КОНТРОЛЬ І ДІАГНОСТИКА ПРОЦЕСІВ ТА СИСТЕМ В ПРИЛАДОБУДУВАННІ

УДК 004.925.8:519.876.5

STRESS-STRAIN STATE SIMULATION OF WELDED PLATE

Tsybulnyk S., Khotsevych M., Tovber A.
National Technical University of Ukraine «Igor Sikorsky Kyiv Polytechnic Institute», Kyiv, Ukraine
E-mail: tsybulnik.s.a@gmail.com

Background. *The most important task in choosing the shape of the structural elements of the ship's hull is to ensure sufficient strength with a small mass. In recent years, great progress has been made in shipbuilding thanks to the use of modern welding methods for connecting parts and components. Ships hulls have become much lighter than when using riveted joints due to the transition to welding. A welded joint can provide optimum corrosion resistance, strength and economy of manufacture. However, it must be remembered that any metal, including stainless steels, may undergo certain changes during welding. Therefore, it is necessary to exercise a reasonable degree of caution during the welding process. In recent years, simulation has become a very popular method for studying processes and phenomena. In addition to mathematical methods, more and more often, authors from all over the world are starting to use simulation modeling as the main research method and compare the results with theoretical or experimental ones. There are many software products that allow you to create geometric models of objects of varying complexity. One such program is SolidWorks, which was chosen to create the geometric models in this study.*

Objective. *Therefore, the purpose of this work is to conduct geometrical modeling (construction of three-dimensional models) and simulation (study of the object by its model) of a metal plate with a weld. Also, a liquid-construction interaction analysis will be carried out to determine the stress-strain state of the metal plate (ship's hull element) with welds of different geometric shapes in the cross-section.*

Results. *In this work, a three-dimensional model of the ship's hull element with welded seams was built. To determine the vector fields of the velocity of water flow and its pressure in the ANSYS software package, a simulation of the load on the hull element was carried out using its constructed geometric model. The analysis of the obtained results made it possible to determine the stress-strain state of the ship's hull element.*

Conclusions. *Three forms of welds are considered, namely: V-shaped, Y-shaped and X-shaped. It is shown that at a flow rate of 5m/s the minimum load is less than at least three times for the X-shaped (double-sided) weld. The maximum loads within the hull element of the ship are almost the same. It is shown that the smallest stress within the welds (without considering the plate) arises when using an X-shaped weld. In further research it is planned to investigate the effect of loads from the water stream on the damaged weld seams.*

Keywords: *weld seam, simulation and geometric modeling, ANSYS, SolidWorks.*

Introduction

Each object has its own operating life, during which the reliability of the design is guaranteed. At the end of this period, the probability of destruction of the structure is high. But objects can be destroyed not only after the expiration of their lifetime. The consequences of premature destruction can be significant material costs and human lives. In order to avoid this, it is necessary to prevent the development of defects [1].

The initial stage of the destruction of the structural elements of any construction or structure is damage. The reasons for their appearance may be, for example, external natural and man-made influences, internal factors, design errors, defects and violation of operating rules. External signs of destruction are cracks. Cracks can appear due to many reasons and have different effects depending on the combination of external and internal loads. The opening of the crack in the elements of the ship's construction can take place in different ways [2]. This is especially true in the conditions of operation of ships, when a crack or defect of a welded joint can lead to damage or breakage of the ship's hull. Therefore, the task of studying the hull plate of the ship is important and relevant to its operation.

When working with metal plates, the welding is one of the methods of connection. Mild steel is a good welding material, and welded joints can provide optimum corrosion resistance, durability and cost-effectiveness. However, it must be remembered that any metal, including stainless steel, may undergo certain changes during welding. Therefore, it is necessary to demonstrate a reasonable degree of caution during welding. This will minimize any harmful effects that may occur. It will also maintain a high degree of corrosion resistance and strength in the welding zone (which is an integral part of the base metal) [3].

In addition to steel, the use of high-strength aluminum alloys, especially for the design and construction of warships, surface ships and high-speed passenger ships, is increasing in the shipbuilding industry. Currently, various welding methods are used for the manufacture of aluminum structures of ships, namely: gas welding, laser welding and friction stir welding, but the most advantageous for the construction of aluminum structures is the friction stir welding technology [2].

If the welding mode is not followed, the structural strength decreases. Therefore, most of the scientific papers are devoted directly to the process of welding. For example, in [2] and [4], the authors reviewed the process of welding various grades of aluminum alloys using the friction stir welding. Friction welding associated with mixing the weld material is an alternative to other welding techniques for aluminum alloys. In the present studies, mechanical properties of friction stir welded aluminum alloys are examined experimentally. The insights developed from the study [2] are documented together with details of the test database. The performed tests and subject-matter literature research [4] indicate that application of FSW method to joining aluminum alloys in shipbuilding is rational.

There is also a list of works devoted to the choice of methods for studying stresses in welds. For example, in [5] were investigated geometrical distortions of two steel plates jointed by metal inert gas welding. The distributions of residual stresses in this welded joint were measured by X-ray diffraction method.

In article [6], the authors state that value of angular distortions in a welded T-joint depends on many parameters, including: weld geometrical dimensions, weld penetration profile, mechanical and physical-chemical properties of material, electric current source. Also, the authors indicate that the most effective method in mitigating welding deformations, apart from straightening, is an appropriate preparation of production process.

In recent years, simulation has become a very popular method of studying processes and phenomena. In addition to mathematical methods, more and more authors from all over the world begin to use as the main method of research simulation and compare its results with theoretical and experimental. So in the article [5] the measured residual stress distributions were compared with residual stress state obtained by means of finite element analysis with using of ABAQUS software and a good agreement was obtained between experimental and analytical data.

In articles [7] and [8] thermo-mechanical finite element analysis has been performed to assess the residual stress in the butt-weld joints of IN718 plates with help of the commercial software package ANSYS, employing 2D plane stress models. The present 2D plane stress analysis results are found to be in good agreement with existing 3D finite element analysis results.

In article [9] residual strains and stresses in a hollow steel beam that had been welded to a D-shaped cross-section have been simulated by plane deformation finite element models and compared with experimental measurements obtained using the neutron diffraction strain-scanning technique. The authors claim that with the assumptions for material properties that were made, the plane deformation model predicts the overall bending of the beam and the overall residual strains and stresses reasonably well.

Consequently, there are many scientific papers devoted to the study of the process of welding directly or mathematical modeling or simulation of loads that arise in welds. Article [9] shows that simulation has been used for research since the 90s of the twentieth century. However, for today there are very few works of joint analysis of fluid-construction for ship’s hull elements.

Formulation of the problem

There is a large variety of software complexes, the main task of which is simulation. Typically, such software systems have different functionality. For example, there are specialized software products for medical simulations. Such programs can not simulate most engineering tasks. However, there are a number of universal software complexes that can be used in many areas of science and technology including shipbuilding.

The most important task when choosing the shape of the structural elements of the ship's hull is to provide sufficient strength at low mass [10]. In recent years, great success has been achieved in this field thanks to the use of modern welding methods for connecting parts. Due to the transition from riveting to welding, the ships hulls became much lighter, mainly due to the removal of connecting corners and riveted joints.

Therefore, the purpose of this work is to conduct geometrical modeling (construction of three-dimensional models) and simulation (study of the object by its model) of a metal plate with a weld. Also, a liquid-construction interaction analysis will be carried out to determine the stress-strain state of the metal plate (ship’s hull element) with welds of different geometric shapes in the cross-section.

Geometric modeling

Geometric models play an important role in the simulation process. They are created in order to:

- to recreate the appearance and basic dimensions of the object;
- to recreate the structure, properties and nature of interaction with external conditions (loads and influences);

- to predict the effects of the impact on the object of external or internal loads;
- simplify the design process.

In any geometric modeling four main stages can be distinguished [11]:

- statement of some geometric problem;
- development of a geometric algorithm for its solution;
- implementation of the algorithm using some tools;
- analysis and interpretation of the results.

In our time, we can talk about three basic methods of geometric modeling - analytical, graphical and computer [11].

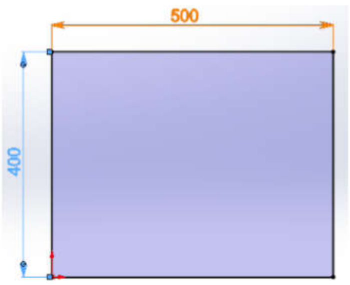
The essence of the analytic method is that the initialized geometric objects and operations carried out on them become accountable for some numerical objects and operations carried out on them. This is accomplished by using appropriate analytical expressions and their transformations.

The graphical method is characterized by the fact that during the implementation of one or another geometric modeling procedure geometric objects are used and the operations carried out on them are interpreted graphically. In this case, appropriate graphic constructions on the plane are used, executed using a ruler, compass and other drawing tools.

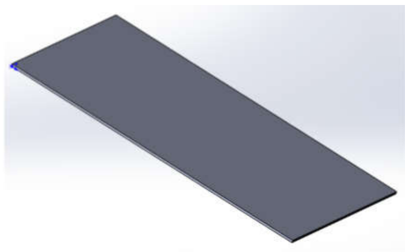
The computer method of geometric modeling assumes that the functions of geometric algorithmization of the process of solution of the initial problem, carried out by a person. However, the necessary graphic constructs in this case are performed using a personal computer using specialized software. Geometric interpretation of the results of graphic constructions, further analysis of this interpretation and the interpretation of its results in categories of source application problems, as before, performs a person [11].

There are many software products that allow you to create geometric models of objects of varying complexity in our time. Some of these programs even have basic simulation modules. One such software is SolidWorks [12], which was chosen to create geometric models in this study.

First, a geometric model of a steel plate was created (Fig. 1). According to [10], such 3mm thick plates are used on boats longer than 12 meters long.



a)



b)

Fig. 1. Boat hull element: a) sketch; b) geometric model

The next step is to build a geometric model of weld seam. Fig. 2 depicts the main forms of welds [13]. Seams of V-, Y- and X- shapes were chosen for the study. This will determine which geometry of the weld seams best accepts the given external load.

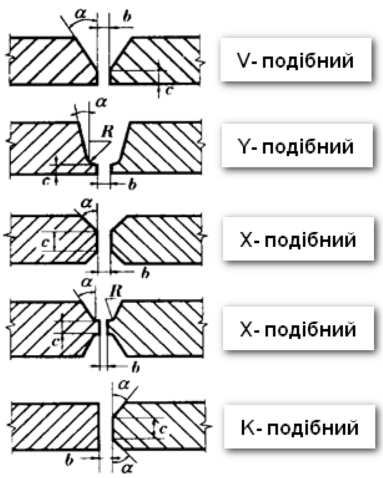


Fig. 2. Forms of welds

Thumbnails and geometric models of selected welds are depicted in Fig. 3 and Fig. 4. After creating the necessary geometric models, they were merged into assembly units. Three assembly units were created according to the types of welds in total. In each of them, the weld is placed along the contour of the metal plate, which acts as an element of the hull of the ship. Also geometric models of welds were cut from geometric models of plates, in order to avoid collisions (mutual penetration of models).

In addition, it is necessary to create geometric models of the liquid (separately for each weld) for liquid-construction interaction analysis. These models are a parallelepiped, the width and length of which coincide with the size of the previously created assembly units. Models of the aquatic environment after the creation are added to the corresponding assembly units. They should be placed in touch with the element of the ship's hull from the side of welds. It is also necessary to cut the geometry of the welds from the geometric model of the aquatic environment to prevent collisions (mutual penetration of the models).

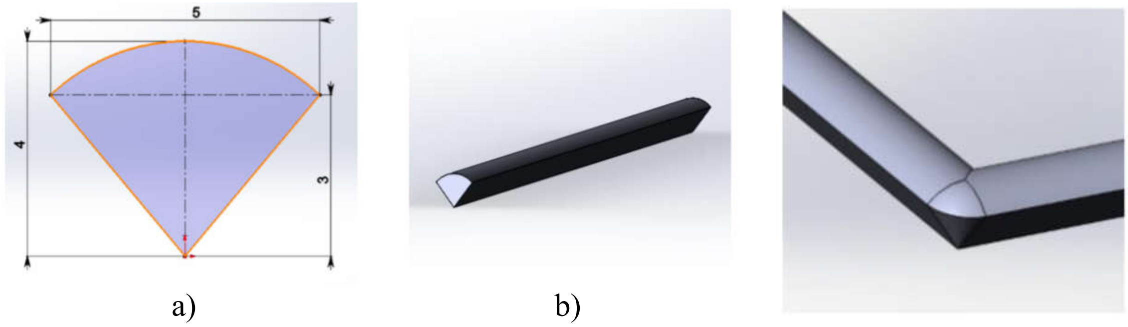


Fig. 3. V-shaped seam: a) sketch; b) geometric model

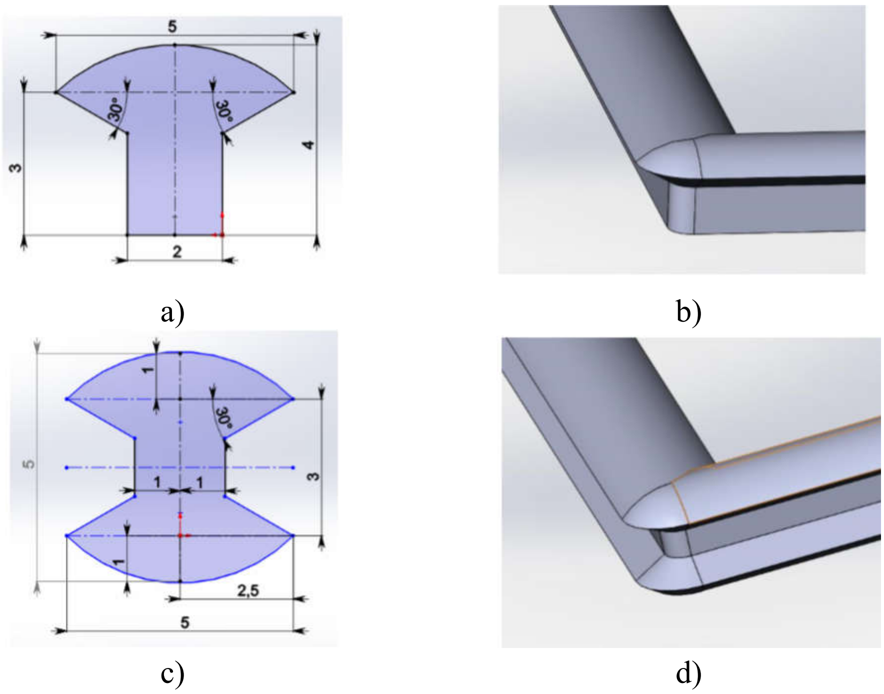


Fig. 4. Weld seams: a) a sketch of a Y-shaped seam; b) geometric model of Y-shaped seam; c) a sketch of a X-shaped seam; d) geometric model of X-shaped seam

After creating all the necessary geometric models, simulation can be performed.

Simulation results

CAE systems use a large set of visualization methods that are classified by the type of mathematical variables: scalar, scalar field, vector field (for example, velocity), and geometry used (three-dimensional or two-dimensional). It is important that the data can be displayed at any stage of the calculation. This allows you to track the dynamics of the modeled process.

Types of models in engineering analysis:

1. Geometric model.

2. Calculation model is a simplified geometric model used for analysis. Simplification of the geometric model occurs by removing those elements that do not significantly affect the results of the analysis. Geometric and calculation models are usually created at the design stage in specialized programs (for example, SolidWorks).

3. The mesh model is a set of nodes and elements on which the calculation model breaks down.

4. Simulation model is a mesh model with given boundary conditions and loads.

To build a simulation model, we use ANSYS Workbench [14], which gives the user a convenient graphical interface to simplify and systematize the necessary actions during modeling.

The simulation of the interaction of the liquid-construction took place in two stages. In the first stage, modeling was carried out with the help of a module of hydro-gas dynamics. A feature of this simulation is finding the value of pressure and flow velocity within the calculated area. The flow rate was chosen as 5m/s for this study. Fig. 5 shows the simulation results in the form of a vector field of speed.

Fig. 5 shows that the flow velocity in the central region of the plate is the lowest (0.45m/s) and increases to the edges, accelerating to 6m/s. The pressure of the water flow has an inverse picture, namely:

the greatest values of pressure is in the central region and decreases to the edges.

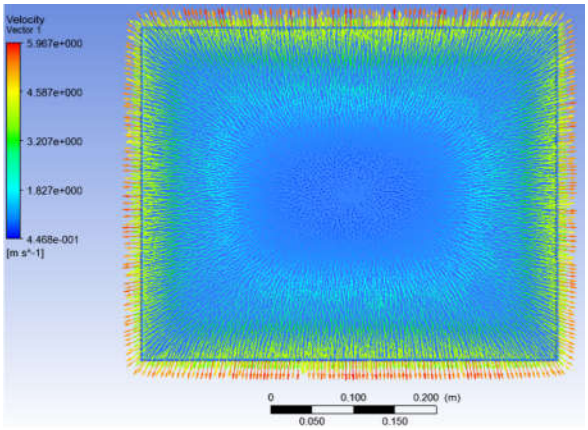
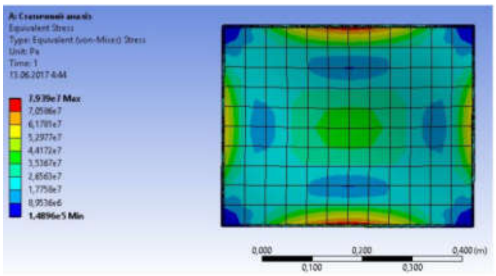
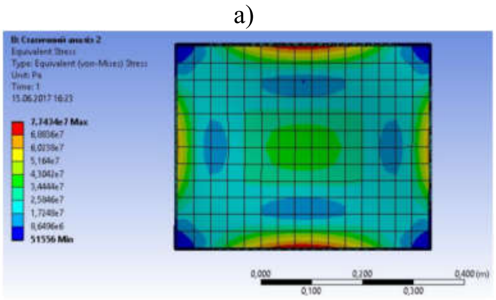


Fig. 5. Vector velocity field

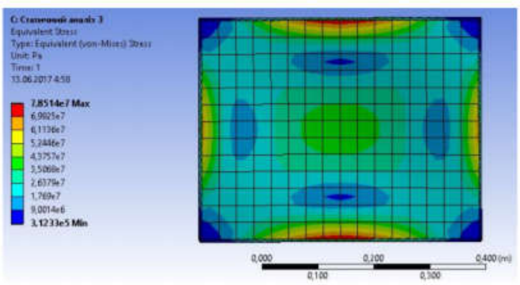
The second stage is a static structural analysis, the result of which is the calculated stress-strain state. Fig. 6 and Fig. 7 show the contour pressure fields of the plate with the weld seam and the weld itself, respectively.



a)



b)



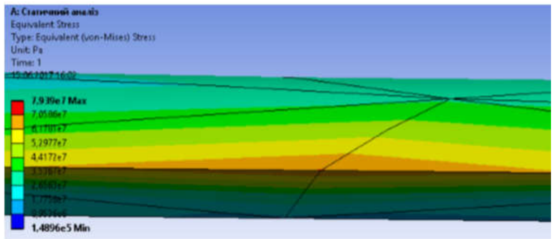
c)

Fig. 6. Contour pressure fields using welds:
a) V-shaped; b) X-shaped; c) Y-shaped

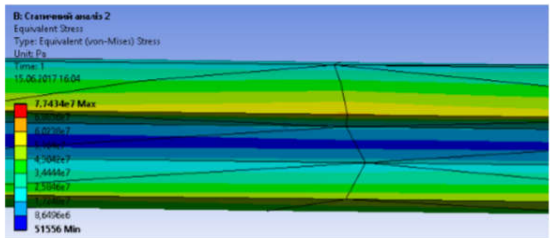
The results of the study showed that within the whole assembly unit (welded plate) the distribution and magnitudes of stresses and deformations are almost identical. The exception is the X-shaped weld seam, in which the minimum stresses and deformations are at least three times less than in other cases.

According to Fig. 7 it can be concluded that the stresses directly on the welds are also smaller in the case of an X-shaped weld seam.

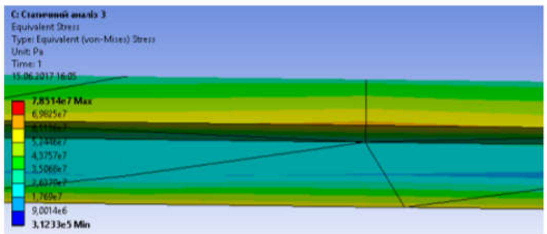
In the future, it is planned to carry out research on the ship's hull element in the case of damaged weld seam. It is also necessary to consider the assembly unit, which will consist of a large number of plates (e.g. four), welded together.



a)



b)



c)

Fig. 7. Contour pressure fields of welded seams: a) V-shaped; b) X-shaped; c) Y-shaped

Conclusions

In this work, a three-dimensional model of the ship's hull element with welded seams was built. To determine the vector fields of the velocity of water flow and its pressure in the ANSYS software package, a simulation of the load on the hull element was carried out using its constructed geometric model. The analysis of the obtained results made it possible to determine the stress-strain state of the ship's hull element.

Three forms of welds are considered, namely: V-shaped, Y-shaped and X-shaped. It is shown that at a flow rate of 5m/s the minimum load is less than at least three times for the X-shaped (double-sided) weld. The maximum loads within the hull element of the ship are almost the same. It is shown that the smallest stress within the welds (without considering the plate) arises when using an X-shaped weld. In further research it is planned to investigate the effect of loads from the water stream on the damaged weld seams.

References

[1] Б. О. Охота, С. А. Цыбульник, "Имитационное моделирование элемента обшивки самолета при многоочаговом повреждении", *Вісник НТУУ «КПІ». Серія приладобудування*, №55(1), с. 93-100, 2018.
[2] J. K. Paik, "Mechanical properties of friction stir welded aluminum alloys 5083 and 5383", *Int. J. of Nav. Arch. and Oc. Eng*, Vol. 1, Issue 1, pp. 39-49, 2009. DOI: https://doi.org/10.2478/IJNAOE-2013-0005
[3] S. Jokosisworo, "Pengaruh besar arus listrik dengan menggunakan elektroda smaw terhadap kekuatan sambungan las butt joint pada plat mild steel", *Kapal*, Vol 6, Issue 2, pp. 118-122, 2012. DOI: https://doi.org/10.12777/kpl.6.2.118-122.
[4] D. Krzysztof, J. Wojciech, "Influence of friction stir welding (FSW) on mechanical and corrosion properties of AW-7020M and Aw-7020 alloys", *Polish Maritime Research*, Vol. 23, Issue 3, pp. 86-90, 2016. DOI: https://doi.org/10.1515/pomr-2016-0036.
[5] V. Monin, T. Gurova, X. Castelo, S. F. Estefen, "Analysis of residual stress state in welded steel plates by x-ray diffraction method", *Rev. Adv. Mater. Sci.*, № 20(2), pp. 172-175, 2009.
[6] J. Kozak, J. Kowalski, "Problems of determination of welding angular distortions of T-fillet joints in ship hull structures", *Polish Maritime Research*, Vol. 22, Issue 2(86), pp. 79-85, 2015. DOI: https://doi.org/10.1515/pomr-2015-0020
[7] M. Jeyakumar, T. Christopher, R. Narayanan and B. Nageswara Rao, "Residual stress evaluation in butt-welded IN718 plates", *IJEMS*, Vol. 18(6), pp. 425-434, 2011.
[8] M. Jeyakumar, T. Christopher, R. Narayanan and B. Nageswara Rao, "Residual stress evaluation in butt-welded IN718 plates", *CJBAS*, Vol. 01, Issue 02, pp. 88-99, 2013.
[9] L. Wikandert, L. Karlssont, M. Nasstromt and P. Webster, "Finite element simulation and measurement of welding residual stresses", *Modelling Simul. Mater. Sci. Eng*., № 2, pp. 845-864, 1994.
[10] Материал и конструкция корпуса [Електронний ресурс]. Доступно: http://www.motolodka.ru/material.htm. Дата звернення: Жовт. 27, 2018.
[11] В. В. Пилюгин, Л. Н. Сумароков, "Геометрическое моделирование", *Матем. моделирование*, Том 6, № 5, с. 21-36, 1994.
[12] SolidWorks [Електронний ресурс]. Доступно: https://www.solidworks.com/. Дата звернення: Жовт. 27, 2018.
[13] Виды сварных швов и соединений [Електронний ресурс]. Доступно: https://lektsii.org/12-77526.html. Дата звернення: Жовт. 28, 2018.
[14] Официальный сайт компании ANSYS Inc. [Електронний ресурс]. Доступно: http://www.ansys.com/. Дата звернення: Жовт. 27, 2018.

УДК 004.925.8:519.876.5
С. О. Цибульник, М. О. Хоцевич, А. Й. Товбер
Національний технічний університет України «Київський політехнічний інститут імені Ігоря Сікорського», Київ, Україна

МОДЕЛЮВАННЯ НАПРУЖЕНО-ДЕФОРМОВАНОГО СТАНУ ПЛАСТИНИ ЗІ ЗВАРНИМ ШВОМ

Найважливішим завданням при виборі форми конструктивних елементів корпусу корабля є забезпечення достатньої міцності при малій масі. В останні роки в кораблебудуванні досягнуто великих успіхів завдяки застосуванню сучасних методів зварювання для з'єднання деталей та елементів. Завдяки переходу до зварювання, корпуси кораблів стали на багато легше ніж при використанні заклепкових з'єднань. Зварне з'єднання може забезпечити оптимальну корозійну стійкість, міцність і економічність виготовлення. Однак необхідно пам'ятати, що будь-який метал, включаючи нержавіючі сталі, може зазнавати певних змін під час зварювання. Тому необхідно проявити розумну ступінь обережності під час зварювання. В останні роки імітаційне моделювання стає дуже популярним методом дослідження процесів і явищ. Окрім математичних методів все частіше автори з усього світу починають використовувати як основний метод дослідження імітаційне моделювання і порівнювати результати з теоретичними чи експериментальними. Існує безліч програмних продуктів, які дозволяють створювати геометричні моделі об'єктів різної складності. Деякі з цих програм навіть мають у своєму складі базові модулі імітаційного моделювання. Однією з таких програм є SolidWorks, яка була обрана для створення геометричних моделей у даному дослідженні. У даній роботі була побудована тривимірна модель елемента обшивки корабля зі зварними швами. Для визначення векторних полів швидкості водяного потоку і його тиску в програмному комплексі ANSYS проведено імітаційне моделювання навантаження на елемент конструкції з викорис-

танням його побудованої геометричній моделі. Аналіз отриманих результатів дозволив визначити напружено-деформований стан елемента обшивки корабля. Розглянуто три форми зварних швів, а саме: V-подібна, Y-подібна та X-подібна. Показано, що при швидкості водяного потоку в 5 м/с мінімальні навантаження при використанні X-подібного (двостороннього) шва менші мінімум в три рази. Максимальні навантаження у межах елемента обшивки корабля при цьому майже однакові. Показано, що найменші напруження в межах зварних швів (без розгляду пластини) виникають при використанні X-подібного шва. У подальшому планується дослідити вплив на пошкоджені зварні шви навантажень від водяного потоку.
Ключові слова: зварний шов, імітаційне та геометричне моделювання, ANSYS, SolidWorks.

С. А. Цыбульник, М. А. Хоцевич, А. И. Товбер
Национальный технический университет Украины «Киевский политехнический институт имени Игоря Сикорского», Киев, Украина

МОДЕЛИРОВАНИЕ НАПРЯЖЕННО-ДЕФОРМИРОВАННОГО СОСТОЯНИЯ ПЛАСТИНЫ СО СВАРНЫМ ШВОМ

Важнейшей задачей при выборе формы конструктивных элементов корпуса корабля является обеспечение достаточной прочности при малой массе. В последние годы в кораблестроении достигнуты большие успехи благодаря применению современных методов сварки для соединения деталей и элементов. Благодаря переходу к сварке, корпуса кораблей стали на много легче, чем при использовании заклепочных соединений. Сварное соединение может обеспечить оптимальную коррозионную стойкость, прочность и экономичность изготовления. Однако необходимо помнить, что любой металл, включая нержавеющие стали, может претерпевать определенные изменения во время сварки. Поэтому необходимо проявить разумную степень осторожности при сварке. В последние годы имитационное моделирование становится очень популярным методом исследования процессов и явлений. Кроме математических методов все чаще авторы со всего мира начинают использовать в качестве основного метода исследования имитационное моделирование и сравнивать результаты с теоретическими или экспериментальными. Существует множество программных продуктов, которые позволяют создавать геометрические модели объектов различной сложности. Некоторые из этих программ даже имеют в своем составе базовые модули имитационного моделирования. Одной из таких программ является SolidWorks, которая была выбрана для создания геометрических моделей в данном исследовании. В данной работе была построена трехмерная модель элемента обшивки корабля со сварными швами. Для определения векторных полей скорости водного потока и его давления в программном комплексе ANSYS проведено имитационное моделирование нагрузки на элемент конструкции с использованием его построенной геометрической модели. Анализ полученных результатов позволил определить напряженно-деформированное состояние элемента обшивки корабля. Рассмотрены три формы сварных швов, а именно: V-образная, Y-образная и X-образная. Показано, что при скорости водного потока в 5 м/с минимальные нагрузки при использовании X-образного (двустороннего) шва меньше минимум в три раза. Максимальные нагрузки в пределах элемента обшивки корабля при этом почти одинаковы. Показано, что наименьшие напряжение в пределах сварных швов (без рассмотрения пластины) возникают при использовании X-образного шва. В дальнейшем планируется исследовать влияние на поврежденные сварные швы нагрузок от водного потока.
Ключевые слова: сварной шов, имитационное и геометрическое моделирование, ANSYS, SolidWorks.

Надійшла до редакції
05 листопада 2018 року

Рецензовано
14 листопада 2018 року