ISSN 2616-5767 Vol.6, Issue 12: 51-55, DOI: 10.25236/AJETS.2023.061208

Research on Finite Element Simulation of the ROV Electronic Cabin Structure Based on ANSYS Workbench

Jikuan Wang[1,a,*], Lianhai Wang[1,b], Yufan Chen[1,c]

[1]*School of Shipping, Shandong Jiaotong University, Weihai, 264200, China*
[a]*2861844943@qq.com,* [b]*wanglianhai@163.com,* [c]*782450545@qq.com*
[*]*Corresponding author*

Abstract: *With the continuous development of the marine economy, the status of electronic cabins in underwater vehicle equipment is ineffable. Firstly, the three-dimensional model of the electronic cabin was established using SolidWorks software, and then the stability and working stress of the pressurized shell of the electronic cabin were simulated by ANSYS software, and it was proved that the electronic cabin fully met the safety requirements for working at a depth of 300m.*

Keywords: *Ocean economy; Electronic cabin; Underwater robots; Structural design; Finite element analysis*

1. Introduction

As human exploration of resources shifts to the ocean, ocean exploration becomes deeper and smarter. With its advantages of low cost, small size, low power consumption, high efficiency and high reliability, underwater robots have become an important tool for human exploration and understanding of the ocean. As one of the key components of underwater exploration equipment, the structural safety and stability of the electronic cabin are essential and an indispensable component.

Jiang Shuguang et al [1] proposed a new design concept and verification method in this paper, which provided a guiding direction for the preliminary research of electronic cabins. Sun Jianxiang et al research on pressure-resistant electronic cabins mainly focused on structural innovation, and proposed a new structural design; Li Jun et al. proposed a more convenient analysis and calculation method in view of the complex calculation of the elastoplastic stability analysis of the pressure-resistant chamber of underwater vehicles. In addition, Su Changqing [2] studied the relationship between the diving depth of the ellipsoidal underwater vehicle and the main scale ratio and weight-to-volume ratio during the design process, which ensured the strength and pressure resistance of the material. Gao Huihui[3] In view of the geometry, material characteristics and external pressure load of the shell, the commonly used numerical method - the Ritz method was used to analyze the buckling analysis of the medium-thickness rotary shell under uniform external pressure.

In summary, ANSYS Workbench has been widely researched and applied in engineering simulation in many industries, and a large number of relevant prediction studies and experimental data show that ANSYS finite element simulation has significant advantages in efficiency and cost compared with traditional statistical and metrology-based methods, and ANSYS parameter tuning optimization in the prediction process is also the focus of determining the prediction results. Therefore, ANSYS Workbench is used to conduct static simulation and modal analysis of the pressure-resistant chamber to prove the safety and stability of the designed pressure-resistant electronic chamber.

2. Research content

As an important carrier of various core components inside the underwater robot, the pressure-resistant cabin is not only the main pressure-bearing component of the entire underwater robot, but also an important guarantee for the normal operation of the underwater robot. When designing the pressure chamber, the stability conditions are first calculated, and the strength, stability and sealing of the underwater robot under the test water pressure are guaranteed to reduce the cost under the premise of meeting the strength and stability of the pressure chamber.

ISSN 2616-5767 Vol.6, Issue 12: 51-55, DOI: 10.25236/AJETS.2023.061208

In this paper, the structural form and basic parameters are determined according to the actual working conditions and lightweight principle, and the finite element simulation software ANSYS Workbench is used to carry out static simulation and modal analysis of the designed pressure-resistant chamber. The study used Solidworks software to build a 3D model, saved it to txt format, and imported it into ANSYS Workbench for pre-processing. In the static structure module of ANSYS Workbench, first ensure the reasonable selection of boundary conditions and the correctness of each parameter, and then click solve to solve the data [4]. Finally, the results were observed and analyzed by ANSYS Workbench software. Ensure the safety and stability of the electronic cabin under working conditions, and provide a theoretical basis for actual processing and manufacturing.

3. Numerical simulation analysis of shallow water ROV electronic cabin structure

3.1. Relevant technical requirements

According to the technical requirements, choose the 7075-T6 model in the aluminum alloy. Since the designed electronic cabin needs to meet the working water depth of 300m, the material needs to withstand an external load pressure of 3.029 MPa. These parameters will be used in further analysis and simulations to determine the material behavior and structural response.

The parameter selection of aluminum alloys in this study is shown in Table 1.

Table 1: 7075 aluminum alloy mechanical performance parameter table

model	Model yield strength(MPa)	Elastic modulus(GPa)	density(kg/m3)	Poisson's ratio	Allowable stress (MPa)
7075	505	71.7	2810	0.33	337

The shape and structure of the pressure-resistant shell of the electronic cabin can be divided into two forms: cylindrical shell and spherical shell. Under normal circumstances, the choice of its shape is mainly determined by the working water depth, the different working water depth, resulting in different pressures, according to experience and convention, to 800 m depth as the limit, more than 800 m to use spherical shell; When the working depth is below 800 m, a cylindrical housing is used[5].

In this paper, the underwater robot has a working limit water depth of 300 meters, and a cylindrical shell is selected.

3.2. ANSYS Workbench electronic cabin numerical simulation steps

In numerical simulation, there are usually two steps: pre-processing and post-processing. The pre-processing steps mainly include importing the geometry of the mold, performing geometry cleaning to ensure model integrity, meshing to generate a suitable mesh, defining appropriate boundary conditions to simulate real-world operating conditions, and finally using ANSYS Workbench to complete the solution [6].

3.2.1. Establishment of finite element model

SolidWorks software was used to create a three-dimensional model of the electronic cabin, with a cylindrical outer shape and an outer and 142 mm inner diameter of 150 m and an inner diameter of 360 mm. These geometric parameters will be used for structural analysis and calculations in subsequent simulations. This is shown in Figure 1

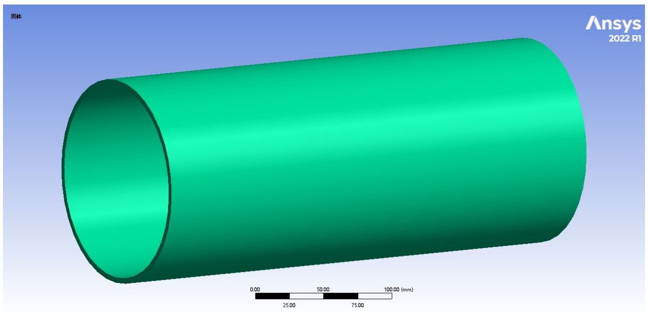

Figure 1: Electronic cabin structure diagram

3.2.2. Geometry cleanup

When the geometry generated by the SolidWorks software is imported into ANSYS Workbench software, some topological relationship defects may occur due to software versions and actual simulated operational technical problems, such as missing surfaces, incomplete connections of lines or surfaces, etc. If not fixed, these defects can affect the quality of meshing [7] and even prevent successful 3D meshing. Therefore, in order to ensure better simplification of geometric surfaces in the simulation area, good quality and continuity in the meshing process are guaranteed. Geometry cleanup is a very important step in pre-processing and deserves attention.

3.2.3 Meshing

When meshing the material used in the stress process, all areas through which the material flows, first use the Model feature of ANSYS Workbench software to automatically mesh. Triangles are selected for the shape of the mesh, and the mesh size is determined according to the characteristics of the model. Precision grids can show more accurate results, and correspondingly higher requirements for computer CPU processing power. To balance computer configuration and analysis accuracy, we employ different sizes and densities in the 2D mesh as needed. Generally, the grid of the main stress zone area is generally the smallest value, because it has the greatest impact on the safety support of the structure, and the meshing of non-critical positions is adjusted according to the computer configuration and actual working conditions [8].

3.2.4 Define boundary conditions

First, select the static structure analysis module and import the TXT file, that is, you can set the boundary conditions. The software will automatically generate the default boundary conditions, change and select more appropriate boundary conditions for different technical requirements [9], and set mesh encryption in key places of the model to facilitate subsequent observation of force deformation results.

3.2.5 ANSYS Workbench solved

ANSYS Workbench research uses Solidworks software for 3D model building, which is saved to txt format and imported into ANSYS Workbench for pre-processing. In the static structure module of ANSYS Workbench, first ensure the reasonable selection of boundary conditions and the correctness of each parameter, and then click solve to solve the data[10]. Finally, the results were observed and analyzed by ANSYS Workbench software. The solution speed is mainly determined by the computer's central processing unit.

Post-processing includes not only the analysis of the results of the simulation analysis, but also the optimization of the product design based on the analysis of the results. Finite element post-processing provides critical information about structural behavior, performance, and optimization direction to guide further design optimization efforts.

4. Analysis of the results

4.1. Strength analysis

First, use Solidworks software for 3D model building, save the model as txt format when completed, then open ANSYS Workbench, click the second section to change the engineering data, and import the model into the geometry for pre-processing. In terms of force settings, fixed constraints are applied at both ends of the model to simulate real-world conditions. At the same time, according to the pressure corresponding to the working depth, according to the requirements of the German Lloyd's Register specification, take the safety factor as 2, calculate the pressure p= 3.029×2= 6.058 MPa, and apply pressure to the outer surface of the shell.

As can be observed in Figure 2, the maximum displacement of the electronic chamber housing is shown to be 0.109 mm under external load, which is exactly the design requirements compared to the wall thickness of 4 mm. Figure 2 shows that the maximum stress of the chamber enclosure is 132.24 MPa, and the yield stress in the material table is 275 MPa, and according to the $\sigma_{max}<\sigma_{0.2}$ criterion, the enclosure meets the strength requirements.

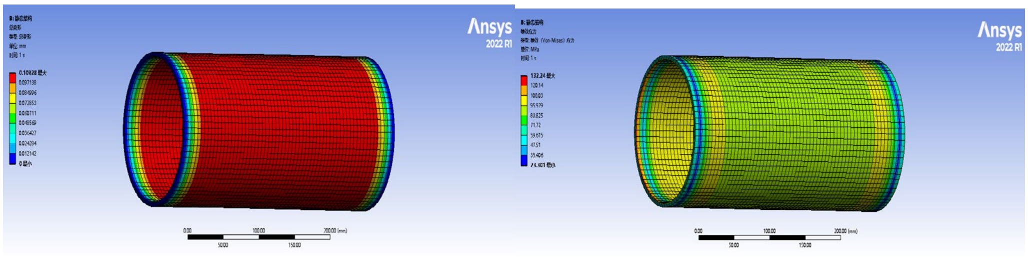

Figure 2: Structural diagram force diagram

4.2. Stability analysis

While ensuring that the strength of the pressure-resistant shell meets the requirements, it is also necessary to pay attention to the stability of the shell. Because in some cases, the shell may become elastically unstable.

During the calculation, the applied load is the calculated pressure of 6.058 MPa. After the eigenvalue buckling analysis, the load coefficient obtained is 2.897, and the theoretical check value pcr = 17.82 MPa. By calculating, the minimum elastic critical pressure is 6.058 * 2.897 = 17.55 MPa, and the two values are very close, that is, the accuracy of the check is verified. As can be observed in Figure 3, when the critical load is reached, the cylindrical pressure-resistant shell has obvious buckling phenomenon, resulting in wrinkles, and the critical value of displacement during instability is 1.032 mm, which is completely in line with the stability requirements of the pressure-resistant shell relative to the wall thickness of the structure.

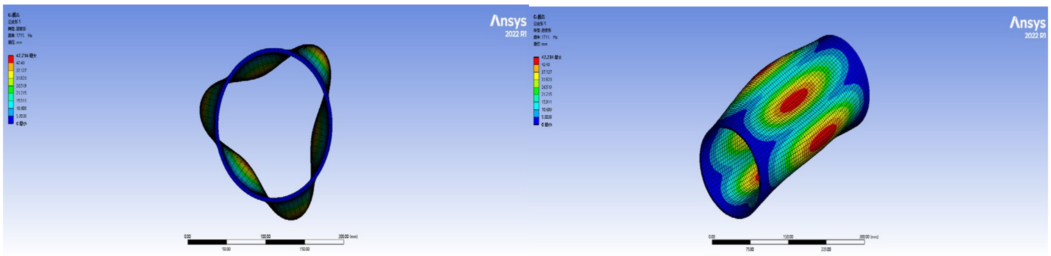

Figure 3: Structure diagram of a framed underwater vehicle

5. Conclusion

After determining the structural form and basic parameters according to the actual working conditions and lightweight principle, this paper uses the finite element simulation software ANSYS Workbench to conduct static simulation and modal analysis of the designed pressure-resistant cabin, and obtains the following conclusions:

①The cylindrical main structure and high-strength 7075- T6 aluminum alloy material not only greatly reduce the underwater flight resistance to the current, but also meet the strength requirements, lightweight design and excellent processing performance.

②Through ANSYS finite element analysis, we ensure that the stress deformation and instability are within a controllable range, meet the stability and strength requirements of the pressure-resistant shell, and provide a theoretical and data basis for the actual processing of the subsequent electronic cabin.

References

[1] Z.W. Bian, P.S. Zhong and J.F. Zhang: Structure Design and CFD Analysis of Flapping Wing Gliding Underwater Robot, Modular Machine Tools and Automatic Processing Technology, Vol. (2017) No.06, p.62-65.

[2] W. Liu, Y.C. Zhou, L. Zhang, et al: Structure optimization of pressure hull for Flying Fish underwater vehicle, Science and Technology Magazine, Vol. (2021) No.07, p.7-8.

[3] H. C. Wang, Q. G. Zhang, H.J Zhao, et al: Structure optimization of pressure hull for Flying Fish underwater vehicle, Low-carbon World, Vol. (2016) No.15, p.261-262.

ISSN 2616-5767 Vol.6, Issue 12: 51-55, DOI: 10.25236/AJETS.2023.061208

[4] X.B. Jin: Zhang: Design of a microminiature underwater robot body structure, Mechanical Engineer, Vol. (2013) No.11, p.47-49.
[5] X. Q. Zhai, J.J. Zhang, and H.R. Wang, et al: Structure design and research of underwater floating crawling robot, Manufacturing Automation, Vol. 43 (2021) No.05, p.42-45+55.
[6] S. J. Jian, C.C. Xhong and P. F. Shang, et al: Structure design and hydrodynamics analysis of an underwater fishing robot, Manufacturing Automation, Vol. 42 (2020) No.05, p.61-66.
[7] J. Tang, S.L. Yang and Z. Qin: Structure Design and Parameter Optimization of Ballast Tank for Underwater Vehicle , Journal of Jiangxi University of Science and Technology, Vol. 41 (2020) No.05, p.88-95.
[8] Y. Gan: Research on Motion Control System Architecture of underwater vehicle (MS, Journal of Harbin Engineering University, China 2007).
[9] C.Z. Wang: Overall Structure Design and Optimization of Autonomous Underwater Vehicle (MS, Jiangxi University of Science and Technology, China 2022).
[10] Q.L. Shen: Structure Design and Analysis of Cabled Underwater Vehicle (MS, Hanzhou Electronic Science and Technology University, China 2017).