Middle East Research Journal of Engineering and Technology
ISSN: 2789-7737 (Print) ISSN: 2958-2059 (Online)
Frequency: Bi-Monthly
DOI: https://doi.org/10.36348/merjet.2025.v05i03.001

Website: http://www.kspublisher.com/
Email: office@kspublisher.com

Investigation into the Utilization of Abaqus Software for the Analytical Modeling and Instruction of Core Strength of Materials Challenges

Phung Tien Duy[1*], Nguyen Thi Thu Phương[1]
[1]Faculty of Engineering Technology – Hung Vuong University – Phu Tho province

Abstract: This study investigates the integration of ABAQUS simulation software into the instruction and analysis of Strength of Materials problems at Hung Vuong University. The primary aim is to enhance pedagogical approaches and facilitate student engagement with complex engineering concepts through dynamic and interactive simulations. ABAQUS empowers learners to visualize and attain a more profound comprehension of mechanical behaviors such as stress distribution, deformation, and displacement in structural elements. By leveraging virtual simulations, students can develop practical and technical competencies without the necessity of physical experimentation, thereby optimizing both time and resource efficiency. Moreover, the software contributes to the optimization of material design, provides a controlled and safe environment for experimentation, and cultivates critical problem-solving abilities. The application of ABAQUS enables students to thoroughly understand key factors influencing material strength and load-bearing performance, while simultaneously equipping them with the professional skills required in modern engineering practice. Overall, the incorporation of ABAQUS into the Strength of Materials curriculum not only elevates the quality of instruction but also bridges the gap between theoretical knowledge and practical application.

Keywords: ABAQUS, Strength of Materials, simulation software, finite element analysis (FEA), design optimization.

Review Paper

***Corresponding Author:**
Phung Tien Duy
Faculty of Engineering Technology – Hung Vuong University – Phu Tho province

How to cite this paper:
Phung Tien Duy & Nguyen Thi Thu Phương (2025). Investigation into the Utilization of Abaqus Software for the Analytical Modeling and Instruction of Core Strength of Materials Challenges. *Middle East Res J. Eng. Technol, 5*(3): 40-47.

Article History:
| Submit: 14.05.2025 |
| Accepted: 13.06.2025 |
| Published: 17.06.2025 |

1. INTRODUCTION

In the context of contemporary education, where rapid advancements in science and technology must align with the practical demands of industry, the innovation of teaching methodologies has become a critical priority. A core element of pedagogical reform lies in enhancing learners' practical competencies while fostering autonomy and creativity throughout the learning process. This not only enables students to comprehend and apply knowledge more effectively but also lays a solid foundation for their future professional development. To achieve this objective, the implementation of advanced teaching approaches—particularly those involving visual simulation—plays a pivotal role.

Visual simulation methods have garnered considerable attention from researchers and have been widely adopted across various academic disciplines, especially in the teaching of engineering subjects at the university level. Such simulations not only allow students to visualize theoretical concepts more intuitively but also provide opportunities for self-directed exploration, experimentation, and engagement with near-real-world models. According to M. T. S. Júnior *et al.* [1], visual simulation significantly enhances students' analytical capabilities and comprehension of complex engineering processes without requiring access to expensive equipment or experimental conditions.

This approach has been successfully applied to subjects such as Descriptive Geometry, Technical Drawing, and Engineering Graphics. Research by L. T. Nguyen *et al.* [2] demonstrated that incorporating simulation in teaching graphical subjects enables students to clearly visualize complex shapes and structures, thereby deepening their understanding of geometric and engineering theories without relying solely on abstract theoretical lectures. Furthermore, other studies have confirmed that the use of simulation in technical disciplines improves students' ability to access and solve real-world problems [1].

In particular, for subjects related to engineering drawing and graphics, simulation methods empower students not only to illustrate technical objects but also to understand the structural and functional factors influencing their design. Simulations of processes such as polyhedral intersections or pulse modulation in electronics help learners conceptualize complex models, thereby enhancing their analytical thinking and creative problem-solving abilities [1].

These studies collectively suggest that visual simulation is not only effective in improving theoretical knowledge retention but also in creating an engaging and intuitive learning environment. This enables students to better observe and internalize the core principles of the subject, ultimately improving their ability to apply theoretical knowledge in practical contexts. This is especially vital in engineering fields, where practical skills and problem-solving capabilities are key determinants of professional success.

Strength of Materials is a fundamental subject within the engineering curriculum, particularly in disciplines such as mechanical, civil, and structural engineering. The course equips students with essential knowledge of stress, strain, and load-bearing capacity of materials under external forces. Beyond foundational theories in solid mechanics, the course also develops students' ability to analyze and solve technical problems related to structural integrity in real-world applications.

Given the rapid pace of technological advancement, integrating software tools into the teaching and research of Strength of Materials has become a critical trend. Among these tools, ABAQUS, a leading finite element analysis (FEA) software, stands out for its robust capabilities in mechanical modeling and simulation. Widely used across industries such as automotive, aerospace, construction, and manufacturing, ABAQUS effectively addresses complex problems involving stress, deformation, and nonlinear phenomena.

Developed by Dassault Systèmes, ABAQUS is a powerful simulation tool that utilizes finite element methods to analyze complex mechanical behavior. It supports not only linear elastic analysis but also nonlinear, dynamic, and thermomechanical simulations, including plastic deformation, material failure, and heat transfer. By applying ABAQUS in the Strength of Materials course, students can simulate realistic engineering problems without relying on time-consuming and costly physical experiments.

Zhang *et al*. [4] found that incorporating ABAQUS into teaching significantly enhances students' ability to visualize and understand mechanical phenomena that are otherwise difficult to convey through theory alone. Simple structural problems—such as axially loaded bars, compressed columns, and 2D/3D deformation scenarios—can be visually simulated, allowing students to grasp both theoretical and physical aspects of these phenomena more effectively.

A study by Nguyen Van Hung *et al*. [5] in Vietnam also confirmed that using ABAQUS in teaching Strength of Materials deepens students' understanding of stress and strain factors, such as load magnitude, geometry, and material composition. These simulations not only streamline the learning process but also allow students to analyze complex scenarios, thereby enhancing their ability to tackle real-world engineering challenges.

One illustrative application of ABAQUS in this subject is the simulation of basic problems involving axial members and compressed structures. Liu *et al*. [6] employed ABAQUS to simulate and analyze axially loaded members in three-dimensional settings, demonstrating the software's accuracy in computing stress and strain under various loading conditions. This enabled students to better understand how loading and geometry affect structural behavior. Additionally, Jiang *et al*. [7] studied the use of ABAQUS in column compression analysis, showing that the software allows both students and engineers to predict structural capacity under compressive loads and simulate plastic deformation and failure modes—capabilities that are difficult to achieve through manual calculation. Consequently, the application of ABAQUS in teaching and research continues to yield positive outcomes, equipping future engineers with the skills necessary to address complex mechanical and structural problems.

2. Research Content

2.1. Simulation and CAE Analysis Methods in ABAQUS

2.1.1. Concept of CAE

CAE, or Computer-Aided Engineering, refers to the use of computer software to simulate the behavior and performance of systems under virtual conditions, with the goal of rapidly identifying solutions, improving product design, or addressing technical challenges. It encompasses simulation, validation analysis, and optimization of products, manufacturing processes, and tools. Currently—and increasingly in the future—CAE systems serve as a critical decision-making support tool for designers during the product development cycle.

With the continuous advancement of science and technology, the application of computer software across various industries has become increasingly prevalent. In particular, engineering disciplines such as mechanical and civil engineering often deal with highly complex models and data sets that cannot be efficiently processed using traditional analytical methods or conventional computing. Thus, simulation and data processing software—capable of delivering fast, accurate results—has become essential in supporting engineers, enhancing work efficiency, productivity, and effectiveness.

While CAD (Computer-Aided Design) focuses on the creation of 2D or 3D models of components and systems, CAE represents the next stage in the digital engineering process. It involves simulating and analyzing the behavior of those models to inform design decisions and refine product performance. CAE plays a crucial role in providing engineers and designers with in-depth information needed to evaluate and resolve complex technical issues.

In CAE practice, three primary numerical methods are typically employed to obtain solutions for mathematical models: the Finite Element Method (FEM), the Finite Difference Method (FDM), and the Boundary Element Method (BEM). Implementing these techniques effectively requires users to have a solid foundation in mechanical engineering and physics, including knowledge of forces, stresses, strains, deformations, and more. This report primarily focuses on the Finite Element Method (FEM)—also known as Finite Element Analysis (FEA)—as the core analytical tool.

FEA is a numerical method, distinct from analytical solutions in theoretical mathematics. It combines principles from mathematics, physics, engineering, and computer science to solve complex problems in structural analysis, heat transfer, fluid and gas dynamics, multibody systems, and electromagnetic fields. These problems often involve partial differential equations with complex variables—such as geometry, loading conditions, or material properties—that are difficult or impossible to solve analytically.

Numerical solution methods provide an effective approach for addressing such challenges. FEM divides the overall problem into smaller, discrete elements, then uses numerical algorithms to approximate solutions within each element. Modern computing power enables these systems of equations to be solved simultaneously, generating near-continuous solutions across the entire model using interpolation and statistical approximation techniques.

Mathematically, FEM is a digital engineering technology used to approximate solutions to boundary value problems defined by differential equations. It utilizes variational methods (i.e., the calculus of variations) to minimize an error function and obtain a stable, approximate solution. Much like how a series of small straight lines can closely approximate a circular curve, FEM works by connecting multiple simple element equations across subdomains (finite elements) to approximate a complex equation over a larger domain.

As a result, FEA involves a computer-generated model in which material properties, loads, boundary conditions, and environmental interactions are defined. The analysis then produces quantitative results—such as stress, strain, displacement, or thermal gradients. FEA is widely used for evaluating the performance of new designs or optimizing and improving existing products.

2.1.2. Applications of CAE in Engineering

The Finite Element Method (FEM) is a transformative technology that has revolutionized various fields. It is not only limited to engineering and scientific disciplines but has also been applied to solve numerous real-world problems. Today, Finite Element Analysis (FEA) is widely utilized across industries such as aerospace, automotive, electronics, heavy industry, oil and gas, and bioengineering. It offers substantial advantages in structural analysis, thermal analysis, and fluid dynamics.

The key advantages of Computer-Aided Engineering (CAE) include the ability to evaluate performance and improve product quality, which leads to reduced R&D costs and shorter time-to-market. CAE facilitates detailed examination of physical quantities such as stress fields, strain, pressure, temperature, and force—enabling engineers to generate innovative and creative design solutions.

For physical phenomena that are difficult or impossible to observe, test, or experiment with using conventional methods, CAE provides a powerful tool for virtual observation and analysis. Today, CAE simulation results are often regarded as essential technical documentation for presenting design concepts and persuading clients or stakeholders.

The FEM offers significant advantages in engineering design and analysis. First, it allows for the analysis and design of arbitrarily shaped structures in a straightforward and efficient manner. It is particularly effective for handling scenarios involving large stresses or deformations—critical conditions in many engineering applications. Additionally, due to the element-wise separation of equations, different regions of a model can be composed of various materials, offering flexibility in design. FEM also supports complex boundary condition management, enabling the simulation of realistic, multifaceted situations.

Moreover, FEM is user-friendly and result-oriented, making it a widely adopted tool among engineers. It can also capture nonlinear behavior due to large deformations or complex material properties. With abundant software resources and comprehensive documentation, FEM has become a flexible and effective numerical method. However, since most CAE tools rely on numerical approximations, results may not be 100% identical to physical reality. Nevertheless, they offer valuable insights through relative comparisons. The accuracy of CAE outcomes depends heavily on factors such as mesh quality and boundary conditions, meaning user expertise and experience are critical.

FEM and FEA essentially refer to the same concept. "FEA" is the more commonly used term in industry, while "FEM" is more prevalent in academic settings. FEA spans numerous areas, including heat transfer, vibration, material strength, acoustics, and more. FEM encompasses various mathematical techniques such as the Galerkin method, weighted residual methods, and other numerical integration schemes, which form the foundation of FEA's mathematical framework. The term "Finite Element Method" (FEM) generally refers to the underlying mathematical theories, often found in academic texts filled with equations and derivations, while "Finite Element Analysis" (FEA) typically refers to the practical application of these methods to real-world engineering problems.

CAE is extensively applied across a wide range of engineering problems, aiding in the optimization of design and the analysis of complex systems. One fundamental application is the dynamic analysis of components in assemblies using FEA to evaluate stress and deformation in structural parts. CAE is also used for thermal and fluid analysis through Computational Fluid Dynamics (CFD), which enables simulation of flow phenomena and heat transfer.

Additionally, CAE supports motion and dynamic analysis of mechanical systems, as well as simulation of mechatronic systems and multi-domain designs. Other applications include control system analysis, process simulations such as casting and stamping, and product or process optimization—ultimately improving efficiency, accuracy, and innovation in engineering design.

2.1.3. Overview of ABAQUS Software

ABAQUS is a comprehensive and high-performance finite element analysis (FEA) software suite, widely utilized in advanced engineering simulations. It is capable of addressing an extensive spectrum of problems, ranging from basic linear analyses to highly complex nonlinear and dynamic phenomena. ABAQUS offers a rich element library that enables the modeling of virtually any geometric configuration and provides an extensive collection of material models that accurately simulate the behavior of diverse structural materials—including metals, elastomers, polymers, composites, and reinforced concrete.

Beyond structural mechanics (e.g., stress, strain, and displacement analyses), ABAQUS extends its capabilities into multidisciplinary domains such as thermal conduction, acoustic analysis, electromagnetic interactions, and electromechanical coupling. This breadth of functionality positions ABAQUS as a versatile platform for multiphysics simulation across various engineering disciplines.

The ABAQUS suite is composed of several core modules, each serving a distinct function:

- ABAQUS/Standard: A robust implicit solver ideal for both linear and nonlinear static and low-speed dynamic analyses. It is particularly well-suited for problems involving complex material behavior, contact mechanics, and thermal-structural coupling.
- ABAQUS/Explicit: An explicit solver tailored for high-speed, transient dynamic simulations—especially suitable for highly nonlinear events such as impact, crash, blast, and rapid deformation. It is widely used in safety engineering and crashworthiness assessment.
- ABAQUS/CAE (Complete ABAQUS Environment): An intuitive graphical user interface that facilitates pre-processing (geometry creation, meshing, boundary condition definition, material assignment) and post-processing (results visualization and interpretation).
- ABAQUS/Viewer: A dedicated post-processing tool for analyzing simulation outputs.
- ABAQUS/Aqua and ABAQUS/Design: Specialized modules for hydro-structural interaction (used in offshore engineering) and design optimization workflows, respectively.

Originally developed in the 1970s by David Hibbitt, founder of ABAQUS Inc., the software was envisioned as a powerful tool for solving real-world engineering problems using the finite element method. Over the decades, ABAQUS has undergone continuous development, evolving from a linear elastic solver into a fully integrated multiphysics simulation platform. In 2005, Dassault Systèmes acquired ABAQUS Inc. and incorporated it into its SIMULIA brand, which continues to expand its capabilities through enhanced interoperability, advanced material modeling, and optimization technologies.

Today, ABAQUS is a cornerstone in both academic research and industrial engineering practice. Its applications span a broad range of sectors, including aerospace, automotive, civil infrastructure, biomechanics, and consumer product design. Through SIMULIA, ABAQUS enables sophisticated simulations such as fatigue life prediction, topology and shape optimization, and real-world performance validation of products under realistic service conditions.

In essence, ABAQUS provides a unified, end-to-end simulation environment, empowering engineers and researchers with state-of-the-art computational tools to model, analyze, and optimize complex physical systems with precision, efficiency, and confidence.

2.2. Application of ABAQUS Software in the Simulation-Based Analysis and Instruction of Fundamental Strength of Materials Problems

2.2.1. Standard Workflow for Solving a Problem Using ABAQUS

A typical Finite Element Analysis (FEA) model in ABAQUS follows a well-defined sequence of steps:

- **Step 1: Model Creation:** This initial phase involves using ABAQUS's modeling tools to construct the geometry. Users can sketch and define components in either 2D or 3D, depending on the nature of the problem. Alternatively, geometry can be imported from intermediate files generated by external 3D CAD software.

- **Step 2: Partitioning the Model:** The model is divided into smaller, more manageable sub-regions to facilitate mesh generation. Partitioning is essential for handling complex geometries, non-uniform load distributions, or when refined post-processing in specific regions is required.
- **Step 3: Assigning Material Properties:** Material definitions, such as elasticity, density, or thermal properties, are assigned to each component in the model to represent their physical behavior under loading.
- **Step 4: Mesh Generation:** A finite element mesh is generated for each part or assembly. Mesh density and element type selection play critical roles in ensuring solution accuracy and convergence.
- **Step 5: Assembly Creation:** Individual components are assembled into a complete model using the Assembly module, allowing the definition of interactions and constraints between parts.
- **Step 6: Step Definition:** Analysis steps are defined to specify the sequence and duration of loading or environmental changes. These steps must be established prior to assigning loads and boundary conditions and are crucial for capturing the physical behavior of the system over time.
- **Step 7: Defining Boundary Conditions and Loads:** Boundary conditions (e.g., fixed supports, constraints) and external loads (e.g., forces, pressures) are applied to the model in accordance with the previously defined analysis steps.
- **Step 8: Simulation Execution:** The model is submitted for analysis using the appropriate ABAQUS solver (Standard or Explicit), depending on the problem type.
- **Step 9: Results Visualization:** Post-processing involves interpreting the output results, such as stress, strain, or displacement fields, through ABAQUS/CAE or ABAQUS/Viewer.
- **Step 10: Exporting and Analyzing Results:** Simulation data is exported for reporting, documentation, or further evaluation. Post-processing may also involve quantitative analysis, such as extracting peak stress or deformation values.

2.2.2. FEA Simulation of a Bending Beam Problem

Axial tension and compression in a prismatic bar represent one of the most fundamental problems in the study of Strength of Materials, particularly within mechanical and structural engineering curricula. This type of loading scenario is also commonly encountered in real-world engineering structures. When a bar is subjected to axial forces—either tensile or compressive—it undergoes elongation or shortening, affecting its load-bearing capacity. The core objective of such an analysis is to determine the maximum stress and displacement values within the member, which are essential for evaluating both the strength and stiffness of the structure [8,9].

Example Problem: Simulate the bending behavior of an I-beam steel section using FEA in ABAQUS.

- Cross-section dimensions: 80 mm × 180 mm × 6 mm × 8 mm
- Length: L = 2000 mm
- Axial Load: P = 2 kN applied at one end of the beam
- Material properties: Elastic Modulus E=210 GPaE = 210 \, \text{GPa}E=210GPa, Poisson's Ratio ν=0.3\nu = 0.3ν=0.3

Step 1: Beam Modeling

The beam is modeled as a static analysis problem in ABAQUS. Each component can be created individually, then assembled using the Assembly module. Geometry is defined precisely to reflect the I-beam cross-section and dimensions.

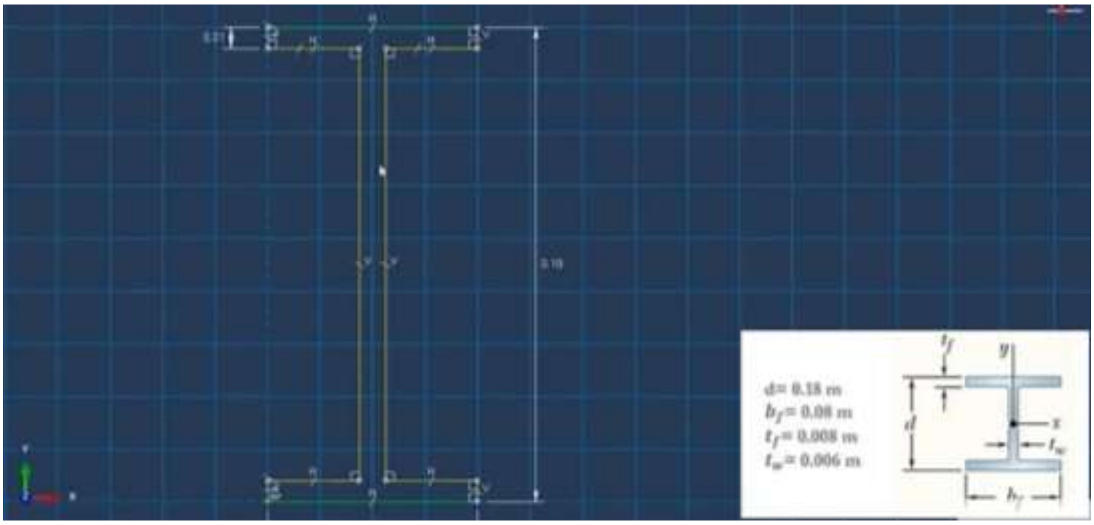

Figure 1: Cross-Sectional Geometry of the Beam Created in the Part Module

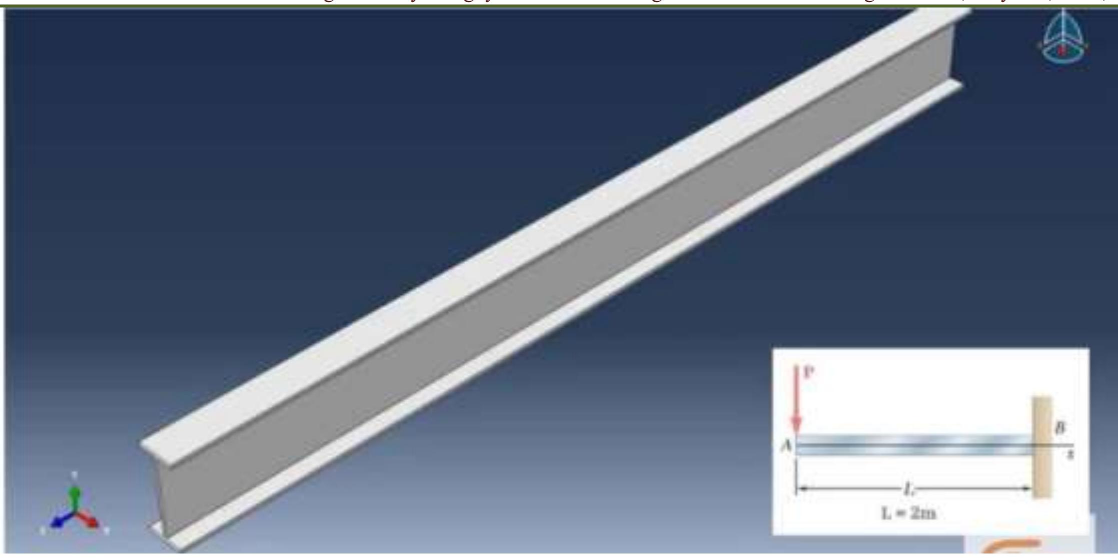

Figure 2: 3D Model Generation

Step 2:

Assign Material Properties The mechanical properties of the selected material can be defined and assigned within ABAQUS using the Property module. This includes specifying material constants such as Young's modulus, Poisson's ratio, yield strength, and other relevant parameters depending on the type of analysis.

Step 3:

Assemble the Structure Components are assembled into a unified structural model within the Assembly module, allowing for the definition of relative positions and connections between parts.

Step 4:

Define the Analysis Step Choose the "Static, General" step for a quasi-static simulation. ABAQUS operates based on a step-based analysis history, where each step represents a distinct phase of the simulation—such as static, dynamic, or transient thermal loading. In its simplest form, a step may consist of a static loading scenario where the applied forces vary gradually over time or load magnitude.

Step 5:

Apply Boundary Conditions and External Loads Define boundary conditions (e.g., fixed supports, symmetry constraints) and apply external loads (e.g., concentrated forces, pressures, distributed loads) corresponding to the physical scenario. These inputs are specified within the context of the previously created analysis step.

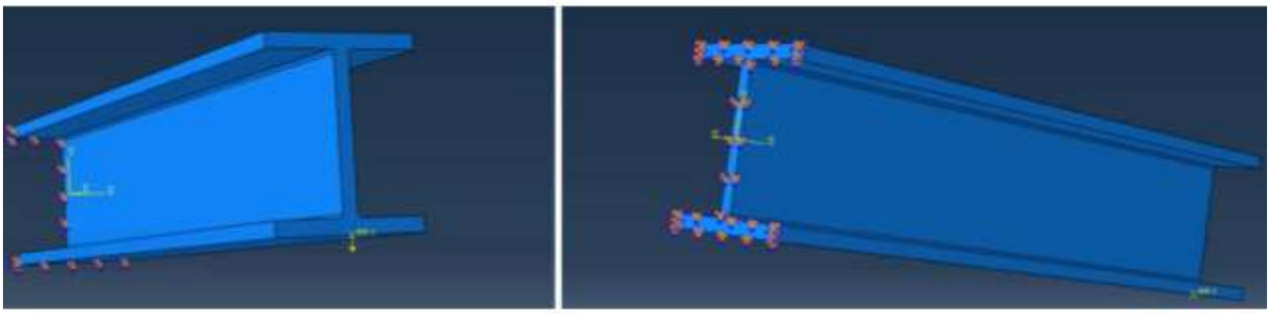

Figure 3: Applying Boundary Conditions and Applying Loads

Step 6: Selecting the Element Type

An FEA model defines a mesh, which is an arrangement of finite elements. A mesh can be generated on a part or an assembly within ABAQUS/CAE. The process of discretizing the geometry into a mesh that represents the finite elements is called meshing.

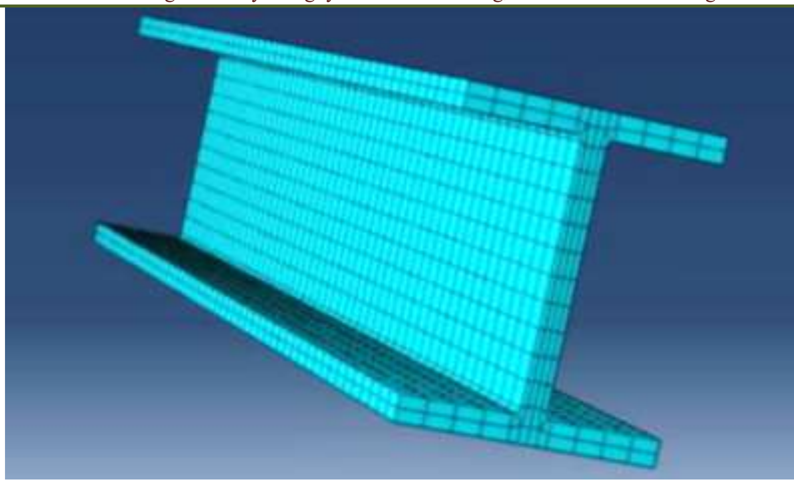

Figure 4: Meshing

Step 7: Results

In the visualization module, you can select the results you want to display. For example, you can choose stress or strain and view these results across all elements of the model. This module also allows you to plot graphs of any variables you wish. The FEA analysis results help engineers evaluate or refine their designs.

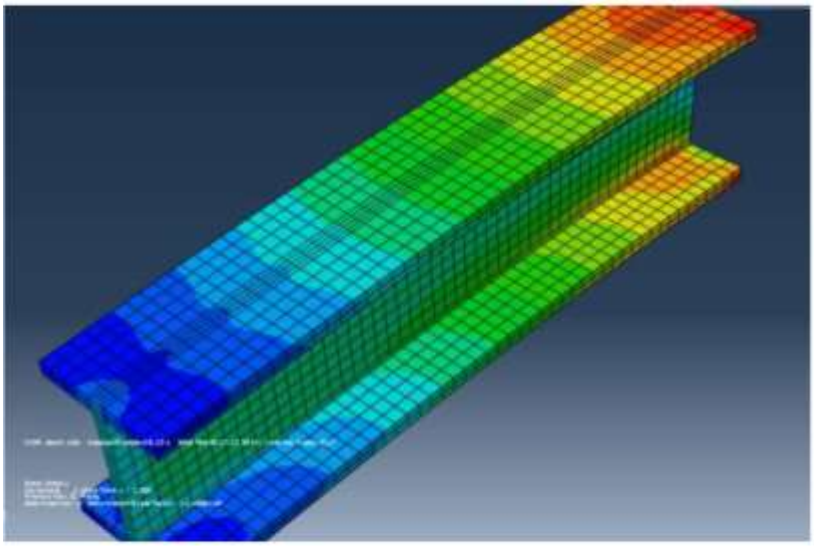

Figure 5: FEA Analysis Results in ABAQUS

The application of ABAQUS software in analyzing and teaching simulation of strength of materials problems at Hung Vuong University has brought significant practical benefits to both instructors and students. This teaching approach opens new learning opportunities, not only deepening students' theoretical understanding but also enhancing their practical skills and creative thinking through complex simulation problems.

One notable advantage is the creation of an engaging learning experience. Using ABAQUS in teaching allows students to interact directly with simulation models, increasing their participation and enthusiasm during the learning process. Instead of merely studying theory abstractly, students can visually observe and experience mechanical phenomena through the software, making lessons more vivid and easier to grasp.

Moreover, ABAQUS supports advanced learning levels. The simulation results of strength problems, such as stress and displacement, are presented visually, helping students grasp complex issues that are difficult to approach through theory alone. Observing these simulated outcomes enables students to better understand how factors like material properties, geometry, and loading conditions influence structural behavior, thereby enhancing their comprehension and ability to apply knowledge in real-world situations.

3. CONCLUSION

Amid the rapid advancement of science and technology, the integration of analysis and simulation software into teaching Strength of Materials specifically, and engineering courses in general, is critically important. The use of ABAQUS in instruction provides students with a clear and vivid understanding of strength of materials problems while simultaneously fostering practical skills and creative thinking. As a result, students are equipped with the essential tools and knowledge to effectively apply the software in real-world engineering tasks after graduation.

REFERENCES

1. M. T. S. Júnior, *et al*. (2020). "Application of simulation for teaching technical drawing." *International Journal of Engineering Education*.
2. L. T. Nguyen, *et al*. (2019). "The role of visual simulation in technical education." *Journal of Engineering Education*.
3. Zhang, X. & Wang, J. (2020). "Improving engineering education through simulation tools." *Simulation & Modeling Practice and Theory*.
4. Zhang, L., Wang, S., & Xu, X. (2017). "Application of ABAQUS in teaching mechanical engineering." *Journal of Engineering Education*.
5. Nguyễn Văn Hùng, *et al*. (2018). "Ứng dụng phần mềm ABAQUS trong giảng dạy môn học Sức bền vật liệu." *Tạp chí Khoa học và Công nghệ*.
6. Liu, Y., Zhang, J., & Zhao, F. (2019). "Finite element analysis of beams under various loads using ABAQUS." *International Journal of Mechanical Engineering*.
7. Jiang, L., Yu, Z., & Li, T. (2020). "Nonlinear analysis of columns under compression with ABAQUS." *Structural Engineering Journal*.
8. Le Quang Minh, Nguyen Van Vuong (2006), *Strength of Materials* (Volume 1), Education Publishing House, Hanoi.
9. Hung Vuong University (2022), Detailed Syllabus of Strength of Materials Course, Mechanical Engineering Technology Training Program.
10. Ngo Tu Thanh (2008), Simulation Methods in Teaching Engineering Disciplines, *Journal of Science and Technology Development*, Vol. 11, No. 10, pp. 114-125.