МОДЕЛЮВАННЯ ТА ОПТИМІЗАЦІЯ В ТЕХНОЛОГІЇ КОНСТРУКЦІЙНИХ МАТЕРІАЛІВ

SIMULATION AND OPTIMIZATION IN TECHNOLOGY OF CONSTRUCTION MATERIALS

DOI: 10.31319/2519-8106.2(49)2023.292550
UDC 538.911

Kletskov Oleksandr, senior lecturer, Department of Higher Mathematics, Physics and General Engineering Disciplines
Клєцков О.М., старший викладач, кафедра вищої математики, фізики та загальноінженерних дисциплін
ORCID: 0000-0003-2587-4647
e-mail: alex.kl87@i.ua

Sakhno Viacheslav, Candidate of Physical and Mathematical Sciences, Associate Professor, Department of Higher Mathematics, Physics and General Engineering Disciplines
Сахно В.М., кандидат фізико-математичних наук, доцент, кафедра вищої математики, фізики та загальноінженерних дисциплін
ORCID: 0000-0002-2314-4547
e-mail: sakhno.v.m@dsau.dp.ua

Dnipro State Agrarian and Economic University, Dnipro
Дніпровський державний аграрно-економічний університет, м. Дніпро

COMPUTER MODELING OF METAL BEAM BENDING WITH DIFFERENT PROFILES

КОМП'ЮТЕРНЕ МОДЕЛЮВАННЯ ЗГИНУ МЕТАЛЕВИХ БАЛОК РІЗНИХ ПРОФІЛІВ

Steel beams are among the most common structural elements in construction and engineering structures. They are used to support and distribute loads, enabling the creation of stable and efficient structures. One of the key aspects in the design of steel beams is their bending behavior.

Research on the bending of steel beams with various profiles holds significant importance for engineers and designers, as it helps understand material behavior and determine optimal dimensions and shapes of structures. Conducting such research involves the use of analytical methods, experimental studies, and numerical models.

One of the outcomes of studying the bending of steel beams is establishing a relationship between the beam's profile, its dimensions, material properties, and load-carrying capacity. This data can be used for developing regulatory documents and standards that govern the design of steel structures. The selection of appropriate beam profiles is a crucial aspect of design. The choice of an optimal profile depends on various factors such as expected loads, beam length, allowable deformations, and material cost. Research allows for the identification of a profile that best satisfies all these requirements.

A comparative analysis between analytical methods and computer modeling revealed that the closest agreement with theoretical values is observed in modeling using Abaqus, while Ansys shows a significant increase in error as the profile shape becomes more complex. The significant error *increase in Ansys is primarily attributed to an insufficient number of mesh elements used to divide the beam according to the finite element method. Therefore, to mitigate substantial deviations from theoretical values when modeling in Ansys, mesh refinement is necessary. One of the research tasks was to perform bending modeling in different software under equal conditions, meaning with the same number of mesh elements. Under these conditions, Abaqus demonstrates an advantage over Ansys.*

***Keywords**: bending, beam, finite element method.*

Металеві балки є одними з найпоширеніших структурних елементів в будівництві та інженерних конструкціях. Вони використовуються для підтримки та розподілу навантаження, що дозволяє створювати стійкі та ефективні споруди. Одним із ключових аспектів проектування металевих балок є їх згин.

Дослідження згину металевих балок різних профілів мають велике значення для інженерів та дизайнерів, оскільки вони допомагають зрозуміти поведінку матеріалів і визначити оптимальні розміри та форму конструкцій. Проведення таких досліджень включає в себе використання аналітичних методів, експериментальних досліджень та чисельних моделей.

Одним з результатів дослідження згину металевих балок є встановлення залежності між профілем балки, її розмірами, матеріалом та здатністю переносити навантаження. Ці дані можуть бути використані для розробки нормативних документів та стандартів, які регулюють проектування сталевих конструкцій. Застосування відповідних профілів для балок є ключовим аспектом в проектуванні. Вибір оптимального профілю залежить від багатьох факторів, таких як очікуване навантаження, довжина балки, припустимі деформації та вартість матеріалу. Дослідження дозволяють визначити профіль, який найкраще задовольняє усі ці вимоги.

Проведений порівняльний аналіз між аналітичним методом та комп'ютерним моделюванням показав, що найбільш близьке співпадіння до теоретичного значення спостерігається при моделюванні в Abaqus, тоді як в Ansys спостерігається значне зростання похибки з ускладненням виду профілю. Значне зростання похибки зумовлене в першу чергу недостатньою кількістю елементів сітки на яку ділить програма балку, згідно методу скінченних елементів. Тож для виправлення значного відхилення від теоретичного значення при моделюванні в Ansys потрібно згущювати сітку. Однією з задач дослідження було проведення моделювання згину в різних програмах при рівних умовах, тобто однаковій кількості елементів сітки, а при таких умовах Abaqus має перевагу перед Ansys.

***Ключові слова**: згин, балка, метод скінченних елементів.*

Problem's Formulation

The creation of stable structures poses the task of choosing the most effective numerical methods and specialized software for modeling. The investigation of metal beam bending is one of the simplest tasks in computer modeling; nevertheless, it allows selecting the most accurate software for conducting more complex calculations and modeling various scenarios, thereby increasing the precision and efficiency of constructing different structures.

Analysis of recent research and publications

The investigation of metal beam bending with different profiles has been the subject of several works [1, 2]. Various authors have conducted computer modeling using diverse software tools [3, 4, 5].

Formulation of the study purpose

The aim of this work is to compare the results of numerical calculations and computer modeling obtained for beams with different profiles. The goal is to select a software tool that most accurately simulates the bending process, enabling the execution of more complex calculations.

Presenting main material

Metal beams are among the most common structural elements used in construction and engineering structures. They are employed for support and load distribution, enabling the creation of sturdy and efficient structures. One of the key aspects in the design of metal beams is their bending behavior.

Bending is a deformation that occurs when a moment is applied to a structural element. It can be an additional load caused by external factors or a result of the beam's own weight. To ensure the safe and efficient operation of steel structures, it is essential to understand how they bend under the influence of loads.

Research on the bending of metal beams with different profiles is of great significance to engineers and designers, as it helps to comprehend material behavior and identify optimal dimensions and shapes for constructions. Conducting such research involves the use of analytical methods, experimental studies, and numerical models.

Analytical methods are based on solving differential equations that describe the material behavior during bending. They allow obtaining analytical expressions for stresses, strains, and beam deflection. These methods are typically used for simple geometric shapes, such as rectangles or circles, and often require assumptions that may not fully consider all aspects of material behavior. As a result, their accuracy might be limited when applied to more complex structures or materials with non-linear behavior. Therefore, for more accurate and comprehensive analysis, numerical methods, such as the finite element method, are often employed to model the bending behavior of metal beams with different profiles. Numerical methods can handle more complex geometries and material behaviors, providing a more accurate representation of the real-world structural response.

Indeed, experimental research involves conducting physical tests on metal beams with various profiles and dimensions. These experiments are typically performed on specialized testing rigs where the loading is controlled, and deformations and stresses are measured. The results of such experiments help validate analytical models and determine the actual behavior of materials.

Numerical models, based on the Finite Element Method (FEM), divide the structure into a finite number of elements, and the equations describing material behavior are solved around these elements. These models allow the investigation of complex geometries and various loading conditions. They serve as powerful tools for predicting structural behavior and optimizing their design.

One of the outcomes of studying the bending of metal beams is establishing the relationship between the beam's profile, dimensions, material properties, and its load-carrying capacity. This data can be utilized in the development of regulatory documents and standards that govern the design of steel structures. The appropriate selection of beam profiles is a crucial aspect of design. Choosing the optimal profile depends on several factors, such as expected loads, beam length, acceptable deformations, and material costs. Research helps determine the profile that best satisfies all these requirements.

One of the crucial aspects of studying the bending of metal beams is understanding the material behavior during this process. Metals have good stability and elasticity, allowing beams to withstand significant loads without permanent damage. However, excessive loading can lead to plastic deformation, reducing stability, and even causing structural failure. Research helps determine the limits of permissible loads where the material remains elastic and assists in defining safety factors to ensure reliability.

Moreover, studying the bending of metal beams enables the development and optimization of specialized structures for various applications. For instance, in bridge design, research helps identify the optimal shape and size of beams to support heavy loads and enhance the longevity of the bridge structure. Similar investigations are also applied in designing buildings, cranes, automobiles, and other engineering structures.

In the modern world, with the advancement of computer technology, numerical methods and modeling are becoming increasingly popular in bending research of metal beams. This allows for complex calculations and modeling of various scenarios, enhancing the accuracy and efficiency of the investigations.

Based on the research on the bending of metal beams, modern regulations and standards are developed for structural design. These norms ensure high levels of safety and reliability in construction and aid in material optimization, reducing costs.

In this study, the problem of beam bending and determination of maximum stresses in a resilient and strong beam of length 100 mm with various cross-sectional shapes was solved. The material of the beam is steel: $E = 2.0 \cdot 10^{11}$ Pa, $\nu = 0.3$. One end of the beam is rigidly fixed, while a load of $p = 1kN$ is applied to the free end. The widely used software tools based on the Finite Element Method, namely Ansys and Abaqus, were employed for computer modeling.

According to the theory [1, 2], during bending in the cross-sectional area of the beams, maximum stresses occur, which can be computed using the following formula:

$$\sigma_{max} = \frac{M}{W_x}; \tag{1}$$

$$W_x = \frac{I_x}{y_{max}}, \tag{2}$$

where M is the bending moment in the cross-section, I_x is the moment of inertia about the principal central axis of the cross-section, which coincides with the neutral axis of the section, and y_{max} is the distance from the x-axis to the furthest fiber of the cross-section.

Let's consider the most common beam profiles:

1. Rectangle with sides b and h. The axial moment of inertia for this profile is:

$$I_x = \frac{b \cdot h^3}{12}. \tag{3}$$

Numerical calculations were performed with values of b = 5 mm and h = 10 mm, which revealed a maximum stress of 1600 MPa. However, computer modeling conducted using Ansys and Abaqus software (Fig. 1) showed stress values of 1306 MPa and 1556 MPa, respectively.

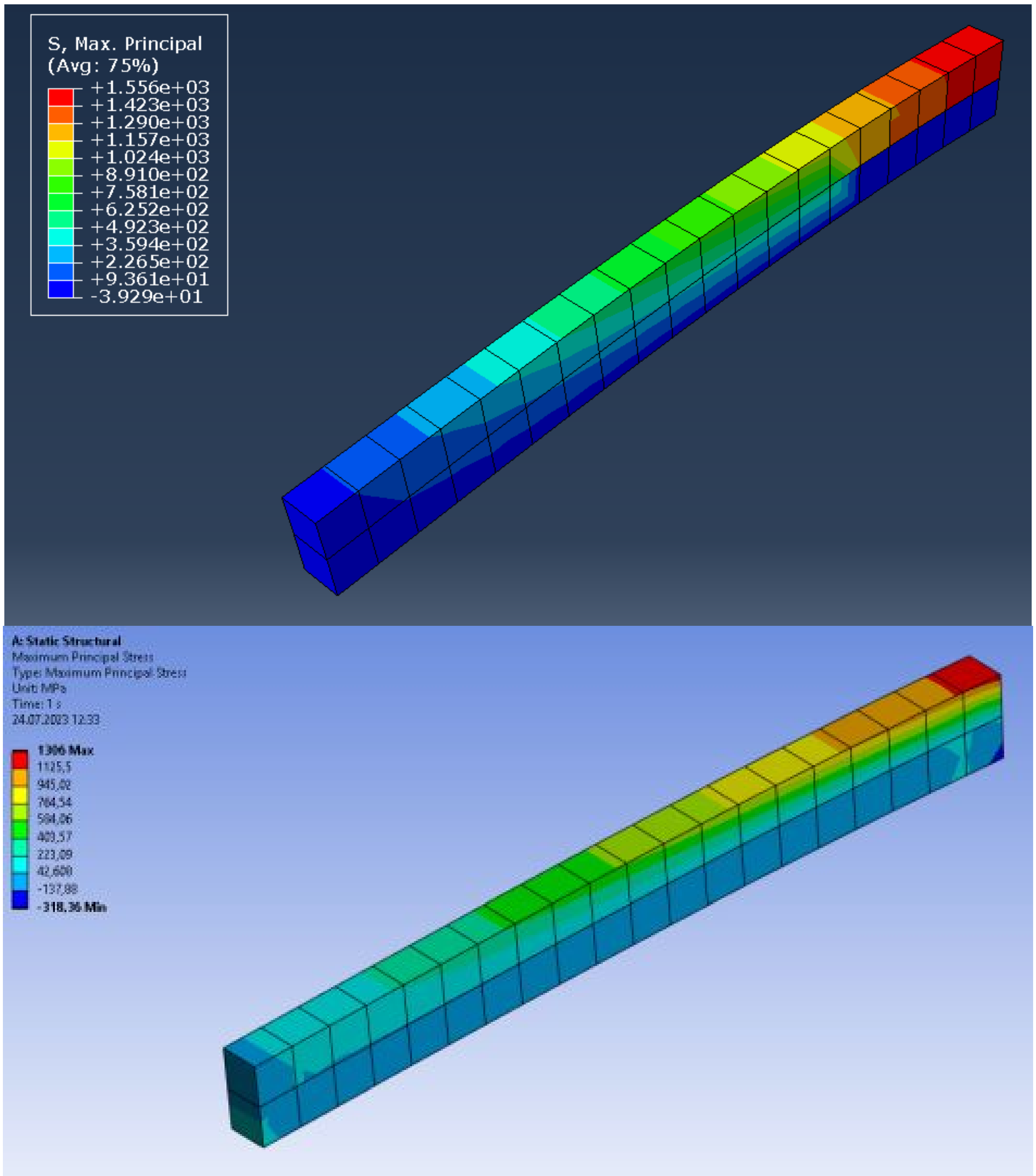


Fig. 1. Model of the deflection of a rectangular beam

2. The beam profile is a rectangle with sides b and h and a rectangular cutout inside with sides b_1 and h_1.

The axial moment of inertia for this profile is given by:

$$I_x = \frac{b \cdot h^3 - b_1 \cdot h_1^3}{12}. \tag{4}$$

Numerical calculations were performed with values of b = 5 mm, h = 10 mm, b_1 = 3 mm, and h_1 = 8 mm, which revealed a maximum stress of 546 MPa. However, computer modeling (Fig. 2) showed stress values of 344 MPa and 513 MPa, respectively.

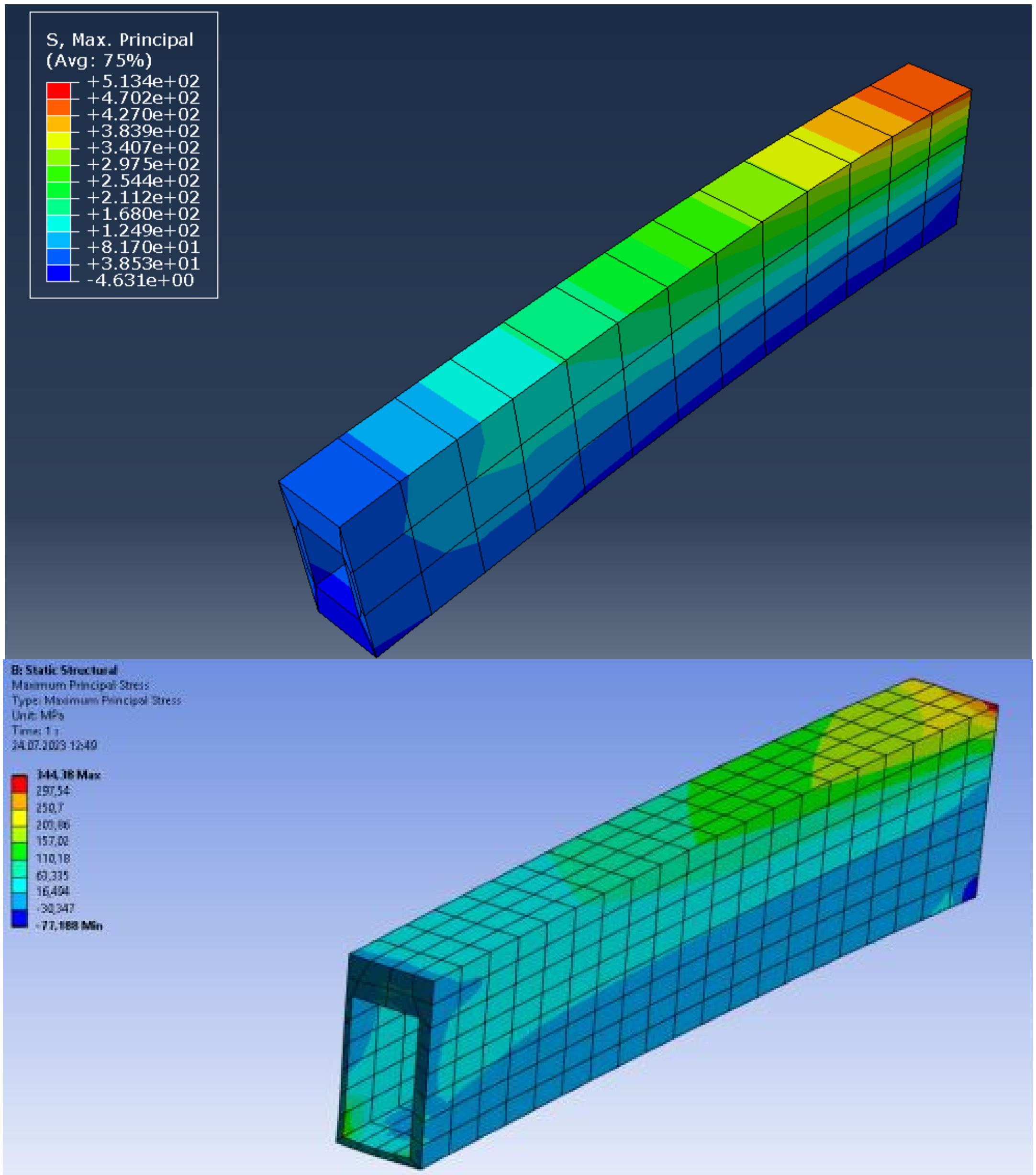


Fig. 2. Model of the deflection of a rectangular beam with a cutout

3. For a circular profile with a radius d, the axial moment of inertia is given by:

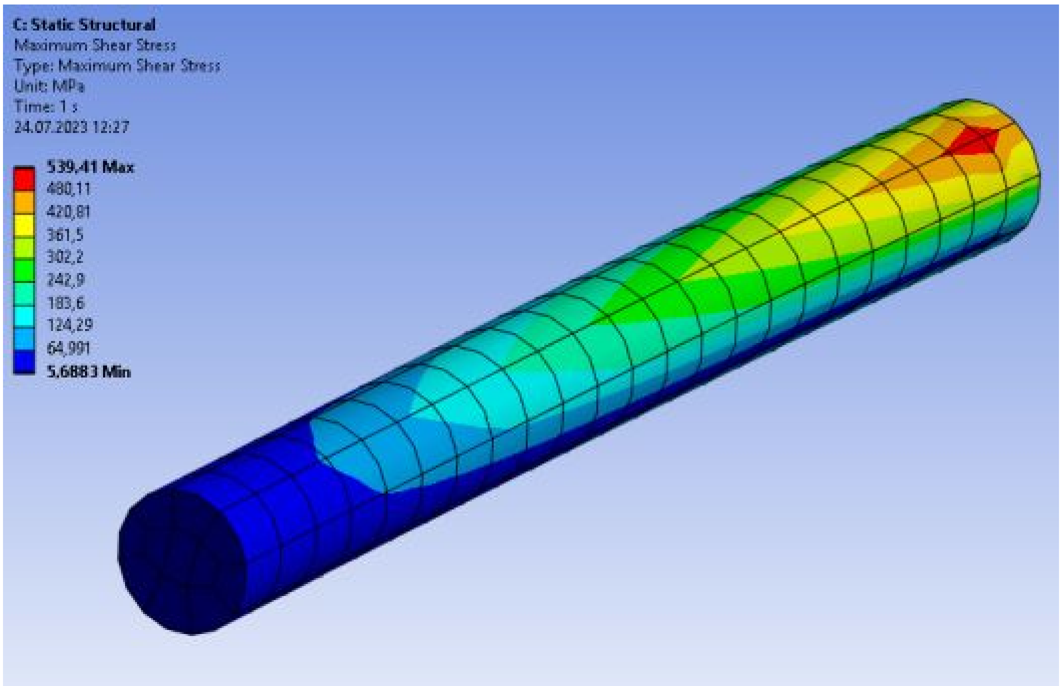
$$I_x = 0{,}049 \cdot d^4. \tag{5}$$

The calculations were performed with a value of d =10 mm, which revealed a maximum stress of 658 MPa. However, computer modeling (Fig. 3) showed stress values of 539 MPa and 686 MPa, respectively.

4. For a pipe with diameters d and d_1, the axial moment of inertia is given by:

$$I_x = 0{,}049 \cdot \left(d^4 - d_1^4\right). \qquad (6)$$

The analytical calculations with values of d=10 mm and d_1=5 mm revealed a maximum stress of 2100 MPa. However, computer modeling using Ansys and Abaqus software (Fig. 4) showed stress values of 1421 MPa and 1967 MPa, respectively.



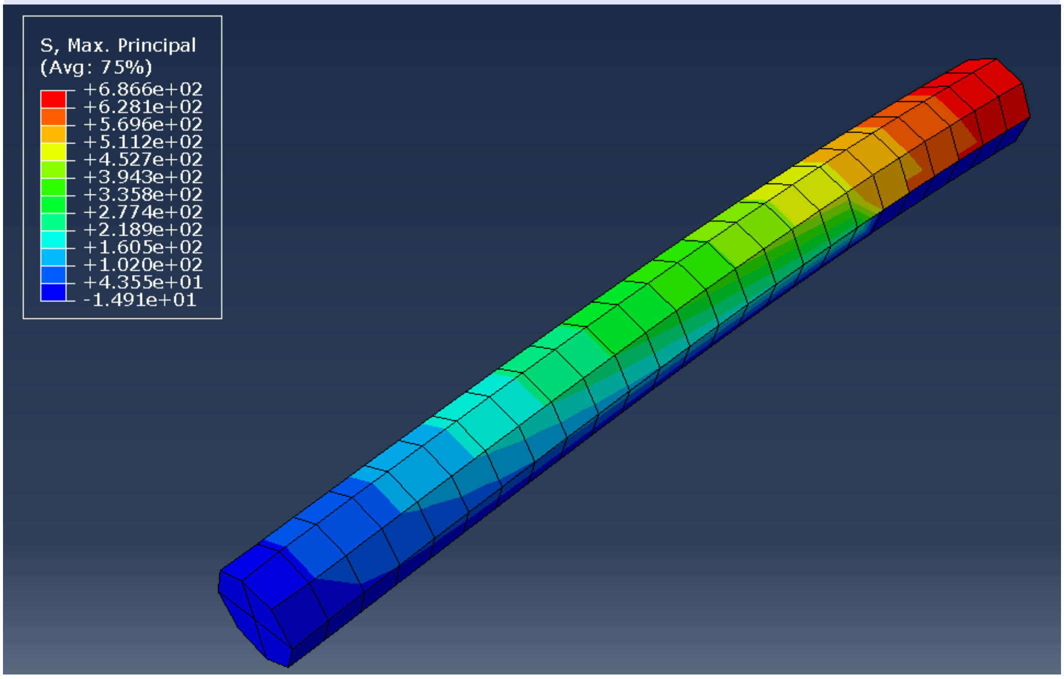


Fig. 3. Model of the deflection of a beam with a circular profile

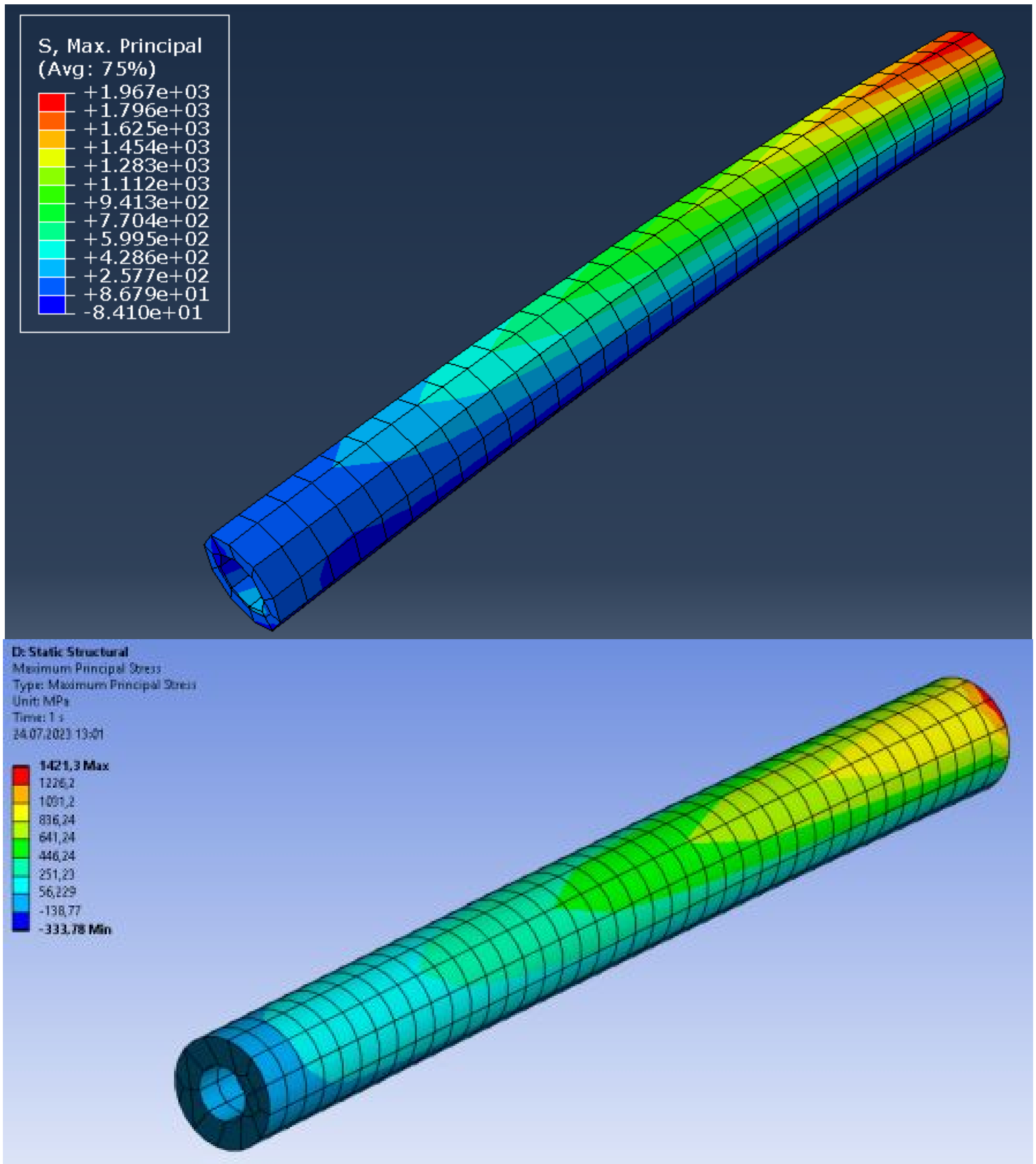


Fig. 4. Model of the deflection of a pipe

5. For a profile in the shape of the letter "L", the axial moment of inertia is given by:

$$I_x = \frac{1}{3} \cdot \left(d \cdot l_1^3 + H \cdot \left(H - l_1^3\right) - \left(H - d\right) \cdot \left(H - l_1 - d\right)^3\right), \tag{7}$$

where H is the height of the upper part and the width of the lower part of the profile, d is the thickness of both the upper and lower parts of the profile, and l_1 is the distance from the bottom of the profile to its center of gravity.

Numerical calculations were performed with values of H=10 mm, d = 5 mm, and l_1 = 5.5 mm, which revealed a maximum stress of 1700 MPa. However, computer modeling using Ansys and Abaqus software (Fig. 5) showed stress values of 1266 MPa and 1511 MPa, respectively.

6. For a profile in the shape of the letter "C", the axial moment of inertia is given by

$$I_x = \frac{1}{12} \cdot \left(B \cdot H^3 - b \cdot h^3\right), \tag{8}$$

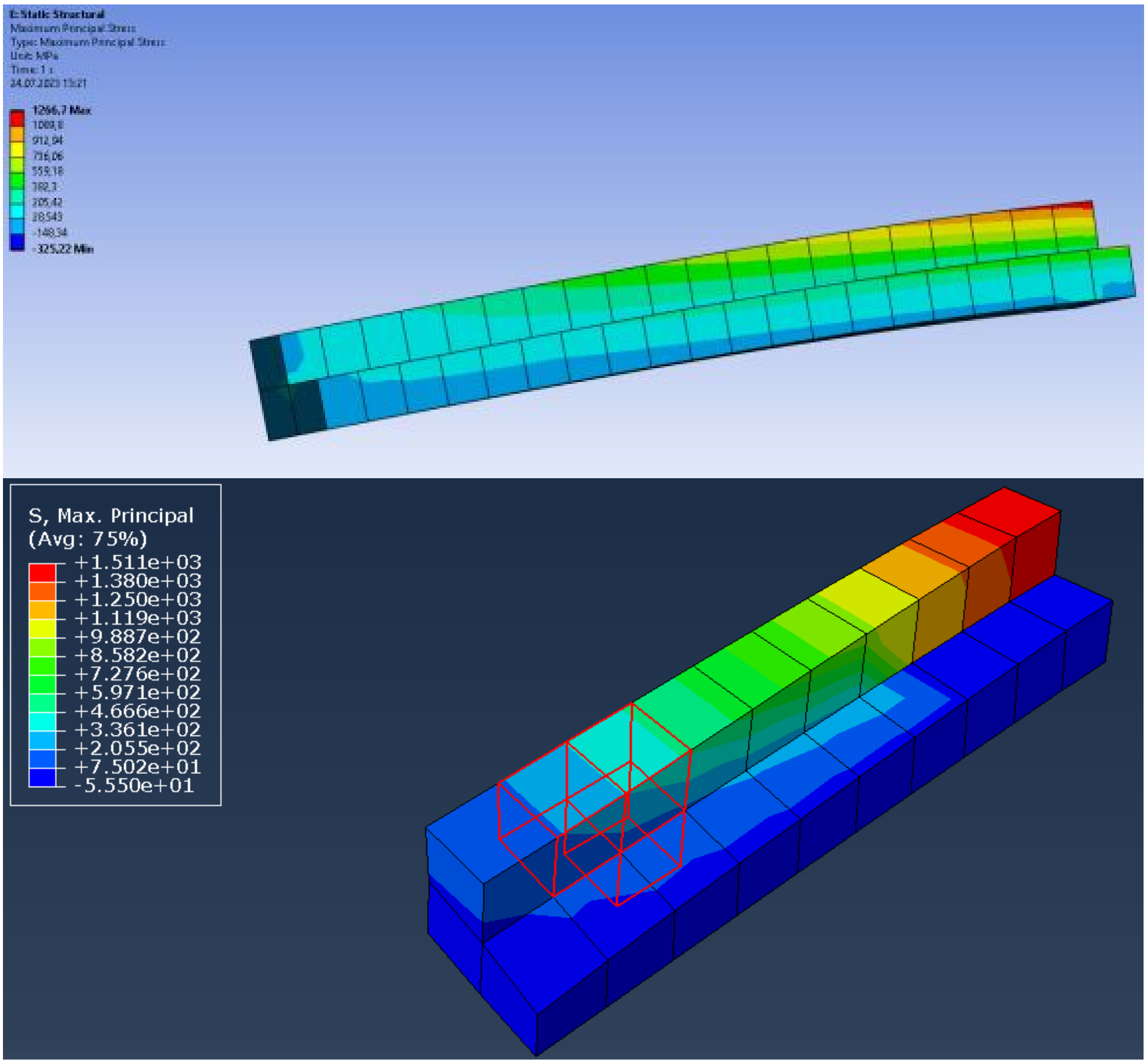


Fig. 5. Model of the deflection of the profile in the shape of the letter "L"

where H is the height of the upper part of the profile, B is the width of the lower part of the profile, h is the depth of the recess in the profile, and b is the width of the recess in the profile.

Numerical calculations were performed with values of H = 10 mm, B = 5 mm, h = 8 mm, and b = 3 mm, which revealed a maximum stress of 3464 MPa. However, computer modeling using Ansys and Abaqus software (Fig. 6) showed stress values of 1903 MPa and 3662 MPa, respectively.

Conclusions

The comparative analysis between the analytical method and computer modeling showed that the closest agreement with the theoretical values is observed when modeling in Abaqus, while Ansys exhibits significant errors as the complexity of the profile increases. The significant increase in errors in Ansys is primarily due to the insufficient mesh density, according to the finite element method. Therefore, to reduce the deviation from the theoretical values in Ansys, it would be necessary to refine the mesh.

One of the research objectives was to perform modeling in different software under the same conditions, i.e., with an equal number of mesh elements. Under such conditions, Abaqus demonstrated an advantage over Ansys.

In conclusion, for modeling mechanical deformations, Abaqus appears to be the most suitable software based on the above-mentioned comparisons and results.

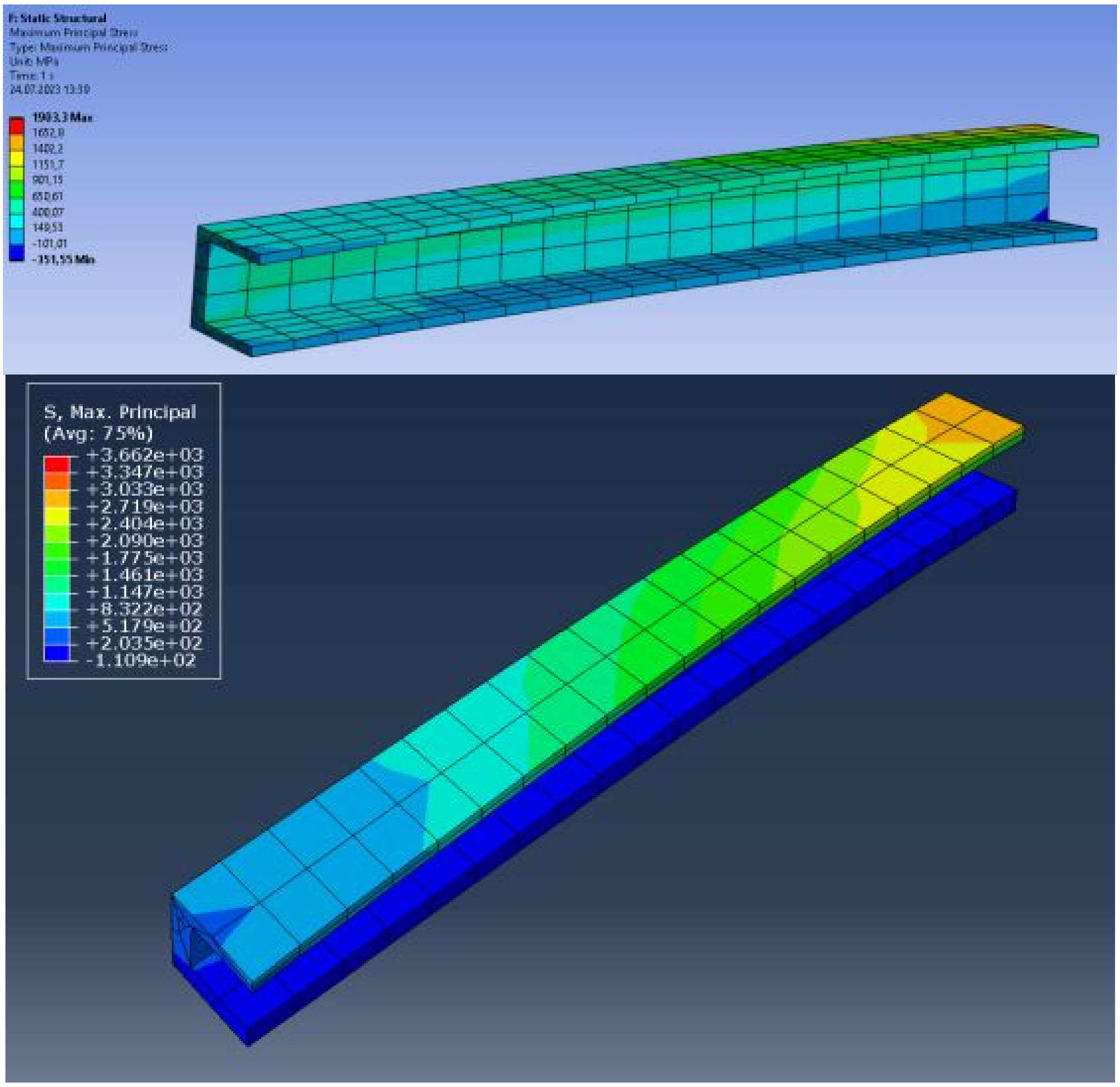


Fig. 6. Model of the deflection of the profile in the shape of the letter "C"

References

[1] Tsurpal, I.A. (2005). Mekhanika materialiv i konstruktsiy [Mechanics of materials and structures]. Kyiv: *Vischa osvita* [in Ukranian]
[2] Kravchuk, V.S, Dashenko, O.F., Kolomiets, L.V., Limarenko, O.M. (2014). Osnovy oporu materialiv [Basics of resistance of materials]. Odessa: *Standart* [in Ukranian]
[3] Yahno, B.O. (2013). Rozrakhunok zvarnykh z'yednan' z vykorystannyam prohramnoho kompleksu Abaqus [Calculation of welded joints using the Abaqus software complex]. Kyiv: *KPI* [in Ukranian]
[4] Becker, A.A. (2006). Introduction to the ABAQUS finite element software. Nottingham: *The University of Nottingham* [in English]
[5] Baraniuk, O.V. (2019). Matematychne modelyuvannya system ta protsesiv [Mathematical modeling of systems and processes]. Kyiv: *KPI* [in Ukranian]

Список використаної літератури

1. Цурпал І.А. Механіка матеріалів і конструкцій: навч. посіб. Київ: Вища освіта, 2005. 357 с.
2. Кравчук В.С., О.Ф. Дащенко О.Ф., Коломієць Л.В., Лимаренко О.М. Основи опору матеріалів: навч. посіб. Одеса: Стандарт, 2014. 322 с.
3. Яхно Б.О. Розрахунок зварних з’єднань з використанням програмного комплексу Abaqus: методичні вказівки. Київ: НТУУ "КПІ", 2013. 64 с.
4. Becker A.A. Introduction to the ABAQUS finite element software. Nottingham: The University of Nottingham, 2006. 28 p.
5. Баранюк О.В. Математичне моделювання систем та процесів: Комп'ютерний практикум. Київ: КПІ ім. Ігоря Сікорського, 2019. 115 с.

Надійшла до редколегії 10.07.2023