FEM analysis of stiffened plate of bulkheads in river ships

I.D. Savu[1], S.V. Savu[1], R. Iacobici[1]; M.R.Gogâldea[3]

[1]University of Craiova, Romania, [2] Polithenica University of Bucharest, Romania

E-Mail: disavu@yahoo.com

Keywords

Bulkhead, stiffening, intensity of restraint, FEM analysis, displacement

1. Introduction

The solutions considered in the design of river ships are hardly influencing the ships' behaviour during the assembling by welding and during the function of the ship. The design should offer to the ship appropriate mechanical resistance, but maintaining a functional elasticity of all the sections considered together in a ship. The welding of the base material can be considered as a thermal aggression applied from the outside to the material. That thermal aggression produces internal stress of the material and specific deformations. Finally, a modification of the dimension and geometry of the welding components occurs. If the joint has low elasticity, then the modifications could lead to cold cracking. Any crack means damaging, maybe even destroying the joint. Due to the external loading applied by the waves, wind, driving manner, cargo (solid or liquid) any ship requests elasticity during its function [1]. The elasticity of a complex structure, as the ship is, it is not simple to calculate or estimate. That is because of the complex geometry, because of the stiffening solution and because of the 3D orientation of the welds. Usually, the evaluation of the stiffness begins with specific calculations applied to a small component of the ship. That will decrease the complexity of the equations involved in the calculations and it increases the precision of the results.

Within the paper it is presented a study of the influence of the structure type on the stiffness of the bulkhead of a river ship. The study consists of the elaboration of geometrical model which will be simulated and analysed, by using the finite elements method, in specific conditions of loading. The task of the study is the analysing of the intensity of restraint of an element of the structure, which is a part of the bulkhead wall. The most important action during building the bulkhead is its erection by welding. Because of that, the paper will present the evolution of the intensity of restraint during the welding together of two panels of the wall.

2. Intensity of restraint method

The intensity of restraint method has been developed to characterise the mechanical load in a welded joint [2-5]. As is well known the local heating that is produced in a welding process is developing shrinkage and stress inside the welded joint [5-6]. The shrinkage due to welding could increase when the local restraint of the plates that will be welded increases. These two conditions, heating and restraining, can produce modification of dimensions and deformations that could become critical for the welded structure.

Within the locally heated region the material expands transversally and longitudinally related to the welding direction. That causes a certain contraction of the gap. Longitudinal to the welding direction, the surrounding cold material provides restraints that can produce stresses and deformations. During cooling, the solidified metal and the expanded and heat-affected region attempt to contract. The contraction is limited by the surrounding cold metal and the restrained contraction results in tensile stress.

The stress inside weld depends on many factors. Two of those factors are the welded joint shrinkage and the stiffness of the welded joint. The force per unit of weld length required to produce a unit transverse groove edge displacement defines the intensity of restraint (figure 1).

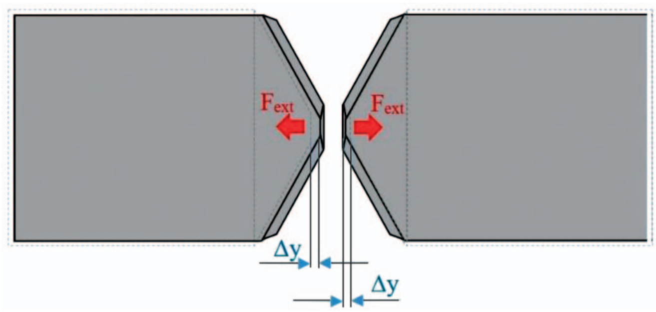

Figure 1. Geometrical elements used to calculate the intensity of restraint.

The simulation of the modification produced by the welding process consists in the application of an external force, Fext, which is similar to the force produced by the welding. Such force acts in the middle of the groove shoulder, is not punctual one, but uniformly distributed on the surface which is the shoulder of the groove and it creates a displacement of each component of the groove. The displacement is on y direction and in figure 1 appears as Δy. The opposition of the material to the external force is considered as intensity of restraint and the indicator can be calculated in FEM analysis.

$$R_y = F_{ext} / (2 \times w \times \Delta y) \quad (1)$$

in which w is the width of the shoulder's surface. It is clear that the intensity of restraint is expressed in kN/mm^2, if the external force is expressed in kN and the dimensions are expressed in mm.

The stress inside weld is longitudinal and transversal. In addition, at common welding processes at steel structures, the transverse contraction at the weld surface is normally larger than in the weld root. This causes a weld angular joint distortion. Furthermore, the movement of the heat source in the welding direction produces an angular distortion perpendicular to welding direction and weld cross section. Altogether the respective restrained distortions may create weld cracks.

3. Geometrical model of the bulkhead and its simulation by using FEM method

The study of the most used designing solutions of the bulkheads (figure 2) helped in the designing of a model of the bulkhead. That element has a relative high stiffness because of the necessity to have a good resistance during exploitation. As already presented, a high stiffness means a high cold cracking susceptibility. Accepting that causality it has been used the element in the analysing of the evolution of the intensity of restraint.

Figure 2. Common shape of the bulkheads walls.

Figure 3 shows the model of the proposed to study element. The accepted dimensions are the same as in the real case. The element models the connection (at the bottom level) between two volume sections of a ship.

To have an appropriate FEM analysis of the model, it had been performed a few initial simulations of the restrictions to decrease the error of the analysing. Plus, the geometry of the model had been analysed in 2D and 3D conception. From this point of view it had been revealed that a 2D analysing involves high errors, up to 30%. Because of that the geometrical model that was used in analysing was a 3D model. Regarding the restriction of the model and regarding the characteristics of the analysing it should be mentioned that specific conditions were used and they will be presented next. The restrictions of the movement were applied to 4 added compression bars (figure 4); the compression bars were meant to model the limited movement of the extremities of the compartment as part of a big and complex structure which the ship is. The application of the force inside the groove was according to the real situation that was given by the used active head. The welding groove was considered to be according to the real groove existing on the ship. The displacement given by the compression bars were corrected when apply in the calculation of the intensity of restraint (figure 4).

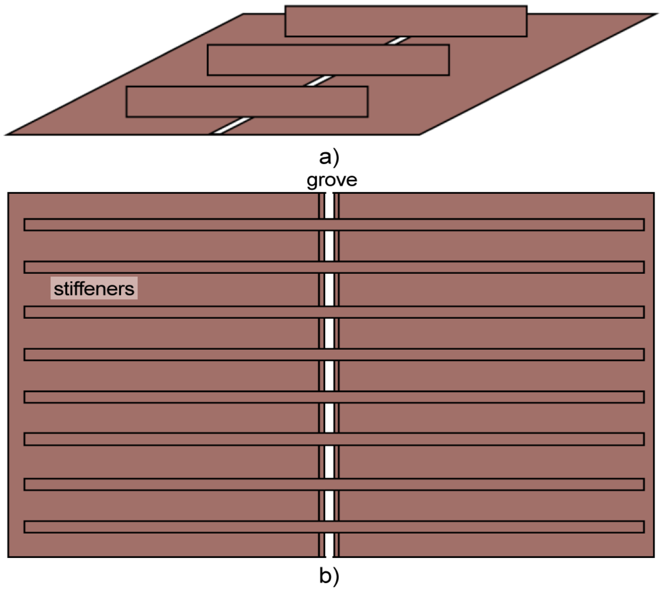

Figure 3. Geometrical model to simulate.

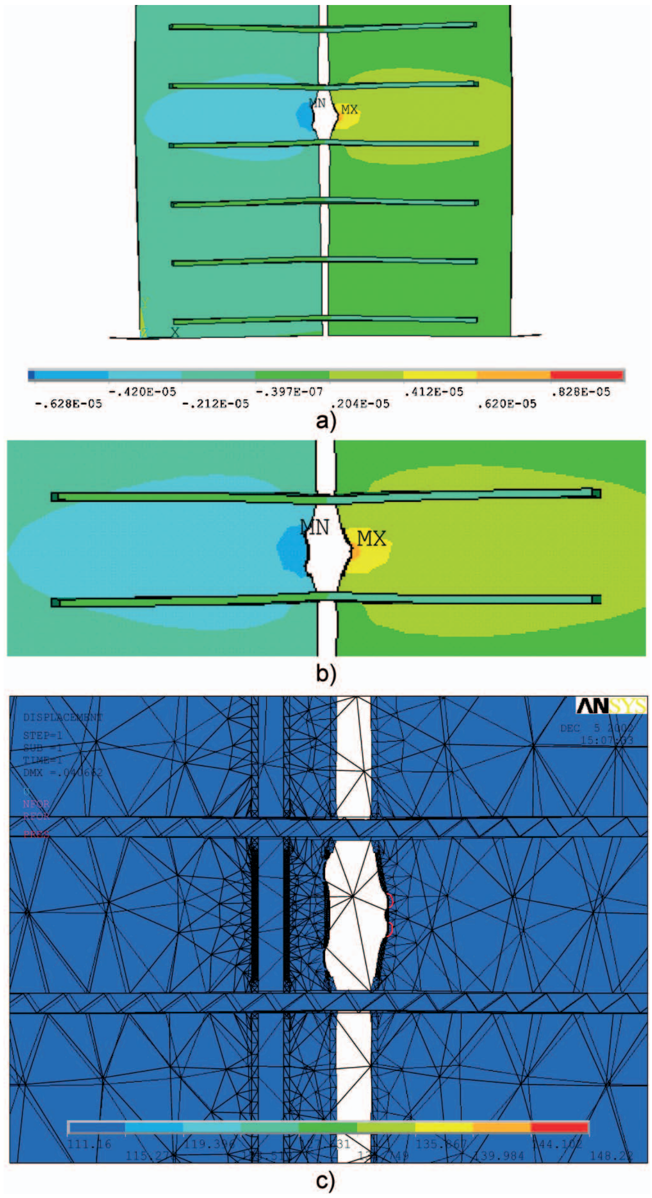

Fig. 4 Geometrical model connected with compression bars (a.) and the displacement error given by the compression bars (b.)

The finite elements that were used in analysing were tetrahedral: Solid Tet 10 node 92.

The results of the calculation of the intensity of restraint for the both method, experimental and FEM analysing are presented in figure 4.

4. Results and discussions

FEM analysis offered evolution of the intensity of restraint during the welding of the elements dedicated to the stiffening of the bulkhead (figure 5).

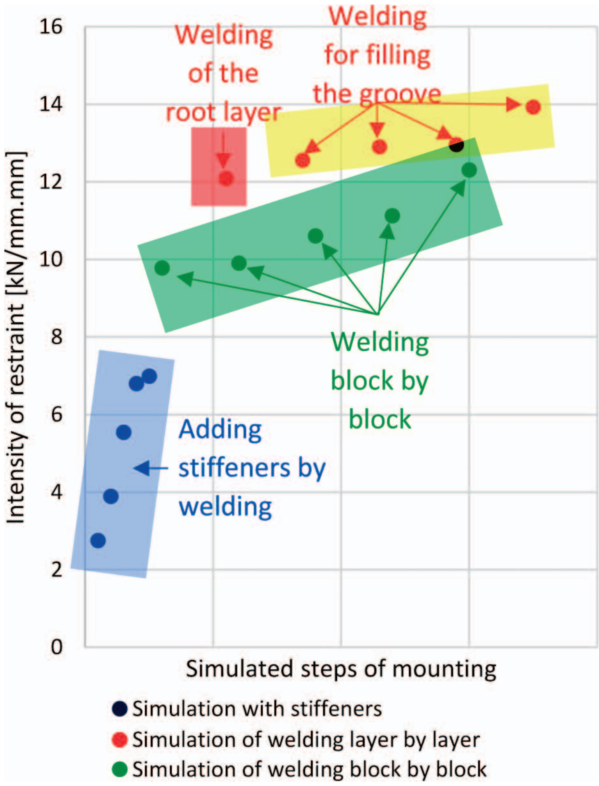

Figure 5. Intensity of restraint as calculated b y FEM simulation.

The applied FEM analysis permits the estimation of the evolution of low complexity welded structure stiffness. To have the information on that evolution, offers to the designer the possibility to choose the most appropriate type of structure during the design of a ship, in order to create a resistant and elastic welded structure.

The graph of figure 5, which was built by using the results that were obtained in the FEM simulation, reveals an increasing of the intensity of restraint during assembling by welding, from 0 up to 14 kN/mm.mm. The increasing of the intensity of restraint has four domains. In the first domain the evolution is almost linear when the number of the stiffeners was uniformly increased. That increasing leads to a decreasing of the assembling elasticity and to an increasing of the cold cracking susceptibility.

Figure 5 shows in the blue area, an almost linear increasing of the intensity of restraint, for a uniform evolution of the building (building by adding new stiffeners on a plane structure). The sharp increasing has a slope that depends on the thickness of the stiffeners, on the density of the stiffeners and on the type of the weld. The last value of the blue area is out of the previous evolutions. That means that the welding of the transversal panels of the bottom compartment is not bringing the same stiffness as the longitudinal bulkheads are. The slope of the increasing of the stiffness is lower in the case of the welding of the transverse stiffeners. In the blue area the increasing of the stiffness is from 2 to 7 kN/mm.mm.

The next area, the red one, is related to the first welding pass. The increasing of the intensity of restraint is similar, in values, as the mounting of the stiffeners (about 5 kN/mm.mm), but the increasing rate is slower. The filling of the groove is presented by the yellow area, which shows lower increasing of the intensity of restraint: from 12.5 to 14 kN/mm.mm. That means that the root pass brings double stiffness comparing to the rest of the 4 welding passes

The green area is related to the welding in blocks of 100 mm length, all 5 passes being done. That manner of welding (by replacing the 5 passes with 5 blocks of complete weld) shows a lower intensity of restraint. The maximum obtained value is comparable to the value obtained after the full root pass (red area).

In the same time, important displacement of the groove were revealed by the analysis.

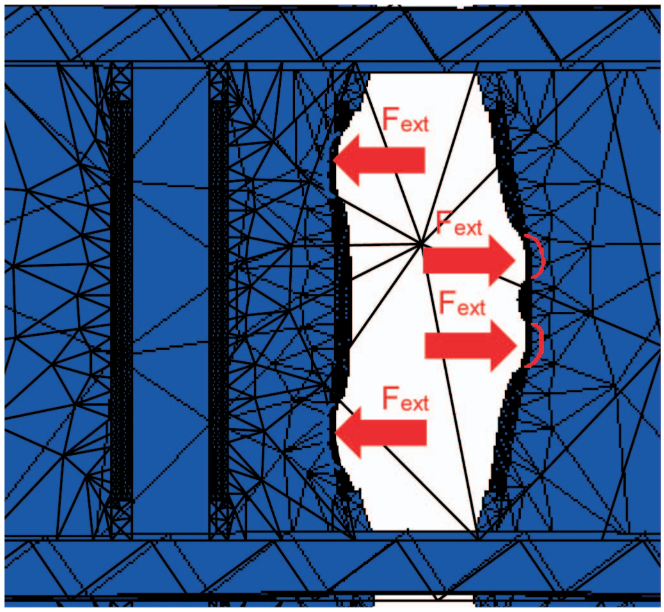

Figure 6. Displacement occurred due to loading.

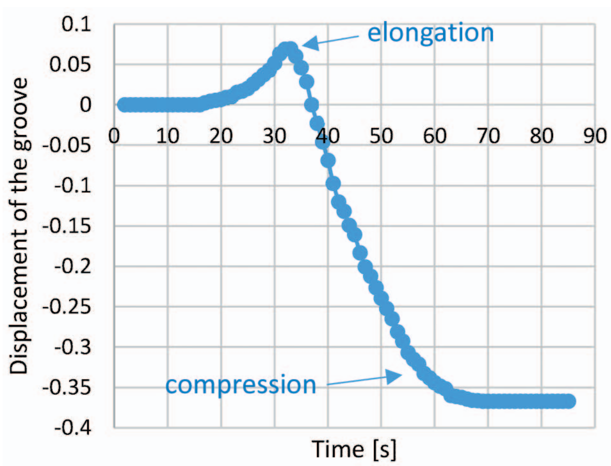

Figure 7. Deformation by welding sequence.

5. Conclusions

FEM analysis can be applied to evaluate the intensity of restraint for a welded structure and subsequently the stiffness of that structure.

The main conclusion is that the stiffening using the longitudinal bulkheads is bringing almost the same influence on the stiffness as the welding of the bottom plates is doing.

Another important conclusion is that the thickness of the bottom plates is a main factor that influences the intensity of restraint of those plates. A high thickness of the plates is hardly reducing the elasticity of the ship and creates the conditions for cold cracking process.

A third parameter that influences the intensity of restraint is the free of stiffeners length of the groove, where the force is applied. The increasing of that part of the groove decreases the intensity of restraint so decreases the stiffness of the structure.

References

[1]. *** - Germanischer Lloyd – Vorschriften für Klassifikation und Bau von stählernen Seeschiffen (Prescriptions for the classification and construction of marine ships of steel), ausgabe 1983, Band III, Kapitel 7A und 7B, with the next versions

[2]. Böllinghaus, Th., Hoffmeister, H., Dangeleit, A. – A scatterband for hydrogen diffusion coefficients in micro-alloyed and low-carbon structural steels, Welding in the world, vol. 35, no.2, pag. 83-96

[3]. Böllinghaus, Th., Hoffmeister, H., Littich, M. – Application of the IRC test for assessment of reaction stresses in tubular joints with respect to hydrogen assisted weld cracking, IIW Doc IX-1900-98

[4]. Kannegießer, Th. - Untersuchungen zur Entstehung schweißbedingter Spannungen und Verformungen bei variablen Einspannbedingungen im Bauteilschweißversuch (Research on the internal stres and deformation given by the welding processes in variable restraints conditions on construction steel), Dissertation, Universität Otto von Guericke, Magdeburg, 24.03.2000

[5]. Ruyter, Erich – Development and assessment of welding procedures for avoiding weld joint cracking in highly restrained offshore steel structures, Disertation, Universität der Bundeswehr, Hamburg, 23.08.1993

[6]. Böllinghaus, Th. – Zur Bestimmung Risskritischer Schrumpfbehinderungen und Wasserstoff-verteilungen in Schweissverbindungen durch Numerische Simulation (On the susceptibility to hydrogen assisted cracking of the welded joints by numerical simulation), Dissertation, Universität der Bundeswehr, Hamburg, 03.05.1995

Calendar of International and National Events

2018			
April 10 - 12	Welding - Fair of Welding Technology and Equipment	Kielce, Poland	http://www.targikielce.pl/en/
April 10 - 13	Exhibition Welding and Cutting	Minsk, Belarus	http://www.minskexpo.com/
Aprilie 26 - 27	The conference "WELDING 2018"	Timisoara, Romania	http://www.asr.ro/index.php/news
June 11 - 13	International conference "Titanium 2018: Production and application in Ukraine"	Kiev, Ukraine	http://pwi-scientists.com/eng/titan2018
June 11 - 15	12th European conference on Non-Destructive Testing	Gothenburg, Sweden	http://www.ecndt2018.com
July 15 - 20	The 71st IIW Annual Assembly & International Conference	Bali, Indonesia	https://www.iiw2018.com/
August 23 - 24	Nordic Welding Conference	Reykjavík, Iceland	http://nwc2018.is/
August 29 - 31	The 4th IIW young professionals international conference YPIC2018	Yutz, France	https://www.ypic2018.com; www.weezevent.com/ypic2018
October 10 - 12	The 4th IIW South-East Europe International Congress	Belgrade, Serbia	http://seeiiw2018.duzs.org.rs/
November 01 - 02	The 9th International Conference "Innovative tehchnologies for joining advanced materials - TIMA18	Timişoara, Romania	http://www.isim.to/tima