THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2016

STATE OF STRESS IN THE HULL OF A BOAT MADE FROM GRP TO ENCHANCE THE RESISTANCE OF BODY STRUCTURE

Mădălina Prodan
“Dunărea de Jos” University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:madalinaprodan_mp@yahoo.com

Vlăduţ-Alexandru Vicol
“Dunărea de Jos” University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:vladutalexandruvicol@yahoo.com

Adrian Caramatescu
“Dunarea de Jos” University of Galati,
Faculty of Naval Architecture
47 Domneasca Street, 800008, Romania,
E-mail:adrian.caramatescu@ugal.ro

Costel Iulian Mocanu
“Dunărea de Jos” University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:costel.mocanu@ugal.ro

ABSTRACT

In naval industry researchers try to discover new lighter and stronger materials but also try to design new shapes to improve the body structure keeping a low weight. This study will try to propose a new shape for better structure or at least a lighter one.

Keywords: FEM, impact, Johnson-Cook

1. INTRODUCTION

Composite materials have emerged as a necessity due to the appearance of new areas – aeronautics, shipbuilding, automotive, military equipment. These were created to meet special properties requirements in terms of special mechanical strength and stiffness, corrosion, resistant to chemical agents, low weight, dimensional stability, resistance to variable shock, wear resistance, insulating properties, etc.

The reinforcement of GRP is glass fiber. The criterion that underlay this is economical, meaning lower cost of production of fibers, obviously taking into account the characteristics of the material. It seems that this material meets the terms of mechanical requirements for the construction of small and medium size ships.

The rapid development of GRP craft was possible because of the advantages the material brings: low weight, low price of raw materials, low price of the end product for a production of large series as well as the possibility of a complex shapes for various items used in vessel construction.

2. THEORETICAL BACKGROUND

Finite element modelling of the boat section was made using Abaqus software.

Abaqus is a suite of powerful engineering simulation programs, based on the finite element method, which can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus contains an extensive library of elements that can model virtually any geometry. It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock. Designed as a general-purpose simulation tool, Abaqus can be used to study more than just structural (stress/displacement) problems.

Before starting to define this or any model, it is necessary to decide which system of units will be used. Abaqus has no built-in system of units. All input data must be specified in consistent units. Some common systems of consistent units are shown in Fig.1.

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2/in^4

Fig.1. Common system of units (Abaqus 6.12 Documentation, 2012)

3. BODY STRUCTURE DESCRIPTION AND ENHANCE PROPOSAL

The actual hull used in this project is property of PLASMA SRL and it is used with acceptance of M. Sc. Eng. Adrian Caramatescu, technical manager (Fig.2). The cross section of the original structure profile is close to a rectangle (Fig.3).

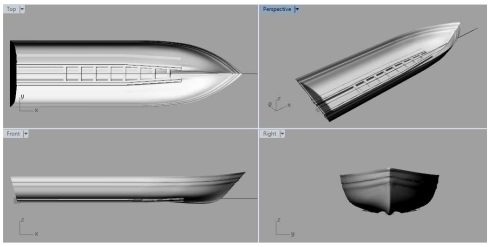

Fig.2. Actual body shape and structure (hull form property of PLASMA SRL)

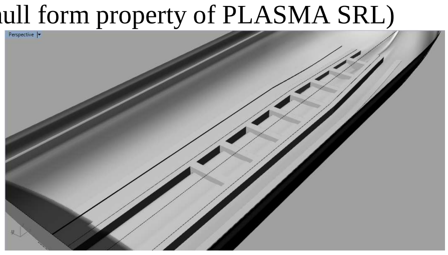

Fig.3. Detail of actual structure

The enhancement proposal consists in changing the structure profile from rectangular to a circular one (Fig.4).

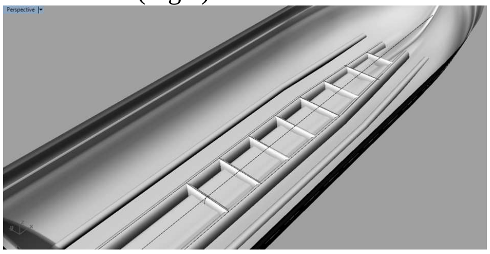

Fig.4. Proposed structure.

All changes, preparation and export in .sat file were made using Rhinoceros 5 with a 90 days demo license. Rhinoceros (known also as Rhino or Rhino3D) is a commercial 3D computer graphics and computer-aided design (CAD) application software.

Rhinoceros geometry is based on the NURBS mathematical model, which focuses on producing mathematically precise representation of curves and freeform surfaces in computer graphics.

For this study it was considered a section of hull of about 1440 mm in the middle part of the boat (Fig.5, Fig.6).

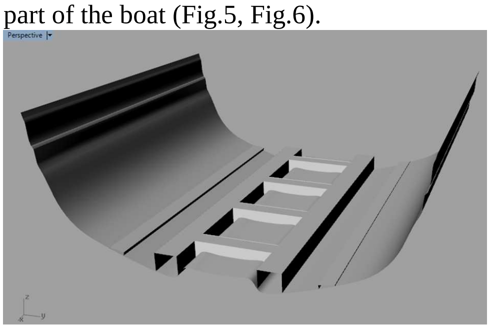

Fig.5. Original structure that will be analyzed.

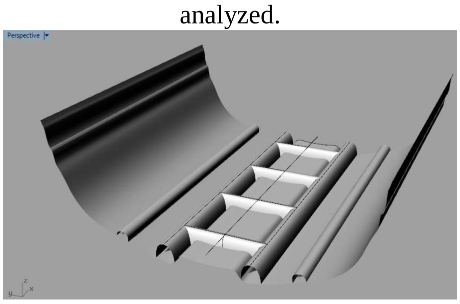

Fig.6. Modified structure prepared for import in Abaqus

3. NUMERICAL ANALYSIS OF GRP BOAT

Finite element modelling of the boat section was made using Abaqus software.

Abaqus is a suite of powerful engineering simulation programs, based on the finite element method, which can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus contains an extensive library of elements that can model virtually any geometry.

It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock.

Designed as a general-purpose simulation tool, Abaqus can be used to study more than just structural (stress/displacement) problems.

Material properties for all elements must be specified. While high-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the Abaqus results is limited by the accuracy and extent of the material data.

In this analysis material properties are shown in Table 1:

Table 1 Material properties.

Property	Value	Unit
Density	1.75e-9	T/mm^3
Young's modulus	9.3	MPa
Poisson's ratio	0.276	-
Thickness	5	mm

Boundary conditions are used to constrain portions of the model to remain fixed (zero displacements) or to move by a prescribed amount (nonzero displacements). In a static analysis enough boundary conditions must be used to prevent the model from moving as a rigid body in any direction; otherwise, unrestrained rigid body motion causes the stiffness matrix to be singular. A solver problem will occur during the solution stage and may cause the simulation to stop prematurely. Abaqus/Standard will issue a warning message if it detects a solver problem during a simulation.

In this case the boundary condition blocks all degrees of freedom on the top sides (where deck should be Fig.7).

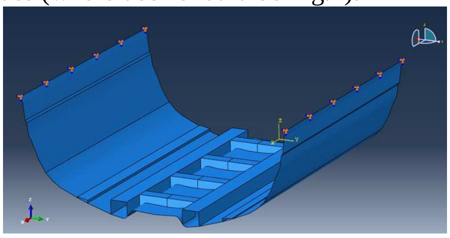

Fig.7. Boundary conditions all degrees of freedom blocked

The load consider for this numerical analysis is a pressure equal to hydrostatic pressure at a depth of 200 mm. Applied pressure will be equal with density of water multiplied by gravitational acceleration multiplied by depth.

- density of water considered at 1000 kg/m3

- gravitational acceleration is considered 9.81 m/s2

- depth 0.2 m

P= 1000 * 9.81 * 0.2 = 1962 N/m2

Modelling is done using mm and from this results that the pressure must be in N/mm2. P = 1962*10-6 = 0.00196 N/mm2.

After applying pressure in Abaqus, geometry will be like in Fig.8.

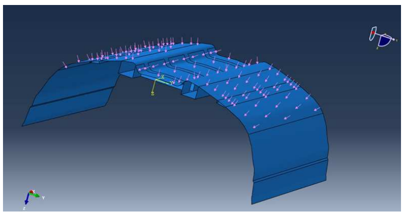

Fig.8. Hydrostatic pressure applied normal to surface.

Finite elements and nodes define the basic geometry of the physical structure which are modelled in Abaqus. Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements - that is, which nodes belong to which elements - comprise the model geometry. The collection of all the elements and nodes in a model is called the mesh. Generally, the mesh will be only an approximation of the actual geometry of the structure.

The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation. The greater the mesh density (i.e., the greater the number of elements in the mesh), the more accurate the results. As the mesh density increases, the analysis results converge to a unique solution, and the computer time required for the analysis increases.

In this case, elements used to mesh the geometry is S4R.

For choosing the optimal dimension of elements it was performed a mesh sensitivity test which consist in running same geometry with different element sizes and register number of elements and maximum stress obtained with those elements. Results of different tests can be seen in Table 2.

Table 2 Mesh sensitivity test results.

Test no.	**Mesh element size**	**No. of elements**	**Max. stress [MPa]**
1	75	1930	20.15
2	50	2977	20.99
3	40	4051	21.61
4	30	5969	22.29
5	25	8844	22.37
6	20	13447	22.04
7	15	21905	21.97

Making a graphic representation of the corelation between the number of elements into the model and the maximum stress, results that the maximum stress is approximately constant. The optimal choice of element size is at the beginning of steady horizontal curve because this will mean closer to real results and reasonable time of running of the numerical analysis (Fig.9).

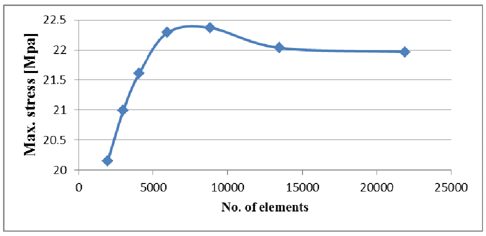

Fig.9. Mesh sensitivity diagram

We can see that the graph flattens up around 13500 elements so the mesh element size for these analysis will be set to 20.

Meshed hull section for both types of structures is obtained like in Fig.10 for rectangular structure and Fig.11 for rounded structure.

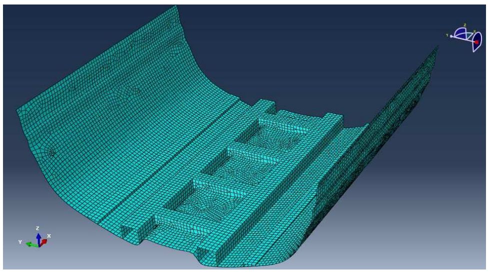

Fig.10. Mesh obtained for actual body structure (13447 elements)

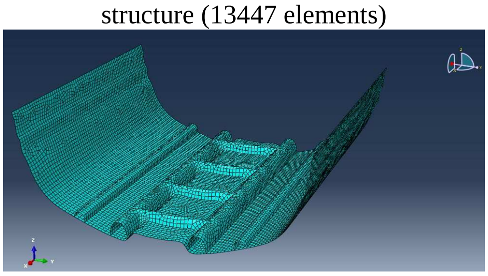

Fig.11. Mesh obtained for proposed structure (12769 elements)

4. RESULTS

Graphical post processing can be found in the Visualization module of Abaqus/CAE. It allows to view the results graphically using a variety of methods, including deformed shape plots, contour plots, vector plots, animations, and X–Y plots.

For rectangular shaped structure, contour plot is represented in Fig.12 and for the proposed structure it can be seen in Fig.13.

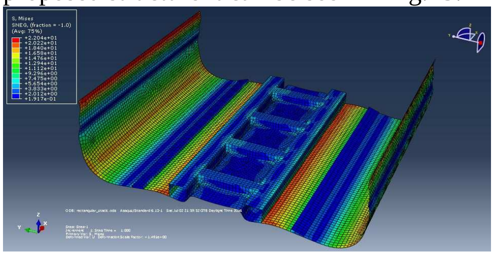

Fig.12. Von Misses stress for rectangular shaped structure

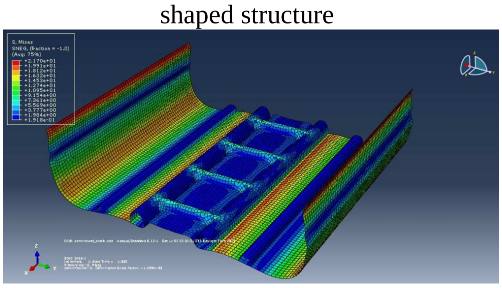

Fig.13. Von Misses stress for circular shaped structure

It is easy to observe that the maximum stress is equivalent in the two study cases, but in the following part it will be compared to the weight difference between the two structures.

The density of the material is 1750 kg/m^3 , the thickening of both profiles is 5 mm and the length for calculation will be 1000 mm. Cross sectional area for rectangle profile is equal to 1340 mm^2 and for circular section is 1100 mm^2. Weight for 1000 mm rectangle profile is 2.35 kg and for 1000 mm of proposed profile is 1.93 kg.

5. CONCLUSIONS

The proposed profile did not offer higher resistance to the body hull of the boat but the weight of the structure, using the circular profile, is reduced by 17.9%.

REFERENCES

[1]. **Costel Iulian Mocanu –** *"Rezistenţa materialelor", second edition, Zigotto Publishing Press, Galaţi, 2007*

[2]. **Leonard Domnişoru,** *"Structural Analysis And Hydroelasticity Of Ships", The University Foundation „Dunarea de Jos" Publishing House Galati, 2006*

[3]. **P.K. Mallick,** *"Fiber-reinforced composites Materials, Manufacturing and Design", third edition, CRC Press, 2007*;

[4]. ***Abaqus,*** *6.12 Documentation*, Dassault Systèmes, 2012.

Paper received on December 30th, 2016