = 1 * 1815.1 = 1815.1 \text{ N}$

d) One end is free, the other end is rigidly fixed.

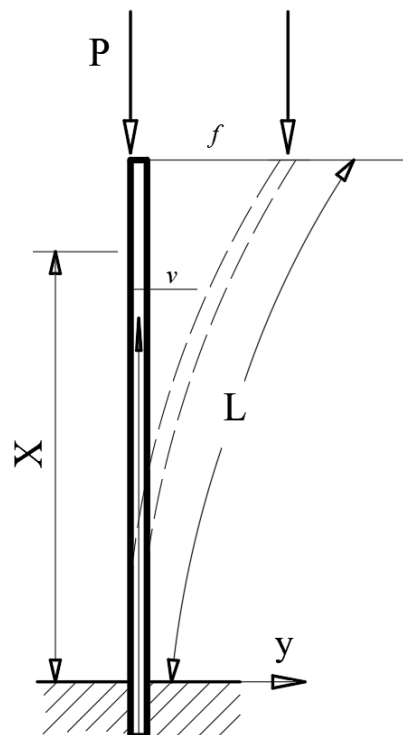


Figure 4.

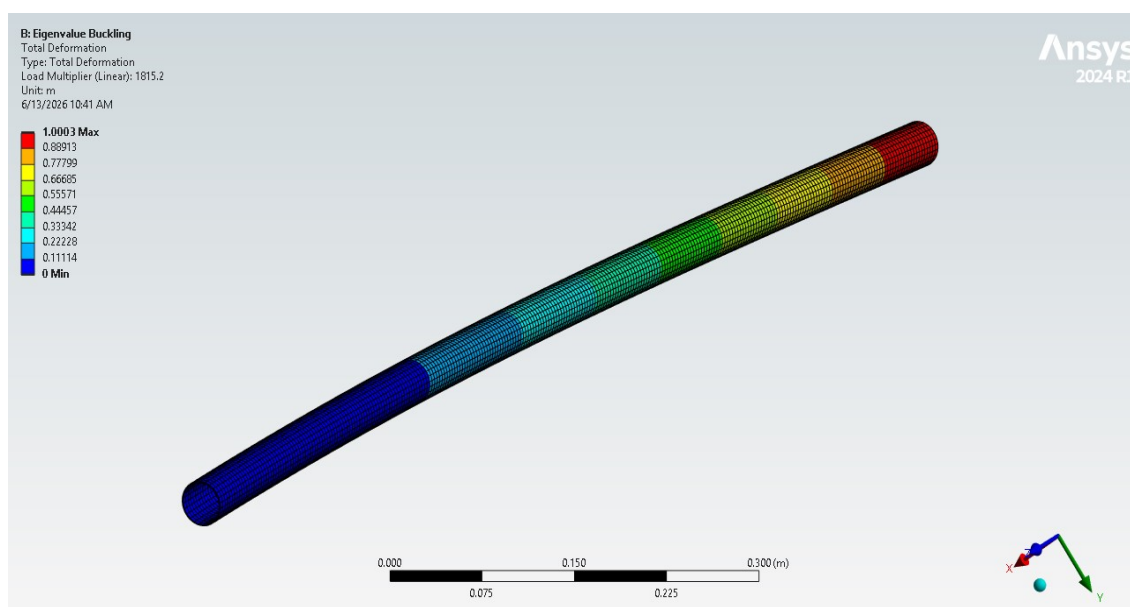


Figure 4. a) Intrinsic value of the strength loss analysis result, strength coefficient = 2.8659×10^5

$$\text{Critical force acting on the structure} - P_{cr} = 1 * 1815.1 = 1815.1 \text{ N}$$

It can be seen from Figures 1-4 that the strain stress spectrum is adequate for fixing the ends of the rod and gives a complete picture, as well as determines the value of the critical compressive force.

CONCLUSION

The application of the finite element method, in particular the ANSYS program, to the issue discussed in the article, as well as to a number of mechanical issues, greatly facilitates research work.

ƏDƏBİYYAT

1. Iskanderov, R.A. Stability and protection of drainage pipelines from environmental influences / R. Iskanderov, L. Mammadova A. Sariyeva // International Journal on “Technical and Physical Problems of Engineering” (IJTPE), Issue 64 Volume 17 Number 3, September 2025, p. 128-137.
<https://www.ijotpe.com/IJTPE/Issues/2020/IJTPE-2025/IJTPE-Issue64-Vol17-No3-Sep2025/14-IJTPE-Issue64-Vol17-No3-Sep2025-pp128-137.pdf>
2. Gulgezli, A. Variational principle for determining the stress-strain state of porous bodies / A. Gulgezli, O. Afandiyev // International Journal of Current Research. Impact factor 2016: 7.086 – 2017. – Vol. 9. – Issue 10. – ISSN 0975-833X.
3. Gulgazli, A.S. Energy condition of plasticity // Journal of Applied Mechanics and Technical Physics. – 2018. – Vol. 59. – No. 2. – p. 310–315. – ISSN 0021-8944.
4. Iskanderov, R.A., Afandiyev, O.Y. Stability of cylindrical coating made of porous material, Engineering Mechanics Scientific and Technical Journal, September 2025 Issue 21 Volume 13 Number 3 p. 46-54.