Study on breaking mechanism of chestnut shell based on finite element analysis

Xueyin Yang 1,a, Yanxing Qi 2,b
1 School of Mechanical and Vehicle Engineering Linyi University Shandong, China
2 School of Automation and Electrical engineering Linyi University Shandong, China
a yangxueyin@lyu.edu.cn, b qiyanxing@126.com

Abstract. Chestnut shell breaking is an important part of chestnut processing. According to the measured physical parameters and geometric dimensions, the solid model of chestnut was established by ANSYS software, and the chestnut was divided into appropriate element grids. The stress and strain of chestnut shell breaking were analyzed along different loading directions. The results show that the y-direction loading is the most labor-saving, and the x-direction loading is the best, so as to improve the quality of chestnut shell breaking It provides a theoretical basis for the further processing of chestnut.

Keywords: Chestnut; mechanical properties; loading direction; finite element analysis.

1. Introduction
In this paper, based on the experimental study on the mechanical characteristics of chestnut shell breaking, the effect and force of chestnut shell breaking are analyzed by using ANSYS simulation software when the force is applied at different positions, and the best position of the force is obtained, which provides a theoretical basis for the development of chestnut shell breaking equipment.

2. Modeling
2.1. Physical parameters and failure criteria of Chestnut
The elastic modulus of Castanea mollissima is determined by experiment, and the shell is made of orthogonal and heterogeneous materials. The specific values are shown in Table 1.

| Tab.1 Elastic modulus of fruit shell |
|------------------------------------|
| Direction | elastic modulus/MPa | shear odulus /MPa | Poisson's ratio |
| X         | 900                 | 740               | 0.30           |
| Y         | 680                 | 690               | 0.30           |
| Z         | 1200                | 820               | 0.29           |

The elastic modulus of chestnut kernel is 120 and Poisson's ratio is 0.32.

*Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.*

Published under licence by IOP Publishing Ltd
The pericarp is mainly composed of cellulose and hemicellulose. The deformation of cellulose and hemicellulose is very small under tension until it is destroyed, so they are regarded as brittle materials, and the failure mode is usually fracture.

2.2. Geometric dimension and solid model of chestnut

The shape of chestnut is very irregular, which can be approximately divided into spherical shape, hemispherical shape and flat shape. According to the experimental analysis, the shape of chestnut has no significant effect on the breaking force, so this paper mainly takes the hemispherical shape as an example. Take 100 samples of chestnut with the same moisture content and the same shape. According to figure 1, the average dimension of chestnut along X, y and Z directions is 30.12mm; the average dimension of Y direction is 30.03mm; the average dimension of Z direction is 18.8mm; the average thickness of shell is about 0.8mm; the average gap is about 0.5mm. The shell of chestnut is close to the quadric surface, the top and bottom of chestnut are not smooth transition, there is stress concentration. The solid model of Castanea mollissima in Figure 2 and the solid model of Castanea mollissima along the symmetry plane in Figure 3 were established.

2.3. Unit selection

Because the surface of chestnut is a curved surface, when selecting the unit type, the unit type of shell and kernel is set as solid95 [3].

2.4. Meshing

First, select the entity to be meshed, and specify the material and cell type of the selected entity. The format of the command is as follows: AATT, material number, real constant, cell number, cell coordinate system

Secondly, set the grid size, command format is: esize, cell grid side length, cell number.

Finally, the grid is divided, and the results are shown in Fig. 4 (a) and (b).
2.5. Element constraints
The chestnut body is symmetrical with respect to the face YZ, so \( \frac{1}{2} \) of the chestnut is taken as the analysis object, so the symmetrical boundary constraint is applied on the face.

YZ, the selected direction is taken as the constraint direction along which direction to load, the area constraint along the opposite direction of Z is applied on the bottom surface of the chestnut, and the concentrated load along the Z direction is applied on the middle surface. See Figure 5.

3. Finite element simulation of loading along different directions

3.1. The load is applied along the X direction
When the load is applied along X, the applied area load is 500N, as shown in Figure 6.

The analysis results are as follows: the shell is analyzed by finite element method, and the equivalent force is shown in Figure 7. It can be seen from the figure that the stress of the tip part and the top of chestnut is the largest at the concentrated stress loading area, followed by the stress near the concentrated stress loading area, and the maximum stress of chestnut shell breaking is 431.604N. The area of stress is
Related to the range of applied area load. The larger stress is mainly distributed in two positions: 1) near the YZ plane of the shell. 2) The applied area is around the load area. In these two parts, chestnut shell is easy to crack. The main reasons are as follows: in the first case, the Y direction is along the texture direction of chestnut shell material, which is easy to split the chestnut shell; in the second case, when the chestnut is loaded along the X direction, it is easy to produce concentrated force in the middle of the shell; in the second case, the shell is compressed by the indenter, which makes the contact part deform and crack. The results show that the cracking condition of finite element analysis is consistent with the test results [4].

It can be seen from the total strain figure 8 of the shell that the concentrated load is produced in the middle of the shell due to the pressure along the X direction on the shell, so cracks are generated along the Y direction near the YZ surface of the shell. The maximum total strain is the sharp part of chestnut, so the structure is special and easy to crack. The maximum deformation is 1.81mm.

![Fig.8 Equivalent total strain Fig.9. Load along Y direction](image-url)

3.2. Apply the load along the Y direction

The constraint and loading along the Y direction of chestnut are shown in Figure 9. The symmetry plane constraint is applied to YZ plane, and the full constraint is applied to the area part of the tip at the bottom. The area load is applied to the top along the Y direction, and the pressure value is 500N. The results are as follows: the shell is analyzed by finite element method, and the distribution of equivalent force is shown in Figure 10. The maximum equivalent stress is 343.549N. The maximum stress is in the top of chestnut, and the larger stress is mainly distributed in the edge of chestnut shell, the top of chestnut and the sharp part of chestnut. The constrained part of chestnut, that is, the sharp mouth part of chestnut, also has a force distribution of about 200N. The main reason why the beak part is easy to deform is that the stress area is relatively small. When the force is applied in the Y direction, the beak part will deform, which is conducive to the shelling of chestnut.
When the chestnut shell is subjected to the force along the Y direction, the maximum deformation occurs at the position of the maximum concentrated force. The strain law basically diffuses around the position of the loading force. The maximum value is 1.33mm, the strain change at the top extends from the stress concentration point in a strip shape to the surrounding of the shell, and the strain change at the bottom mainly occurs in the sharp mouth part of the shell. When the deformation of the shell increases to a certain extent, the deformation can not be restored, then the shell cracks in the most deformed part. With the further deepening of the deformation, the cracks gradually expand, and finally most of the shell cracks. The equivalent strain is shown in Figure 11, which mainly occurs at the top, bottom, symmetry plane and edge of the shell. These parts are the stress concentration points and are easy to crack.

3.3. Apply the load along the Z direction

![Fig.12 Load along Z direction.](image_url)  ![Fig.13 Equivalent stress nephogram](image_url)
When simulating along z direction, the constraint parts of chestnut are as follows: ① symmetry plane YZ of chestnut, apply symmetry plane constraint to it, as shown in Figure 12, s part in the figure is the symbol of symmetry plane constraint. ② The plane part of the shell along the Z direction, that is, the XY plane, is applied with a pressure load of 500N on the symmetry plane along the Z direction. There is a reaction force along the Z direction on the XY plane.

When the shell is loaded along the Z direction, the equivalent stress of the finite element analysis is shown in Figure 13. In the figure, the maximum stress is 748.917N when the shell is loaded along the edge line. The stress value in the area close to the sideline is the second, and the stress distribution area is like an ellipse. The stress direction is mainly extended to the top and bottom ends of the shell. The whole stress region has a clear direction. The maximum displacement in Z direction is 0.94mm at the loading position of concentrated force, and the distribution of displacement decreases in Z direction at the loading position of concentrated force; the XY plane of fruit shell is affected by the reaction force in Z direction during the loading process, so the displacement also changes.

The total strain of chestnut is shown in Figure 14. The maximum total strain is 0.4897mm in the concentrated force distribution, and the light blue part is shown in the figure. Because of the concentrated force, the strain and strain are the maximum, resulting in pits in the shell, and then cracks in the shell, and the size distribution of strain gradually decreases along the Z direction. At the same time, the XY plane also produces phase change. The strain of this plane is about 0.2177mm.

3.4. Analysis of loading results along different directions

In the finite element simulation test, the loads applied along the three different directions are all area loads with the size of 500N, and the constraint situation is the constraint position of the fixture to the shell during the simulation test [5]. It can be concluded from the ANSYS analysis that the minimum shell breaking is required when loading along the Y direction, it is not easy to constrain the shell, and the crack direction of the shell is not very obvious, so it is not desirable. When loading along the X direction, the stress value is moderate, and it is easy to impose area constraints on the shell. When the shell is broken, the crack has obvious direction, and the range of stress and strain is relatively large, so the shelling effect is the best, which is consistent with the conclusion of the experimental analysis. Although the shell breaking force is the largest when loading along z direction, the displacement and strain of the shell are very small. After analysis and comparison, the shelling effect is the best when loading along X direction.

4. Conclusion - Comparison [4] between compression test and finite element analysis

When the load is applied, the nonlinear analysis is adopted because the contact analysis is between the shell and the kernel. First, set the loading time and loading step, the specific setting is: nsubst, 6,90,1, that is, the loading time of chestnut is 90s, and the loading step of each load is 6S. Take the y-direction as an example, the relationship curve between the stress and the convergence value in the 90th second after loading is shown in Figure 15, where crit is the force on the shell in the actual loading, and L2 is
the value automatically assigned to the shell by the finite element method. It can be seen from the figure that the value assigned to the shell by the finite element method and the trend of the stress and strain during the shell test are based on the convergence within a certain period of time. This is the same.

**Fig.15** Time stress diagram along Y direction

**Tab.2** Comparison of test value and finite element analysis value

| Loading direction | comparison | test value | ANSYS value | ratio |
|-------------------|------------|------------|-------------|-------|
| X                 | F/N        | 610.99     | 431.60      | 1.42  |
|                   | S/mm       | 1.97       | 1.81        | 1.09  |
| Y                 | F/N        | 590.64     | 343.55      | 1.72  |
|                   | S/mm       | 2.51       | 1.33        | 1.89  |
| Z                 | F/N        | 761.95     | 748.92      | 1.02  |
|                   | S/mm       | 1.04       | 0.94        | 1.11  |

In the compression test, the comparison of shell breaking force and deformation under three different loading directions is shown in Table 2. From the analysis of the ratio after comparison, (1) the difference of displacement is the biggest when loading along the Y direction. The reason is that the sharp part of the shell is actually softer than other parts of the shell, which is easy to deform. In the finite element analysis, it is assumed that the material of the whole shell is the same, so there is a certain gap; 2) when loading along the Y direction, it is not easy to position the chestnut, and it is difficult to apply the load. It is easy to cause the chestnut to slide, so the displacement value is increased in the actual test. In fact, the gap between the shell and the kernel is not the same everywhere in the experiment, and the gap between the shell and the kernel is different with different moisture content; in ANSYS modeling, the equal gap distribution is assumed between the shell and the kernel, which is also the reason for the larger ratio. (2) The ratio of reanalysis force value and test value are larger than the results of ANSYS analysis, and the results of Z direction are the most similar, basically consistent. The second is the ratio in X direction, and the worst is in Y direction. The reason is mainly related to the material characteristics of the shell. When loading along the Y direction, the shell at the top of the shell is softer than that at the tip, and the shell is thinner. The part contacting with the mechanical sensor is the top, so the value of the shell breaking force measured at the top is larger than that analyzed by ANSYS. On the whole, the results of finite element analysis and experimental analysis are almost the same, so when studying the mechanical characteristics of chestnut shelling, we can use the finite element virtual simulation analysis without the restriction of external conditions to provide a theoretical basis for chestnut shelling.
References

[1] Yang Xueyin, Qi Yanxing. Preliminary study on mechanical shelling technology of Chinese chestnut. Agricultural Equipment and Vehicle Technology, 2006 (5): 42-43

[2] Yang Xueyin. Study on chestnut shell breaking mechanism based on finite element method. Master Dissertation of Huazhong Agricultural University, 2005.5

[3] YANG X Y, QI Y X. Factors affecting the cracking force of chestnut. College of Food & Biotechnology, 2007 (5): 56-60. (in Chinese with English abstract)

[4] Yang Xueyin, Qi Yanxing. Study on mechanical properties of chestnut shell. Journal of Zhengzhou Institute of Light Industry, 2006(8):61-64

[5] Xue Yin. Yang. The Study on the Drying Characteristic of Chinese Chestnut[J], Advanced Materials Research, 2013, 781-784:1474-1478