Available online www.jocpr.com

Journal of Chemical and Pharmaceutical Research, 2014, 6(11):669-673

Research Article

ISSN : 0975-7384
CODEN(USA) : JCPRC5

Analysis of the harvesting robot arm modal based on CAE

Jian Zhou[1], Zhimin Yang[1] and Songlin Chen[2]

[1]*Machinery and Electrical Engineering College of Centre South University of Forestry Science and Technology, Changsha, P. R. China*
[2]*Hunan Dizhi Middle School, Changsha, P. R. China*

ABSTRACT

In order to improve the harvesting efficiency of forest-fruit, the structure properties of the harvesting robot arm was introduced. Based on the modal analysis theory, the harvesting robot should have more reasonable structure and dynamic performance, and a finite element model of the harvesting robot arm was established by utilizing the software of ANSYS Workbench. Three kinds of typical pose were selected, further more, the preceding six steps natural frequency and mode were obtained. Finally, the weak parts of the robot arm were found out and relative improving suggestion was put forward, which laid foundation for the optimized design.

Keywords*:* Harvesting robot arm; ANSYS Workbench; Model analysis

---

INTRODUCTION

Because of its' strong seasonal characteristics,high labor intensity,etc,forest-fruit harvesting operation has become the most important link in the process of forest-fruit production, and the harvesting labor force has accounted for the entire production process's 35%～45%of the labor force[1,2].At present,there aren't much people who engage in research on fruit harvest mechanization technology in China,for various reasons,compared with foreign advanced level,there was still a large gap in Chinese research level.According to the new design of the forest-fruit picking robot,the cantilever crane was studied as the object,and the finite element modal of different position in the robot cantilever crane was analyzed by using ANSYS Workbench software,thus the natural frequency and vibration type could be gotten, and the weak link cantilever crane might be found out,thus it can lay the foundation for further optimization design of cantilever crane[3,4].

EXPERIMENTAL SECTION

PICKING ROBOT ARM SHELF STRUCTURE

Mechanism diagram of picking robot arms is shown as Figure 1,which has two degrees freedom of mechanism,namely,they are the level and vertical linear motion bases.The main arm 1 overall length was 720mm,and the ratio of the middle hinge joint was 1:5 ;The main arm 2 length was 600mm;The main arm 3 length was 780mm,and ratio was the same as middle joint.The connection strap length was 130mm.The main arm 1,the main arm 2,the main arm 3 and the connecting pieces formed a parallelogram mechanism.It was the characteristics of the arm frame that the uprignt base plate was stationary and the horizontal base moved a unit distance,the main arm 3 end moved 6 units distance along the horizontal in the same direction;When the horizontal base was still while the upright base moved a unit distance,the main arm 3 end moved 5 units distance along the vertical opposite direction.The structure characteristics could be good to reduce the difficulty of the robot program controlling and save the calculating time,so it also might improve the response speed of the robot.

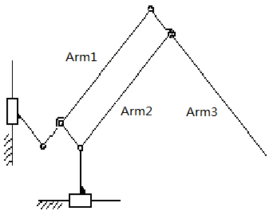


Fig.1 The schematic diagram of harvesting robot arm

In order to improve the jib structure strength and stability,in the meanwhile,for ensuring the end bearing installation surface which was always kept in the horizontal direction,and two groups of parallelogram mechanism in the main arm were added,so the robot arm of the final three-dimensional entity model was shown as Figure 2.

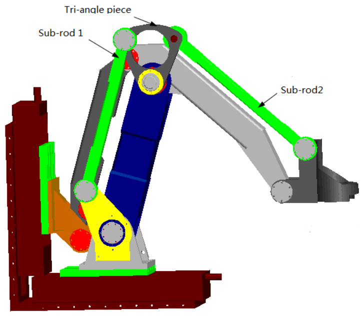


Fig.2 The model of harvesting robot arm

FINITE ELEMENT MODAL ANALYSIS THEORY

For getting a continuous quality structure with finite element,modal was analyzed.Firstly,the structure was dispersed into finite elements model to find out the element stiffness matrix[K] and the element mass matrix [M];Secondly,according to the node DOF serial number, the element stiffness matrix and mass matrix were set to get the total stiffness matrix{K} and the total mass matrix {M};Finally,small damping structure of linear dynamic system could use composite damping [C] to obtain the vibration differential equation of structure(equations) [5].

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F(t)\}$$ Formula（1）

In equation (1), $\{\ddot{u}(t)\}$ was the acceleration vector ,$\{\dot{u}(t)\}$was the velocity vector, and $\{u(t)\}$ was the displacement vector.$\{F(\mathrm{t})\}$ was the overall load vector.

The modal analysis of structure required that structural inherent frequency and vibration type parameter should be solved,but the inherent frequency had nothing to do with the external load,and the effect of damping on the inherent frequency and vibration type was very small.So it could be neglected.The formula(2) could be obtained.

$$\left(-\omega_i^2[M]+[K]\right)\{u\}=0$$ Formula (2)

Formula (2) was the systematic no-damp free vibration equation of the matrix expression, ω_i was natural frequency of the free vibration.From formula (2), n-th degree polynomial of ω_i^2 could be obtained,and polynomial roots (eigenvalues) namely were the natural frequency of the model.When the polynomial roots were substituted in type (1),we could find the feature vector,thus the mode shapes of the structure could be obtained at a given frequency.

THE FINITE ELEMENT MODEL ESTABLISHMENT

As shown in Figure 2,picking robot arm frame was mainly composed of horizontal and vertical bases,the connecting piece,pull rod seat plate,the main arm,subtie,triangle,end supports,the pin shaft and the end cover.Although ANSYS Workbench has a very powerful function in the finite element calculation,but its function in the aspect of modeling isn't enough strong.Therefore,we used Pro/E software to establish a three-dimensional entity model of robot arm,then data exchange of the model could be imported into ANSYS Workbench through software interface[6].Because the three-dimensional entity model was more complex,it contained too many components.If the parts were not simplified,and all were input into ANSYS Workbench to be calculated directly,the finite element mesh generation would be very complicated,at the same time,it would increase the calculation amount and affect the calculation precision.Thus the model should be simplified before calculation.From the analysis theory of finite element model,we know that the natural frequency of the structure and the main mode were mainly related with the quality and stiffness distribution.Therefore,the simplification of the model was to ensure the overall quality and the quality distribution on the whole structure,after the geometric characteristics and mechanical properties of the arm frame were taken into consideration synthetically,the specific treatment of the model steps were as follows:(1)The little effecting parts such as bolt holes,auxiliary holes,chamfering angles,fillets,etc,were all simplified.(2)The parts which have interference fits with the pin and shaft were all handled by "form a new part" in ANSYS Workbench,so they became an integral part.(3)Step two also suited the screw joint parts.(4)Bearing end cover was relatively lighter,and they had little effect on the stiffness of the structure,so the bearing covers effecting on the arm were ignored.(5)The bearing transmission relationship was simplified,and the joint deputies were simulated as the pin shafts and holes[3,4].The simplified model was shown as Figure 3.

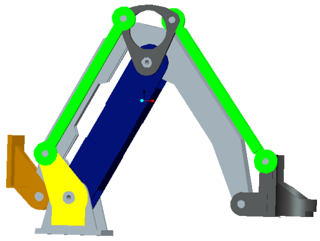


Fig.3 Simplified model of harvesting robot arm

Before the modal was analyzed in ANSYS Workbench,first of all,the materials needed choosing,because the arm frame parts were composed of Q345 and 45# steel,therefore the "Engineer data" module of the ANSYS Workbench was chosen the default "Structural Steel".According to the introduction of the front jib structure,picking robot arm was a two-degree-freedom structure,also the modal analysis was required before adding fixed constraints on the base surface to simulate the sliding table which was locked by motor,also including the case which degree-freedom was constrained.According to the actual situation,all rotary joints which had relative movement would be regarded as the "Revolute" connection. It is very important to divide grids for finite element analysis,whether it was good or bad,it would have great impact on the result of analysis and calculation,the arm frame,which was the final effect of finite element mesh,was shown as Figure 4,there were 52768 nodes and 6958 units altogether.The hexahedral meshes were accounted for 86%,and the overall meshes quality was better.

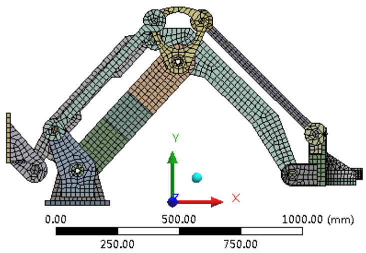
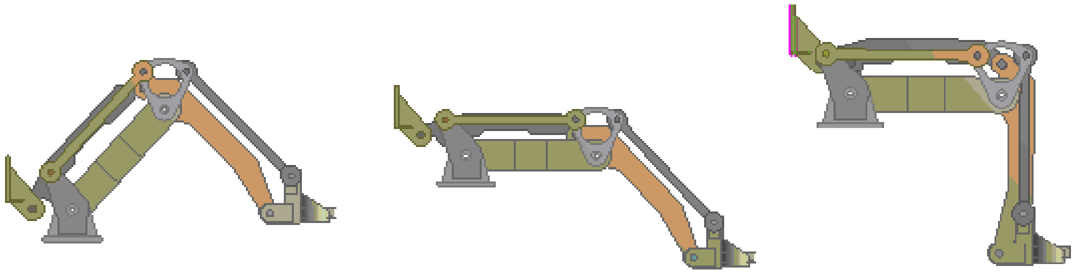


Fig.4 The finite element meshes of robot arm

The overall coordinate system direction of finite element was defined as bellows:The X axis was horizontal to the right,and the same to the level of the base direction;the Y axis was vertical upward direction,and the same to the uprignt base;the Z axis was vertical surface outward.

Picking robot arm was designed for picking the fruit,such as camellia oleifera.For the whole modal characteristics of picking robot arm could be understood more comprehensively, after taking the robot arm factors into consideration synthetically,which include the arm extension size,arm parts interference,the chassis height,the growth of plants,as well as the terminal holder attitude,etc,then three typical postures of the arm frame's finite element modals were calculated as bellow:(1)The horizontal position had characteristics which the horizontal base, the upright base plate,the straight line of main arm 3 end point were hinged joints,they were all in the horizontal position,in the meantime,the height of the main arm 3 end was 1300mm above the ground.(2)The feature of the main arm 3 was to stretch the furthest position,it also maintained the horizontal position,and the angle of main arm 2 with main arm 3 is 135 degree,the height of the main arm 3 end was 840mm above the ground.(3)The feature of main arm 3 was to stretch the lowest position,which maintained a horizontal position and was perpendicular to the main arm 2,the height of the main arm 3 end was 650mm above the ground.Three kinds of typical poses were shown as Figure 5.



a) b) c)

Fig.5 The typical positions of robot arm

RESULTS AND DISCUSSION

The resonance frequency which robot arm frame mainly happened was in low frequency stage,and six modes of the arm were obtained by ANSYS Workbench,the modes could basically meet the requirements of vibration characteristics of arm frame.The six natural frequencies and modes situation of the robot arm were shown as Table 1.

Tab.1 The preceding six steps natural frequency and mode of the harvesting robot arm（Unit：Hz）

Mode order	Horizontal position	Furthest stretching position	Lowest stretching position	Vibrating type
1	33.81	32.302	36.597	End support vibrating along the X direction
2	41.529	40.01	42.412	End support vibrating along the Z direction
3	65.744	64.556	67.275	End support vibrating along the Y direction
4	92.858	90.427	93.997	Triangle plate swinging along the Z direction
5	132.83	131.61	137.87	Sub-rod 1 rotating along the Y direction
6	164.27	162.53	168.51	End support rotating along the Y direction

The calculated results showed that the six modal frequencies of the robot arm frame were between 32~170Hz,and the end support's deformation was larger,its' stiffness was smaller,the side pull rod and triangular block also occurred the deformation,the main reason was these parts were thinner relatively,the solving methods were to increase their thickness or change better materials to reduce the deformation.Compared with the analysis results of arm frame in the three typical postures,they showed that the modal natural frequencies of the arm frame were closer to the corresponding positions,and the vibration modes were basically same.In general,the inherent frequency of the arm was relatively low,which showed that the integer rigidity of the arm structure was low,it could be solved to improve the final integer rigidity through the design which changed the arm frame structure to get the higher strength.

CONCLUSION

The modal analysis of picking robot arm was carried out by ANSYS Workbench,and the modal characteristics in the different positions were gotten,then we found that the deformations of end bearing of the arm frame,the sub-rod and a triangular block were larger,thus the stiffness was smaller,which could be reduced by increasing the parts' thickness or changing good materials.In terms of jib's general structure,its' natural frequency was lower,which showed that the whole arm shelf structure stiffness was lower relatively,the solving way was the better design to improve the characteristic of the arm structure.

Acknowledgement
Aid program for Science and Technology Innovative Research Team in Higher Educational Institutions of Hunan Province;Conditional innovating project of science and technology department of Hunan province (2012TT2048).

REFERENCES

[1]Yang Zhimin,Zhou Jian,Li Lijun,Cheng Xueliang,*Journal of agricultural mechanization research*, **2013**, 12:56-58, 62.
[2] Horvath E, Sitkei G, *Transactions of the ASAE*,**2005**,48,19-25.
[3] Castro-García S, Gil-Ribes J A, Blanco-Roldán G L,*Transactions of the ASABE*,**2007**,50,727-732.

[4] Castro-Garcia S, Blanco-Roldan G L, Gil-Ribes J A,*Trees-structure and Function*, **2008**,22, 795-802.
[5] Polat R, Gezer I, Guner M,*Journal of Food Engineering*, **2007**,79, 1131-1135.
[6] Jian Zhou, Lijun Li,Lei Lei. *Journal of Chemical and Pharmaceutical Research* ,**2014**,6,2746-2751.