Finite Element Analysis for the Lifting Platform of a Self-Propelled Multi-Functional Machine Used in Orchard

Bai Zongchun¹², Lv Xiaolan¹ and Xia Liru¹

¹ Institute of Agricultural Facilities and Equipment, Jiangsu Academy of Agricultural Sciences, 50 Zhonglin St., Nanjing, 210014, P.R. China
² Key Laboratory of Urban Agriculture, Ministry of Agriculture, Shanghai, 200240, P.R. China

Email: vipmaple@126.com. Tel: +86-025-84390493, Fax: +86-025-84390435.

Abstract. Weight reduction is frequently concerned for general structure design as it can save energy in the life cycle of a machine. A small self-propelled multi-functional working machine for orchard use was designed. The lifting platform of the machine is analyzed using the finite element method for the purpose of structure optimization. By carrying out the finite element analysis, the inner stress for each component that forms the lifting platform was discussed under the designed maximum working load. Based on the relationship between the calculated von Mises stress in the structure and the yield strength of the material, the static strength of the component was verified and structure optimization suggestion was given. According to the FEA result, the overall structure of the lifting platform designed on the basis of theoretical calculation is statically safe, except for one of the long shaft sleeve. The dimension for most of the parts should be reduced since these components had an abundant static strength.

1. Introduction

For the past few years, the change in fruit tree cultivation mode in China requires more and more mechanized operations in orchard. However, the technical research in orchard working machine lags behind, which restricted the development of orchard greatly[1]. As the orchards in China are widely distributed in hilly and mountain area with complicated terrain, it is necessary to develop a small and multi-functional orchard working machine for the improvement of mechanization in orchard operation. Although some orchard machine products are readily available in western countries, e.g., the SOUIRREL lifting platform developed by an Australian company named CRENDON[2] and the Windegger Picking Platforms developed by a British company named N. P. SEYMOUR[3], their wheeled walking mechanism is not suitable for operations in complex mountains. To solve this problem, Zhu developed the first self-propelled multi-functional orchard working machine with tracks as its walking mechanism which can carry goods, trim branches of the tree and collect fruits[4]. Comparable machines with hydraulic lifting platform were also developed by Sun from agricultural university of Hebei and Liu from agricultural university of Hunan[5,6]. With abilities of trimming, weeding, picking and agricultural chemicals spraying, the moving platform designed by Wang can accept remote control[7]. To make the machine more accommodate complex landform, Wang designed a lifting platform with levelling mechanism included[8]. In addition of the works stated above, there are many other works reported about the orchard working machine. However, the general
shortcoming of these machines is mono-functioning with large size of the structure. Besides, the operation of the machines is inconvenient.

The author tried to design a small self-propelled multi-functional working machine for orchard use, and the prototype machine is built. In this paper, the lifting platform of the machine is analyzed using the finite element method for the purpose of structure optimization.

2. Overall Structure of the Machine

Overall structure of the orchard working machine was illustrated in Fig. 1, including the crawler chassis, power unit, lifting platform, console and the heavy-duty platform. The power unit consists of an engine, storage battery, variable pump and a hydraulic motor.

![Overall structure of the orchard working machine](image1.png)

**Figure 1.** Overall structure of the orchard working machine

The principle diagram of the machine system was shown in Fig. 2. According to the function of each part, the machine can be divided into the control system, the power system, the hydraulic system and the mechanical structure. The power system provides power output for the machine’s driving force, hydraulic pressure force and power for charging the battery. By control the hydraulic system and the power system, the control system is responsible for the move of the machine, rise and fall of the lifting platform and change in the hydraulic pressure.

![Principle diagram of the machine system](image2.png)

**Figure 2.** Principle diagram of the machine system

3. Finite Element Model of the Lifting Platform
3.1. Structure and Materials

Figure 3 showed the 3D model of the lifting platform when completely rose up, which consists of dozens of mechanical parts marked by Arabic numbers (hidden components not included). All the parts are made of Q235A steel (E235A steel in ISO 630 ~ 1995) except for the shaft parts and shaft sleeve parts which are made of 45 steel (ASTM 1045 steel), e.g., component 2, 6, 20, etc. Mechanical parameters of the two materials were shown in Table 1.

| Material No. | Elasticity modulus /GPa | Poisson's ratio | Yield strength /MPa |
|--------------|-------------------------|-----------------|---------------------|
| Q235A        | 212                     | 0.238           | 235                 |
| 45           | 209                     | 0.269           | 515                 |

Figure 3. 3D model of the lifting platform

3.2. Finite Element Model

A general static analysis of the lifting platform was carried out employing the finite element code ABAQUS v6.13. As the stress characteristics in the pedestal of the lifting platform may be of no interest, the pedestal-related parts were removed to simplify the analysis, e.g., parts from number 9 (number convention is shown in Fig. 3) to number 19, 22, 23, etc. The finite element model of the assembly were shown in Fig. 4, which also showed the total size of the lifting platform. To mesh the parts, the C3D8R element (an 8-node linear brick with reduced integration) was prior used, otherwise the C3D10 element (a 10-node quadratic tetrahedron) was used. Allowing for the stress concentration, a finer mesh was used around holes. The contact between moving components were constrained by surface-to-surface contact with finite sliding, e.g. the contact between part 2 and part 6. Besides, the contacted surface was assumed to be frictionless. If there is no relative motion between two components with contact, e.g. contact between part 4 and part 7, then the “tie” constraint was used on the faces. The axis of the short shaft assembled in the hole of part 1 and in the hole of part 5 was constrained only allowed to rotate around z-axes, as well as the axis of part 20. As the design load lifting platform is 200 kg, to analyze severe loading situation, a concentrated force of F equals 2000N.
was loaded near a corner of part 8, illustrated in Fig. 4. The distances between the loading position and the adjacent two edges are 150 mm.

![Finite element model of the lifting platform](image)

**Figure 4.** Finite element model of the lifting platform

### 4. Results and Discussion

#### 4.1 General Result of Stress Characteristics

Figure 5 showed contour plots of von Mises stress distribution in the lifting platform. It can be seen that, generally, in the structure the stress may not exceed 100 MPa for most of the parts. However, for the shaft sleeve part 2, it can be seen that the stress level is significantly high, especially at the two ends, which may be over 400 MPa. Besides, the stress distribution in the supporting member, e.g. part 1, around the hole is also conspicuous. Therefore, it implies that these parts may be the mechanically weakness of the lifting platform which need to be further investigated.

![Contour plots of von Mises stress distribution in the lifting platform](image)

**Figure 5.** Contour plots of von Mises stress distribution in the lifting platform /Pa
4.2 Static Strength of Key Components

The von Mises stress distribution in the shaft sleeve part 2 was shown in Fig. 6, which showed that the maximum stress existed in the shaft sleeve near the loading side. The stress level can be as high as 534.8 MPa and it is a little higher than 515 MPa, the yield strength of the material (refer to Table 1). Therefore, the shaft sleeve may need to be enhanced for further improvement of the platform. For instance, the static strength can be improved by appropriate heat treatment or thicken the wall of the sleeve. As a contact pair, the von Mises stress distribution in the shaft part 6 was shown in Fig. 7. It can be seen that the stress level is pronounced than other parts showed in Fig. 5. Although the maximum stress could be 186.5 MPa, which appears in a different side in the shaft compared to the shaft sleeve, it is still safe for the structure allowing for the yield strength of the material. Moreover, the size of the shaft can be further reduced because the stress level is far below the yield strength.

Figure 6. Von Mises stress distribution in part 2 /Pa

Figure 7. Von Mises stress distribution in part 6 /Pa

As mentioned above, the stress distribution in the supporting member around the hole is also remarkable. A detailed stress characteristics in part 1, part 3, part 4 and part 5 were shown in Fig. 8. It can be seen that, in general, the higher stress level mainly exist in the surrounding material of the hole in the middle of these parts, whereas at the two ends the stress increase is not notable around the holes. Besides, a comparison between the four components showed that the highest stress level of approximately 138 MPa was found in part 1, which is far from the loading point compared to part 5 in z direction. This may be attributed to a higher torque due to a bigger arm of force. In addition, the
comparison also showed that the stress is higher in part 1 than that in part 3, and higher in part 5 than that in part 4. This is because the stress concentration caused by holes is frequently discussed when the structure is subjected to a tension load, whilst under compression this phenomenon is not significant. In the present model, based on the knowledge of theoretical mechanics it may be found that generally part 1 and part 5 are under tension load, and part 3 and part 4 are under compression load. The static stress characteristics in the supporting members showed that in most region the stress level is far lower than the yield strength of the material (refer to Table 1), even for the maximum stress the value of which is just about half of the yield strength. Therefore, it may suggest that the size of these components could be further optimized for weight reduction of the machine, especially the cross-section size of the four components far from the middle hole, and the overall size of part 3 and part 4.

Figure 8. Von Mises stress distribution in the supporting members /Pa

Figure 9. Von Mises stress distribution in the short shafts /Pa
Allowing for that the short shafts assembled at the bottom of part 1 and part 5, as well as at the top of part 3 and part 4, may suffer great shear forces, the von Mises stress distribution in the four components were shown in Fig. 9. It can be seen that, although these shafts are subjected to shear load, the stress distribution in the shafts is not as serious as expected. The result according to the finite element analysis (FEA) reveals that generally the maximum stress may not exceed 12 MPa under the design load of the lifting platform. Hence, the static strength of the short shaft is sufficient. On account that the shaft material has a yield strength of 515 MPa, the diameter of these shafts can be therefore further reduced to make the lifting platform lighter.

4.3 Structure Design Optimization

By carrying out the finite element analysis, the inner stress for each component that forms the lifting platform can be exhibited immediately under the overall working load. Based on the relationship between the calculated von Mises stress in the structure and the yield strength of the material, the static strength of the component can be verified and the dimension of the structure can be optimized accordingly. According to the FEA result, the overall structure of the lifting platform designed on the basis of theoretical calculation is statically safe. However, the FEA result also implies that the dimension for most of the parts should be reduced since these components had an abundant static strength.

For the shaft sleeve (part 2), the static strength should be improved by thickening the wall of the sleeve, if not by changing the heat treatment technology or replacing the material, whereas for the shaft (part 6) assembled in the sleeve, the diameter could be further reduced. For the supporting members (part 1, part 3, part 4 and part 5), the size could also be further reduced for weight reduction of the machine, especially the cross-section size of the four components far from the middle hole, and the overall size of part 3 and part 4. Besides, for the four short shafts that assembled in the supporting members and the rest parts of the lifting platform, the cross-section size in relation to the loading direction could also be reduced substantially.

5. Conclusions

Weight reduction is frequently concerned for general structure design as it can save energy in the life cycle of a machine. Based on the knowledge of theoretical mechanics, the initial dimension of the lifting platform could be designed for each component and the prototype machine could be built. By carrying out the finite element analysis, the inner stress for each component that forms the lifting platform was exhibited under the design load. Based on the relationship between the calculated von Mises stress and the yield strength of the material, it showed that the weakest part is the long shaft sleeve used in the middle of the supporting members. The designed parameter according to theoretical calculation for the shaft sleeve may be destructive near the middle hole. On the contrary, the static strength for most of the components is sufficient. For the purpose of weight reduction, therefore, the dimension for most of the machine parts could be reduced.

Acknowledgement

This research was financially supported by the Jiangsu Agricultural Science and Technology Innovation Found (Grant No. CX(15)1045) and the Foundation of Key Laboratory of Urban Agriculture(No.UA201707), Ministry of Agriculture, P.R. China.

References

[1] J.L. Zhang, S.W. Chen, X.W. Zhuang, et al. 2014. Research Status of Picking Machinery Used in Orchards on Southern Hillside, Forestry Machinery & Woodworking Equipment. 42 (9): 4-7. (In Chinese)
[2] CRENDON. 2017. Squirrel 536 LB SD. Online. http://www.crendon.com.au/squirrel656lb.htm. (Accessed August 2017)
[3] N. P. SEYMOUR. 2017. Windegger Picking Platforms. Online. http://www.npseymour.co.uk/ Windegger.htm. (Accessed August 2017)

[4] H.T. Zhu. 2011. Improved Design of LG-1—a Machine Used in Orchard, Xinjiang Agricultural Mechanization. 28 (5): 44-44. (In Chinese)

[5] Z.J. Sun. 2012. Design and Research of Crawler Multifunctional Orchard Operating Platform, Agricultural University of Hebei, Hebei. (In Chinese)

[6] D.W. Liu, F.P. Xie, X. Li, et al. 2015. Design and experiment of small lifting platform in orchard, Transactions of the Chinese Society of Agricultural Engineering. 31 (3): 113-121. (In Chinese)

[7] Z. Wang. 2011. Small multi-function remote control power platform, Agricultural Equipment & Technology. 37 (3): 39-39. (In Chinese)

[8] J.C. Wang. 2013. Design on Hanging Operating Platform for Hilly Orchard, Agricultural University of Shandong, Shandong. (In Chinese)