Modeling and Simulation of Tensile Experiment and Teaching Application in Higher Vocational Colleges

Meiqin Liang[1], Xiaozhan Li[2], Shang Wang[3*]

[1]*College of Automation Engineering, Beijing Polytechnic, Beijing, China*
[2]*Institute of Engineering Technology, University of Science and Technology Beijing, Beijing, China*
[3]*School of Automotive Engineering, Beijing Polytechnic, Beijing, China*
First author: liangmeiqin@bpi.edu.cn
**Corresponding author: wangshang@bpi.edu.cn*

Abstract

The problem of poor teaching effect of tensile experiment in Material Mechanics course in higher vocational colleges has existed for a long time. In order to solve it, the research team put forward a method of using modeling and simulation to obtain materials such as pictures, animations, so as to improve the teaching effect. A 3D finite element model for tensile experiment was established and conducted calculations on a computer. The simulated animations visually showed the necking and cracking behavior of the specimen under tension, and the Mises stress nephograms clearly showed the stress distribution of the specimen at different time points. In addition, a large amount of data has been calculated. The Mises stress nephograms and animations were used as course materials of Material Mechanics. The beneficial effects of simulation materials are mainly reflected in: (1) the unfamiliar concepts, such as neck contraction and fracture, are easier to be understood by students, and the teaching quality is improved; (2) it stimulates students' interest in learning; (3) the model simulation and demonstration are done on a computer, which not only has no danger, but also saves the experimental cost. The research idea should be applied in more courses of higher vocational education.

Keywords: *tensile experiment, teaching application, simulation, finite element, modeling.*

1. INTRODUCTION

Material Mechanics is not only a required course for engineering majors such as machinery, automobile, and rail transit but also an important basic course of many specialties. The learning effect of Material Mechanics will directly affect follow-up related professional courses, such as Mechanical Design Principle, Mechanical Manufacturing Engineering and Material Processing Engineering [6][11].

At present, the contents of Material Mechanics textbooks in domestic higher vocational colleges are mostly based on that of undergraduate colleges. Because course content is numerous and difficult, and many theorems and formulas need complex mathematical derivation processes, students generally feel great academic pressure and have no interest in learning [4][7].

Experimental teaching is an important way to further study and consolidate classroom content. Many concepts of Material Mechanics, such as necking, stress concentration and moment of inertia, are esoteric and abstract, and need to be understood and consolidated with the help of experimental teaching. The learning ability of vocational college students is worse than that of undergraduate students. Therefore, experimental courses are more important to students in vocational colleges. However, the problem of poor teaching quality of Material Mechanics experiment course in vocational colleges has existed for a long time [8].

For instance, the equipment in many mechanics laboratories is old and small, which is difficult to meet the teaching needs. Many vocational colleges have only one experimental equipment. Only one teacher operated and dozens of students observed around the experimental equipment. Only three or four students in the front row can see the whole experimental process clearly. Even if the school has multiple devices, teachers in most cases do not let students use them for safety reasons [6].

Tensile experiment is the first and very important experiment of Material Mechanics. The experimental teaching quality not only affects the follow-up teaching

A. El-Hashash et al. (Eds.): IEIT 2022, ACSR 100, pp. 316–323, 2023.
https://doi.org/10.2991/978-94-6463-058-9_53

content, but also affects students' interest and initiative in learning.

In previous stages, scholars [2][9][10] used finite element modeling and simulation to solve some tensile and necking problems in practical engineering, but there was little research on modeling and simulation of tensile experiment teaching.

In order to improve the teaching quality of the experiment course and improve students' interest in learning, the research team established a 3D finite element model. The model conducted calculation on a computer, and a lot of Mises stress nephograms and demonstration animations were obtained. These digital materials were used in the classroom of Material Mechanics and achieved good teaching results.

The research logic of this paper is shown in Figure 1.

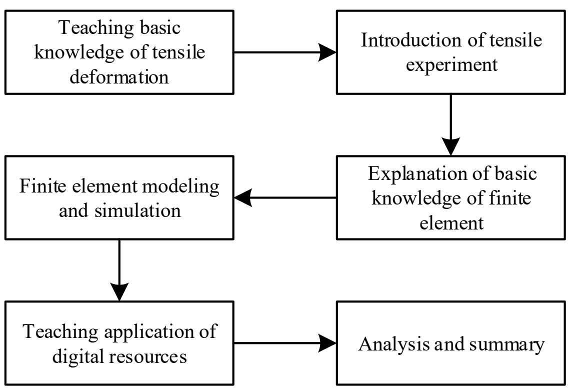


Figure 1: The research logic diagram of the paper. (figure credit: original)

2. INTRODUCTION OF TENSILE TEST EQUIPMENT

A tensile experiment apparatus is as shown in Figure 2. A tensile sample in the middle is clamped by upper and lower clamping devices. The lower clamping device is fixed. A sensor is provided for the lower clamping device for measuring pulling forces on the sample at different times. Under the action of a motor or a hydraulic device, the upper clamping device can move slowly upward at a set speed, so as to exert upward pulling force on the sample.

The tensile sample is shown in Figure 3. The middle of the sample is an observation area of tensile deformation (test part). The two ends of the sample are areas for clamping.

When movement speed v of the clamping device is a fixed value, strain of the sample at any time is:

$$\varepsilon = vt/L \tag{1}$$

The diameter of the sample is measured, and the cross-sectional area of the sample can be calculated. The pulling force at any time is recorded by the sensor. A stress at any time is:

$$\sigma = 4F/\pi D^2 \tag{2}$$

The strain (ε) and stress (σ) data at different times are calculated according to the above two equations to obtain a strain/stress curve relationship of the tensile sample.

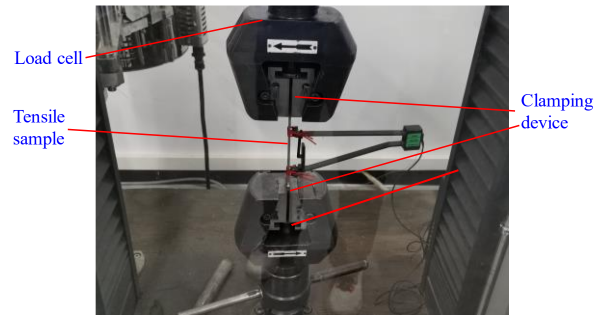


Figure 2: The tensile experiment and the tensile sample. (photo credit: original)

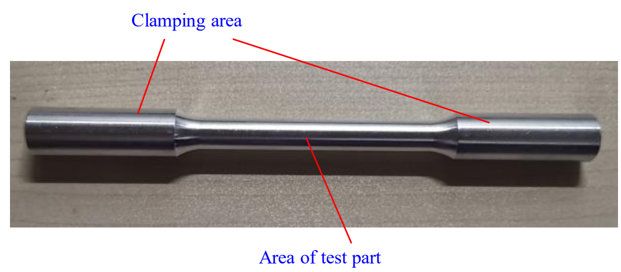


Figure 3: The structure of the tensile samples. (photo credit: original)

In traditional experiments, strain (ε) and stress (σ) cannot be seen intuitively. The application of finite element modeling and simulation can get rich animation and cloud images. As a supplement to classroom teaching, these materials can help students understand the relevant knowledge.

Some basic theories of finite element were distributed to students in the form of after-class reading materials. It would lay a foundation for the application of simulation materials.

3. INTRODUCTION OF FINITE ELEMENT THEORY

The finite element theory is to simulate a real system (geometry and load conditions) by using the method of mathematical approximation. The finite element model discretizes the continuous structure into finite elements, and sets finite nodes in each element. In this way, the complex continuum is treated as a collection of elements connected only at the nodes. Based on the variational principle of mechanics and related formulas, the equations for solving node unknowns are established. Using the solving process of equations on the computer, the problem of infinite degrees of freedom in continuous domain is actually transformed into the problem of finite degrees of freedom in discrete domain [3][5].

The equilibrium equation of finite element theory is:

$$\begin{cases}\frac{\partial \sigma_x}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+X=0\\ \frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \sigma_y}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+Y=0\\ \frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \sigma_z}{\partial z}+Z=0\end{cases} \tag{3}$$

The geometric equation of finite element theory is:

$$\begin{cases}\varepsilon_x=\frac{\partial u}{\partial x}\\ \varepsilon_y=\frac{\partial v}{\partial y}\\ \varepsilon_z=\frac{\partial w}{\partial z}\end{cases} \tag{4}$$

And:

$$\begin{cases}\gamma_{xy}=\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\\ \gamma_{yz}=\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\\ \gamma_{zx}=\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\end{cases} \tag{5}$$

The constitutive equation of finite element theory is:

$$\begin{cases}\sigma_x=\lambda e+2G\varepsilon_x\\ \sigma_y=\lambda e+2G\varepsilon_y\\ \sigma_z=\lambda e+2G\varepsilon_z\end{cases} \tag{6}$$

And:

$$\begin{cases}\tau_{xy}=G\gamma_{xy}\\ \tau_{yz}=G\gamma_{yz}\\ \tau_{zx}=G\gamma_{zx}\end{cases} \tag{7}$$

The boundary condition equation of finite element theory is:

$$\begin{cases}l\sigma_x+m\tau_{yx}+n\tau_{zx}=\overline{X}\\ l\tau_{xy}+m\sigma_y+n\tau_{zy}=\overline{Y}\\ l\tau_{xz}+m\tau_{yz}+n\sigma_z=\overline{Z}\end{cases} \tag{8}$$

Thanks to the rapid development of computer technology and increasingly intelligent and integrated finite element software in recent decades, the establishment of model and the operation of the above equations became feasible.

ABAQUS is a widely recognized and powerful finite element analysis software. It can analyze complex mechanical problems, especially can simulate highly nonlinear complex problems. It can not only analyze the mechanics and multiple physical fields of a single part, but also do system-level research. Moreover, ABAQUS is widely used by enterprises and research institutes in various countries because of its excellent analytical ability and reliability in simulating complex systems [1][5].

The flow of ABAQUS modeling and calculation is shown in Figure 4.

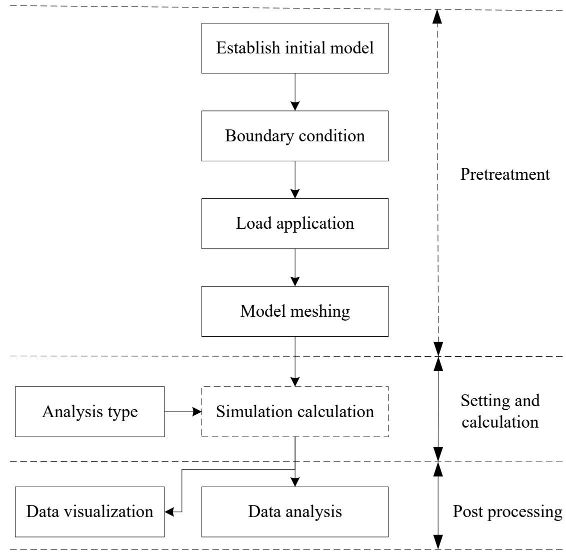


Figure 4: Flow chart of modeling and calculation. (figure credit: original)

4. FINITE ELEMENT MODELING

4.1. ABAQUS Modeling

As stated earlier, the sample includes the test part in the middle and the clamping part at both ends. The test part is used to measure the stress/strain of the sample; the clamping part is used for clamping and applying tensile force to the sample by an experimental device. In order to reduce unnecessary calculations, only the test part is considered in finite element modeling.

The test part is a cylinder with a diameter of 10 mm and a length of 50 mm. The axis of the test part is the X-axis, and the geometric center is located at the coordinate origin. The clamping part is a cylinder. The axis of the clamping part is the X-axis, with a chamfer on one side. The chamfer is 45°, the length along the X-axis is 2 mm, and the other side of the chamfer is connected with the test part. The diameter of the clamping part is 14 mm and the total length is 20 mm, as shown in Figure 5.

Setting of material property of the sample: density of the material ρ=7.9×10^3 kg/m^3, elastic modulus E=210 GPa, and Poisson's ratio λ=0.3. In order to better simulate the necking and fracture of the sample, set fracture strain value ε_0=1.4 and failure display value x_0=0.04 mm in ABAQUS software.

The element type of finite element model is chosen as C3D8R. To improve the calculation speed and final simulation accuracy of the finite element model, local mesh of the model is refined, as shown in Figure 5. Among them, the minimum grid size is set to 1 mm, the smaller grid size 2 mm, and the maximum grid size 4 mm.

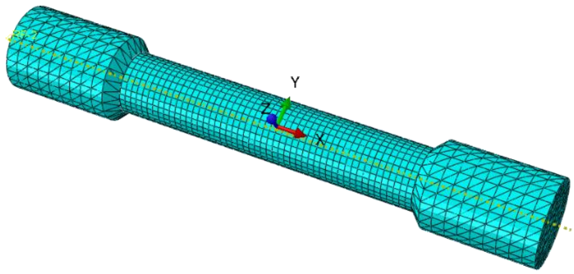


Figure 5: Finite element model after meshing. (figure credit: original)

The constraint of finite element is set as six degrees of freedom full constraint on the leftmost face of the model (x=0 face). Apply velocity to the rightmost face of the model (x=100 mm face).

The direction of speed is X-axis positive direction, and the size of speed v=0.05 mm/s. The time of the "Step" is set to t=20 s, and the "H-output" is set to T=20 times.

5. SIMULATION CALCULATION AND ANALYSIS

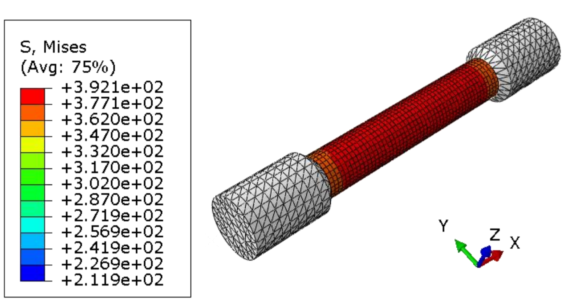
The model is simulated on a computer, and Mises stress nephograms of a finite element model at different times are obtained.

5.1. Necking and Cracking

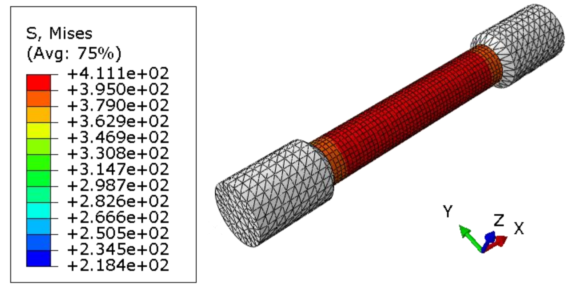
The variation of stress distribution, necking and cracking at different time were studied.

Figure 6 are results of an initial stage of model simulation. Figure 6(a) is a Mises stress nephogram when t is 1 s, at which time the maximum Mises stress is 392.1 MPa. Figure 6(a) is a Mises stress nephogram when t is 2 s, at which time the maximum Mises stress is 411.1 MPa.

As can be seen from Figure 6(a) and Figure 6(b), stress distribution of the tensile sample is very uniform at the initial stage of simulation.

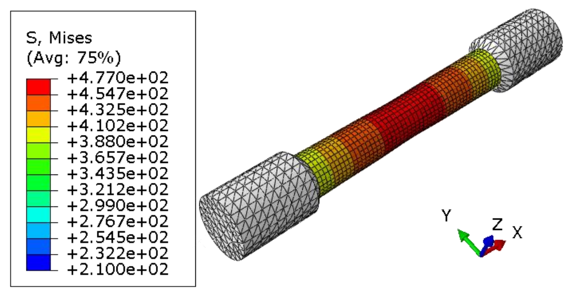


(a)

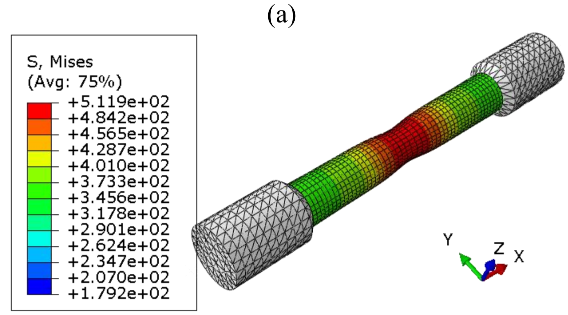


(b)

Figure 6: Mises stress nephograms at different times: (a) t=1 s; (b) t=2 s. (figure credit: original)



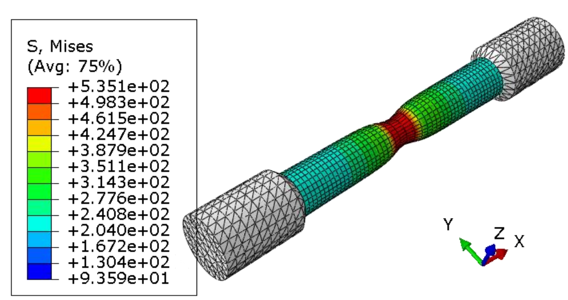
(a)



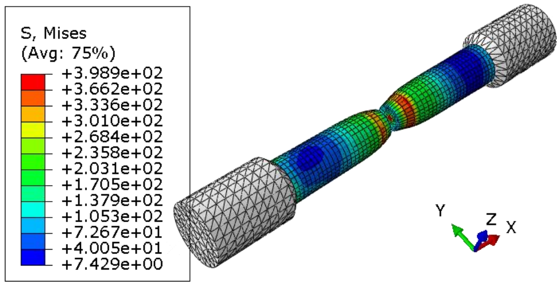
(b)

Figure 7: Mises stress nephograms at different times: (a) t=9 s; (b) t=13 s. (figure credit: original)

Figure 7 are results of a middle stage of model simulation. Figure 7(a) is a Mises stress nephogram when t is 9 s, at which time the maximum Mises stress is 477.0 MPa. Figure 7(b) is a Mises stress nephogram when t is 13 s, at which time the maximum Mises stress is 511.9 MPa. Comparing Figure 6 with Figure 7, it can be seen that stress distribution of the sample in the middle stage of simulation is severely non-uniform and a significant necked area begins to appear. Stress concentration occurs in the necked area, and the area farther away from the necked area has less stress.



(a)



(b)

Figure 8: Mises stress nephograms at different times: (a) *t*=17 s; (b) *t*=18 s. (figure credit: original)

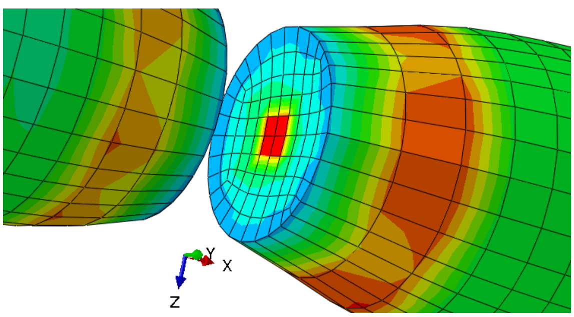


Figure 9: Morphology observation of tensile break surface. (figure credit: original)

Figure 8 are results of a final stage of model simulation. Figure 8(a) is a Mises stress nephogram when *t* is 17 s, at which time the maximum Mises stress is 535.1 MPa. It is apparent that the necked area has been very significant when *t* is 17 s, and the diameter of the necked area is only half the diameter of the sample. Figure 8(b) is a Mises stress nephogram when *t* is 18 s, at which time the sample is snapped. Since the sample is snapped, the stress within the sample is released, at which time the maximum Mises stress of the finite element model is directly reduced to 398.9 MPa. Furthermore, the minimum stress of the model is only 7.249 MPa, which is close to zero.

5.2. Analysis of Fracture Section

As shown in Figure 9, the cross section of the necked area at the end of model simulation (*t*=20 s) is observed.

As can be seen in Figure 9, the cross-section appears to be a "tunnel shape". The cross section is high in periphery, low in center, and is centrosymmetric overall. Stress distribution of the cross section is non-uniform. The stress is large in center, small in periphery, and is centrosymmetric overall. Compared with the initial state of the model of Figure 5, it can be seen that after the sample is snapped, a part at the cross section is substantially elongated in the X-axis direction (axial direction).

5.3. Analysis of Maximum Mises Stress

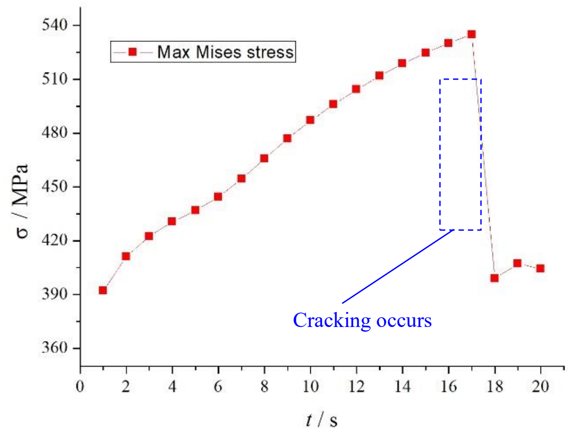


Figure 10: Maximum Mises stress statistics at different times. (figure credit: original)

The maximum Mises stress of the model at different times is calculated, and statistical results are as shown in Figure 10. As can be seen from Figure 10, the stress of the sample at an initial stage of tension increases dramatically, and the maximum stress of the sample when *t* is 1 s has reached 392.1 MPa. Before fracture, the maximum Mises stress of the sample progressively increases over time. Overall, the maximum stress of the sample does not vary greatly after *t*=1 s. Since the stress is already large, the sample is continuously plastically deformed, attenuating the increase of the stress. Between *t*=17 s and *t*=18 s, the sample is fractured (as shown in Figure 8), and the maximum Mises stress is rapidly reduced from a maximum of 535.1 MPa to 398.9 MPa. By the end of simulation, the maximum Mises stress is increased (407.1 MPa) and then decreased (404.5 MPa), but does not vary greatly overall. The maximum Mises stress of the model varies since a small scale of metal flows to occur to the fracture after the sample is fractured.

5.4. Results

As mentioned above, the research team established the finite element model of tensile experiment, and obtained rich Mises stress nephogram and a large number of experimental data through simulation calculation. Mises stress nephograms clearly and intuitively show the stress distribution (the redder the color, the greater the stress) of the sample under different tensile forces. The experimental data can be called directly from ODB. These materials and data provide convenience for the teaching reform of Material Mechanics.

6. TEACHING PRACTICE AND ANALYSIS

As is known to all, students in vocational colleges are poor in learning and understanding. Because of abstract concepts and many knowledge points, the teaching quality of Material Mechanics is not good for a long time. The front-line teachers of Beijing Polytechnic have carried out teaching and discussion of Material Mechanics for many times. In the discussion, all the teachers said that the teaching of stretching experiment was very difficult. It takes a lot of time for students to thoroughly master the concepts of stress concentration, necking, and fracture and so on. Pictures, animations and other materials that can intuitively show stress/strain are urgently needed for the courses.

The finite element model used for tensile experiment was established by this paper, and simulation calculation was carried out on a computer, and thereby a large number of pictures, animations and data were obtained. These digital resources have been applied in teaching practice of Material Mechanics. The teaching practice shows that the animations intuitively show the behavior of specimen neck contraction and fracture, and it is easy for students to understand. Teachers' explanations are more relaxed, and teachers have more communication with students with the help of animation.

A questionnaire survey was conducted among students and teachers, among which there are 67 students. The questions involved in the questionnaire are as follows:

Q1: have the materials obtained by simulation enhanced your interest in learning?

Q2: have the materials obtained by simulation more clearly shown necking and fracture of the sample?

Q3: have the materials obtained by simulation helped you grasp knowledge points more solidly?

Q4: have the material obtained by simulation helped you with your test scores?

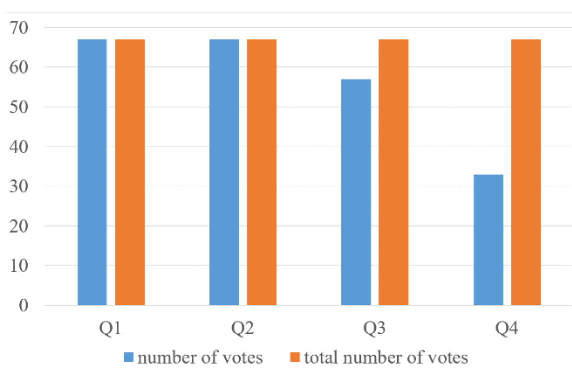


Figure 11: Statistics of questionnaire survey. (figure credit: original)

The statistical results of the questionnaire are shown in Figure 11. As can be seen from Figure 11, all students believe that the application of materials obtained by simulation improves their interest (67 votes). And all students believe that the application of these materials clearly shows necking and fracture of the sample (67 votes). However, the number of students that voted for Q3 was 57, accounting for 85.1%. Q4 received only 33 votes, accounting for 49.3%.

In order to further improve teaching quality, the research team communicated with the students. Some students who are against Q4 believe that Material Mechanics is very difficult to learn. There is no denying that these digital materials can help them more intuitively understand the laws of necking, fracture and stress change. However, these had little effect on test scores. Learning and mastering of this part still requires a lot of time, and need to be completed through traditional learning modes such as reading notes and doing homework. The students' views were endorsed by the research team. A lot of practice shows that the tensile experiment is a difficult teaching point of Material Mechanics in vocational colleges. The solid mastery of a strange, complex and abstract knowledge mainly depends on the traditional teaching process, including pre-class preview, listening carefully in class, taking notes, doing exercises and summarizing. This process is not only a scientific process of learning and memory for human brain, but also a mainstream teaching mode in higher vocational colleges.

The classroom teaching and after-class self-study of Material Mechanics are very boring, so it is easy to distract students. The application of the digital materials obtained by finite element simulation intuitively shows the necking that is difficult to be clearly observed in ordinary classes, and obviously improves students' interest in learning. From the perspective of safety and cost, finite element technology is also worth popularizing. The simulation is completed on a computer, which not only has no danger, but also saves the experimental cost.

There is another thing worth mentioning. One student is particularly interested in simulation technology, and he plans to study ABAQUS software seriously after he graduates and enters the undergraduate program. There is no doubt that these are the words all teachers want to hear. This also fully shows that the teaching application of finite element simulation technology in this paper is successful. The research idea of this paper should be applied for more courses of higher vocational colleges.

7. CONCLUSIONS

The research team put forward a method of using finite element modeling and simulation calculation to obtain materials such as pictures, animations, so as to improve teaching quality. The digital material obtained from finite element simulation has been applied to teaching practice and achieved good teaching results.

Firstly, the model simulation obtains clear demonstration animations, which can repeatedly show necking and fracture of the sample under external load. These digital materials make it easier for students to understand abstract and obscure concepts.

Secondly, the Mises stress data at different moments are obtained by simulation and displayed in different color forms (the redder the color, the greater the stress). These digital materials obviously enhance students' interest in learning.

Thirdly, the model simulation is done on a computer, which reduces the experimental cost and does not cause any danger.

The progress of technology in various fields has greatly promoted the construction of vocational education curriculum resources. Teachers in higher vocational colleges should constantly learn new technologies and strive to improve the quality of teaching. The research idea of this paper should be applied in more courses of vocational education.

ACKNOWLEDGEMENTS

The research of this paper is supported by the Project of Beijing Office for Education Sciences Planning (Grant No. CCDB2020135 and No. CGDB21208), and the Project of China Vocational Education Association (Grant No. ZJS2022YB024).

REFERENCES

[1] Barbero E J. (2013). Finite element analysis of composite materials using AbaqusTM[M]. Boca Raton: CRC press.

[2] Du X, Jin L, Ma G. (2014). Numerical modeling tensile failure behavior of concrete at mesoscale using extended finite element method. J. International Journal of Damage Mechanics, 23(7): 872-898.

[3] Khennane A. (2013). Introduction to finite element analysis using MATLAB® and abaqus[M]. CRC Press.

[4] Liu J. (2012). Exploration on teaching reform and innovation of material mechanics[C]//Advanced Materials Research. Trans Tech Publications Ltd, 591: 2208-2211.

[5] Rao S S. (2017). The finite element method in engineering[M]. Butterworth-heinemann.

[6] Wang S. (2022). Teaching research of material mechanics aimed at stimulating vocational college students' interest based on simulation technology. J. Scientific and Social Research, 4(3): 77-83.

[7] Wang S, Ma J, Liu H, et al. (2021). Exploration on teaching practice of material mechanics course for the run-through external cultivation in vocational college[C]//2021 4th International Conference on Humanities Education and Social Sciences (ICHESS 2021). Atlantis Press: 1406-1410.

[8] Wang T, Luo H F, Wang J, et al. (2016). Exploration and practice of material mechanics teaching reform. J. Experimental Science and Technology, 14(3): 116-118.

[9] Wisselink H H, van den Boogaard A H. Finite element simulation of the stretch-forming of aircraft skins[C]//AIP Conference Proceedings. American Institute of Physics, 2005, 778(1): 60-65.

[10] Ye J, Yan Y, Li J, et al. (2018). 3D explicit finite element analysis of tensile failure behavior in adhesive-bonded composite single-lap joints. J. Composite Structures, 201: 261-275.

[11] Zhang J W, Zhu M J, Zhang L W. (2011). Teaching research and practice on material mechanics with the core of practical cases. J. Procedia Engineering, 15: 4256-4260.