

Simulation Analysis of a Cover System Based on a Parametric Modeling Approach

Zhanhexiang Zhang^{1,2}

¹713th Research Institute of China State Shipbuilding Corporation Limited, Zhengzhou, Henan, China

²Henan Key Laboratory of Underwater Intelligent Equipment, Zhengzhou, Henan, China

Abstract: The cover opening system is a key component of underwater equipment, its structural reliability is closely related to the stability and safety of the internal environment of underwater devices. During engineering design, finite element models often need to be modified repeatedly as geometric dimensions change. For a system with multiple components and complex contact relationships, such modifications usually require manual reconstruction of geometry, assembly constraints, interaction definitions and boundary conditions, which reduces modelling efficiency and increases the possibility of operational errors. To improve the efficiency of structural iteration, a Python-based parametric finite element modelling approach is developed for a cover system. According to the geometric characteristics and load-bearing paths of the cover, hinge, supports, and bolts, six key design parameters are defined and embedded into the modelling scripts. Secondary geometric features are automatically calculated through geometric relationships, enabling the model configuration to be updated consistently when input parameters are changed. Using the Abaqus scripting interface, the proposed approach realizes automatic part generation, assembly positioning, contact definition, dynamic pressure loading, boundary condition assignment, and finite element simulation. A physical model is used as a reference case to verify the applicability of the method. The simulated stress response fluctuates with the pressure load, with high-stress regions mainly concentrated at the cover plate lug transition area and the hinge-lever connection. The maximum hinge stress approaches the material yield strength, indicating a limited strength margin, whereas the base and bolts remain

at relatively low stress levels. The proposed approach reduces repetitive modelling operations and provides an effective approach for the structural design and optimization of cover systems.

Keywords: Parametric Modeling; Finite Element Analysis; Abaqus; Python

1. Introduction

The cover system is a critical component of underwater operating equipment and plays an important role in maintaining the internal environmental stability of underwater devices. It operates under harsh working conditions, and the performance of each component directly affects the overall reliability. Therefore, mechanical performance evaluation and structural design of the cover system have clear engineering value.

However, engineering design often involves multiple iterations. The cover system contains numerous components with complex connection relationships. Once geometric dimensions are adjusted, the simulation setup needs to be reconfigured accordingly. This not only increases the repetitive workload but also raises the risk of operational errors, thereby limiting the efficiency of dimensional design. To address this problem, this paper proposes an automated approach for parametric finite element modeling and simulation analysis. The proposed method can significantly reduce repetitive work during model iteration and improve design efficiency and accuracy. It can therefore support more efficient design and optimization of the cover system.

2. Design Parameter System

Design parameters form the foundation of the parametric modeling method. Based on the geometric configuration of each component, key geometric dimensions of the design targets

are selected as design parameters and embedded into the parametric modeling program. By adjusting these parameters, the geometric configuration of the corresponding component can be modified, and the subsequent simulation settings can be updated automatically.

The main idea in constructing this design parameter system is to focus on the primary load-bearing components. The inner surface of the cover is the direct action surface of the impact load at the opening, and its area directly affects the overall load borne by the structure, as shown in Figure 1. Therefore, parameters related to the surface area (radius, arch height, and cover thickness) are selected as design parameters.

The spacing between the two connecting supports on the outer surface of the cover affects the load-bearing condition of the hinge and also controls the inner diameter of the hinge. The position of the rear support is considered in the same way. Therefore, the support spacing is included as a design parameter, as shown in Figure 2.

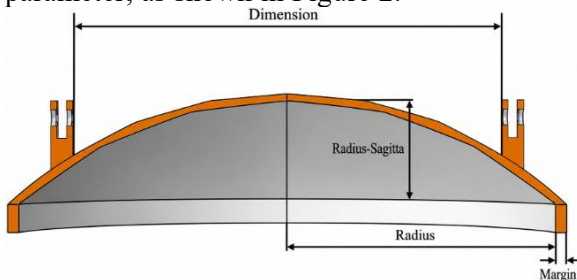


Figure 1. Schematic Diagram of Cover Design Parameters

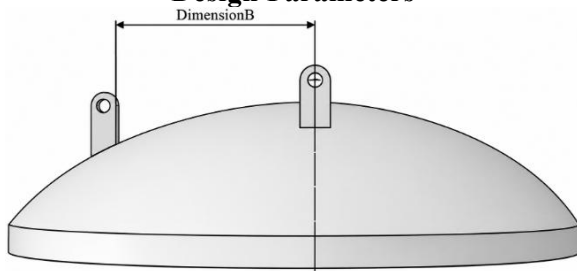


Figure 2. Schematic Diagram of Rear Support Design Parameters

Within the constructed parameter system, the design parameters are not completely independent; rather, they exhibit certain geometric associations. Establishing a reasonable parameter progression logic can reduce parameter redundancy and facilitate the adjustment of component assembly relationships. For instance, the ordinate of the mounting position of the connecting support is

no longer defined by an independent parameter. Instead, it is treated as a secondary parameter, calculated from the main dimensions of the cover via geometric relationships. In this way, the support position updates automatically when the relevant design parameters are modified.

Let the ordinate and abscissa of the connecting support be γ and χ , as shown in Figure 3. Based on geometric relationships, the following is derived:

$$\chi = \frac{Dimension}{2} \tag{2}$$

$$R = Radius + Margin \tag{3}$$

$$(y + Sagitta)^2 + \chi^2 = R^2 \tag{3}$$

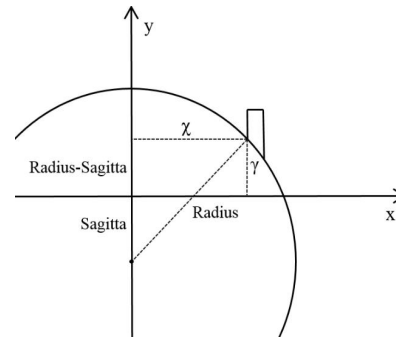


Figure 3. Schematic Diagram of Support Coordinate Positions

Thus, by substituting the calculation formulas for the abscissa and ordinate of the connecting support into the parametric modeling program, the coordinates of the support can be parameterized, eliminating the need to define it as an independent design parameter.

The design parameters of the hinge are shown in Figure 4. The straight segment length of the hinge (HingeLength) determines the equivalent moment arm when the pressure field load is transmitted from the cover to the vicinity of the hinge rotation axis; it thus has a considerable influence on the structural load-bearing capacity. The arc radius of the hinge can be calculated from the support mounting spacing parameter and does not require an additional independent parameter.

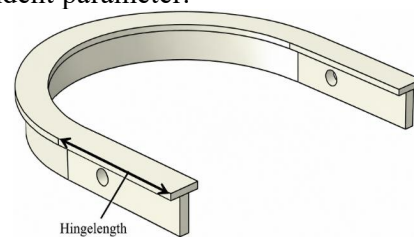


Figure 4. Schematic Diagram of Hinge Design Parameters

Table 1. Parameters of the Cover System

Model parameter	Symbol
Cover arch height	Sagitta
Cover thickness	Margin
Cover support mounting position	Dimension
Cover inner diameter	Radius
Hinge body length	Hingelength
Rear cover support mounting position	DimensionB

In summary, focusing on the key design objects in the project (the cover and the hinge) and based on the simplified finite element model of the cover system, this paper defines six design parameters, as listed in Table 1.

3. Parametric Modeling

In the geometric modeling stage, the parametric modeling script first calculates the coordinates of key control points that define the profile, based on the design parameters. These control points are then used as references, and the corresponding feature commands are applied to connect them. After trimming, a closed cross-sectional profile is formed. Three-dimensional solid bodies are subsequently generated through operations such as revolving and extruding. Part of the code is presented below Algorithm 1:

Algorithm 1. Geometric modelling (excerpt)

```
# Control point coordinates
s1.Spot(point=(0.0, -Sagitta))
s1.Spot(point=(0.0, Radius-Sagitta))

# Generate arc segments
s1.ArcByCenterEnds(center=(0.0, -Sagitta),
point1=(0.0, Radius-Sagitta), point2=(Radius, -Sagitta), direction=CLOCKWISE)

# Trim and connect segments
s1.autoTrimCurve(curve1=g[6],
point1=(Radius+Margin, -Sagitta))
s1.Line(point1=(0.0, Radius-Sagitta+Margin),
point2=(0.0, Radius-Sagitta))

p.BaseSolidRevolve(sketch=s1, angle=360.0)
```

Similar to interactive point-and-click operations, parametric modeling programs also require corresponding identification methods to pick targets. When creating a sketch, the keywords in the program segment involve four types of elements: geometry entities, vertices, dimensions, and constraints. Each time a new geometric feature is generated, the software automatically assigns it a number, which

serves as an identifier for positioning in subsequent feature operations.

The cross-section of the cover body is a shape cut from an offset circle. It should be noted that a vertical construction line and a horizontal construction line are automatically generated during sketch creation, numbered 0 and 1 respectively. Subsequent geometric curves are numbered starting from 2. As shown in Figure 5 and Figure 6, the first created vertical reference line is numbered 2, and the next created horizontal construction line is numbered 3. When a later generated curve passes through the construction lines, it is split into two segments. At this point, if a trim operation is performed, the entity sequence changes according to the position of the trim point: the original curve No. 4, after trimming, has its remaining segment renumbered as 5.

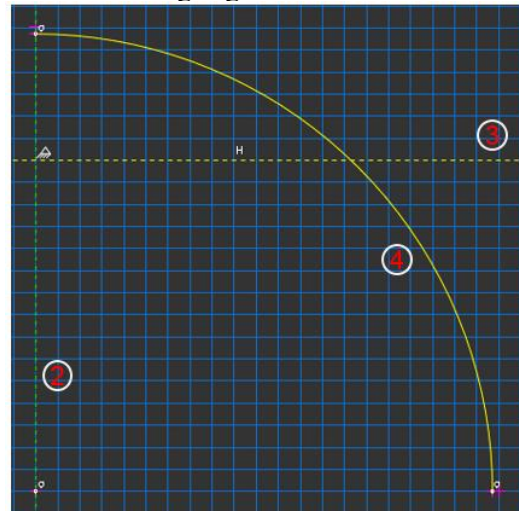


Figure 5. Schematic Diagram of Geometric Sequence Number Change (a)

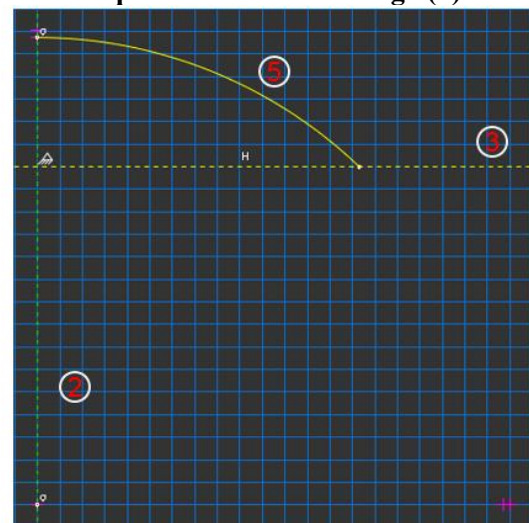


Figure 6. Schematic Diagram of Geometric Sequence Number Change (b)

In the assembly stage, this paper takes the hinge as the main body and assembles the components in a top-down sequence. After importing the components into the assembly interface, their positions and orientations need to be adjusted. Specifically, a geometric feature (such as an edge, face, or point) of the reference component is first identified by its geometric sequence number. It is then matched with the corresponding feature of the component to be adjusted. Constraints such as parallelism, coplanarity, and coaxiality are applied, thereby completing the assembly. Part of the code is shown below Algorithm 2:

```
Algorithm 2. Assembly modelling (excerpt)
# Import assembly components
a1 = mdb.models['Model-1'].rootAssembly
a1.Instance(name='uxinggaiban-1', part=p,
dependent=ON)
a1.Instance(name='uxingzhuti-1', part=p,
dependent=ON)

# Coplanarity constraint
f11 = a1.instances['uxinggaiban-1'].faces
f12 = a1.instances['uxingzhuti-1'].faces
a1.FaceToFace(movablePlane=f11[8],
fixedPlane=f12[17], flip=ON, clearance=0.0)

# Parallelism constraint
a1.ParallelFace(movablePlane=f11[173],
fixedPlane=f12[3], flip=OFF)

# Coaxiality constraint
a1.Coaxial(movableAxis=f11[131],
fixedAxis=f12[7], flip=ON)

# Linear pattern
a1.LinearInstancePattern(instanceList=('luoshuan-1', 'luoshuan-2', 'luoshuan-3'), direction1=(-1.0, 0.0, 0.0), direction2=(0.0, -1.0, 0.0), number1=1, number2=2, spacing1=95.0, spacing2=2*Holeord)
```

After the components are assembled, the mechanical interaction relationships among different parts need to be further defined. This is often one of the more tedious procedures in a complex model with multiple assembled components, while the use of parametric modelling can improve this aspect of modelling efficiency.

The batch assembly of bolts and the definition of contact interactions can better demonstrate the efficiency advantage of the parametric modelling method. In a conventional simulation process, contact properties need to be assigned separately to the interfaces

associated with each bolt. Once the geometric dimensions are updated, the assembly operation, contact surface selection, repeated screen selections, model rotation, component hiding, and other manual operations usually have to be performed again. With the parametric modelling method, when the bolt assembly position or specification changes, the program can automatically update the target surfaces for bolt assembly according to the input parameters, based on the previously matched geometric features. In this way, a consistent connection logic of the model can be maintained under different parameter combinations. Part of the code is as follows Algorithm 3:

```
Algorithm 3. Interaction settings (excerpt)
# Tie constraint between the bolt bottom surface and the base
side1Faces1=
s1.getSequenceFromMask(mask=('[#8 ]', ), )
mdb.models['Model-1'].Tie(name='Constraint-19', master=region1, slave=region2, adjust=ON)

# Contact interaction between the bolt and the inner surface of the threaded hole
side1Faces1=
s1.getSequenceFromMask(mask=('[#10 ]', ), )
mdb.models['Model-1'].SurfaceToSurfaceContactStd(name='Int-12', master=region1, slave=region2, sliding=SMALL, thickness=ON, interactionProperty='IntProp-1')

# Contact constraint between the bolt and the upper surface of the support
side1Faces1 =
s1.getSequenceFromMask(mask=('[#8 ]', ), )
mdb.models['Model-1'].SurfaceToSurfaceContactStd(name='Int-23', createStepName='Step-1', master=region1, slave=region2, interactionProperty='IntProp-1')
```

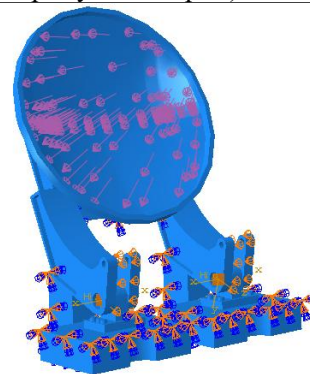


Figure 7. Schematic Diagram of Load and Boundary Condition Settings
For the load and boundary condition settings, a

time-varying dynamic load is applied to the inner surface of the cover to simulate the working condition of the hatch cover system in the open state [11], while the bottom support is fixed, as shown in Figure 7.

4. Simulation Analysis Results

Using the dimensions of a physical model as the reference, the parametric modelling program can automatically complete model generation and simulation calculation after the corresponding dimensional parameters are entered. Under the action of the alternating pressure field load, the generated finite element model shows a more pronounced response to load fluctuation at the root of the lug and in the transition region between the lug and the cover, as shown in Figure 8. The equivalent stress in this region is further extracted, and its time-history curve is obtained, as shown in Figure 9.

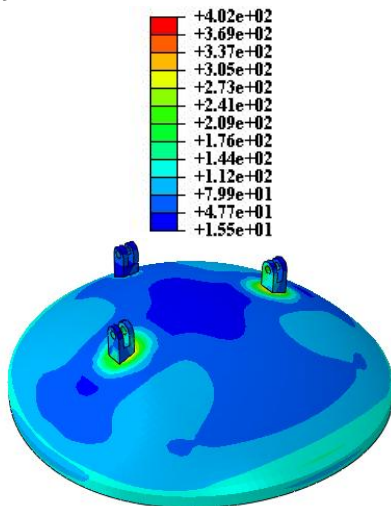


Figure 8. Stress Contour of the Cover at the Moment of Maximum Stress (t = 0.36 s)

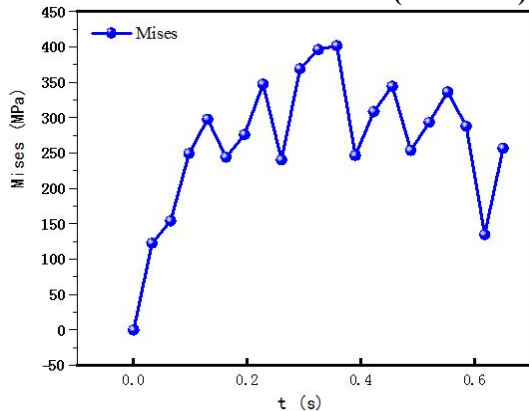


Figure 9. Stress-Time Curve of the High-Stress Region of the Cover

The stress curve of the cover plate shows an overall fluctuating upward trend. When the

analysis time approaches the middle stage, the load amplitude is close to its maximum value, and the peak stress in this region reaches 402 MPa. Except for the main peak, the average stress level in this region remains between 250 MPa and 350 MPa for a relatively long period. The load-bearing state is relatively stable and meets the allowable strength requirement of the structure.

The hinge is connected to the lug of the cover through a connecting pin and is also linked to the lever, forming a moving mechanism. The high-stress region of the hinge is mainly distributed at the transition position between the hinge and the lever, as shown in Figure 10. The stress peak also appears in the middle stage of the analysis time, as shown in Figure 11. The maximum stress of the hinge is approximately 597 MPa. High stress levels appear repeatedly at several time points and are very close to the material yield strength of 600 MPa, indicating that the structural strength margin of the hinge is relatively small under the present simulation condition.

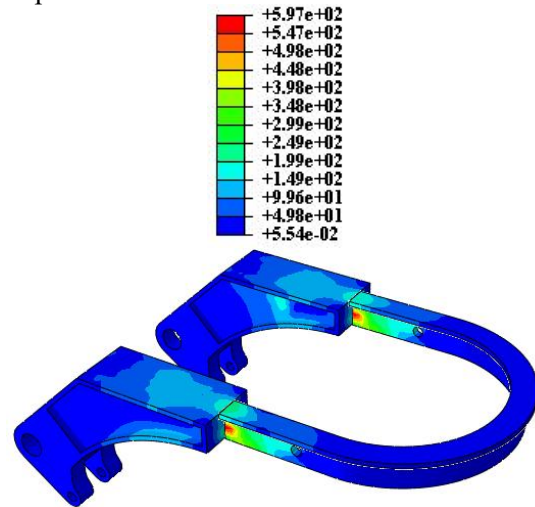


Figure 10. Stress Contour of the Hinge at the Moment of Maximum Stress (t = 0.36 s)

The stress contour of the support bolt at the moment of maximum stress is shown in Figure 12. The high-stress region is mainly located on the bottom surface of the bolt head and the side surface of the bolt body, indicating that the overall system tends not only to tilt backward but also to undergo a certain degree of torsion. This tendency is transmitted to the bolt region through the cover–hinge–support load transfer path. As can be seen from the curve in Figure 13, under the pressure field condition at the cylinder opening, the stress curve of the bolt also fluctuates with the variation of the

pressure field load. However, the overall stress level of the support bolt is relatively low, and the safety margin is relatively high.

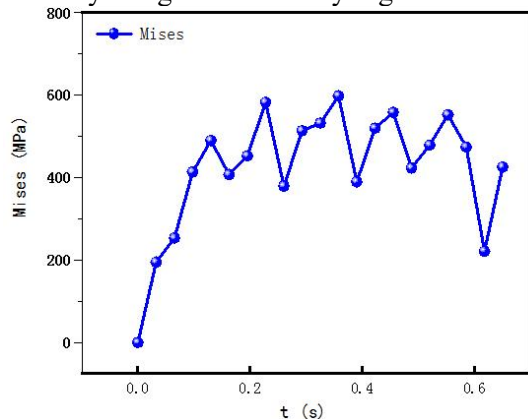


Figure 11. Stress-time Curve of the High-Stress Region of the Hinge

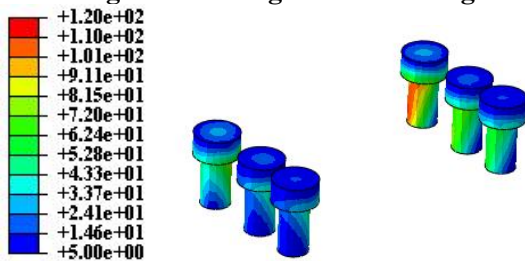


Figure 12. Stress Contour of the Support Bolt at the Moment of Maximum Stress ($t = 0.36$ s)

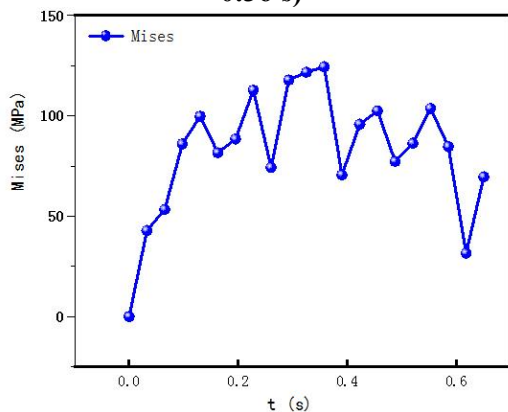


Figure 13. Stress-Displacement-Time Curve of the High-Stress Region of the Bolt

5. Conclusions

In this study, a parametric finite element modelling and simulation process based on Python scripts is established for a certain type of hatch cover opening system. By extracting the key dimensional parameters of the main structural components, including the cover, hinge, support, and bolts, and embedding them into the modelling program, the program automatically carries out geometry generation, component assembly, contact definition, load

application, and boundary condition setting. After the dimensional parameters of the physical model are entered as reference inputs, the program can automatically generate the corresponding finite element model and complete the simulation calculation. This indicates that the proposed method can reduce repeated modelling operations during dimensional iteration and improve the efficiency of verifying the mechanical performance of different structural schemes.

The simulation results show that, under the alternating pressure field load, the overall stress response of the hatch cover opening system fluctuates with the variation of the load. The high-stress regions are mainly concentrated at the transition region of the lug of the cover and the connection position between the hinge and the lever. The peak stress in the high-stress region of the cover reaches 402 MPa, and the overall load-bearing state remains relatively stable, meeting the allowable strength requirement of the structure. The maximum stress of the hinge is approximately 597 MPa, which is close to the material yield strength of 600 MPa. This indicates that the structural strength margin of this part is relatively small and should be further considered in subsequent structural optimization. In comparison, the stress levels of the base and support bolts are relatively low, with a higher safety margin. The proposed parametric modelling method can provide a reference for rapid modelling, simulation verification, and structural optimization of cover opening systems.

References

- [1] Hu J H. Research on automatic generation method of gear contour based on parametric modeling. *Modern Manufacturing Technology and Equipment*, 2026, 62(5): 186-188.
- [2] Dong S J, Yang A R, Yuan W, Zhang L Z, Fang Y F, Shen X L. Parametric modeling and optimization design of typical turbine air-cooled blade. *Journal of Aerospace Power*, 2025, 40(3): 20230316.
- [3] Apellániz D, Vierlinger R. Enhancing structural design with a parametric FEM toolbox. *Steel Construction*, 2022, 15(3): 188-195.
- [4] Cheng J, Zhou Z, Yang S, et al. Parametric modeling method of double-channel

- turbine volute with arbitrary area ratio. *International Journal of Automotive Technology*, 2025, 27(2): 1-16.
- [5] Li S, He B, Wang G, Li T, Zhan Z H, Rui X T. Dynamics modeling and simulating of the underwater launch system under the impact load. *Acta Mechanica Sinica*, 2025, 41: 524653.
- [6] Ji W H, Sun W, Ma H W, Zhang Y, Wang D, et al. Parametric finite element modeling and topology optimization for stress reduction of spatial pipeline based on beam/solid element coupling. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 2025, 239(8): 2864-2881.
- [7] Liu J L. Automated design method of mechanical parts based on parametric modeling. *Machinery Industry Standardization & Quality*, 2025, (12): 51-53, 70.
- [8] Deng L F. A modeling method for corrugated bulkheads FE model based on parametric technology. *Scientific and Technological Innovation*, 2026, (4): 1-4.
- [9] Yan L Y. Research on part modeling method based on computer-aided design and manufacturing. *Metalforming Equipment & Manufacturing Technology*, 2024, 59(6): 86-89.
- [10] Zhang W Q, Tian C L, An Y K, Li S. Parametric modeling method of marine propeller. *Internal Combustion Engine & Parts*, 2022, (16): 52-55.
- [11] Peng Y C, Chen X H, Wu Y D. Parametric modeling and buckling analysis of reinforced spherical shell based on ABAQUS. *Advances in Aeronautical Science and Engineering*, 2022, 13(5): 123-130, 170.