THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620, e-ISSN 2668-3156
DOI: https://doi.org/10.35219/AnnUgalShipBuilding/2024.47.18
2024

THE INFLUENCE OF THE CONSTRUCTIVE SHAPES OF ALUMINUM PROFILES ON IMPACT PHENOMENA

Gabriel POPESCU
University "Dunarea de Jos" of Galati,
Faculty of Naval Architecture, Galati, Domneasca
Street, No. 47, 800008, Romania,
E-mail: gabriel.popescu@ugal.ro

ABSTRACT

The purpose of this study is to analyze the results obtained from the impact for aluminum profiles in four constructive variants, two with connections and two with bevels.The forms significantly influence the results from a quantitative and qualitative point of view, making it possible to recommend the choice of the type of profile.

Keywords: shapes, impact, deformations, tensions

1. Introduction

In the field of ship hydrodynamics, Ship description

In the shipbuilding, automobile and aeronautical industries, aluminum profiles are frequently used, manufactured profiles, not laminated ones. This study proposes the analysis of the influence of the shapes of four types of profiles in the case of their response to impact.

The proposed profiles can be used as external protection bars in the industries mentioned above.

Table 1 shows the shape characteristics of the four profiles.

Table 1 Main characteristics of the shapes

Shape number	The geometric variant of the shape
Shape 1	Connection with a radius of 15mm
Shape 2	Connection with a radius of 20mm
Shape 3	Bevel 15 mm
Shape 4	Bevel 20 mm

2. 3D models

The creation of the study models was done in SolidWorks using the specific commands for generating the sketches of the four variants and then extruding them to obtain the three-dimensional models. It should be noted that all four versions had as their initial 2D shape the same rectangle, which in the first two versions was processed with connections of 15 and 20 millimeters, respectively, and the last two were chamfered with the same values, 15 and 20 millimeters.

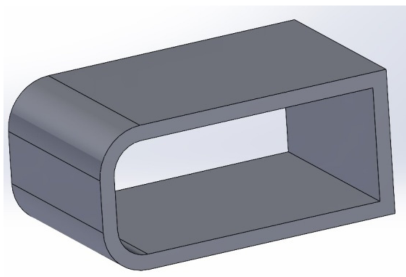

Fig. 1 Model 1 Radius 15mm

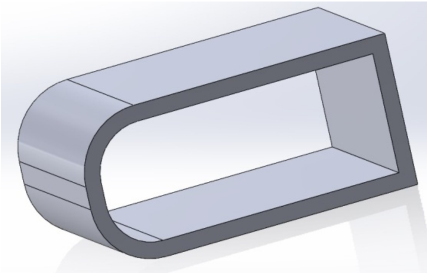

Fig. 2 Model 2 Radius 20mm

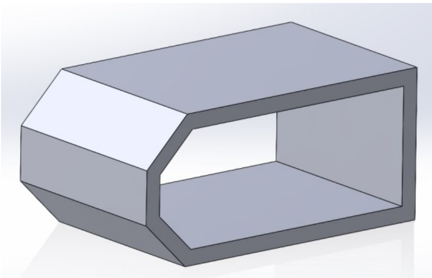

Fig. 3 Model 3 Chamfer 15mm

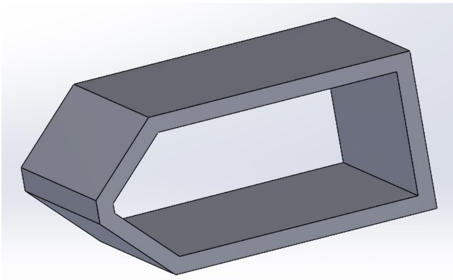

Fig. 4 Model 4 Chamfer 15mm

3. Defining the analysis conditions

The Solidworks calculation system has the possibility of analysis through simulation and impact. One of the options is the drop test.

As this type of simulation is a dynamic one, that is, a development over time of the analysis, the equation of the general movement is expressed by the sum of the inertial forces Fin(t), the elastic forces Fel(t) and the damping forces Fam(t). The resultant expresses the impact force.

The solution of such an equation can be done in two methods of integration as a function of time, implicit as well as explicit methods.

In the case of the explicit method, the stiffness matrix does not have to be decomposed and for this reason, there are economic runs in terms of time that translate into a reduction of the necessary resources.

The time step must be defined so that it has a value lower than the critical one. This condition is imposed because for a correct result the solution must be convergent.

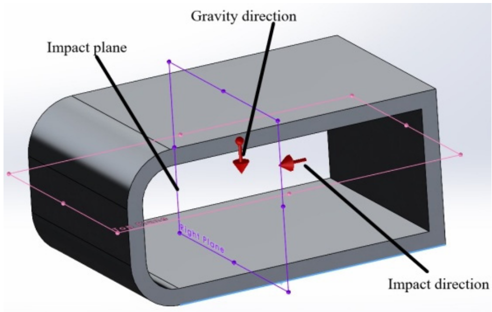

Fig. 5

For these models, the same 6061 Alloy material with the following characteristics was selected

6.9e+10	Elastic Modulus	N/m^2
0.33	Poisson's Ratio	N/A
2.6e+10	Shear Modulus	N/m^2
2700	Mass Density	kg/m^3
124084000	Tensile Strength	N/m^2
55148500	Yield Strength	N/m^2

4. Results and discussion

The runs of the four models for the same impact speed of 100 m/s (formula 1 cars) are presented in the following figures containing for each case the displacements and stresses on the model as well as the graphs of the evolution of these values during the analysis time for the same node in the flat area of impact.

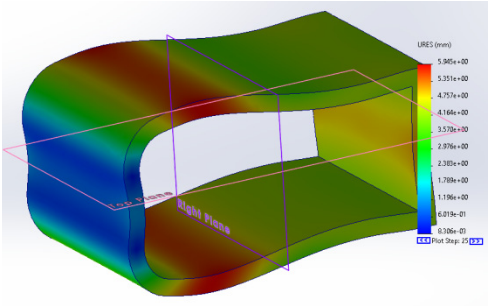

Fig. 6 Displacements on 25 step Model 1

Results for model 2

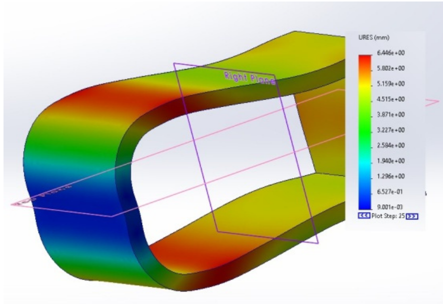

Fig.10 Displacements on 25 step Model 2

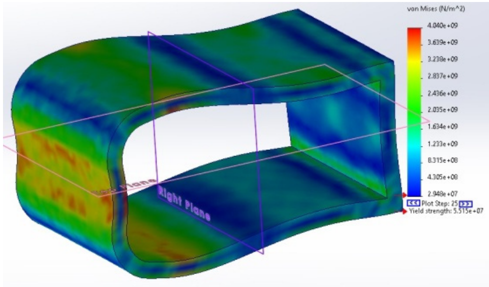

Fig.7 Stresses on 25 step Model 1

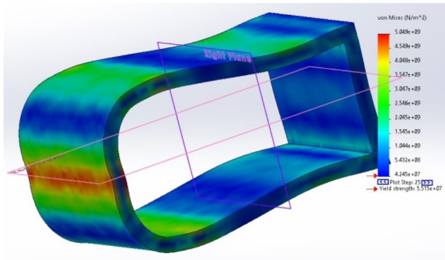

Fig.11 Stresses on 25 step Model 2

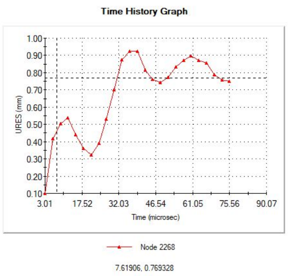

Fig. 8 Displacement time evolution Model 1 (node 2268)

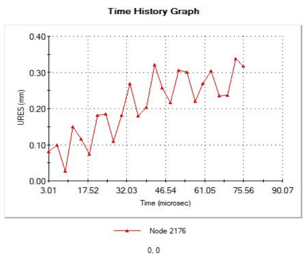

Fig. 12 Displacement time evolution Model 2 (node 2176)

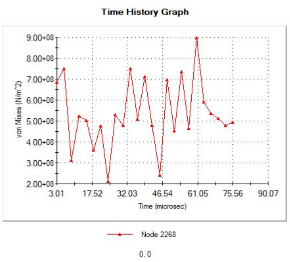

Fig. 9 Stresses time evolution Model 1 (node 2268)

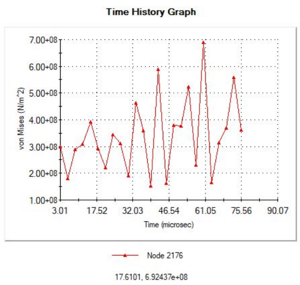

Fig. 13 Stresses time evolution Model 2 (node 2176)

Results for model 3

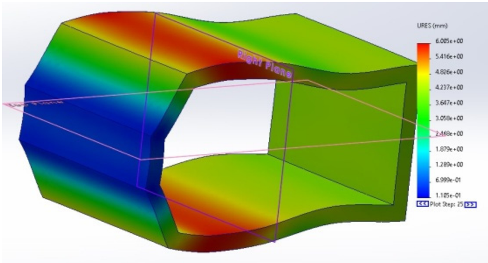

Fig. 14 Displacements on 25 step Model 3

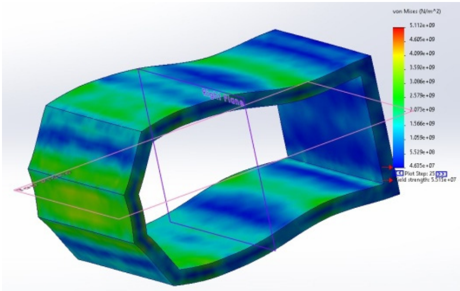

Fig. 15 Stresses on 25 step Model 3

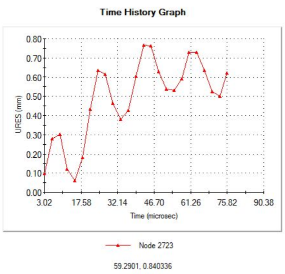

Fig. 16 Displacement time evolution Model 3 (node 2723)

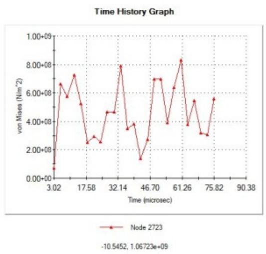

Fig. 17 Stresses time evolution Model 1 (node 2723)

Results for model 4

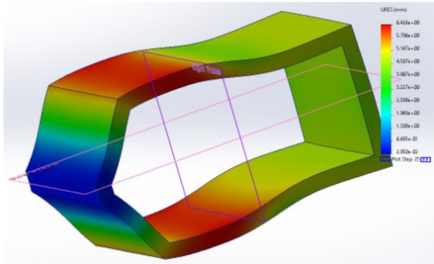

Fig. 18 Displacements on 25 step Model 4

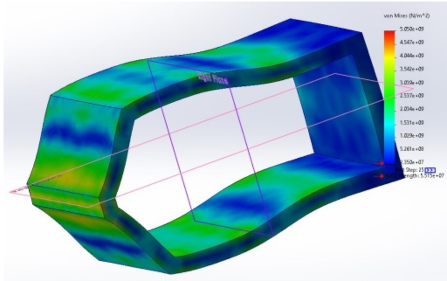

Fig. 19 Stresses on 25 step Model 4

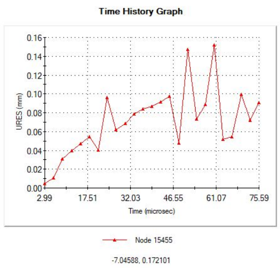

Fig. 20 Displacement time evolution Model 4 (node 15455)

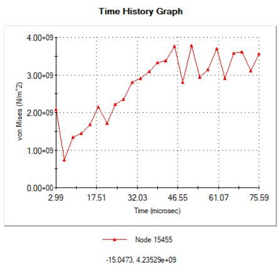

Fig. 21 Stresses time evolution Model 4 (node 15455)

4. Discussion

For an analysis and interpretation of the results, table 2 was created, which contains for each model the maximum value for the displacement of the non in the impact zone as well as the value of the maximum tension for the same node.

	Model 1	Model 2	Model 3	Model 4
Disp.	0.92	0.34	0.77	0.145
Stress	8.9+08	6.8+08	8.3+08	3.57+09

Disp. [mm] , Stress[N/m^2]

For a successful analysis, the analyzed nodes are presented in figures 22, 23, 24, 25, all having in common the flat area of impact. Being models with different geometries and the discretization being automatic, the number of the node where the displacements and stresses were presented have different numbers.

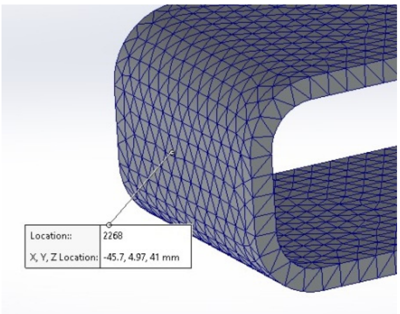

Fig. 22 The analysis node Model 1

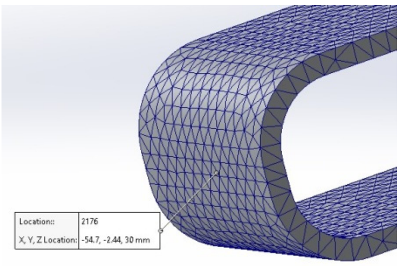

Fig. 23 The analysis node Model 1

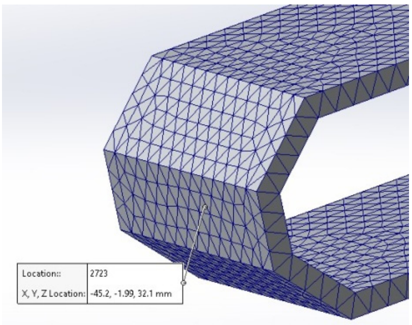

Fig. 24 The analysis node Model 3

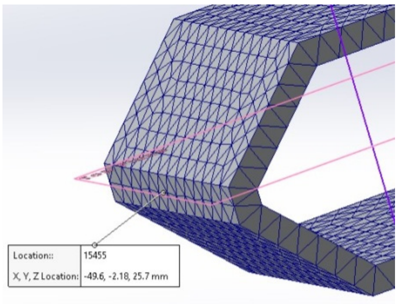

Fig. 25 The analysis node Model 4

5. Concluding remarks

For the interpretation of the results, comparative graphs were made with the values of the resulting displacements.

In order to study the influence of the shapes on the displacements, comparative graphs were generated between the two models with connections figure 26, figure 27, between the two models with bevels,figure 28, figure 29, as well as between the models with connections and bevels of the same value figure 30 and figure 31.

In essence, by increasing the connection radius from 15mm to 20mm, the frontal impact surface is reduced.

In essence, by increasing the connection radius from 15mm to 20mm, the frontal impact surface is reduced. The same change in geometry is recorded by changing the texture from 15mm to 20mm.

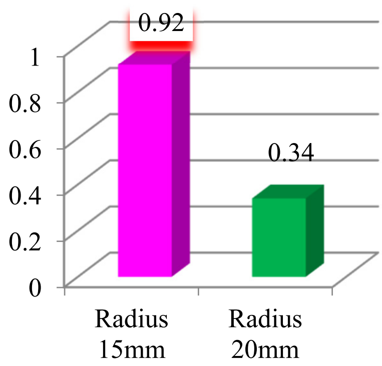

Fig. 26 Displacements Model 1 Model 2

In the case of the comparison of the displacement values between the two models with connections, a decrease in their value by 63.04% is found.

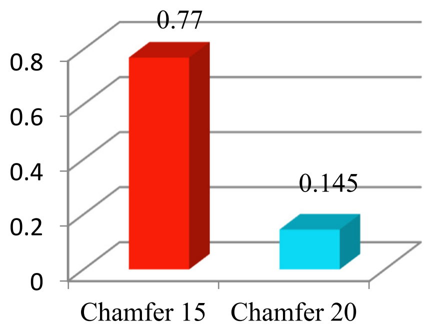

Fig. 27 Displacements Model 3 Model 4

In the case of the comparison of the values of the displacements between the two models with tapering, a decrease in their value is found by 81.16%.

In order to be able to appreciate the geometric solution with the effect of decreasing displacements, models 1 and 3, respectively 2 and 4, are compared.

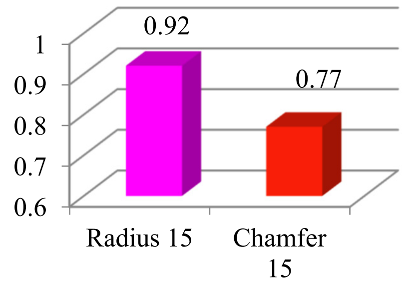

Fig. 28 Displacements Model 1 Model 3

In the case of the comparison of displacement values between models 1 and 3, a decrease of 16.30% is found

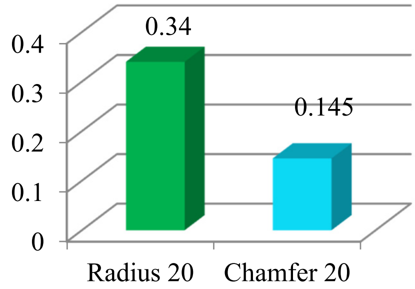

Fig. 29 Displacements Model 2 Model 4

In the case of the comparison of displacement values between models 2 and 4, a decrease of 57.35% is found.

It is found that the most important decrease in displacements is recorded between models 3 and 4. By increasing the value of the texture from 15 to 20mm, the decrease is 81.16%. Thus, it is recommended to choose the geometric version of model 4.

.

REFERENCES

[1] Solid Solutions Ireland, "*SOLIDWORKS TEACHER TRAINING MANUAL*",2020.

[2] M. Nechita, *"Barge's study under impact loads*," IOP Conference Series Materials Science and Engineering, 2019 .

Paper received on November 14th, 2024