ISSN: 2414-1895 DOI: 10.6919/ICJE.202102_7(2).0022

# Damage Analysis of Engineering Ship Salvage Structure based on Finite Element Simulation

Daobing Zhang[1], Xiong Hu[1], Xiaomei Tao[2], Bing Wang[1] and Dejian Sun[1]

[1]Logistics Engineering College, Shanghai Maritime University, Shanghai 201306, China;

[2]Shanghai JianQiao University, Shanghai 201306, China.

## Abstract

**Taking an engineering ship in a salvage operation as the research object, the finite element method is used to establish a finite element model of the whole ship. Two typical working conditions of salvage, hoisting and placement are studied, and the influence of ship length on hull placement deformation is analyzed. The results show that in the salvage and lifting conditions, due to the unscientific setting of the lifting position, the local stress of the lifting point reached 1810 MPa, which far exceeded the allowable stress value of the structure, causing damage to the structure; in the placing condition, uneven mud on the shore The surface causes uneven weight support of the ship structure, resulting in insufficient structural strength in the middle position, resulting in damage to the entire ship's waist. This research has important significance and reference value for ship deformation theory and ship salvage engineering.**

## Keywords

**Structural Damage; Lifting; Finite Element; Ship Salvage; Strength.**

## 1. Introduction

At present, with the continuous and rapid development of the international shipping industry, accidents such as collisions, reefs, fires and other accidents have caused ship damage to occur from time to time, and reflecting a significant increase trend [1]. Such kinds of accidents will directly cause damage to the ship structure, and serious consequences may be caused such as cargo leakage, environmental pollution, and casualties. If not handled in time or improperly, it will even lead ship to break and sink as a whole [2-3]. The salvage of a ship is an important task after an accident occurs. If the salvage method is improper, it is likely to cause secondary damage to the hull and cause greater economic losses. Therefore, the implementation of a scientific and reasonable salvage plan is of great significance for maintaining the integrity of the hull structure and backtracking the cause of the accident [4].

At present, research on ship deformation at home and abroad is mainly concentrated on the analysis of ship structure, loading conditions, wind and wave loads and other factors [5-7]. There is no relevant literature on the deformation analysis of the ship salvage process, and most of the relevant literature is concentrated in design and safety analysis of salvage vessel. In literature [8], the author analyzes and summarizes the basic principles and key technologies in the design and realization of floating cranes to salvage sunken ships. In literature [9], the author numerically simulates the lifting process of the sunken ship based on the multi-body hydrodynamics software AQWA, and analyzes the motion response of the 1000t full-rotation crane ship "Chang tian long" in the sunken ship salvage operation, and proposes a crane ship Safety assessment method for salvage operations of floating and sinking ships. In literature [10], the author analyzed the static analysis and hydrodynamic dynamic analysis methods of the bow lifting operation in the "Shiyue" shipwreck salvage project, and obtained the

ISSN: 2414-1895 DOI: 10.6919/ICJE.202102_7(2).0022

corresponding lifting force, ship motion and other related data; and compared and analyzed the actual operation results, to verify the rationality of the analysis method. In literature [11] the used finite element analysis software to establish a finite element model of the salvage boom structure of a 250t inland river crane, and evaluated the structural strength and stability of the salvage boom under operating conditions through a direct calculation method to determine Stability and safety of the structure and performance of the pole.

In this paper, an engineering ship in a salvage operation is used as the research object. The finite element method is used to establish a finite element model of the whole ship. The method combines the qualitative analysis of basic mechanics and the quantitative analysis of finite element direct calculation to refer to and compare drawings and data. On the scene pictures, two typical working conditions of salvage, lifting and placement are studied, and the influence of the length of the ship's support on the deformation of the hull is analyzed. It has important meaning and value to scientific research and engineering practice.

The paper is organized as follows: Section II introduces the finite element modeling of the hull. The modeling and analysis of lifting conditions is introduced in Section III. The modeling and analysis of static placement conditions is introduced in Section IV. Finally, the conclusion of this paper is given in Section V.

## 2. Finite element modeling of the hull

The analysis object of this paper is an engineering ship with a length of about 40 meters and a no-load displacement of 357t. The ship's propulsion shaft system is single-engine single-propeller propulsion. The main engine is B&W 12K98MC-C. The propulsion shaft system is mainly supported by two oil-lubricated stern bearings and consists of two intermediate bearings, two thrust bearing supports and main engine bearings. The ship structure adopts steel material and horizontal frame type. When the engineering ship was docked at a certain wharf, it turned over when it encountered high tide. The overturned ship was subsequently salvaged. The lifting scene diagram is shown in Figure 1.

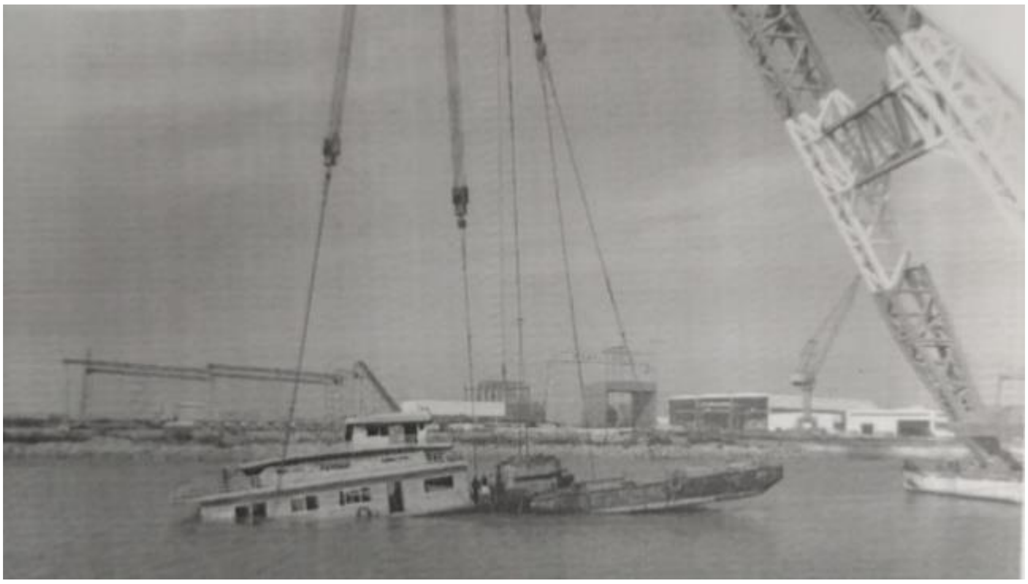

Figure 1. On-site lifting diagram of the ship

During the first salvage process, the overall structure of the hull was in good condition, but in the position A of the hull (10131mm from the middle of the ship, along the bow direction) deformed sharply and caused cracks, as shown in Figure 2. After repairing the cracks at this local location, the hull was lifted out of the water and placed on the soil on the shore. The hull structure was damaged by the waist on the second day, and the damage diagram of the structure is shown in Figure 3.

ISSN: 2414-1895
DOI: 10.6919/ICJE.202102_7(2).0022


Figure 2. Position A structure destruction

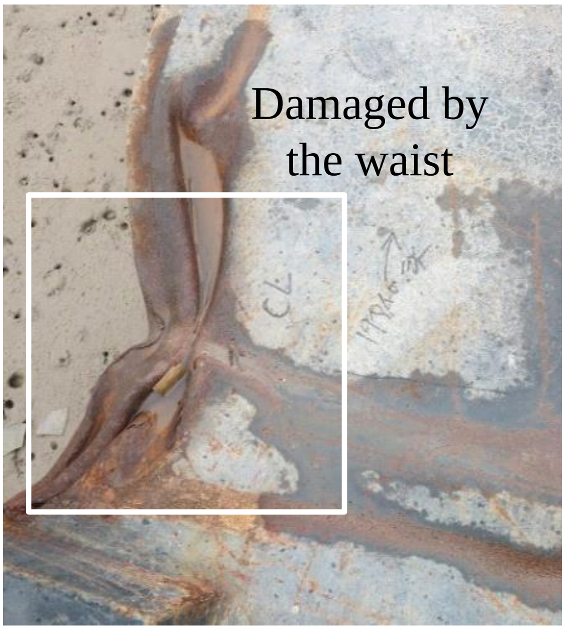


Figure 3. The diagram of damaged by the waist

The finite element software ANSYS is used for finite element modeling of the entire ship. The finite element model of the entire ship is shown in Figure 4.

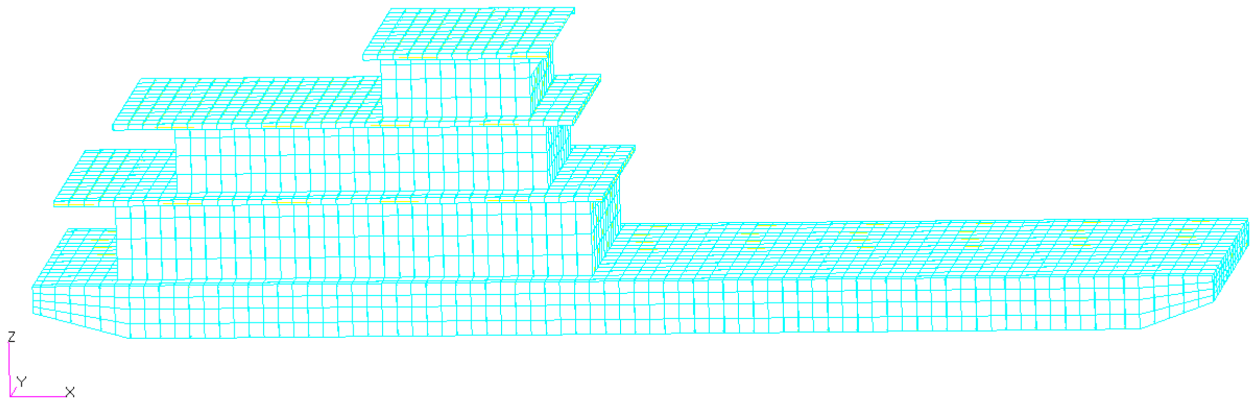

Figure 4. Three-dimensional finite element model of the whole ship

ISSN: 2414-1895
DOI: 10.6919/ICJE.202102_7(2).0022

According to the above information, the damage of the structure at position A is related to the lifting and salvage operation, and the structural damage at the mid-ship position is closely related to the support of the ground frame on the land after the hull is taken out of the water. For further analysis, two working conditions of lifting and static placement are set respectively, and the ship is simulated and analyzed by finite element simulation. A series of drawings such as general arrangement drawings, basic structure drawings, profile drawings, and main cross-section drawings of the engineering ship can establish a finite element three-dimensional model corresponding to the structure, as shown in Figure 4. The whole ship finite element method uses inertial release technology to directly calculate the hull structure [12-13]. This will help the calculation results that are more reasonable and in line with the actual situation, so that the strength of the ship's structure can be analyzed and evaluated more reasonably.

## 3. Modeling and analysis of lifting conditions

In the process of ship salvage, the ship is equipped with 4 ropes. The lifting position is shown in Figure 1. The no-load displacement of the engineering ship is 357 tons. According to the salvage situation, the crane on the ship is kept separate from the hull, except that the crane weighs 32 tons, so the actual lifting weight is 325 tons. According to the lifting with 4 ropes, the average bearing weight of each rope is 81.25 tons. That is, 81.25x1000x9.8=796250 N. In the actual lifting process of the ship, some ropes must bear a force greater than 796250N. According to the lifting design description, the average magnitude of the above-mentioned force is added to the vicinity of the actual lifting position. The finite element simulation is shown in Figure 5.

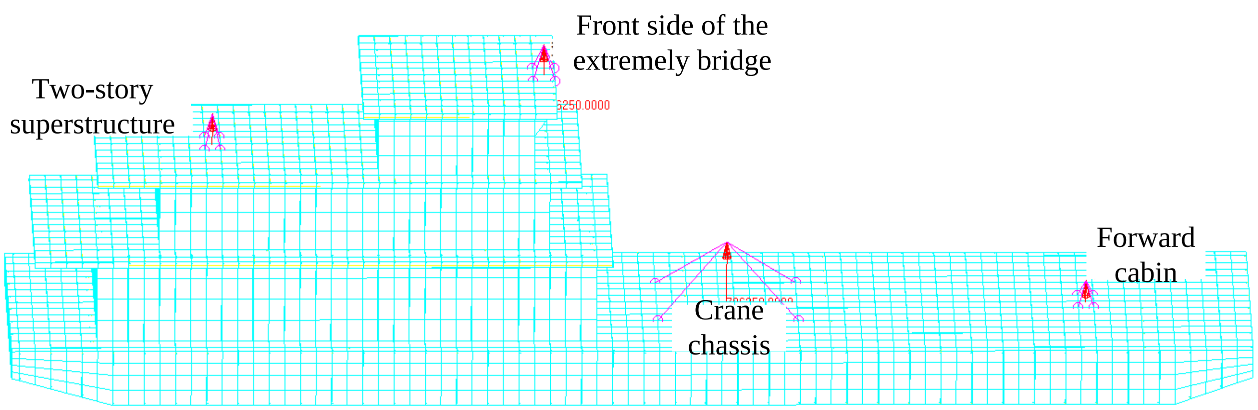


Figure 5. Schematic diagram of the finite element model of the load position

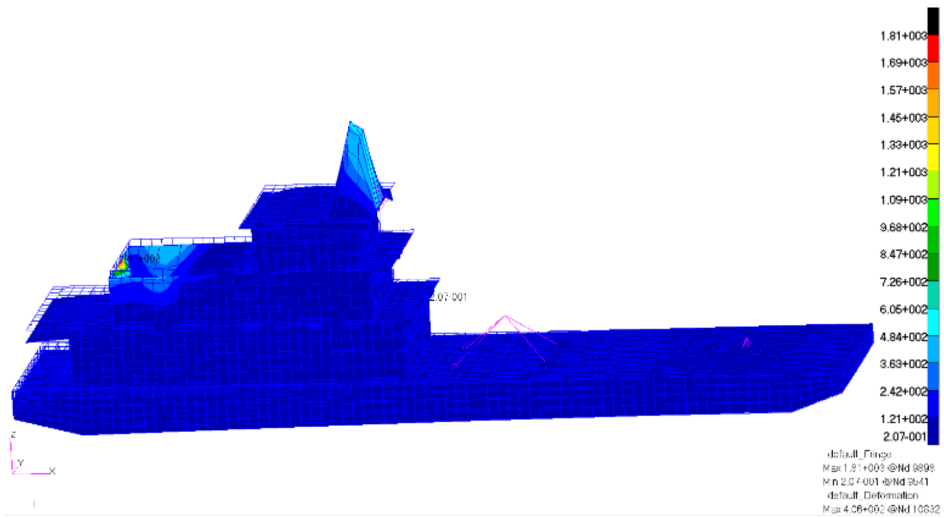


Figure 6. Finite element simulation diagram

ISSN: 2414-1895
DOI: 10.6919/ICJE.202102_7(2).0022

Figure 7. Site map after the actual ship is damaged

Perform finite element calculations on the above model to obtain the deformation and stress results of the hull. As shown in Figure 6. It can be seen that greater stress will be generated in the second deck area, top deck area and front end area of the hoisting position, and the maximum concentrated stress value reaches 1810 MPa, which is far greater than the allowable 235 MPa for general steel structures; The maximum stress value in the area reached 677MPa, and the maximum stress of the main hull was 468MPa, both exceeding the allowable stress value of the structure. What is certain is that the structure here will be destroyed. The greater the stress generated by the structure, the greater the damage it will suffer. In the middle area of the ship, the superstructure deck area, especially the second superstructure deck, stress concentration will cause structural deformation. This is consistent with the actual survey photos shown in Figure 7. In Figure 7, the second deck of the hull has undergone a large torsional deformation.

## 4. Modeling and analisis of static placement conditions

After the ship is hoisted and placed on land, under its own gravity, if there is no suitable fulcrum support, it will have a destructive effect on the hull structure. Therefore, based on the use of finite element to simulate the lifting conditions, this section simulates the static placement conditions based on the finite element to analyze the cause of the fracture in the middle of the hull.

### 4.1 Finite element modeling and calculation

After field surveys, the placement plane of the ship after salvage is not horizontal and has a certain slope. The ship is not in complete contact with the ground, and the length of support in the direction of the ship's length is limited. Select a lay-up length and perform the whole ship finite element simulation on the hull, as shown in Figure 8.

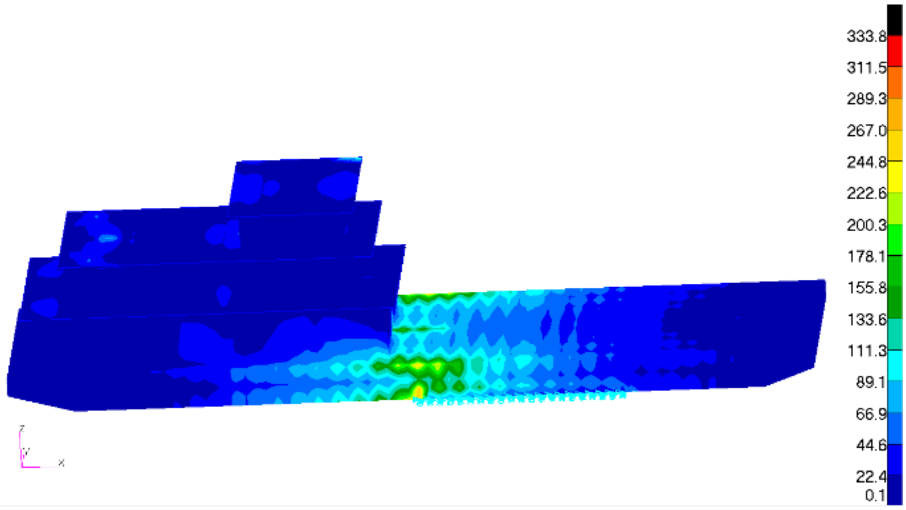


Figure 8. Schematic diagram of the finite element model of the ship's structural damage

ISSN: 2414-1895 DOI: 10.6919/ICJE.202102_7(2).0022

According to the results of finite element simulation, the maximum stress area of the hull is calculated to be located in the middle area of the hull. The maximum stress in the middle area is 333.8 MPa. It is the high stress in this area that causes the hull to crack, which is consistent with the site map shown in Figure 9.


Figure 9. Site map of structural damage in the ship

### 4.2 Support length impact analysis

After many tests and calculations, it is found that during the static placement of the ship, if the support position is near the center of gravity, the structural stress of the ship can be effectively reduced. According to this conclusion, considering that the hull of the project is made of ordinary steel, the yield limit is 235 MPa. The center position is located 1.5 meters to 3 meters behind the front end wall of the first-story superstructure (hereinafter referred to as the front end wall). At this time, the stress on the support structure is 198MPa, which is the minimum support length that supports the weight of the ship without causing damage. The finite element simulation is shown in Figure 10. If you continue to shorten the length of the support at this time, that is, from 1.5 meters to 2.5 meters behind the front wall, through the finite element simulation shown in Figure 11, the maximum structural stress at this time is 246.4 MPa, even if the allowable coefficient is not considered, it will still exceed the yield stress of ordinary marine steel will cause stress concentration in the hull.

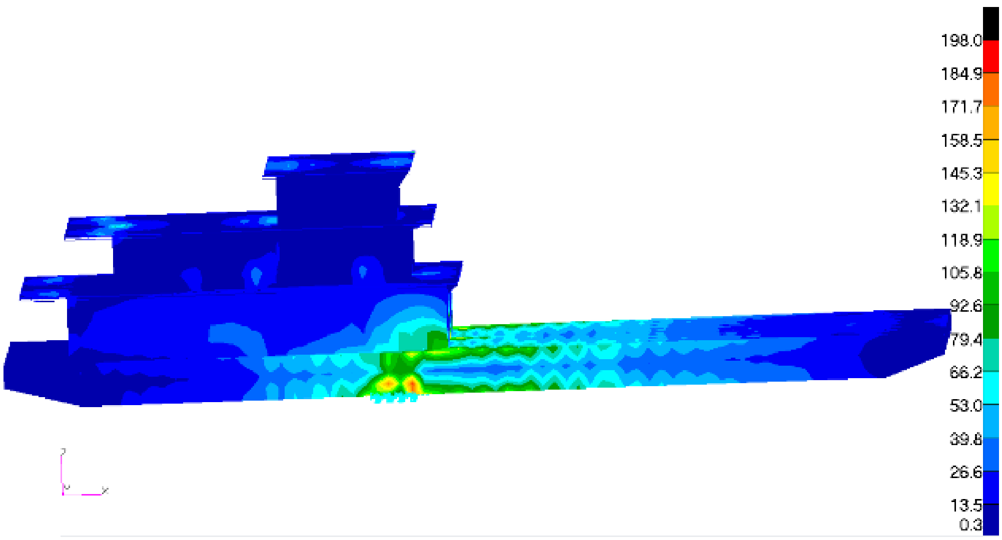


Figure 10. Structural stress with support length of 1.5 meters

ISSN: 2414-1895 DOI: 10.6919/ICJE.202102_7(2).0022

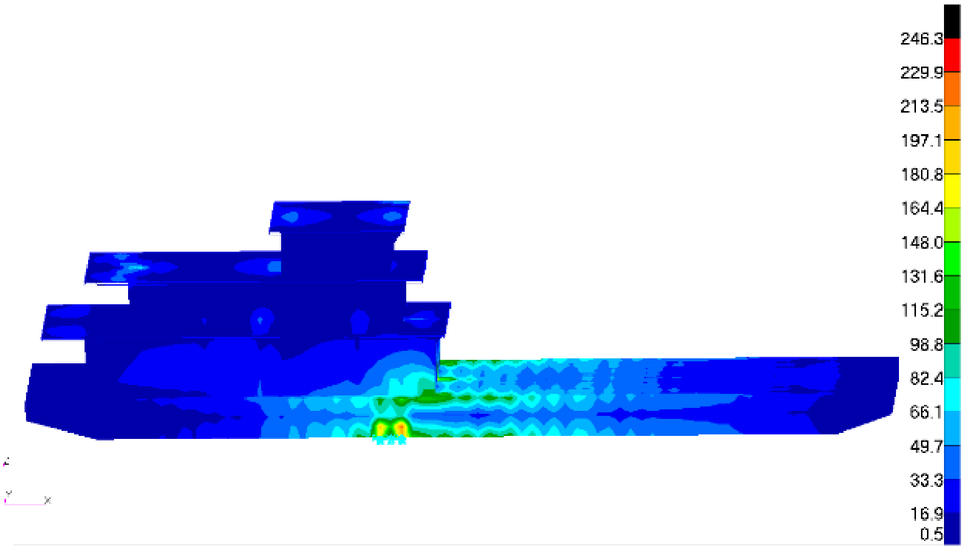


Figure 11. Structural stress with support length of 1.0 meters

After field surveys, the placement plane of the ship after salvage is not horizontal and has a certain slope. The ship is not in complete contact with the ground, and the length of support in the direction of the ship's length is limited. Select a lay-up length and perform the whole ship finite element simulation on the hull, as shown in Figure 8.

Taking the position of the ship's center of gravity as the starting point, finite element simulation is used to calculate the stress change curves when one-way and two-way supports are added, as shown in Figures 12 and 13, respectively. Figure 12 shows the stress change curve when the support is added along the stern direction. With the front wall of the first superstructure as the origin, the support of each rib is added in sequence along the stern direction. The overall structural stress shows a decreasing trend, increasing to the third When the rib position reaches the 5th rib position, the stress is significantly reduced, and the supporting effect is obvious; from the 5th rib position to the 10th rib position toward the stern, the maximum stress on the ship structure decreases sharply, and then continues to the first rib position. When there are 20 ribs, the stress change is gentle and the support effect is not obvious; Figure 13 shows the stress change curve when the support is increased in both directions. It can be seen that when one rib is added along the bow and stern at the same time, the structural stress is also reduced overall, but the effect is shown in a broken line. This is related to the structure of the ship itself. Its structure is an ordinary horizontal frame and a strong frame, alternately arranged. When the support is a strong frame, the structural stress is reduced significantly; otherwise, it is not obvious.

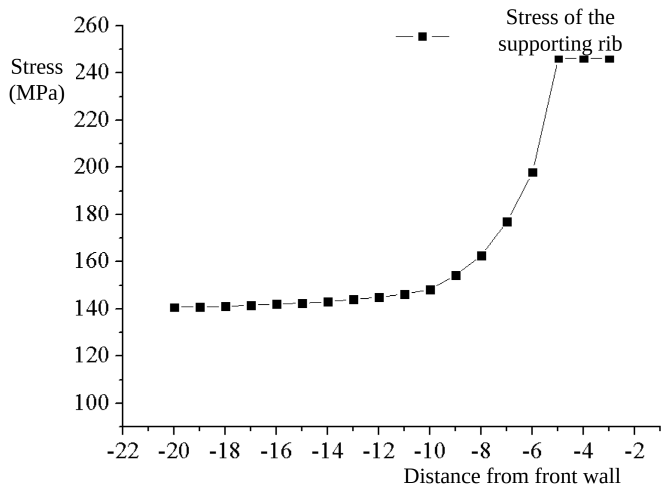


Figure 12. Corresponding stress for adding support along the stern direction

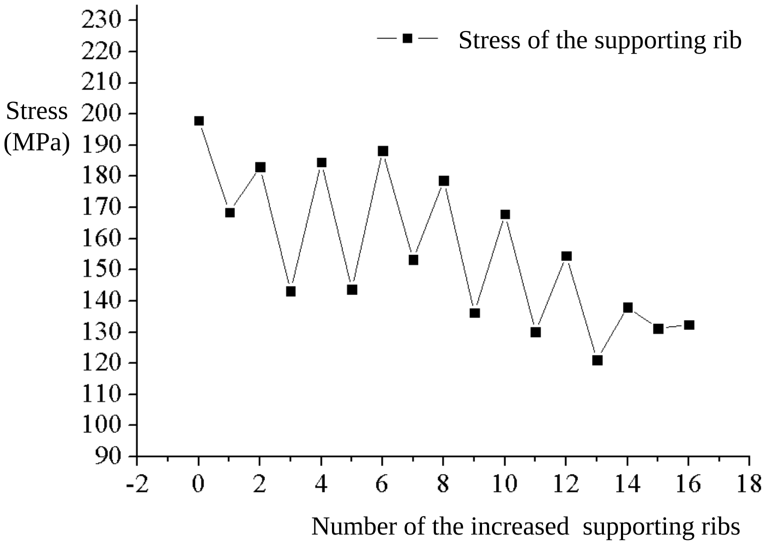


Figure 13. Corresponding stresses for simultaneous application of supports along the bow and stern directions

## 5. Conclusion

In this paper, an engineering ship in a salvage operation is used as the research object, and a finite element model of the whole ship is established by the finite element method. Two typical working conditions of salvage, hoisting and placement are studied, and the influence of ship length on hull placement deformation is analyzed. After calculation and analysis, the following conclusions are obtained:

(1) During the salvage and hoisting process, due to the improper number of slings and the hoisting position, high stress is generated in the local area of the hoisting position and the entire midship area, accompanied by large deformation of the entire ship area. Among them, the maximum stress has exceeded the allowable stress, resulting in permanent and irreversible plastic deformation and even breakage of the corresponding position structure. Considering that the bearing capacity of the rope must exceed the above average force, the actual situation will produce greater stress and plastic deformation or fracture, which will cause greater damage to the structure.

(2) During the static placement of the ship after salvage, insufficient ship support length will cause severe structural stress and cause cracks and damage to the hull. If the support conditions are limited, the position of the support needs to be fully calculated, and the position close to the center of gravity is a better choice. When the supporting conditions are good, the relationship between the supporting length and the structural stress can be established, and the optimal structural supporting scheme can be selected.

## Acknowledgments

This research received specific grant from National High Technology Research Development Plan (2013AA041106), National Natural Science Foundation (31300783), China Postdoctoral Science Foundation (2014M561458).

## References

[1] Han Junsong, Liu Ao, Cheng Yichun. Cause Analysis on Ship Collision Accidents in Laotieshan Channel and Associated Countermeasure Study, China Maritime, 2020(08): 24-27.

[2] Wang Siqi, Zhang Zhiming, Zheng Pengjun. Evaluation method based on Bayesian network for completeness of emergency plan of ship pollution accident, JOURNAL OF NINGBO UNIVERSITY (NSEE), 2020, 33(04):97-102.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202102_7(2).0022


[3] Lin Zhidan. Analysis on the Causes and Lessons of Ship Strike Bridge Accident, Highway Traffic Technology (Applied Technology Edition), 2020,16(06):311-313.

[4] Wang Zhao, Weikang, Xuhan. On the development of my country's offshore lifting and salvage operations and the development of large-scale crane vessels for basic equipment, China Plant engineering, 2020(12): 219-220.

[5] Wen Lu, Li Yawei, Du Zhipeng. Deformation Distribution and Transfer Characteristics of Hull Structure Under Underwater Explosion, Journal of Wuhan University of Technology, 2020,44(03):570-575.

[6] Zhang Shengdong, Yang Xiuying, Huang Qiang. Influence of Hull Large Deformation under Wave Load on Stern Tube Bearing Loading, SHIP ENGINEERING, 2018,40(S1):69-74.

[7] Zhao Yu. Numerical simulation of ship hull deformation under impact loading, SHIP SCIENCE AND TECHNOLOGY, 2018, 40(02):1-3.

[8] Wang Daoneng. Design and realization of floating crane ship salvaging sunken ship, China Water Transport, 2019, 19(10):17-18+50.

[9] Lin Qizhen, Ai Jun, Wang Bin. Safety analysis of crane ship salvage operation based on AQWA, China Water Transport. Waterway Technology, 2017(06):11-18.

[10] LiuYui, Sun Liping, Chen Shihai. Calculation and Analysis of Bow Lift in SEWOL Salvage Project, SHIPBUILDING OF CHINA,2017,58(03):207-220.

[11] Pang Juan, Xie Yonghe. Finite Element Strength Analysis of Salvage Boom of 250 t Floating Crane, Journal of Zhejiang Ocean University(Natural Science), 2016,35(02):155-159.

[12] Wu Zhonghua. Application of inertial release in the hull structure design, SHIP SCIENCE AND TECHNOLOGY, 2016, 38(06):1-3.

[13] Yan Yuquan. Application of Inertia Release in the Structural Design of Lifting Platform, Lifting and transporting machinery, 2020(14):47-50.