International Journal of Control and Automation
Vol.8, No.8 (2015), pp.385-398
http://dx.doi.org/10.14257/ijca.2015.8.8.36

Light-Weight Study of Aircraft Deicing Vehicle Based On Ansys

Min Yuan[1], Peng Zhang[2]

[1] *School of Mechatronic Engineering, Southwest Petroleum University, Chengdu, 610050, China*

[2]*School of Civil Engineering and Architecture, Southwest Petroleum University, Chengdu, 610050, China*

[1]*Email: 421436821@qq.com (Min Yuan)*

Abstract

Firstly this article briefly describes the technology of topology optimization. Then the design space of frame of aircraft deicing vehicle is established and the model of finite element is gained by meshing. Loads and DOF constraints are applied based on the analysis of working conditions. The results of topology optimization are attained by calculation. On the basis of aforementioned results, the analysis is made. Secondly after a brief introduction of the theory of modal analysis, the modal analysis for the aircraft deicing vehicle is made. Then topology optimization takes the maximization of the 1st mode frequency as the objective function on the basis of modal analysis. Finally, the second design is carried out according to the results of topology optimization, and a summary of the thesis is concluded.

Keywords: *Light-weight; Aircraft Deicing Vehicle*

1. Background And The Theory Of Topology Optimization

1.1. Background

Aircraft deicing vehicle (hereinafter, “deicing vehicle”) is a kind of ground equipment for airport. Its main function is to jet special liquid on the surface of aircraft to clear the accumulation of ice, and also to meet the needs of aircraft cleaning and maintenance. The demand for the stability of working is higher. The frame of deicing vehicle is the main part that is applied force. Meanwhile, it is the platform for installation of the main components of deicing vehicle. The vast majority of components and assemblies of the deicing vehicle are fixed to the frame.The function of the frame is supporting and connecting of the main parts of the deicing vehicle. Also the frame is exposed to various loads from both inside and outside the car. The design of the frame structure largely determines the static and dynamic characteristics of the vehicle. Therefore the objective of the study is to design the frame met the strength and lightweight requirements. And the study is necessary and important.

1.2. Introduction To Finite Element Modeling

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

ISSN: 2005-4297 IJCA

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation).

1.3. Technology of Topology Optimization

At Present homogenization method, artificial materials and evolutionary structural optimization are applied more widely in topology optimization techniques. The core idea of technology of topology optimization is to seek the best material distribution within a given design space. So that the resultant structure can both satisfy given constraints and objective function. In Ansys topology optimization is based on the mathematical model of variable density method. Density of the element is the same in the model. The design variable is the density of each element. Topology optimization takes reducing compliance or improving natural frequency of the modal as the objective function in Ansys. In recent years, with the rapid development of computer technology, the research of topology optimization is also more extensive and intensive. The topology optimization is combined with the finite element method to solve practical problems in the process of product development. Theoretical study of topology optimization has become mature gradually, but the practical applications also require further exploration. In addition, the organic integration of the topology optimization techniques and CAD systems is a problem deserving research. Topology optimization is the most general branch in structural optimization. When using topology optimization it is desired to find where in the design space it is optimal to place material. The design space is a geometrical space where material placement is allowed. A design variable in topology optimization has in theory a discrete value (material or void), and every point in the design space is a design variable. In theory this means that any structural shape can be achieved within the design space. In practice, this is not the case because of the FE-discretization that has to be made for computational reasons.

2. Topology Optimization Under A Single Working Condition

2.1. The Establishment of the Finite Element Model

The pure bending condition and bending-torsional condition is generally considered in statics analysis of frame. The road in airport that deicing vehicle works on is very smooth. Operating speed is at 5km / h or so. At this time frame is in pure bending condition. Static analysis and topology optimization is solved in pure bending working condition with full load. The material of frame is 16Mn. Its properties are listed in Table 1.Vertical DOF of four wheels, longitudinal DOF of two front wheels and transverse DOF of the left front wheel and the left rear wheel is constrained in pure bending condition. And all other degrees of freedom are released. The distribution of load mass is listed in Table 2. The arm of deicing vehicle is lifted from the front bracket under working condition. The load owing to the upper-car is equivalent to the vertical force and the equivalent moment. The equivalent vertical force and moment is 24500N and 62400N.m respectively. Engine is loaded as concentrated loads. The quality of the rest is loaded uniform. The remaining part of the mass is loaded evenly. It takes 9.8m2/s as the value of the acceleration of gravity.

Table 1.Material properties

Density	Modulus	Poisson's ratio	Yield limit
7.82×10^{-6}kg/mm^3	206Gpa	0.28	345Mpa

Table 2.The Distribution Of Load Mass

The cab (including the driver)	500kg
Engine, hydraulic pump and bracket	1000kg
Fuel tank (including oil)	500kg
Hydraulic tank(including oil)	500kg
Boiler (including radiator)	1900kg
The tank for deicing fluid (including liquid)	10000kg
Big arm, small arm and rotating platform	1300kg
Operating room (including the operator)	200kg

The finite element model of frame is established in Ansys Workbench environment. Wherein, in order to make full use of frame space and lower center of mass, a portion of the de-icing fluid tank was placed in a closed space formed by the frame structure of the middle layer. Engine and the hydraulic tank, respectively, is arranged on the right and left side of the middle frame. The design described above makes the layout more compact.

2.2. The Analysis For Optimized Results

Loads and DOF constraints are applied on the frame in Ansys Workbench environment. Total deformation displacement contour shown in Figure 1 and the equivalent stress contour shown in Figure 2 are obtained by calculation in the software.

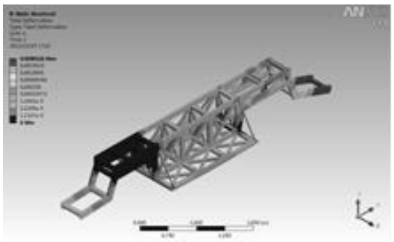

Figure 1: Total Deformation Displacement Contours

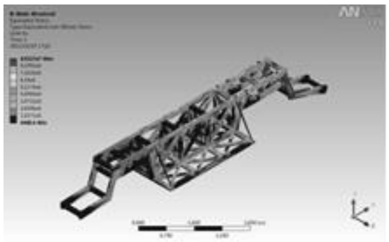

Figure 2: Equivalent Stress Contours

Maximum stress areas of frame are mainly concentrated in the junction of rear suspension mounting groove section and boiler installation segment by Figure 2 and Figure 3. The value of maximum stress is 65.1MPa and far below the yield limit of frame material of 16Mn. The areas of maximum deformation are mainly concentrated in the end of the frame and the value of maximum deformation is 3mm.The deformation of Engine mounting bracket is required smaller. The value of maximum deformation is 2mm, which is far less than the reference value set by automotive styling test procedures. To sum up, the safety factor of the frame is relatively high. Accordingly, there is a big space for improvement and optimization.

2.3. Establishment Of The Design Space

Topology optimization space frame should be set up in front of the topology optimization. The function of the frame and the arrangement of the components on the frame must be fully considered in the process of building. Also, from the perspective of manufacturing, optimizing space area should be built as big as possible in order to realize the possible maximum optimization. Material and quality is concentrated in the middle of the frame. The installation of the front and rear suspension is involved with the middle section. Therefore under the premise of keeping basic framework, topology optimization space is mainly built on the middle frame. And Local topology optimization space is built on front and rear suspension mounting section of the frame. Entity unit or sheet unit are established between vertical and horizontal beam as optimization space. Vertical and horizontal beam is classified as non-optimized space.

2.4. Optimization Calculation And Analysis Of The Result

According to the topology optimization space, finite element model is established in Ansys Workbench. Constraints and loads are applied on the frame. And related parameters are set in the topology optimization module of the software. The results of topology optimization are shown in Figure 3 by calculation.

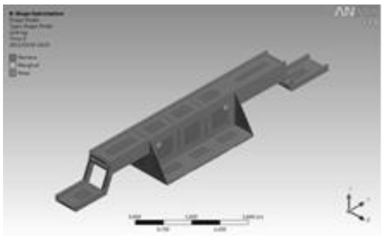

Figure 3: Contours of Topology Optimization

Based on the results of the analysis, some conclusions can be summarized:

1) Judged from Figure 3, the topological shape of the frame is regular. The boundary of topology results is relatively clear. So the specific optimized structure is more easily to be designed.

2) Material removal areas are mainly concentrated in the middle of the frame. Diagonal strut is removed by topology optimization. The distribution of beam and vertical beam appears obviously.

3) Some parts of the optimization are not reasonable because too much material is removed such as the junction of cab mounted section and suspension mounting section. Structure can't be connected together so that transmission of forces and moments can't be finished. These areas must be paid particular attention in the process of second design.

2.5. Secondary Design Of The Frame

The results of topology optimization provide a theoretical model without considering the actual function of the frame and manufacturing constraints. Frame must be secondary designed based on the theoretical model. Diagonal strut is removed by topology optimization. The arrangement of unreasonable is adjusted according to the topology cloud. Torsional flat is added in the rear section of the frame. Stringer in front and rear of the frame is strengthened. Longitudinal beam is designed to be the structure of double layer. Closed loop structure was adopted to disperse stress as much as possible and avoid stress concentration. Whether accord with the design requirements the frame generated by secondary design should be analyzed by the finite element calculation.

Loads and DOF constraints are applied on the frame in Ansys Workbench environment. Total deformation displacement contour shown in Figure 4 and the equivalent stress contour shown in Figure 5 are obtained by calculation in the software.

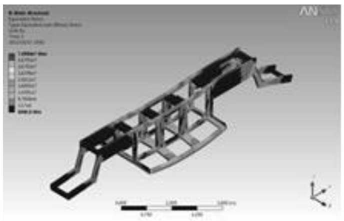

Figure 4: Total Deformation Displacement Contours

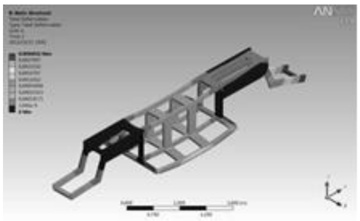

Figure 5: Equivalent Stress Contours

The areas of maximum deformation are also mainly concentrated in the end of the frame and the value of maximum deformation is 3.4mm. The deformation of Engine mounting bracket is required smaller. The value of maximum deformation of engine mounting bracket is 2mm. Maximum stress areas of frame are also mainly concentrated in the junction of rear suspension mounting groove section and boiler installation segment. The value of maximum stress is 76.9MPa. The quality of frame reduces from 2300 kg to 1656 kg by topology optimization. Light-weight effect is obvious. Strength is satisfied while the quality is reduced.

3. Topology Optimization Under Multiple Loading Conditions

3.1 .Establishment of the Design Space

The essence of the topology optimization is to seek the best distribution of material in a given design space. Therefore, the design space of the frame should be established first. Considering the layout of the vehicle and the connection of various components with the frame, the space of design should be as large as possible in order to play the biggest role of topology optimization. The design space is divided into the part used for optimization and the part not used for optimization. The stringers determine the basic structure of the frame. And the main components of the vehicle and the frame are connected on the stringers. So the non-optimized area is defined on the stringers, and the optimized area is defined between the two stringers. Three-dimensional digital model of design space as shown in Figure 6 is completed in Solidworks software.

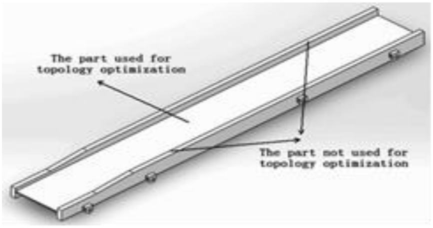

Figure 6: The space for design.

3.2. Meshing Of The Design Space

In this paper, Ansys as a universal CAE software is selected as a tool for topology optimization. This software can be run in classic and Workbench environment independently. Ansys Workbench can be seamlessly connected with most of the three-dimensional modeling softwares, but topology optimization can be realized only under single working condition in Ansys Workbench. Although topology optimization under multiple loading conditions can be accomplished in Ansys classic environment, most of the three-dimensional digital models can be imported into Ansys classic environment by the conversion for file format. Losses generated in the conversion may make negative influence on the results of topology optimization. The type of analysis should be selected to Shape Finder in Ansys Workbench. Then the model of finite element that is established in Ansys Workbench should be transferred to classic environment. And the topology optimization based on minimum compliance is to be continued in Ansys classic environment. This article innovatively combines the two environments to improve the accuracy of the results. The process of meshing is shown in Figure 7.

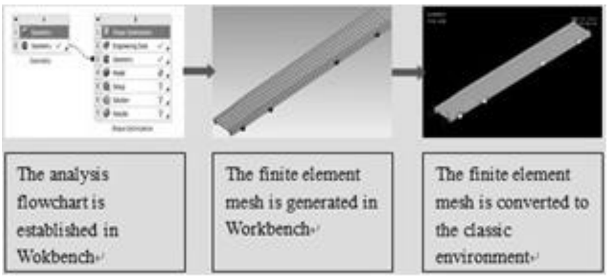

Figure 7: The Process of Meshing Finite Element

3.3. Basic Load

The basic load refers to the loads of frame applied from the own weight (including the frame itself) above suspension. This kind of loads is not depended on the working conditions. The material of frame is 16Mn. The load owing to the upper-car is equivalent to the vertical force and the equivalent moment. It takes 9.8m2/s as the value of the acceleration of gravity. The distribution of load mass is listed in Table 3.

Table 3: The Distribution of Load Mass

The cab (including the driver)	500kg
Engine, hydraulic pump and bracket	680kg
Fuel tank (including oil)	360kg
Hydraulic tank(including oil)	280kg
Boiler (including radiator)	1200kg
The tank for deicing fluid (including liquid)	6000kg
Big arm, small arm and rotating platform	1760kg
Operating room (including the operator)	200kg

3.4. The Analysis Of Working Conditions

The deicing vehicle's working conditions is more complex than the common truck. As a kind of ground equipment for airport, the deicing vehicle travels on the road in good conditions. The vehicle is parked in state of deicing. The lifting of the upper-car is likely to cause fatigue damage in the position of the frame connected with the rotating platform.

On the other hand, the deicing vehicle needs transition. At this moment, just like the general road truck, the dangerous condition of the frame which damages to frame more seriously is the bending-torsional condition or the emergency braking condition. The bending-torsional condition which is likely to cause the frame distorted. And this condition takes place on uneven road when the four wheels are not in the same horizontal planet. In the emergency braking condition, the value of deceleration is larger. So this condition is easy to cause shear failure of frame.

Based on the above analysis, the frame is constrained on the various conditions according to the Table 4.

Table 4: Constraints of the Various Conditions

Working condition	The method for constraining and loading
Deicing operation	Constraining vertical DOF of four wheels, longitudinal DOF of two rear wheels and transverse DOF of the left front wheel and the left rear wheel; The equivalent vertical force and moment is 19208N and 27733N.m respectively.
Bending-torsional condition	Constraining vertical and longitudinal DOF of the left front wheel and vertical DOF of the right rear wheel; The equivalent vertical force and moment is 19208N and 2680N.m respectively. The value of dynamic load coefficient takes 1.3
Emergency braking condition	Constraining longitudinal DOF of four wheels and vertical and transverse DOF of two rear wheels. The equivalent vertical force and moment is 19208N and 2680N.m respectively. The value of braking deceleration takes 5.88m^2/s. The value of dynamic load coefficient takes 1.2.

3.5. The Calculation of Topology Optimization

Loads and DOF constraints are applied on the frame in Ansys classic environment as described in Table 2 and Table 3. There are three kinds of working conditions. Accordingly there are three load steps. All load information must be deleted after a load step is written, and then the next load step can be written. The objective function is set to of the minimum compliance on the three conditions. And the volume removal ratio is set to 50%. By 30 times iterative calculation, the pseudo-density contour in Figure 8,the iterative curve of compliance in Figure 9 and the iterative curve of volume constraint in Figure 10 are drawn.

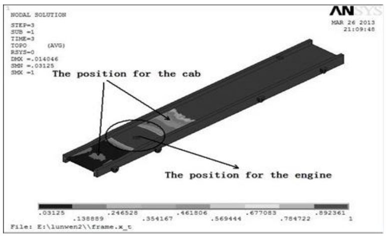

Figure 8: The Pseudo-Density Contour

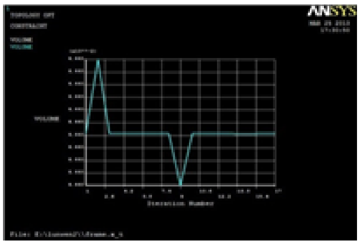

Figure 9: The Iterative Curve of Compliance

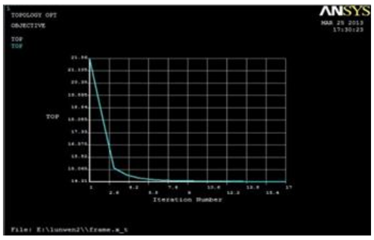

Figure 10: The Iterative Curve of Volume Constraint

3.6. The Analysis for Optimized Results

The results of topology optimization are sensitive to load conditions. Different load conditions may make a big difference in the results. By the Figure 8, the results of topology optimization have clear boundaries. There is obvious distribution of beams. The elements of smaller density are mainly located in the position of the frame for the cab. Obviously these elements are considered for removal in the design process. On the contrary, the elements in the position of the frame for the engine are denser. The material in this part should be retained in the design process.

Concluded from Figure 9, the compliance of the frame decreases with the increase of the number of iterations. The iterative curve of compliance converges in the 17th iterative calculation. Meanwhile, the compliance makes the minimum and the results of topology optimization are effective. What the Figure 10 reflects is the change of volume of the part used for the optimization in the iterative process. The volume is 0.0888m3 before optimization. After optimization, the volume is reduced by fifty percent to 0.0444 m^3.

4. Topology Optimization Based On Modal Analysis

4.1. The Theory of Modal Analysis

Dynamic characteristics mainly include the natural vibration frequencies, damping ratios and mode shapes. Frame is a multi-DOF system, so there must be multiple modes. Modal analysis is a modern method that is used to study the dynamic characteristics. Modal analysis is also an application of the system identification method in the field of engineering vibration.The typical equation for solving undammed modal is a classic eigenvalue problem. And the problem can be described using the following formula

$$[K]\{\theta i\}=\omega_i^2[M]\{\theta_i\} \qquad (1)$$

[K]-stiffness matrix ;$\{\theta_i\}$-the i-th mode vector; ω_i-the natural frequency of the i-th mode; [M]-the mass matrix.

By using the technology of modal parameter identification, modal frequencies, damping ratios and mode shapes are obtained. Scientific means is provided for depth study of dynamic characteristics, analysis of dynamic weakness and optimization design of structure.

Mathematical model of topology optimization based on modal analysis can be described as follows.

$$min\ f(\mu_i) \quad (2)$$

$$s.t. \int \mu_i d\Omega \leq \alpha V \quad (3)$$

f-natural frequency; μi-pseudo-density of the i-th element, 0≤μi≤1; α-removal ratio of volume; V-volume of design space.

4.2. Establishment of the Finite Element Model

Establishing solid model in Ansys is complicated. In General, solid model of the design space is created in professional 3D design software. Then the model is imported to Ansys for the next step. Solid model of the design space is transformed to parasolid of the general-purpose formats. Parasolid is a modeling module with clear boundary. Solid model, universal element model and free-form surface model can be supported by parasolid.16Mn is selected as the material for frame.

In order to realize the function for topology optimization in Ansys, the type of element should be selected for two-dimensional plane, three-dimensional block or shell. The model can only have the following type: SOLID2 and SOLID82 of two-dimensional plane; SOLID92 and SOLID95 of three-dimensional block; SHELL93 of shell. According to the actual situation of this article, SOLID92 is chosen for the type of element.The element numbered one can be performed optimization calculation. Otherwise, the optimization calculation is not executed. Under the premise of maintaining the basic structure of the frame, solid elements that used for optimization are established between boundary beams of the frame. Accordingly these elements are numbered one. Oppositely the boundary beams are not used for optimization, and the number of these elements is set to greater than two.

4.3. Modal Analysis

The frame of deicing vehicle is mainly affected by low order mode. Therefore this article only extracts the first six modes of the design space (excluding rigid mode). The data are obtained by the calculation in Ansys. In order to make the analysis more scientific, the above data are plotted as the modal frequency and displacement curve as shown in Figure 11.And each vibration mode shape is shown in Figure 12.

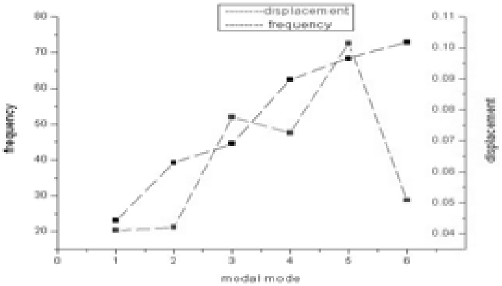

Figure 11: The Modal Frequency and Displacement Curve

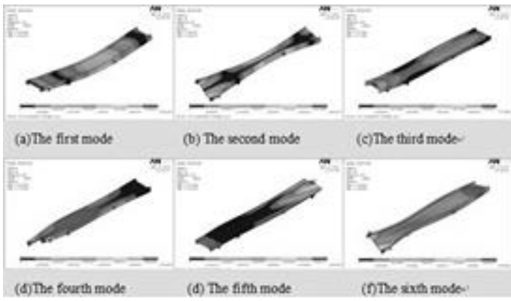

Figure 12: Each vibration mode shape

First-order vertical bending is appeared in the 1st vibration mode. And larger displacement occurs in the front section and the central section of the frame. First-order torsional vibration is appeared in the 2ed vibration mode. And the maximum displacement occurs in the front of the two stringers. Second-order vertical bending is appeared in the 3rd vibration mode. And partial transverse bending is appeared in the front section and the rear section of the frame. The 4th vibration mode is local vibration mode. And in this mode local torsional vibration in the front of the frame is the main modal characteristics. Just like the 4th vibration mode, the 5th vibration mode is local vibration mode too. And local transverse bending vibration is the main modal characteristics in this mode. Overall second-order torsional vibration is appeared in the 6th vibration mode. Maximum value of the displacement is 0.101449, which takes place in the 5th mode at the middle of the back-end of the frame. So the last beam is the weakness of the entire frame. Although the value of maximum displacement in 1st mode is minimum, the mode frequency is relatively low. Resonance is most likely to occur at this time. Obviously the 1st mode frequency is also regarded as the weakness of the dynamic characteristics.

4.4. Topology Optimization

The maximization of the 1st mode frequency of the frame is set to the optimization target. And the removal ratio of volume is set to 50%. By 30 time's iterative calculation, the pseudo-density contour in Figure 13 and the iterative curve in Figure 14 are drawn.

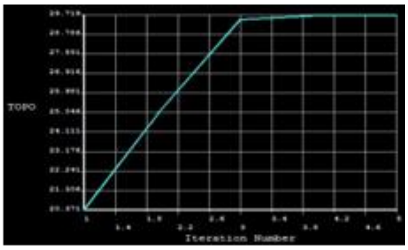

Figure 13: The Pseudo-Density Contour

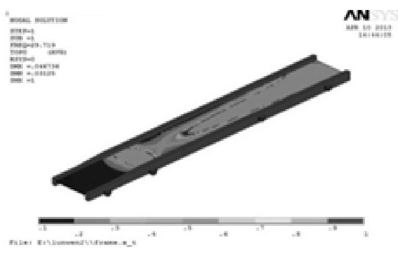

Figure 14: The Iterative Curve

Judged from Figure 13 the topological shape of the frame is regular. The boundary of topology results is relatively clear. So the specific optimized structure is more easily to be designed. And the retained material is mainly concentrated in the engine mounting section and rear section of the frame. These parts withstand greater loads under the influence of the engine vibration. The iterative curve converges in the 5th iterative calculation, which indicates that the topology result is valid. In the iterative process the value of the 1st mode frequency gradually rises from 20.2071HZ to 29.719HZ.

4.5. Suggested Design

Abstract design recommendations are given by topology optimization. Of course, this is just still stuck in the conceptual design phase. In order to get the specific structure, the second design of the frame must be made. Considering the function and manufacturing processes, the second design is carried out according to the results of topology optimization. First of all beams is designed to be intensive in the rear section of the frame. And then the engine mounting section is strengthened. The frame of deicing vehicle generated by the second design is shown in Figure 15.

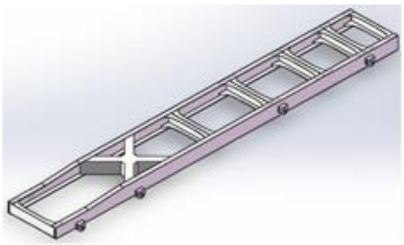

Figure 15: The Frame Generated By the Second Design

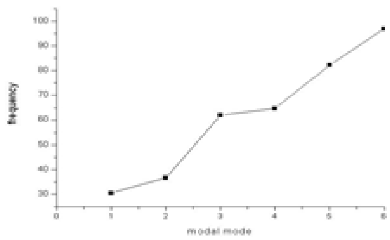

Figure 16: The Modal Frequency Curve

The modal analysis of the frame generated by the second design is made. And parameters should be set the same as the phase before optimization. The first six modes frequency is graphed in Figure 16.By Figure 16, the value of the 1st mode frequency increased from 23.123HZ to 30.486HZ by 32% compared to the phase before optimization. So the effect of optimization is obvious. Idle speed of the engine is 750r/min, corresponding to the excitation frequency of 37.5Hz; Common working speed is 2000-2200r/min, corresponding to excitation frequency of 67-73Hz; the value of the excitation frequency caused by roughness of the road is less than 20Hz.The 1st mode frequency is lower than the idling frequency but higher than the excitation frequency from road surface. Also the overall modal and the range of common operating frequency is not overlapped. Concluded from the above analysis, the dynamic characteristics of the frame meet the design requirements.

5. Summary and Conclusions

Ansys Classic and Workbench environment are innovatively combined in the process of meshing. This innovation can effectively reduce the loss in the process of model transformation and improve the accuracy of the results. Complying with the design requirements, the shape from the results of topology optimization can effectively shorten the design cycle in the conceptual design. Topology optimization under multiple loading conditions is realized in the paper. In the case of considering the frame manufacturing process and the total arrangement, how to carry manufacturability analysis and design a frame for the requirements is worth researching.

The first six modes frequency is gained by modal analysis. The weaknesses of the frame are pointed out through the analysis of the results. These weaknesses are the basis of the topology optimization. The maximization of the 1st mode frequency of the frame is set to the optimization target. And effective results of topology optimization are calculated by iteration. The second design is carried out according to the results of topology optimization. And the modal analysis of the frame generated by the second design is made. The analysis demonstrates that the frame generated by the second design meets the demands for the dynamic characteristics.

6. Acknowledgments

Authors gratefully acknowledge the financial support provided by the National Natural Science Foundation of China (50974105) and the Research Fund for the Doctoral Program of Higher Education of China (20105121110003).

7. References

7.1. Journal Article

[1] Dongshen Lv.A static research of bus frame based on Hyperworks. Machinery design&manufature, **2011**, 3(3). (In Chinese)
[2] Xiangman Zhou. Topology optimization design for CNC gear shaper machine tool bed based on Ansys.Coal mine Machinery, **2010**, 6(31). (In Chinese)
[3] Jian Zhang.Optimal design of some truck frame based on FEA. Machinery design&manufature, **2012**, 5(5). (In Chinese)
[4] Zhang C D,Song S M.Forward Position Analysis of Nearly General Stewart Platforms[J].Sptial Mechanisms and Mechanical.**1992**,45:84-84.
[5] Ibrahin S.R. and Mikulcik E.C.A Method for the Direct Identification of Vibration Parameters from the Free Response,The shock and Vibration Bullein,Vol,4,**1977**.
[6] Runlian Jiang, Yaohua Wang.The Numerical Simulation of Equivalent Static Pressures Distribution for the Aircraft Tire Burst Jets Based on MATLAB. International Journal of Control and Automation, **2014**,7(1) :379-390

7.2.Book

[7] Chen Jiarui, Automobile Construction, China Machine Press, Beijing, **2007**
[8] Xie Longhan, Structural and Dynamic Analysis in Ansys, Publishing House of Electronics Industry, Beijing, **2012**

Conference Proceedings

[9] Lam Tim Fai.FEA Contact Element Technique And Its Application in Metal Lead Forming Failure Analysis[C].Proceedings of 9th IPFA **2002**,561:47-50.
[10] Hongjun Yang,Gangyan Li.Research on Automatically Leveling Control System for Vehicle-borne Platform with High Accuracy[C].Proceedings of the **2006** IEEE／ASME International Conference on Mechatronic and Embedded System Applications(MESA06)：432-436.
[11] Zhang Zhen wei, Chen Chong, Jiao Ruobin, Hu Rongrong. Advanced Materials Research,**2013**, 644:101-104.Publiser:Trans Tech Publications.

Authors

Min Yuan received his master's degree in School of Transportation and Automotive Engineering of Xihua University in 2014. He is currently pursuing a doctor's degree majoring in School of Mechatronic Engineering of Southwest Petroleum University. His research interests include theory of reliability, FEA, structural vibration, and modal analysis.

Peng Zhang received his doctor's degree in mechanics in Harbin Institute of Technology. He completed postdoctoral research in mechanical post-doctoral mobile stations at Harbin institute of technology. Now he is a doctoral supervisor in Southwest Petroleum University. His research interests include failure analysis and reliability of building structure and systems, evaluation of residual strength for oil and gas pipelines, residual life prediction and risk management technology.