



Based on the Diversity Exploration of ABAQUS-Driven Engineering Mechanics Courses

Ping Gan*, Jing Liu

Nanchang Institute of Science & Technology, Nanchang, Jiangxi, China

*117444666@qq.com, 2461303213@qq.com

Abstract. As a theoretical technical basic discipline, engineering mechanics is closely connected with engineering technology. Its theorems, laws and conclusions are widely used in engineering technology in various industries, and they are indispensable technical basic courses for engineering majors. It not only plays a role in connecting the basic and future courses and professional courses, but also an important basis for solving practical engineering problems. By organically combining ABAQUS with engineering mechanics, students can better understand the theoretical knowledge and computational problems in the curriculum. This teaching method concretizes the abstract mechanical problems, and through post-processing analysis, enables students to have a deeper understanding and grasp of the relevant problems. This not only enhances the students' enthusiasm for learning, but also improves the interest and practicality of the course.

Keywords: Engineering mechanics; ABAQUS; finite element analysis.

1 Introduction

Engineering mechanics course is a required course for non-mechanical engineering majors such as electrical engineering, telecommunications, environment, chemical industry and big data, and it is an important basic engineering course. Learning engineering mechanics courses helps to improve students' ability to analyze and solve problems [1]. At present, the teaching methods of colleges and universities focus too much on theoretical knowledge, and basically adopt multimedia teaching methods [2]. This way mainly focuses on formula derivation, theoretical indoctrination and exercises, which makes the original abstract theory more difficult to understand, leading to students' loss of interest in the subject. In class, students often ignore thinking and only cram before the exam, and soon forget what they have learned after the exam, which has a negative impact on the following course learning. In addition, too much attention to grades and ignoring practical operation and knowledge expansion is not conducive to improving students' practical operation skills [3].

In view of these problems, this project plans to organically combine ABAQUS software with engineering machinery students, aiming to improve students' mechanical quality and application and creative ability, and promote the integration of inter

© The Author(s) 2024

M. Yu et al. (eds.), *Proceedings of the 2024 5th International Conference on Big Data and Informatization Education (ICBDIE 2024)*, Advances in Intelligent Systems Research 182,

https://doi.org/10.2991/978-94-6463-417-4_22

disciplinary disciplines [4]. Because the finite element modeling is very vivid, students can compare the finite element calculation results with the theoretical formula results [5]. In addition, because engineering mechanics is mainly rigid body calculation and small deformation calculation, it is not difficult to use finite element software to calculate, so it is easier for students to master finite element calculation software for numerical calculation after class. Through this method, students can not only better understand the basic principles and concepts, but also enhance the ability to analyze and solve problems, stimulate their interest in learning, and further improve the efficiency of classroom teaching [6]. At the same time, cultivate students to explore the spirit of physical problems, stimulate their strong desire to solve problems.

2 ABAQUS Overview of the software

ABAQUS Is one of the world's most advanced large-scale general-purpose finite element analysis software, with a rich material constitutive model and accessible development platform. It is widely used in machinery manufacturing, petrochemical, aerospace, automotive transportation, national defense, water conservancy and hydropower, civil engineering, biological engineering, electronic engineering, energy, mining, shipbuilding and household appliances industry and scientific field [7]. ABAQUS It has an excellent reputation in technology, quality and reliability, and can analyze and calculate the complex linear and nonlinear problems of engineering. Since its entry into China, more and more enterprises, universities and research institutes have begun to use ABAQUS for product research and research [8]. The analysis process includes 3 main steps:

First, the pre-processing part includes creating a model, defining material properties, applying load, dividing grids and so on.

Second, the simulation calculation uses ABAQUS / Standard or ABAQUS / Explicit to solve the numerical model defined in the input file, which usually runs in a later table mode [9].

Third, in the post-processing stage, the relevant data and images of displacement, stress, strain and so on, can be used to calculate and evaluate the calculation results, and to evaluate the visualization module of ABAQUS / CAE or other post-processing software in the graphical environment. Including color stress and strain contour map, animation demonstration, deformation map and X-Y curve map. This paper takes the bending deformation of pressing rod as an example, and introduces the role of applied ABAQUS in engineering mechanics analysis, to cultivate students' applied innovation ability and better adapt to the needs of society and enterprises.

3 ABAQUS Teaching examples

3.1 Modeling

The pressing bar structure is shown in Figure 1 and table 1, the structural material is steel, using work-shaped section steel, the material parameters are: elastic modulus E

=206 GPa, Poisson ratio $\nu=0.31$, lower yield strength $\sigma_s=296\text{MPa}$, density $\rho = 7800 \text{ kg/m}^3$, pressing bar length $l=5\text{m}$, Q235 steel $\lambda P=101$, length coefficient $\mu = 2$, pressure $F_P = 200 \text{ kN}$.

Table 1. I-beam cross-section dimension parameters

Parameter	Size (mm)
Overall height	190
Abdominal height	170
Abdominal thickness	6
Wing margin width	200
Wing edge thickness	10

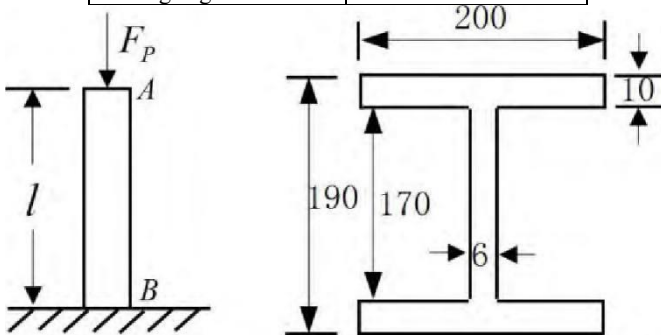


Fig. 1. Schematic diagram of pressure rod force and cross-section dimension

3.2 Apply the load

In the Create Loads dialog box, we first need to select the "force" type in the category. Then, in the Type available for the selected analysis step list, we need to select Shell. Next, switch to the graphic window and select the edge of the work-shaped cross section as the target for the applied load. Continue to open the Edit Load dialog box and enter the load size of 200000N and set the orientation perpendicular to the cross section. After completing the above steps, we set the load.

3.3 Grid division

Meshing is a crucial step in engineering analysis that involves transforming the building model into a series of discrete entities. The precision and analysis time of this transformation process are directly influenced by the quality of the meshing. In order to ensure the accuracy and efficiency of the analysis, we used 3D four-junction surface thin shell for mesh division, and sand glass control mechanism, selected finite membrane strain cell type. This division method ensures the accuracy and reliability of the analysis results, while optimizing the computational efficiency, As shown in Figure 2.

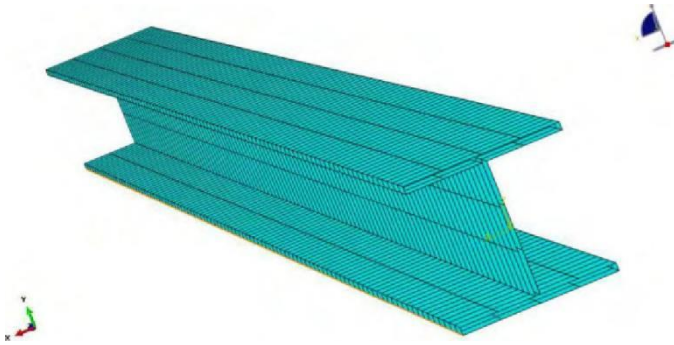


Fig. 2. Model grid division

3.4 Network inspection

To ensure the continuity of the solution process and the accuracy of the results, we must conduct a network inspection. In this process, focus on shape, size, and analytical inspection. Specific indicators include but are not limited to aspect ratio, average aspect ratio, worst aspect ratio and maximum side length, etc. The current number of explicit grid cells is 232000, and the analysis shows error 0 (0%) and warning 0 (0%). Therefore, the next step of analysis can be carried out smoothly. However, if an error or warning is found to be zero during the inspection, the grid should be redivided or local refinement adjusted.

3.5 Operation analysis

Solving the FFE model is an important engineering analysis task. In the ABAQUS software, model information can be input in the form of an INP file and then solved using ABAQUS Standard or ABAQUS Explicit. Create an INP file that contains all the instructions and data needed to build a finite element model. When writing INP files, grammar rules need to follow ABAQUS to ensure the accuracy and completeness of model information. Open the Job module in the ABAQUS software and submit the INP file for analysis and calculation. In this module, you can interactively submit jobs, monitor the analysis process, and view the analysis results. Jobs can also be debugged and optimized to ensure the stability and accuracy of the solving process. When the solution is complete, click the "Results" button to enter the visualization module. In this module, visual images of the solution by viewing and analyzing the results. Observe the deformation, stress distribution and strain, and the results are analyzed and evaluated.

3.6 Reprocessing analysis

The post-processing system provides a graphical presentation of the FEM model as well as a graphical representation of the analytical results. The user can obtain details of the model and the results from the output data. To better present the results, the

post-processing uses the form of cloud maps or X-Y charts, whose result types include, but are not limited to, deformation, stress, strain, displacement, vector / tensor, material orientation, and animation. Taking the stress cloud map as an example (Figure 3), we can clearly see that the middle part of the I-beam bears the maximum stress, and the stress gradually decreases from the middle of the pressure bar to both ends of the winding bar. In the bending deformation vector diagram (Figure 4), we can determine the deformation displacement direction of the pressing bar by observing the color and direction of the arrow. In particular, in the middle of the pressure rod, the displacement reaches a maximum, which means that an elastic buckling deformation occurs at this site.

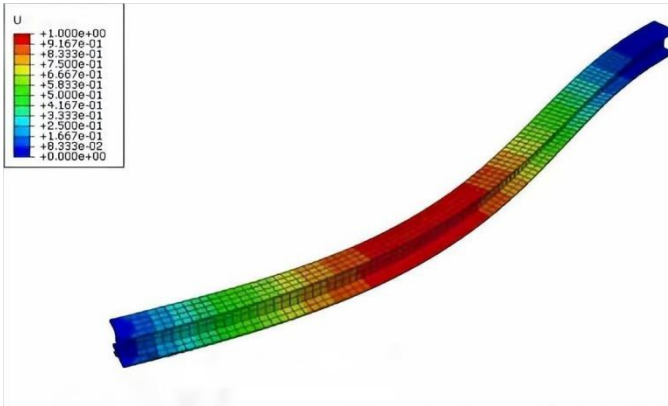


Fig. 3. Cloud diagram of pressing lever bending stress

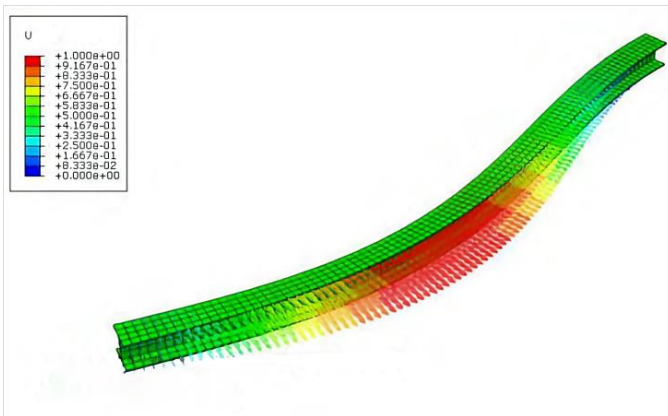


Fig. 4. Vector diagram of pressure bar bending deformation

3.7 Stress concentration problem

Plate strip with a round hole or incision, when stretched in the local area near the round hole or incision, the stress will be quickly reduced and tend to be uniform. This

phenomenon of increasing stress concentration in local areas due to shape mutation is called stress concentration. In class, students feel that the phenomenon of stress concentration is more abstract and difficult to understand. The author instructs the students to use ABAQUS software after class to calculate the drawing of the two ends of the strip with holes, and observe the change of the stress cloud map around the hole. The author instructs the students to use ABAQUS software after class to calculate the drawing of the two ends of the strip with holes, and observe the change of the stress cloud map around the hole. For example, consider a rectangular slat with 1 length 00 mm, 10mm wide and 10 mm diameter hole of 2 mm in the middle of the slat, with a uniform distribution line load $q=10000 \text{ N / m}$ applied at both ends of the slat. The elastic modulus of aluminum alloy $E = 69 \text{ GPa}$ and Poisson ratio $\nu=0.3$. Due to the calculation model structure and the applied load symmetry, one half of the model was taken for calculation, and symmetric boundary conditions were applied on the pore-containing side of the calculation model, as shown in Figure 5. Using the four-node bilinear plane strain quadrangle unit (CPE 4 R) simulation strip, when the number of grids $N=20210$, the calculation results converge. Figure 6 shows a diagram of the Mises stress cloud of the rectangular member. Through observation, the students can find that the stress on the black midline at the hole edge increases sharply; while the stress change is minimized at the side position of the member. Students can intuitively detect the stress concentration around the holes. In addition, the author also proposed that students can observe the influence range of the stress concentration phenomenon around the hole, which can also be easily learned by observing the finite element calculation cloud map. Through the calculation by themselves, students understand the phenomenon of stress concentration and the scope of influence.



Fig. 5. Model diagram of finite element calculation of hole-containing rectangular slab bars under tensile load

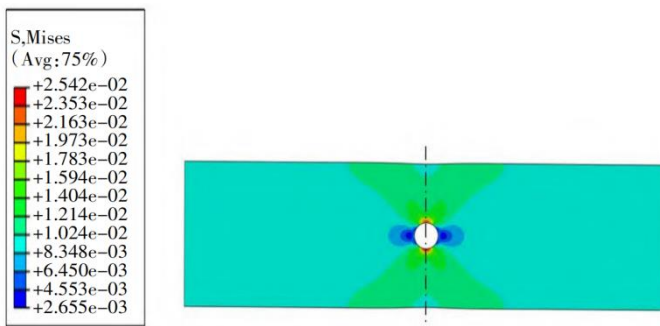


Fig. 6. Cloud diagram of the Mises stress of the component subjected to tensile uniform load

3.8 The ison of theoretical calculation and model analysis

When components or structure on several loads at the same time, if the effect of the load (binding force, internal force, stress, displacement) do not influence, the influence is negligible, then the total effect of the load alone of the effect, this is the superposition principle, is in the important method of beam bending deformation. In teaching, some students have questions about the bending deformation of the superposition method, and wonder whether the result of direct superposition is accurate. The author instructs the students to use ABAQUS software to calculate a finite element example of the bending deformation of the superposition principle, so as to intuitively understand the bending deformation of the beam by the superposition principle.

Determine the I steel cross section size, calculate the inertia distance I_Z and cross section area A .

Inertial distance:

$$I_Z = I_{Z0} + a^2 A \quad (1)$$

Flexibility:

$$\lambda = \frac{\mu l}{i} \quad (2)$$

Radius of inertia:

$$i = \sqrt{\frac{I_Z}{A}} \quad (3)$$

It is concluded that: $I_Z = 34889833 \text{mm}^4$, $A = 5020 \text{mm}^2$. Take flexibility $\lambda = 120 > 101$ (Q235 steel flexibility λ_P). So the model is a large flexibility bar, and the Euler formula is used to calculate the critical load.

Euler's formula:

$$F_{PCr} = \sigma_{cr} \cdot A = \frac{\pi^2 E}{\lambda^2} \cdot A \quad (4)$$

$F_{PCr} = 708.77 \text{kN}$, $F_P = 200 \text{kN}$, $F_{PCr} > F_P$.

Therefore, theoretical calculations show that the rod undergo elastic flexion deformation, which is consistent with the analysis results of ABAQUS model.

After class, students can discuss the rationality of transforming the concept and formula into finite element calculation examples, the finite element calculation of ABAQUS software, course knowledge points and issues related to the course at any time, which can be fully discussed between teachers, students and students. If more students are confused, the teacher can send the ABAQUS calculation model to the wechat group, and explain to the students how to transform the corresponding concept or formula into the finite element calculation example, so that the students can understand the concept or formula.

4 Analysis of teaching effect

Compared with the traditional classroom teaching, ABAQUS software can assist students in the "engineering mechanics after class" to make students intuitively understand the theoretical formulas and concepts of the course. This teaching method can make students to rely more on themselves to solve the difficult problems, can improve students' self-study ability, improve students' ability to transform theoretical formulas and concepts into practical examples, but also to improve students' practical ability. Through their own efforts, students solve the course problems and master the course knowledge, which also improves their confidence in learning. Therefore, ABAQUS software in the "engineering mechanics" after-class auxiliary teaching, combined with classroom teaching, can better make students master the course knowledge. In practice, we found that this teaching method is widely welcomed by the students. Compared with the previous students, the students who used classroom teaching combined with ABAQUS software for after-school assisted teaching significantly improved the average score of the test, thus proving the effectiveness of this teaching method.

5 Conclusions

In the course of "engineering mechanics", the author guides the students to use ABAQUS software to assist the teaching practice. The students' general feedback can better understand the concepts and formulas in the course, broaden the way to learn this course well, and improve the course examination results. It can be seen that ABAQUS finite element model simulation analysis is used to expand the visibility, practicality and interest of engineering mechanics teaching. The buckling deformation of the model is calculated through pre-processing, solution analysis and post-processing process, and the analysis results are animated to present the specific process of bending deformation of the pressing rod. To effectively enhance students' interest in learning engineering mechanics theory and improve the learning atmosphere, which is higher than the actual mechanical derivation efficiency. In addition, this teaching method can improve students' self-study ability, as well as students' practical ability. The teachers should find out the problems in and after class in time, put forward solutions to the problems and practice and explore, so as to ensure the teaching quality of the course, and provide many convenient conditions for the practical application and innovative design in the enterprise work in the later stage.

Acknowledgment

Teaching Team of Engineering Mechanics Course (JXTD-PY-04).

Reference

1. Jin Xiaoqing, Gong Shiyu, Fang Zhouyu, et al. Thinking on the teaching reform of engineering mechanics based on students' perspective. *College Education*, 2023, pp. 127-129.
2. Wang Tao. Research and practice of Improving the Teaching quality of Engineering Mechanics. *Equipment Manufacturing technology*, 2019, pp. 187-189.
3. Pan, F., Wang, Y., Hu, Y., & Huang, C. (2022, January). Application of finite element analysis software ABAQUS in civil engineering course teaching. In *2022 3rd International Conference on Education, Knowledge and Information Management (ICEKIM)* (pp. 355-358). IEEE.
4. Liu Qingsong, Yang Fubin, Sun Cultivation. Research and Practice to improve offline classroom Participation through Online Teaching- -Taking the "Engineering Mechanics" course as an example. *Equipment Manufacturing technology*, 2022, pp. 188-191.
5. Jamshidi, R., & Milanovic, I. (2020, June). Adding a Simulation Module to a Primarily Experimental Mechanical Engineering Course. In *2020 ASEE Virtual Annual Conference Content Access*.
6. Hu Zhichao. Optimization and reform practice of experimental courses of engineering mechanics. *Equipment Manufacturing Technology*, 2016, pp. 257-258, 261.
7. Feng Wenjie, Li Haiyan. Application of numerical simulation software in the teaching of vibration mechanics. *Mechanics and Practice*, 2021, vol. 43, pp. 632-639.
8. Cress, J., & Thomas, P. (2023, October). Methods for Integrating Visual Communication in an Applied Finite Elements Modeling Course. In *Proceedings of the 41st ACM International Conference on Design of Communication* (pp. 210-214).
9. Lv Chang. The "Student-centered" teaching paradigm reform practice- -Take "Engineering Mechanics and CAE" as an example. *Equipment Manufacturing technology*, 2022, pp. 188-191.

Open Access This chapter is licensed under the terms of the Creative Commons Attribution-NonCommercial 4.0 International License (<http://creativecommons.org/licenses/by-nc/4.0/>), which permits any noncommercial use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license and indicate if changes were made.

The images or other third party material in this chapter are included in the chapter's Creative Commons license, unless indicated otherwise in a credit line to the material. If material is not included in the chapter's Creative Commons license and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder.

