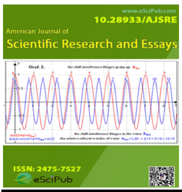
Research Article **AJSRE (2019), 4:19**



American Journal of Scientific Research and Essays (ISSN:2475-7527)

Simulation calculation and research of double layer metal rubber vibration isolator

Shengzhe Jin, Guohe Jiang, Gang Wu, Biwen Chen, Xiangyi Zou

Merchant Marine College, Shanghai Maritime University, Shanghai, 201306, China.

ABSTRACT

Marine diesel units will produce certain vibration and noise at work, which will affect the normal operation of machinery and crew's physical and mental health. It is a common solution to equip diesel units with vibration isolators, so it is very necessary to optimize the design of vibration isolators. Based on diesel engine set with double layer metal rubber vibration isolator, for example, to establish the geometric model of the vibration isolator with Solidworks, and importing it into the finite element software called ANSYS Workbench to make simulation calculation, get the vibration isolator in forward loading, reverse load and radial load displacement nephogram under three working conditions and stress nephogram, get the maximum double metal rubber vibration isolator deformation, the maximum stress value and the location of maximum stress. The results show that the maximum stress of the double layer metal rubber isolator is less than the yield limit of the material, which meets the work requirements. The stress concentration is easy to occur in the center of the bottom base of the vibration isolator, that is, fatigue failure is easy to occur. Under the condition of radial loading, the maximum deformation of the case is large. The maximum deformation and stress of the improved case are significantly reduced, which is of guiding significance to the optimal design of the vibration isolator.

Keywords: Double layer metal rubber vibration isolator; Finite element; ANSYS; Stress concentration; Optimization design

***Correspondence to Author:**
Shengzhe Jin
Merchant Marine College, Shanghai Maritime University, Shanghai, 201306, China.

How to cite this article:
Shengzhe Jin, Guohe Jiang, Gang-Wu, Biwen Chen, Xiangyi Zou. Simulation calculation and research of double layer metal rubber vibration isolator. American Journal of Scientific Research and Essays, 2019 4:19

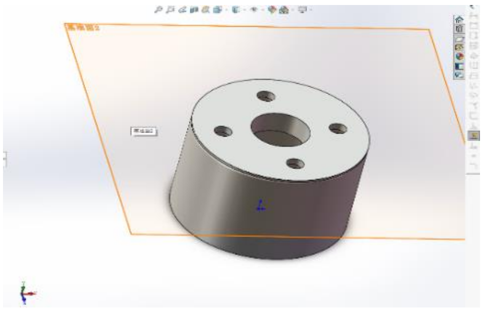
eSciPub LLC, Houston, TX USA.
Website: http://escipub.com/

Vibration is a common phenomenon in life. When a ship sails at medium and low speeds, the vibration and noise generated by the operation of its mechanical equipment will not only reduce the comfort of the ship, affect the crew's work efficiency and health, but also accelerate the ship structure and its machinery. Fatigue damage of equipment, affecting the normal use of equipment on board, and increasing maintenance costs. Therefore, the vibration control technology of marine equipment has gradually become a research hotspot in the field of ship engineering [1]. The rubber vibration isolator is widely used in the field of ship vibration reduction and shock resistance because of its adjustable outer shape structure and stable three-way stiffness [2]. The metal rubber vibration isolator comprises a metal rubber component, which is a friction damping material which is formed by a knitting machine by a knitting machine in a certain form. When the vibration isolator is subjected to the crushing impact, the internal wire generates the slip dry friction and consumes energy, thereby functioning as a buffer for damping. This kind of material has the characteristics of corrosion resistance, high temperature resistance, low temperature resistance and aging resistance. Therefore, it has been widely used in aerospace, weapon equipment and marine diesel engines. It also has practical significance for enhancing the maintenance of equipment and improving the reliability of equipment. [3]

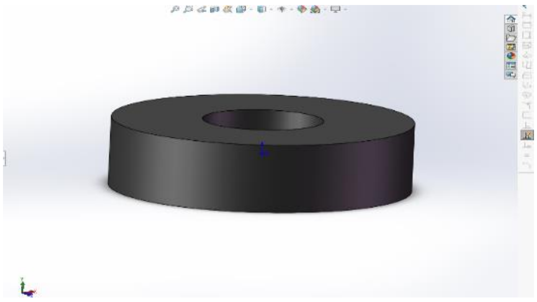
In this paper, using the finite element analysis software ANSYS Workbench, make the finite element analysis of the forward loading, reverse loading and radial loading of the double-layer metal rubber isolators for the diesel engine set of Xinhaie ship is used to accurately describe the stress distribution. It has certain guiding significance for the optimal design of the vibration isolator

1 model overview of double layer metal rubber isolator model overview

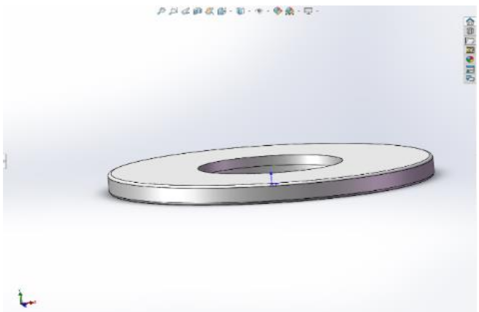
The double-layer metal rubber isolator for the Xinhaie ship diesel engine set consists of a case, a central axis, a metal gasket, two identical wire rubber blocks and a base.（As shown in Figure 1）. Its design parameters are：The housing has a diameter of 60 mm, a height of 40 mm, a wire rubber block outer diameter of 50 mm, an inner diameter of 20 mm, a thickness of 15 mm, a metal gasket outer diameter of 56 mm, an inner diameter of 25 mm, a thickness of 4 mm, and a vibration isolator base diameter of 100 mm. The maximum working load at the top of the housing and at the bottom of the center shaft is 1400N.

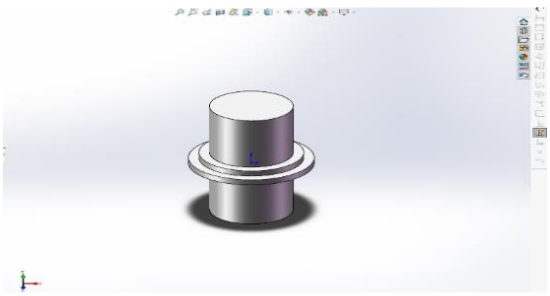


case



identical wire rubber block





metal gasket central axis

base Sectional view

Figure 1. Model overview

2. Finite Element Simulation and Computational Analysis of Double Metal Rubber Isolators

Because the Design Modeler model of ANSYS Workbench is not convenient, this paper uses Solidworks2018 software as a tool to geometrically model the double-layer metal rubber isolators, save the model as an IGS format that ANSYS can recognize, and then import the file into ANSYS Workbench and analysis it.

2.1 Introduction of ANSYS

ANSYS（General-purpose，Finite-element Program for structural，Hea-transfer and Static-eleCtromagnetic Analysis. The software was created by SASI, a world-renowned mechanical analysis expert and professor of the University of Pittsburgh, by J Swanson, and introduced in 1970.

ANSYS can be used for finite element static, dynamic, thermal, linear or nonlinear analysis of 2D or 3D systems. ANSYS performs thermal stress analysis by directly connecting the output of the thermal analysis to the input of the structure. In addition to its powerful analytical capabilities, ANSYS also includes pre-processing, post-processing, and support for a wide range of graphical display facilities. Model and load generation pre-processing and its inspection are included in ANSYS. Both the mesh and the external load are automatically generated and can be verified by a graphical display. Each command has a master file. ANSYS post-processing supports many graphics functions-hidden lines, cross sections, XY-drawings, distortion shapes, and contour maps of stress and temperature. ANSYS post-processing also includes a database language so you can selectively check the results. And ANSYS can interface with many CAD systems [5].

2.2 Definition of material properties

We chose the Static Structural Analysis System for static analysis. After importing the model into the Geometry module, define the properties of the material in the Engineering Data module. Metal rubber is a simulated product of natural rubber. The metal rubber component is made of metal wire and does not contain any natural rubber, but has the same elasticity and porosity as natural rubber. Its internal mesh is similar to the macromolecular structure of natural rubber. The metal rubber material is also a nonlinear dry friction damping material with good deformation self-recovery ability. It has a memoryless resilience with good three-dimensional nonlinear components. In addition, it has excellent deformation self-recovery ability, ie memory recovery force. The recovery ability of metal rubber consuming device has nonlinear hysteresis effect, and its damping component has viscous damping. There is also dry friction damping [6]. The material of the wire rubber block is not in the material library. We need to redefine it. The elastic modulus is 20GPa and the Poisson's ratio is 0.4. The vibration isolating shell, the middle shaft and the base are all made of structure. Steel, which can be directly selected in the material library, has a density of 7.85g/(cm)^3, an elastic modulus of 210GPa, and a Poisson's ratio of 0.3. The yield strength is 250 MPa and the allowable force is 86 MPa.

2.3 Meshing

Once the geometry model has been created, it must be meshed for finite element analysis. This step is an important part of finite element processing. The quality of the meshing affects the accuracy and speed of the analysis. In general, the more nodes and units, the better it is to improve the calculation accuracy, but it also increases the CPU time and storage space of the calculation. Therefore, in the case of ensuring that the original structural geometry and mechanical properties and calculation accuracy are guaranteed, we must minimize the number of unit nodes and reduce the computational workload [7]. In this paper, the method of automatic meshing is adopted. The Relevance is 100, and the Relevance Center selects the Fine Fine option. The Element Size is set to 0.5 mm to generate a mesh. as shown in picture 2.

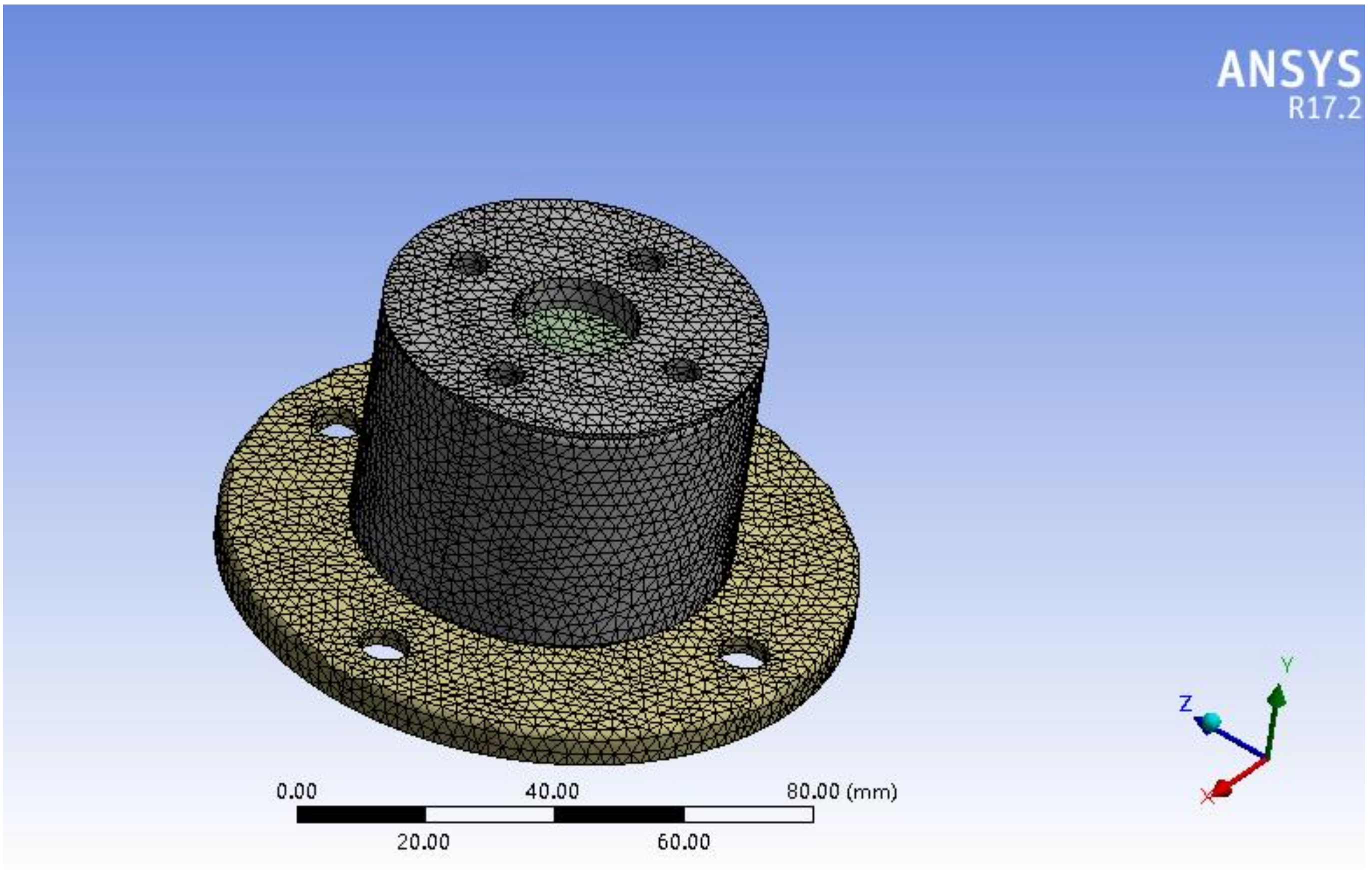
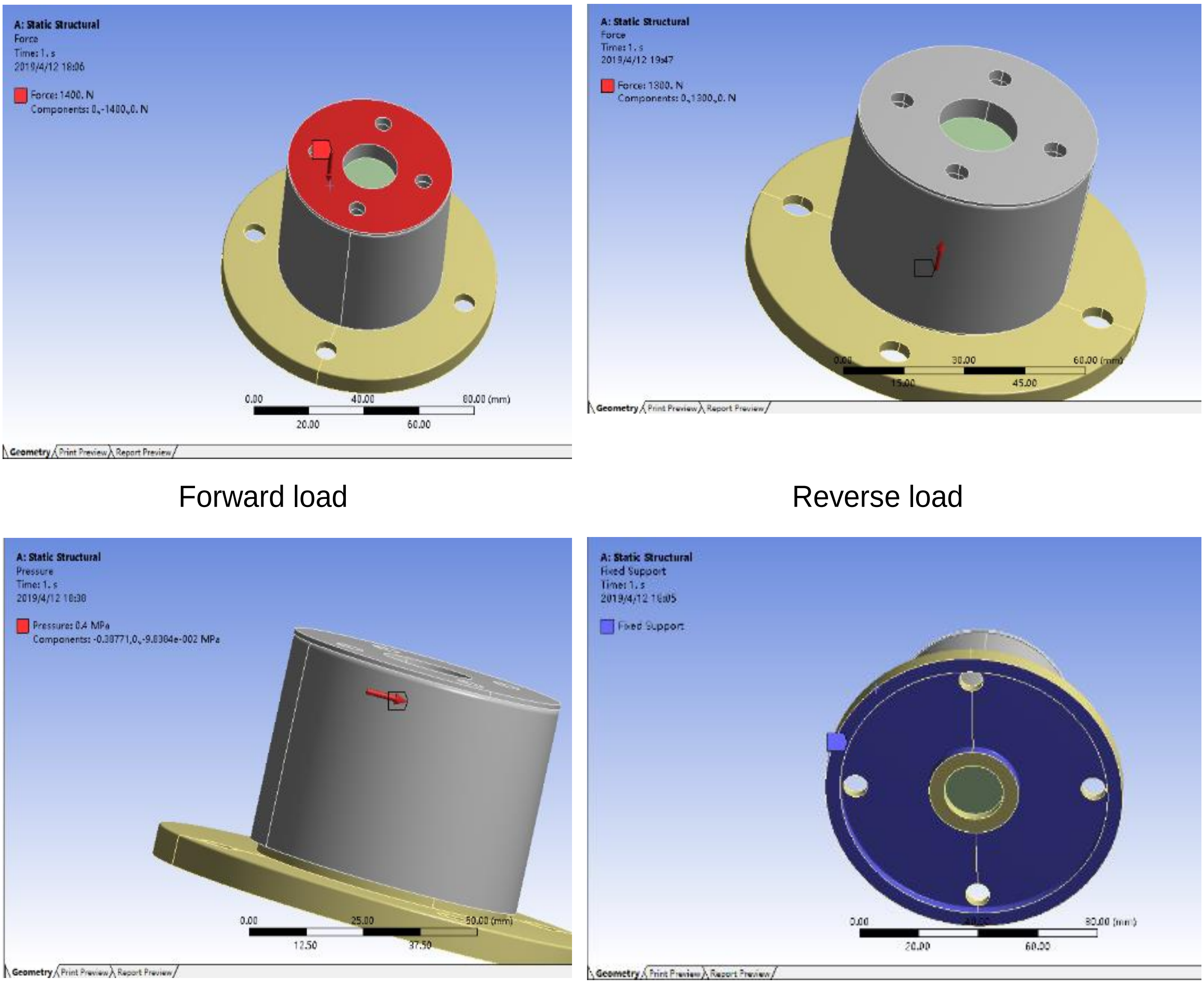


Figure 2. Meshing

2.4 Applying constraints and loads

The main purpose of finite element analysis is to examine the response of a structure or component under certain load conditions. Therefore, specifying a reasonable load in the analysis is a critical step [5].

In this paper, the three-layer metal rubber isolators are subjected to three loading analysis: the first is to apply a forward load to the top of the isolator casing, the size is 1400N; the second is to apply a reverse load to the bottom of the central axis, the size is 1300N; the third time is to apply a radial load to the center shaft, the size is 0.4MPa. Because the base of the isolator is fixed, a fixed constraint is applied to the bottom of the isolator base when the load is applied three times. The load and constraints are shown in Figure 3.



Forward load

Reverse load

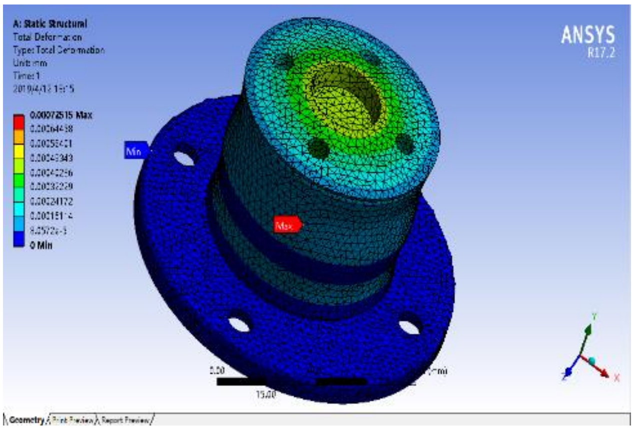
Radial load

Bottom constraint

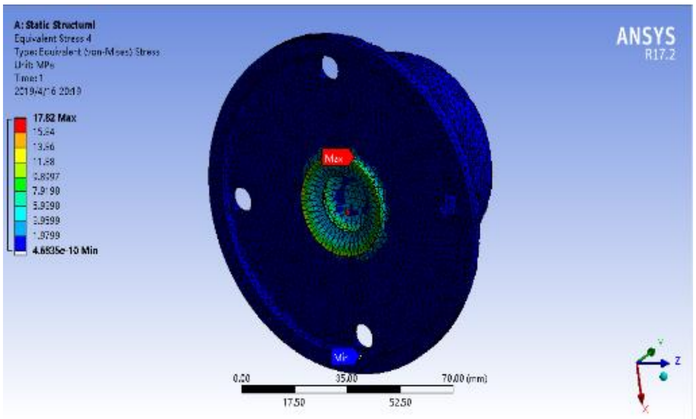
Figure 3. Load and constraint

2.5 Calculation results and analysis

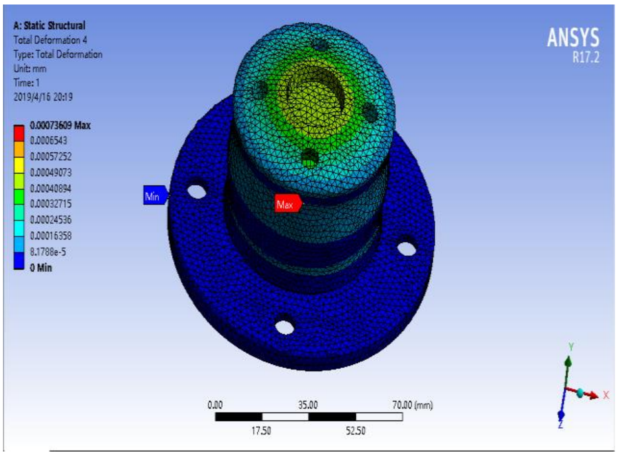
After applying the load and the constraint three times, the total deformation and the equivalent stress are selected as the calculation results, and the solution is performed three times in sequence to obtain the displacement cloud map and the stress cloud map in three cases, as shown in Figure 4.



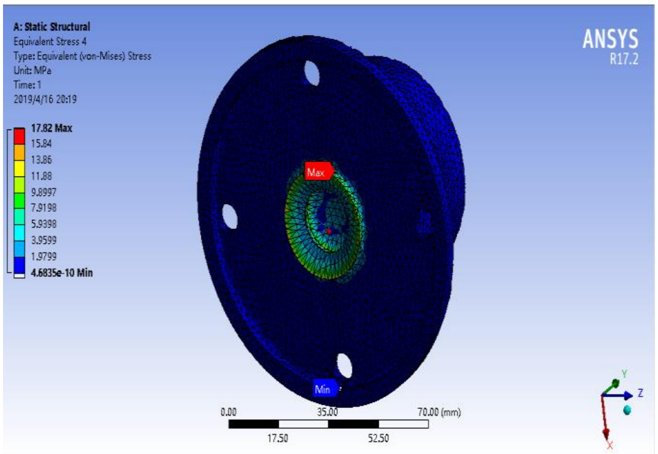
Displacement nephogram of forward load



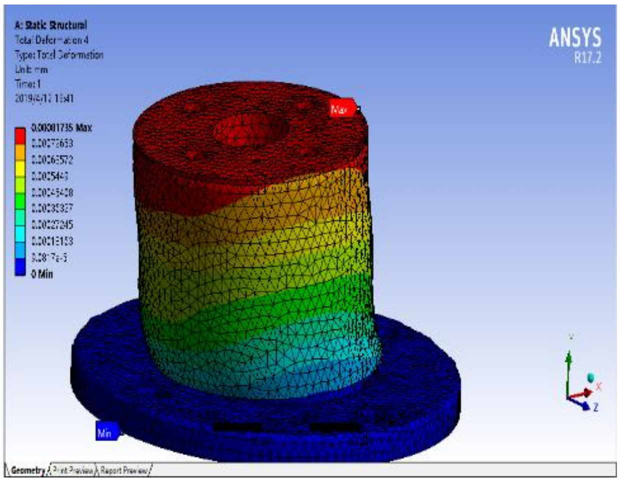
Stress nephogram of forward load



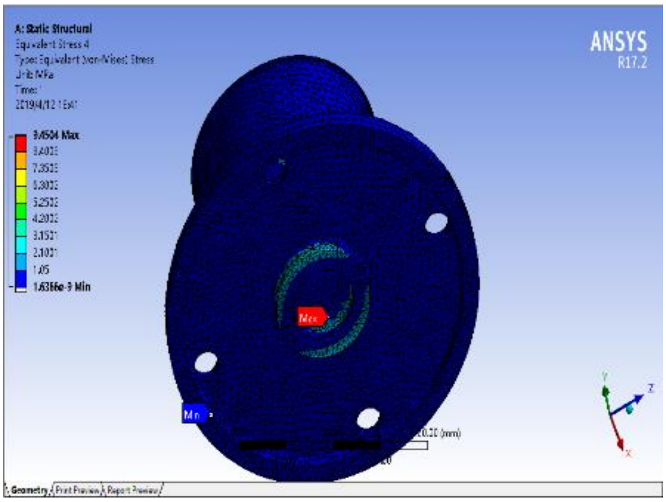
Displacement nephogram of reverse load



Stress nephogram of reverse load



Displacement nephogram of radial load



Stress nephogram of radial load

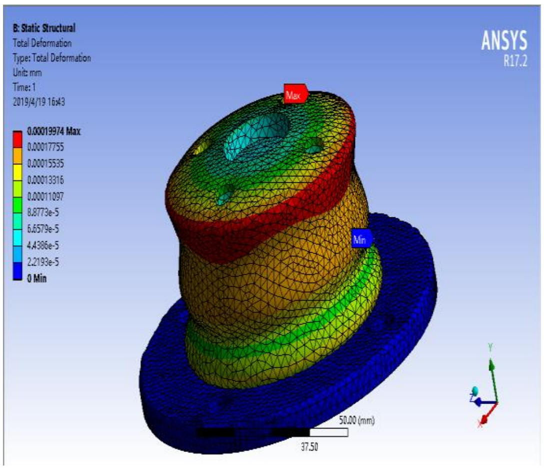
Figure 4. Nephogram of displacement and stress

According to the calculation results obtained in Fig. 4, the maximum deformation and maximum stress of the double-layer metal rubber isolator and the positions where the maximum deformation and the maximum stress occur are analyzed, as shown in Table 1.

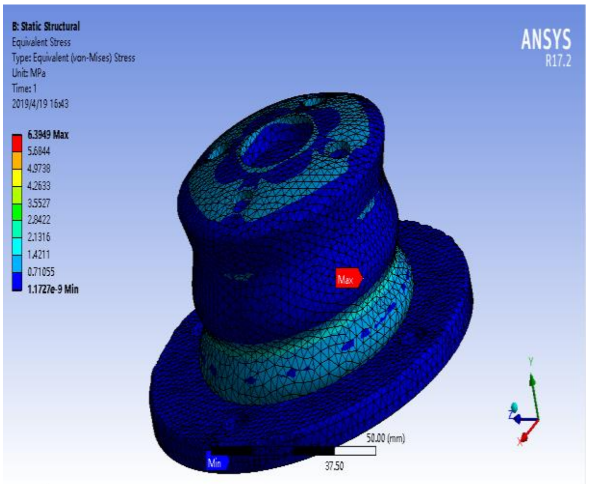
Table 1. Analysis result

Type of load	Max-def/mm	Max-str /MPa	Position of max deformation	Position of max stress
Forward load	7.25×10^{-4}	17.38	Central of the base	Central of the base
Reverse load	7.36×10^{-4}	17.82	Central of the base	Central of the base
Radial load	8.17×10^{-4}	9.45	Edge of the top case	Central of the base

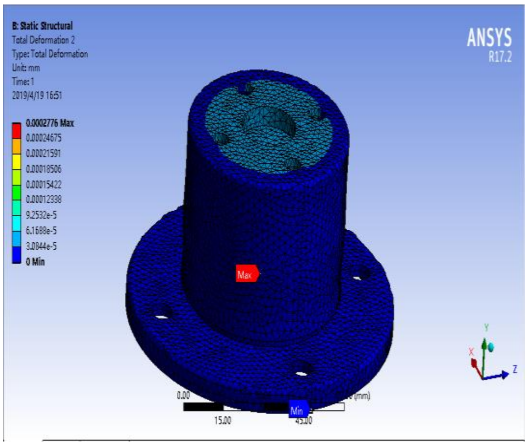
As can be seen from the contents of Figure 4 and Table 1, the vibration isolator base is subject to fixed constraints, total deformation is nearly 0 mm. Under forward, Since the inside of the metal rubber component is a spatial network formed by the intertwining of the wires, the maximum stress is large under the action of the forward load; Under the reverse load, since the radius of the bottom surface of the middle shaft is small, the force area is small, so the maximum stress is large; Under radial load, the top edge has a large amount of deformation due to the lack of chamfering or rounding at the top edge of the housing. The maximum stress under three working conditions is 17.82 MPa, which is less than the allowable force of the material 86 MPa, which is within the safe range; The maximum stress in all three working conditions occurs in the center of the base of the vibration isolator, that is, the center of the base is prone to stress concentration, which is prone to fatigue failure, which basically conforms to the actual engineering application.



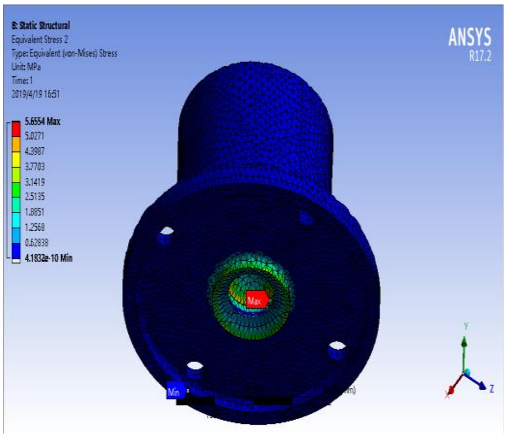
Displacement nephogram of forward load



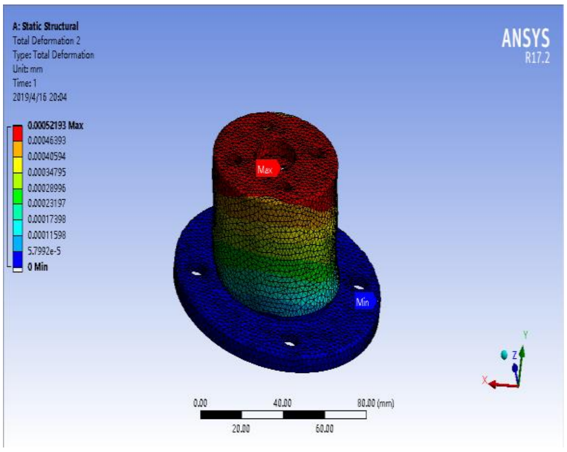
Stress nephogram of forward load



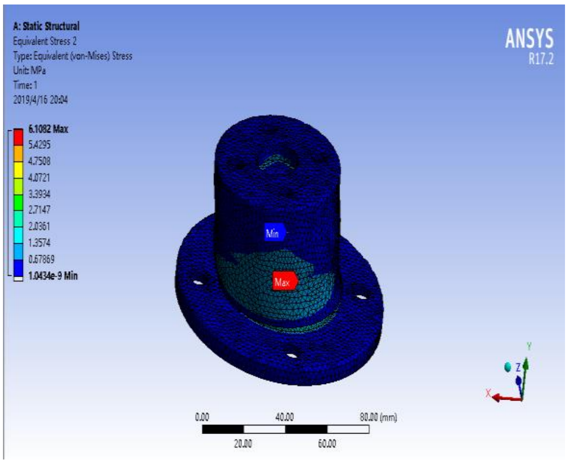
Displacement nephogram of reverse load



Stress nephogram of reverse load



Displacement nephogram of radial load



Stress nephogram of radial load

Figure 5. Improved nephogram of displacement and stress

After the top edge portion of the casing is rounded, a forward load, a reverse load, and a radial load are applied. The total deformation and maximum stress improvement under three conditions are shown in Fig. 5.

From the calculation results obtained in Fig. 5, the improved maximum deformation and maximum stress and the positions of maximum deformation and maximum stress are analyzed, as shown in Table 2.

Table 2

Type of load	Maximum deformation /mm	Maximum stress /MPa	Position of maximum deformation	Position of maximum stress
Forward load	1.99×10^{-4}	6.39	Edge of the top case	Central of the base
Reverse load	2.78×10^{-4}	5.66	Central of the base	Central of the base
Radial load	5.22×10^{-4}	6.11	Center of the top of the case	Central of the base

As can be seen from the contents of Figure 4 and Table 1, the vibration isolator base is subject to fixed constraints, total deformation is nearly 0 mm. The maximum deformation and maximum stress under three conditions are greatly reduced. The maximum stress under the working condition is 6.39 MPa, which is still less than the allowable force of the material 86 MPa, which is within the safe range; The maximum stress in all three conditions occurs in the center of the base of the vibration isolator, that is, the stress concentration is easy to occur in the center of the base, but the value has been greatly reduced, and the fatigue failure phenomenon can be basically avoided.

3. Summary

Based on the Static Structural Analysis System of ANSYS Workbench software, this paper analyzes the structural statics of the double-layer metal rubber isolator used in the Xinhaie ship's diesel engine set, and obtains the total deformation and stress distribution of the isolator. The results show that the strength of the double-layer metal rubber vibration isolator can meet the working requirements, the maximum stress of the material is less than the allowable force, meet the design requirements, and the maximum deformation and maximum stress under three working conditions after the rounding of the top edge of the case Both have been greatly reduced, providing valuable data information and a certain guiding significance for optimizing the design of the vibration isolator.

REFRENCES：

1. Zou Xiangyi, Jiang Guohe, Sun Xiucheng, Zhang Chunlei.Characteristics Analysis and Optimi-

zation of Vibration Isolation for Diesel Engine System with FEM[J].Navigation Of China,2015, 38(01):14-17+27.

2. Liu Yong, Cao Jinyi, Lv Zhiqiang, Yang Xue.Finite Element Analysis of Polyurethane Isolator Used in Diesel Engine Set[J].Chinese Journal Of Applied Mechanics,2018,35(01):161-165+234-235.
3. Yang Kunpeng, Fan Wenxin, Cao Cuncun, Sang Manxian, She Yong.Dynamic Mechanical Modeling and Parameter Identification of Metal-rubber Materials[J].Mechanical Science and Technology for Aerospace Engineering,2017,36(12):1830-1833.
4. Wang Shulu, Bai Hongbai, Lu Chunhong, Lu Haixing.Finite Element Analysis of Metal Rubber Vibration Isolator Used in Diesel Engine Set Based On ANSYS Workbench [J].Journal of Ordnance Engineering College,2015,27(05):75-78.
5. Sun Liping. Finite Element Analysis of Ship Structure [M].Harbin：Harbin Engineering University Press,2013
6. Wang Yajie, Zhao Yagebai, Ren Xiaolong, Huang Xinyu, Wang Xuan, Han Jingyi.Study on the Constitutive Relationship of Metal Rubber[J].ShanXi Architecture,2019,45(04):29-31.
7. Gao Yaodong, Su Fucun, Li Zhen, Yin Ming.30 Examples of Mechanical Engineering Applications Based on ANSYS Workbench [M].Beijing：Publishing House of Electronics Industry，2013：56