Strength analysis and lightweight research of a fertilizing and soil covering vehicle

Heng-Hui Sun¹, Zheng-Yong Zhang¹,*, Yang Liu¹, Hai-Ming Xu¹ and En-Wei Chen²

¹ Institute of Intelligent Machines, Chinese Academy of Sciences, Hefei, 230031, P. R. China
² School of Mechanical Engineering, Hefei University of Technology, Hefei, 230009, P. R. China

*Corresponding author e-mail: sunhenghui@iim.ac.cn

Abstract: In this paper, parametric modeling is carried out for the frame part of a kind of fertilizing and soil covering vehicle to define boundary conditions such as load, constraint, etc. when the frame is under the working condition of normal full load. ANSYS software is used to produce finite element model of frame, and to analyze and solve the model, so as to obtain stress and stain variation diagram of each part of frame under working condition of normal full load. The calculation result shows that: the structure of frame is able to meet the strength requirement, and the maximum value of stress is located at joint between frame and external hinge, which should be appropriately improved in thickening way. According to the result of finite element, the scheme with size optimization is employed to design the frame in lightweight way. The research result of this paper provides the theoretical basis for the design of frame of fertilizing and soil covering vehicle, which has deep theoretical significance and application value.

1 Introduction

For long time, the farming in China was seriously dependent upon manual operation. The example is that manual furrowing, fertilizing, soil filling was required during deeply applying fertilizer by hand. This method does not only waste time and energy with high labor intensity and low working efficiency, is but also difficult to control the fertilizing amount and easy to form waste, so is not able to meet the requirement of modern farming.

With the mechanization and scale development of agricultural production, the problems including low working efficiency of manual operation, more waste formed during furrowing fertilizing and low fertilizer efficiency during farming are required to solve rapidly. Based on the combination with
theoretical analysis and experimental test, a kind of fertilizing and soil covering vehicle with design and research diversification, broader application range and higher technological content is capable of conducting continuous operations including furrowing, fertilizing, and so on instead of manual work. This research trend has become a focused issue attracting agricultural machine researchers [1~6].

Finite element method is an approximate value analysis method used to solve complicated issues during engineering analysis. Simplification of mechanical model based on frame structure and load-carrying capacity feature of fertilizing and soil covering vehicle, and finite element analysis conducted by ANSYS[8] is an effective method for assessment of structural rationality, and provides technical base for combination with experimental test and taking improvement measures in the future [8~11].

In this paper, we make parametric modeling for frame part of a kind of fertilizing and soil covering vehicle to guide parametric model into finite element analysis software ANSYS workbench, so as to carry out finite element analysis to this model and obtain stress and strain distribution diagram of frame structure. In accordance with finite element result, frame structure and connecting styles of each part are improved, and the frame has been designed in lightweight way using scheme of size optimization. The research result of this paper, offering theoretical foundation for design and lightweight of frame, indicates high theoretical significance and application value.

2 Finite element analysis model of frame

2.1 Establishment of model

During establishing three-dimensional model for frame of a kind of fertilizing and soil covering vehicle, the close matching of model and practice is not only considered, the feasibility of model calculation is also considered. According to accuracy principle of frame structure and design calculation model, parametrical model is partly simplified when establishing the model, and geometric features less impacting on mechanics property analysis result of overall frame, such as some holes, chamfers, bending, etc. in initial model of frame are required to simplify.

1.Hopper of fertilizer tank; 2.Fertilizer tank support; 3. Base plate support; 4. Hinge connecting with wheel support; 5. Hinge connecting with tractor;

Fig. 1: Finite element model of frame

The frame model of fertilizing and soil covering vehicle established in ansys is shown in Fig. 1. The frame of fertilizing and soil covering vehicle is of space beam structure which is welded with square steel tubes, rectangular steel tubes and steel plates, with structural steel material (density $7.84 \times 10^3 \text{Kg/m}^3$, elasticity modulus $E=2.068 \times 10^5 \text{MPa}$, Poisson’s ratio $\mu=0.3$).
2.2 Handling of load and working conditions constraint

As shown in Fig. 1, the load borne by this frame mainly come from gravity applied by full-load hopper of fertilizer tank, which presents as concentrated load perpendicular to frame plane. When load of hopper of fertilizer tank is full, the load is 2000kg, which is expressed as Load A IN ANSYS model.

In Fig. 1, part 4 means the hinge connecting with wheel support, through which fertilizing and soil covering vehicle connects with wheel to pass the load of full-load hopper of fertilizer tank to ground. Part 5 means the hinge connecting with tractor, through which fertilizing and soil covering vehicle connects with tractor to pull fertilizing and soil covering vehicle forward by tractor. In this paper, part 4 and part 5 are set as fixed constraints, which is indicated as constraint B–F in ansys model.

In addition, before establishing model, dimensioning system of all input data is required to unify. Material property is homogeneous and isotropic.

2.3 Grid partition

In this paper, finite element analysis model established for the frame of a kind of fertilizing and soil covering vehicle is used to analyze the strength of this frame, and conduct lightweight design on this basis. In the frame, fertilizer tank support and base plate support, as the main carrier and load passing mechanism, are required to be selectively analyzed with respect with strain. Therefore in ANSYS, solid element is selected to model for frame of a kind of fertilizing and soil covering vehicle, hexahedral element is selected to conduct precise partition on fertilizer support and base plate support, and tetrahedron element is selected to conduct partition on other secondary element, in order to save calculation resource.

For grid partition, method of 8-node linear solid hybridization element (C3D8H) is used with grid seed density 0.05m. The size of established model is same as solid size, and established material is same as solid material which is structure steel.

According to this method, there are totally 2859740 nodes and 1665026 elements formed, as shown in Fig. 2.

3 Analysis result

Based on the static load calculation of ANSYS program, equivalent stress distribution cloud chart of frame is obtained as shown in Fig. 3. The maximum stress point shown in figure is located at joint between frame and external hinge with a stress up to 50MPa, so this position should be improved in
thickening way. The stress at joint between fertilizer tank hopper and fertilizer tank support is concentrated, which should be considered. Because of $\sigma_{\text{max}}=50\text{MPa} < [\sigma]_{12}$, the maximum stress is less than permissible stress of material and meets the strength requirement.

Fig. 3: Stress distribution cloud chart

Fig. 4: Strain distribution cloud chart

As shown in Fig. 4, under the condition of full-load fertilizer tank hopper, there is a tiny frame deformation by force, and the maximum deformation point is located at lowest point of fertilizer hopper with a maximum value 0.3mm, which does not impact the quality of fertilizing operation. If prevention of more deformation at damper on lower part of hopper is required, it is suggested to add rib at damper on lower part of hopper.

4 Lightweight design of frame

Finite element model analyzed based on static and finite element analysis result indicates that under full-load fertilizer tank hopper, this frame structure bears less stress, and there is a more design margin for frame strength. In accordance with finite element result, we employ size optimization method to conduct lightweight design for frame to lower the weight of frame and overall vehicle.

Size optimization is an optimization design program which makes the overall volume of frame minimum to obtain lightest quality by optimizing the volume of vehicle frame. According to specific design requirement, the frame of fertilizing vehicle researched in this paper has a main structure
difficult to change, therefore sectional dimension (wall thickness of steel tube) of each beam forming fertilizer tank support and base plate support in frame structure is selected as design variable. In accordance with practice, the thickness of steel tube used for frame is \((0.6–6.0) \pm 0.1\) mm, so the steel tube wall thickness as design variable selected in ANSYS is limited within a certain range, namely \(1 \leq x_i \leq 6\), in which \(x_i\) is the thicknesses of 2 groups of steel tubes of key beams, \(i=1, 2\).

### 4.1 Confirmation of objective function and constraint function

In ANSYS, the frame model quality of a kind of fertilizing vehicle is conducted with parameterization, and set as objective function \(\min F(x)\) and the quality of frame of overall vehicle as \(F(x) = V(x) \rho\), where: \(V(x)\) is frame volume in m\(^3\), and \(\rho\) is density of material in kg/m\(^3\).

Load borne by frame of fertilizing vehicle during working is mainly self gravity load of full-load fertilizer tank hopper with perpendicular direction as the generated maximum displacement direction. In order to ensure the rigidity of overall vehicle and safety performance, the perpendicular direction of frame during working is required to be within safe range, making a definition of constraint function as \(d_{\text{max}}(z) \leq d\), in which \(d_{\text{max}}(z)\) is the maximum displacement amount in perpendicular direction produced during working in mm; \(d\) is maximum equivalent stress produced during working in MPa; \(\sigma_s\) is material yield limit and \(\sigma_s = 235\) MPa.

Load borne by frame of fertilizing vehicle during working is mainly self gravity load of full-load fertilizer tank hopper with perpendicular direction as the generated maximum displacement direction. In order to ensure the rigidity of overall vehicle and safety performance, the perpendicular direction of frame during working is required to be within safe range, making a definition of constraint function as \(d_{\text{max}}(z) \leq d\), in which \(d_{\text{max}}(z)\) is the maximum displacement amount in perpendicular direction produced during working in mm; \(d\) is displacement permitting value produced allowably by frame in perpendicular direction, and in this research \(d = 1\) mm.

### 4.2 Analysis for size optimization result

After setting change range for relevant design variables in ANSYS S Workbench, conduct size optimization analysis for frame model to obtain one group of optimal reference point, with the optimal result as shown in Table 2.

| Variable name                             | Before opt. | Aft. Opt. |
|------------------------------------------|-------------|-----------|
| Overall volume of frame \((V)\)/m\(^3\)   | 0.1445      | 0.1115    |
| Overall quality of frame \((M)\)/kg       | 1144.1      | 883.5     |
| Max. stress under full-load condition \((\delta)\)/mm | 0.3         | 0.7       |
| Max. displacement under full-load condition \((\sigma)\)/MPa | 50          | 70        |
| Wall thickness of fertilizer tank support beam \((x_1)\)/mm | 12.0        | 6         |
| Wall thickness of base plate support beam \((x_2)\)/mm | 12.0        | 5         |

As shown in table 1, after size optimization design, overall frame quality reduces from 1144.1kg to 883.5kg by 260.6kg, with a reduction rate up to 22.72%, which shows significant lightweight effect.

ANSYS static load calculation is used to generate equivalent stress distribution cloud chart after optimization of frame structure, as shown in Fig. 5. The maximum value of stress shown in figure is 70MPa, but the maximum stress of frame is still less than permissible stress of material due to \(\sigma_{\text{max}} = 50\) MPa \(< [\sigma][12]\), therefore meets the strength requirement.
5. Conclusion

1) The space static analysis for plough frame using ANSYS finite element is capable of better reflecting stress and deformation of frame, and may be the feasible method for frame structure design.

2) For the fertilizing and soil covering vehicle under condition of full-load fertilizer tank hopper proposed in this paper, the maximum stress of frame is less than permissible stress of material and meets the strength requirement. The point with maximum stress is located at joint between frame and external hinge with the value up to 50MPa, but still less than permissible stress, so it meets strength requirement, and this position should be improved in thickening way. The stress at joint between fertilizer tank hopper and fertilizer tank support is more concentrated and should be considered.

3) In accordance with finite analysis result, after size optimization design, overall frame quality reduces from 1144.1kg to 883.5kg by 260.6kg, with a reduction rate up to 22.72%, which shows significant lightweight effect and realizes lightweight target. The research result of this paper provides the theoretical basis for the design of frame of fertilizing and soil covering vehicle, which has deep theoretical significance and application value.

Acknowledgement

This research was financially supported by the Science and Technology Extension Project of agricultural comprehensive development in Ningxia, “The design and development of precise operating machine for the green production of Chinese wolfberry”, during 2015~2017. We also acknowledge for the support of the Research of Agricultural Science and Technology in Ningxia (No. XD-XMBW-2016001); the Science and Technology Service Network of Chinese Academy of Sciences (No. KFJ-SW-STS-144); Construction of test and maturation platform for complete sets of hardware and software in Agricultural Internet of things (Shanghai agriculture promotion word (2016) No. 2-5-11).

References

[1] Zeng S, Tang H F, Luo X W, et al. Design and experiment of precision rice hill-drop drilling machine for dry land with synchronous fertilizing [J]. Transactions of the Chinese Society of Agricultural Engineering, 2012, 28(20): 12-19. (In Chinese).

[2] Su Z H, Lan F, Li Z M, et al. Current Situation Analysis for Furrowing and Fertilizing Machine used in Domestic and Overseas Orchard [J]. Farm Machinery, 2014, 15:134-137. (In Chinese).

[3] Ji J B. The design and research of mountainous citrus orchard ditching and fertilizing
machine[D]. HuBei: Huazhong Agricultural University, 2013. (In Chinese).

[4] Zhu Y R. Design of Orchard’s Machine Structure with Ditching and Deep Fertilizing[J]. Journal of Agricultural Mechanization Research, 2017, 39(12):102-106. (In Chinese).

[5] Wang R L, Pan G G, Wei F L, et al. Design and Experiment on Fertilizing Machine Combined with Deep Tilling and Ridging[J]. Journal of Agricultural Mechanization Research, 2016, 1:90-94. (In Chinese).

[6] Zhang Q, Wang W, Liao J A. Design of Fertilizing Machine’s Furrow Opener in Jujube Orchard and Finite Element Analysis[J]. Journal of Agricultural Mechanization Research, 2016, 4:23-28. (In Chinese).

[7] Gong W T. Design and Test Research for Furrowing, Fertilizing and Seeding combining Machine[D]. NanJing: Nanjing Agricultural University, 2012. (In Chinese).

[8] Li L M, User Guide of ANSYS Finite Element Analysis [M]. Beijing: Qinghua University Press, 2005. (In Chinese).

[9] XU G F, HU G Z, CHENG P F, et al. The digital modeling and the motion simulation of the hand-held portable rice transplanter [J]. Journal of Agricultural Mechanization Research, 2015(4): 27–31.

[10] LI X, XU W M. Tomato translates seedling machine, the chassis finite element method analyses based on the ANSYS[J]. Machinery Design & Manufacture, 2010(1): 56–58. (In Chinese).

[11] LONG K, ZUO Z X. Structural design of main bearing cap based on topological optimization and shape optimization[J]. Transactions of the Chinese Society for Agricultural Machinery, 2008, 39(4): 152–156.

[12] Liu Hongwen. Ed. Mechanics of Materials [M]. Higher Education press, 1992.