

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI

Publicat de

Universitatea Tehnică „Gheorghe Asachi” din Iași

Volumul 72 (76), Numărul 2, 2026

Secția

CONSTRUCȚII DE MAȘINI

DOI: 10.2478/bipcm-2026-0016

FINITE ELEMENT ANALYSIS OF CYLINDRICAL SPECIMENS IN TENSILE STRESS UNDER THE INFLUENCE OF STRESS CONCENTRATORS

BY

**GABRIEL-MADALIN MURARU^{1,2}, ANA-MARIA COMĂNICI³ and
CIPRIAN-IONUȚ MORĂRAȘ^{1,*}**

¹“Gheorghe Asachi” Technical University of Iași, Faculty of Mechanical Engineering,
Iași, Romania

²National Institute for Research and Development in Construction, Urban Planning and
Sustainable Spatial Development, Iași, Romania

³“Gheorghe Asachi” Technical University of Iași, Faculty of Machine Manufacturing and
Industrial Management, Iași, Romania

Received: April 16, 2026

Accepted for publication: June 15, 2026

Abstract. This paper presents a finite element analysis (FEA) of cylindrical specimens subjected to a tensile load, focusing on the influence of stress concentrators and geometric variations. Three types of specimens were studied: a reference specimen with a diameter of 8 mm, a specimen with a stress concentrator located at the midpoint of the gauge length, and a specimen with a diameter increased to 10 mm. The simulations were performed in ANSYS under linear static conditions. The numerical results indicate that the maximum equivalent elastic strain reached values of approximately 0.0031 for the reference specimen and 0.0039 for the specimen with a stress concentrator, highlighting a significant increase in local strain. The maximum values of the equivalent stress obtained were 659.92 MPa for the 8 mm specimen, 782.27 MPa for the specimen with a stress concentrator, and 627.36 MPa for the 10 mm specimen. These

*Corresponding author; *e-mail*: ciprian-ionut.moraras@academic.tuiasi.ro

© 2026 Gabriel-Madalin Muraru

This is an open access article licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International License (CC BY-NC-ND 4.0).

results clearly demonstrate stress amplification in the presence of geometric variation of the sample's cross section. The study confirms that stress concentrators significantly influence stress distribution and represent critical points for fracture initiation in tensile specimens.

Keywords: FEA, tensile test, concentrator shape.

1. Introduction

The mechanical behavior of materials subjected to tensile loads is significantly influenced by the presence of geometric discontinuities, commonly referred to as stress concentrators. Features such as notches, grooves, or abrupt changes in cross-sectional area lead to localized stress amplification, which can initiate failure even when the nominal stress remains below the material strength. Therefore, understanding and quantifying the effects of stress concentration is essential in the design and evaluation of engineering components.

The theoretical foundations of stress concentration were established in the studies of (Pilkey and Pilkey, 2008) who analyzed the stress distribution around elliptical holes. Recent studies have focused on numerical approaches for evaluating stress concentration factors, highlighting the effectiveness of finite element methods in accurately capturing stress gradients near notches (Bhosle *et al.*, 2025). Recent research has focused on improving the understanding and prediction of stress concentration factors in notched components using both analytical and numerical approaches (Zappalorto and Lazzarin, 2017).

Furthermore, various approaches combining analytical, numerical, and experimental methods have been developed to investigate stress concentration effects. For instance, (Glinka, 1985) proposed energy-based methods for evaluating stresses near notches, while more recent works have demonstrated strong agreement between FEM predictions and experimental measurements for isotropic materials.

With the advancement of numerical methods, finite element analysis (FEM) has become a powerful and versatile tool for analyzing stress and strain distributions in complex structures. FEM allows for accurate modeling of real geometries, boundary conditions, and material behavior, overcoming many limitations of classical analytical solutions. As highlighted by (Cook *et al.*, 2002) FEM provides reliable results for linear elastic problems, especially when appropriate discretization and boundary conditions are applied.

Despite the extensive body of research, the combined influence of specimen geometry—such as diameter variation—and the presence of localized stress concentrators on tensile behavior remains an area of practical interest. In particular, evaluating how these factors affect stress distribution, strain

localization, and failure initiation is important for optimizing specimen design and improving the interpretation of tensile test results.

The aim of this paper is to investigate, through finite element analysis, the mechanical response of cylindrical tensile specimens with different geometrical configurations. The study considers three cases: a reference specimen with a diameter of 8 mm, a specimen incorporating a stress concentrator at the midpoint of the gauge length, and a specimen with an increased diameter of 10 mm. The analysis focuses on the evaluation of equivalent stress, elastic strain, and total deformation fields, followed by a comparative assessment of the results in order to quantify the influence of geometric variations on stress distribution and deformation behavior.

2. Materials and Methods

In this study, a carbon steel with 0.45% carbon content (C45) was used. Numerical simulations were performed using ANSYS 2025 R2 software. The material properties considered in the analysis included a Young's modulus of 2.1×10^5 MPa and a Poisson's ratio of 0.26, assuming isotropic linear elastic behavior.

Three cylindrical geometries were modeled, with diameters of 8 mm and 10 mm, as well as a specimen with a stress concentrator. The analyses were carried out using the Static Structural module available in ANSYS.

The geometrical models were created in SpaceClaim, as shown in Fig. 1. The tensile test conditions were defined according to the ISO EN 6892:2010 standard.

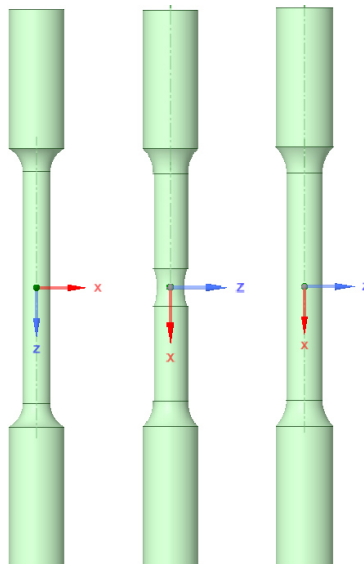


Fig. 1 – Set of specimens for Ansys testing.

3. FEM model for an 8 mm diameter specimen

The first specimen was subjected to a tensile test over its entire gauge length, with a length of $L_1 = 80$ mm and a diameter of $d_1 = 8$ mm. Figure 2 shows the loading and support conditions used in the finite element analysis of the tensile specimen, performed in the ANSYS software.

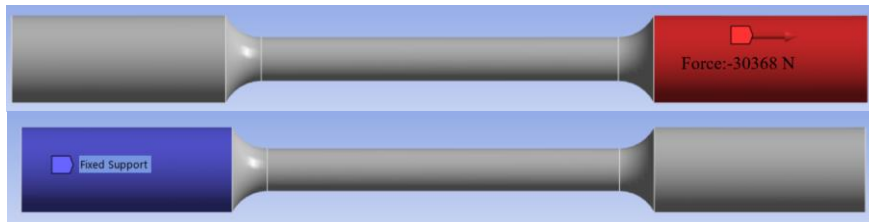


Fig. 2 – Boundary and loading conditions at model for an 8 mm diameter.

The finite element model was discretized using quadratic tetrahedral elements. For the specimen, we chose a discretization of 2 mm. Figure 3 shows the number of nodes is 33.974 and the number of elements is 22.372.

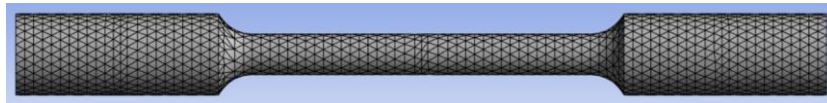


Fig. 3 – Mesh model.

Figure 4 illustrates the maximum equivalent elastic strain for tensile test specimen with stress concentrator over the entire calibration length is equal to 0.0031.

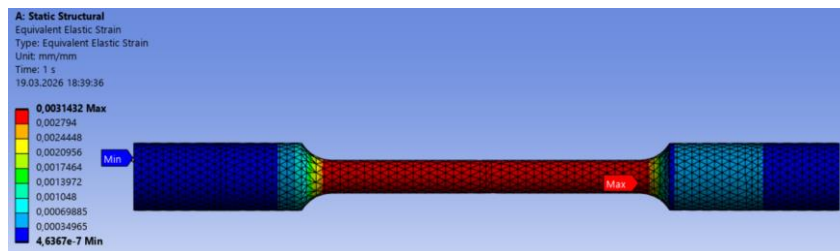


Fig. 4 – Equivalent Elastic Strain.

Figure 5 presents the maximum equivalent stress for tensile test specimen with stress concentrator over the entire calibration length is equal to 659.92 MPa.

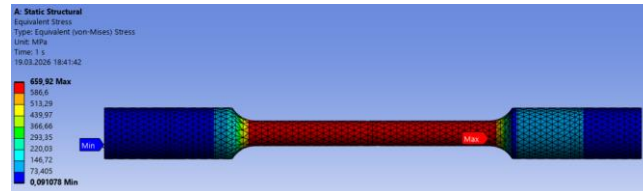


Fig. 5 – The distributions of the von Mises stress.

Figure 6 presents the maximum total deformation for tensile test specimen with stress concentrator over the entire calibration length is equal to 0.24 mm.

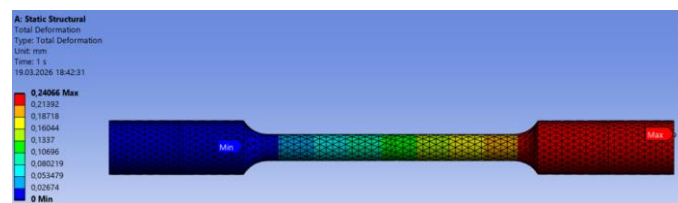


Fig. 6 – Total Deformation.

4. FEM model for the specimen with a stress concentrator

The second specimen subjected to tension test with a stress concentrator in the middle had a length $L_2 = 80$ mm and a diameter $d_2 = 7$ mm. It failed at $F = 27.580$ MPa. Figure 7 shown the breaking force for tensile specimen with stress concentrator in the middle.

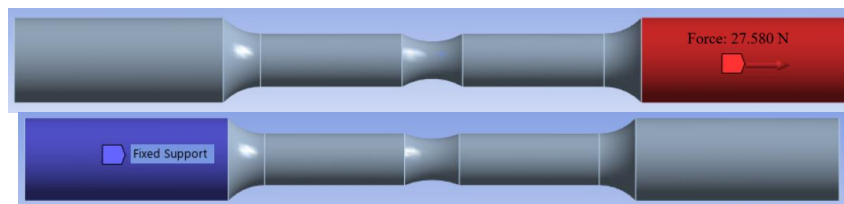


Fig. 7 – Boundary and loading conditions for the concentrator specimen.

Figure 8 highlights the portion of the specimen where the finite element discretization was applied.

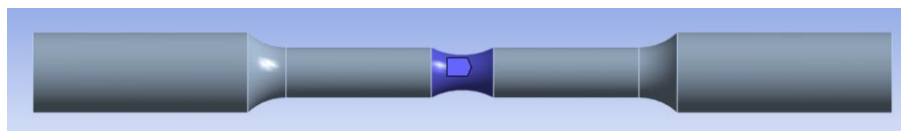


Fig. 8 – Local mesh in the stress concentrator region of the specimen.

A refined mesh was generated in the reduced gauge section of the specimen to accurately capture the stress distribution in the critical region. The finite element model was discretized using quadratic tetrahedral elements with an element size of 1.5 mm. The final mesh, presented in Figure 9, composed of 103.044 nodes and 70.471 elements.

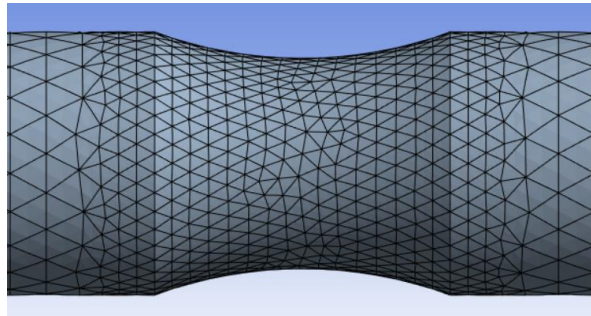


Fig 9 – Mesh model.

Figure 10 illustrates the maximum equivalent elastic strain for tensile specimen with stress concentrator in the middle is equal to 0.0039.

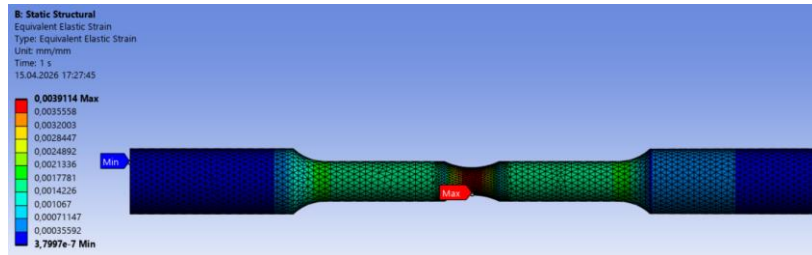


Fig. 10 – Equivalent Elastic Strain.

Figure 11 presents the maximum equivalent stress for tensile specimen with stress concentrator in the middle is equal to 782.27 MPa.

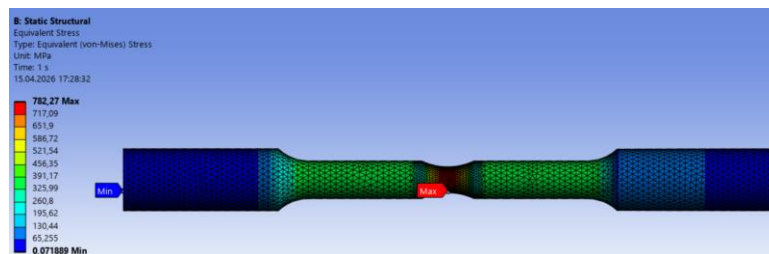


Fig. 11 – The distributions of the von Mises stress.

Figure 12 presents the maximum total deformation for tensile specimen with stress concentrator in the middle is equal to 0.17 mm.

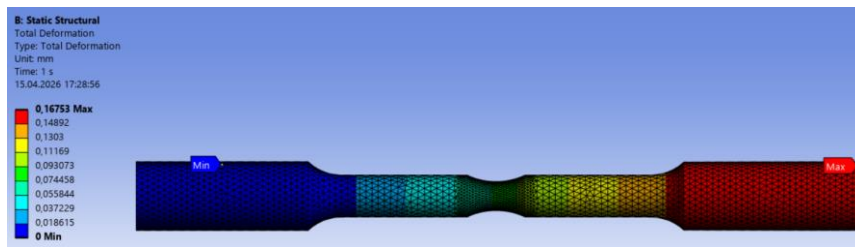


Fig. 12 – Total Deformation.

5. FEM model for a 10 mm diameter specimen

The third specimen tensile tested with a tensile testing had a length $L_3 = 80$ mm and a diameter $d_3 = 10$ mm. Figure 13 depicts the maximum load sustained by the 10 mm wide specimen prior to failure during tensile testing.



Fig. 13 – Boundary and loading conditions for a 10 mm diameter sample.

A quadratic tetrahedral mesh with an element size of 1.5 mm was generated for the specimen. The final finite element model, shown in Figure 14, comprises 37.300 nodes and 24.683 elements.

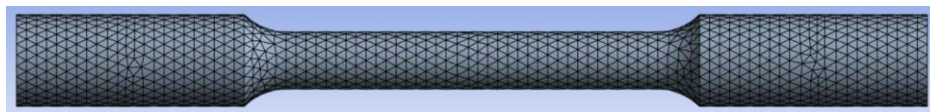


Fig. 14 – Mesh model.

Figure 15 illustrates the distribution of the maximum equivalent elastic strain in the 10 mm diameter tensile specimen at a tensile strain of 0.0031.

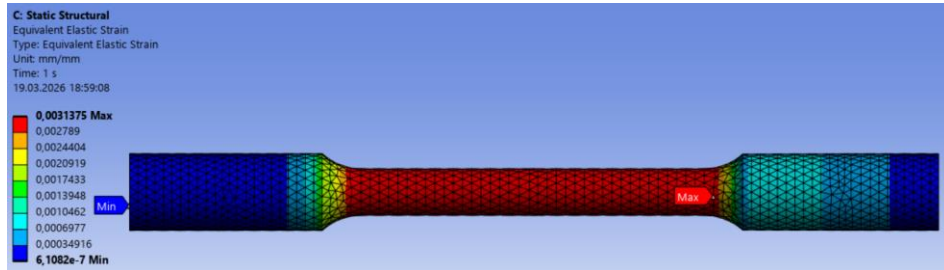


Fig. 15 – Equivalent Elastic Strain.

Figure 16 illustrates the maximum equivalent stress obtained for the 10 mm diameter tensile specimen, with a value of 627.36 MPa.

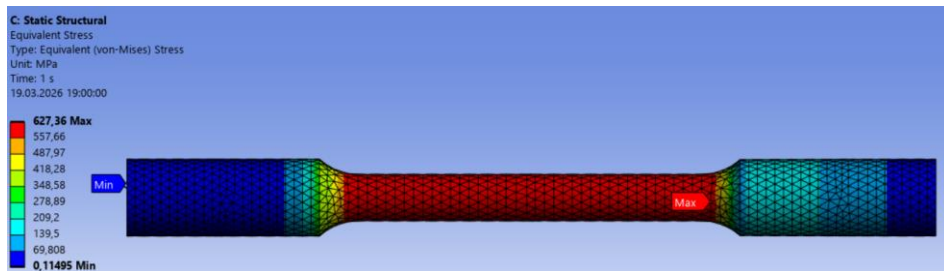


Fig. 16 – The distributions of the von Mises stress.

Figure 17 presents the maximum total deformation of the 10 mm diameter tensile specimen, with a value of 0.25 mm.

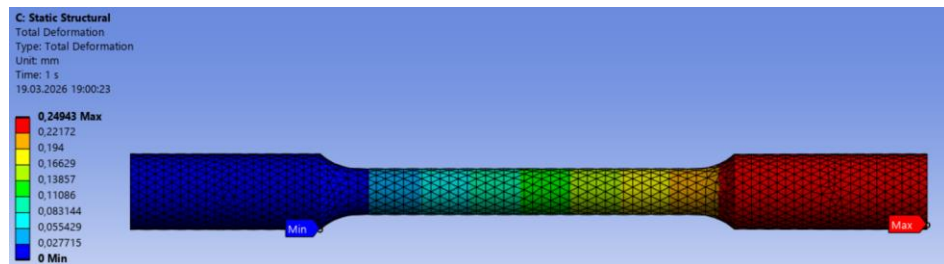


Fig. 17 – Total Deformation.

Table 1 summarizes the experimental tensile stresses and the corresponding von Mises stresses obtained from the finite element analysis.

Table 1
Stress analysis results

	Diameter [mm]	Area [mm ²]	Force [N]	σ_{UTS} [MPa]	$\sigma_{\text{von Mises}}$ [MPa]
Specimen one	8	50.27	30.368	604.1	659.92
Specimen two	7	38.46	27.580	716.56	782.27
Specimen three	10	73.54	45.023	573.25	627.36

6. Conclusion

This study highlights the role of geometry in determining the stress distribution in tensile test specimens. In the case of specimens without stress concentrators, the stress distribution remains stable and uniform, indicating predictable mechanical behaviour characteristic of tensile loading. The maximum von Mises stress occurs in the transition (fillet) zones of the cylindrical specimens, where geometric discontinuities naturally act as stress concentrators.

The introduction of a stress concentrator significantly alters this behaviour, leading to a marked redistribution of stresses and the emergence of critical zones where the equivalent stress values reach approximately 782.27 MPa, which is considerably higher than in the reference case 659.92 MPa. Furthermore, the obtained von Mises stresses exceed the experimental ultimate tensile strength. This indicates that local maximum stresses initiate crack formation, followed by a redistribution of stresses across the cross-section due to the development of plastic deformations. On the other hand, increasing the specimen diameter leads to a decrease in the stress level to 627.39 MPa, highlighting the direct influence of the cross-sectional area on the load-bearing capacity. Overall, the results underscore the fact that it is not only the overall stress level that is relevant, but especially how the geometry controls the local stress distribution, determining the critical zones and the failure mode.

Acknowledgements. The authors would like to thank Prof. Paul Doru Bârsănescu, Department of Mechanical Engineering, Mechatronics, and Robotics, “Gheorghe Asachi” Technical University in Iași, Romania, for his helpful advice and recommendations.

REFERENCES

- Bhosle S.M., Bhosale S.S., Mahadik S.C. et al., *Exploring research trends in use of finite element analysis for optimization of stress concentration factor in bars with fillets*, Discover Mechanical Engineering, vol. 4 (2025).
- Cook R.D., Malkus D.S., Plesha M.E., Witt R.J., *Concepts and Applications of Finite Element Analysis*, 4th ed., Wiley (2002).
- Glinka G., *Energy Density Approach to Calculation of Inelastic Strain-Stress Near Notches and Cracks*, Engineering Fracture Mechanics, vol. 22, no. 3, pp. 485–508 (1985).
- Pilkey W.D., Pilkey D.F., *Peterson's Stress Concentration Factors*, 3rd ed., Wiley (2008).
- Zappalorto M., Lazzarin P., *Stress concentration factors for notched components: recent developments and applications*, Theoretical and Applied Fracture Mechanics, vol. 91, pp. 1–20 (2017).

ANALIZA CU ELEMENTE FINITE A PROBELOR CILINDRICE SUB INFLUENȚA CONCENTRATORILOR DE TENSIUNI

(Rezumat)

Prezenta lucrare prezintă o analiză cu elemente finite (FEA) a unor epruvete cilindrice supuse unei sarcini de tracțiune, concentrându-se asupra influenței concentratorilor de tensiune și a variațiilor geometrice. Au fost studiate trei tipuri de epruvete: o epruvetă cu un diametru de 8 mm, o epruvetă cu un concentrator de tensiune situat în punctul median al lungimii de măsurare și o probă cu diametrul de 10 mm. Simulările au fost efectuate în ANSYS în condiții statice liniare. Rezultatele numerice indică faptul că deformația elastică echivalentă maximă a atins valori de aproximativ 0.0031 pentru proba de referință și 0.0039 pentru proba cu un concentrator de tensiune, evidențiind o creștere semnificativă a deformației locale. Valorile maxime ale tensiunii echivalente obținute au fost de 659.92 MPa pentru proba de 8 mm, 782.27 MPa pentru proba cu un concentrator de tensiune și 627.36 MPa pentru proba de 10 mm. Aceste rezultate demonstrează în mod clar amplificarea tensiunilor în prezența variațiilor geometrice ale secțiunii transversale a probei. Studiul confirmă faptul că concentratorii de tensiune influențează semnificativ distribuția tensiunii și reprezintă puncte critice pentru inițierea rupei în probele supuse la tracțiune.