Proceedings of the PRADS2013, pp. 575~582
20-25 October, 2013
CECO, Changwon City, Korea

# Optimisation Methodology Applied to Ship Design

Amirouche Amrane[1)], Abbas Bayatfar[1)] and Philippe Rigo[1)]

[1)]ANAST, University of Liege, Belgium

## Abstract

Optimization with mathematical algorithms can be very helpful to find the best solution (minimum weight, minimum cost, maximum inertia, …). Typically, finite element analysis (FEA) tools are used in ship structural assessment. But, to build a FEM model from a CAD one is not easy and needs a big amount of manual work. This paper presents an innovative optimization workflow by which the following steps are automatically carried out, without any manual intervention. First, from the 3D CAD model, an idealized CAD model is created by the idealization module to take into account the FEM needs. Then, the idealized CAD model is transferred to the FEM tool. After that, the FEM model is meshed, loaded and solved. The obtained results (stress, volume etc.) are transferred to the optimizer. The optimizer evaluates the values of the objective function and the constraints previously defined and modify the design variables (plate thickness and the stiffener scantling) to create a new structural model. After several iterations, the optimum solution is evaluated.

## Keywords

Structural optimization; ship structure; BESST; scantling; FEM.

## Introduction

The present work has been done in the framework of the European Project BESST "Breakthrough in European Ship and Shipbuilding Technologies". The research leading to these results has received funding from the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 233980.

The optimization process developed on the present work is presented on the following steps. The 3D CAD model is transferred from the CAD software to the idealization module. The idealization module will generate a simplified geometry which belongs to the FEM needs and then the idealized CAD model is transferred to the FEM tool to create a meshed and loaded structural model. After solving, the results (stress, displacement, volume etc.) are transferred to the optimizer. The optimizer evaluates the values of the objective function and the constraints previously defined and modify the design variables (plate thickness and the stiffener scantling) to create a new structural model. After FEM solving, the results (stress, displacement, volume etc) are transferred again to the optimizer.

The softwares AVEVA Marine (Bohm, 2010; Doig 2009 & 2010) and FORAN are used as CAD software. So, an idealized geometry is created and transferred to ANSYS (FEA tool) to build the FEM model. For the optimization process, the modeFRONTIER platform is used. This platform has a full library of algorithms for both single and multi-objective optimization (genetic algorithms …) and allows easy coupling to ANSYS.

As a case study, the scantling optimization is performed for a typical deck structure for local optimization. Structural and geometrical requirements are imposed.

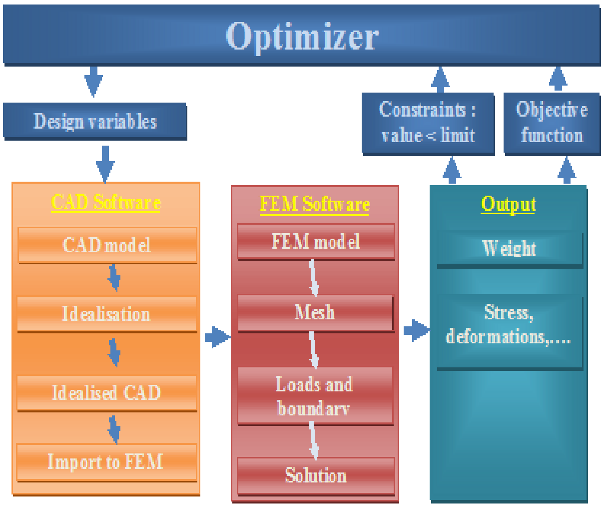

**Fig. 1: Optimisation Workflow**

## Case Study

The model studied is a deck structure shown in Fig. 2. The structure is constituted by deck Plate, longitudinal girders, transversal frames, longitudinal stiffeners and two longitudinal walls connected to the deck structure.

The meshed structure is shown in Fig. 3. Plate, girders and frames are modelled with shell elements. The longitudinal stiffeners are modelled with beam elements.

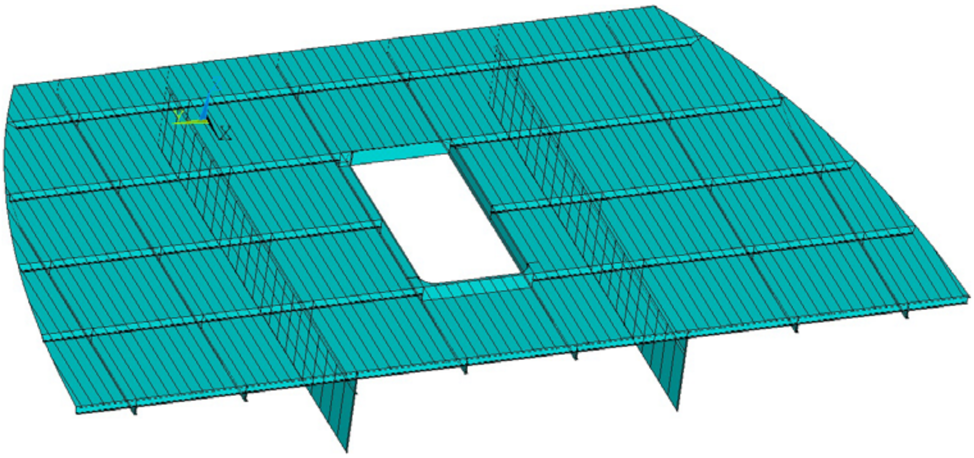

Fig. 2: Deck Structure

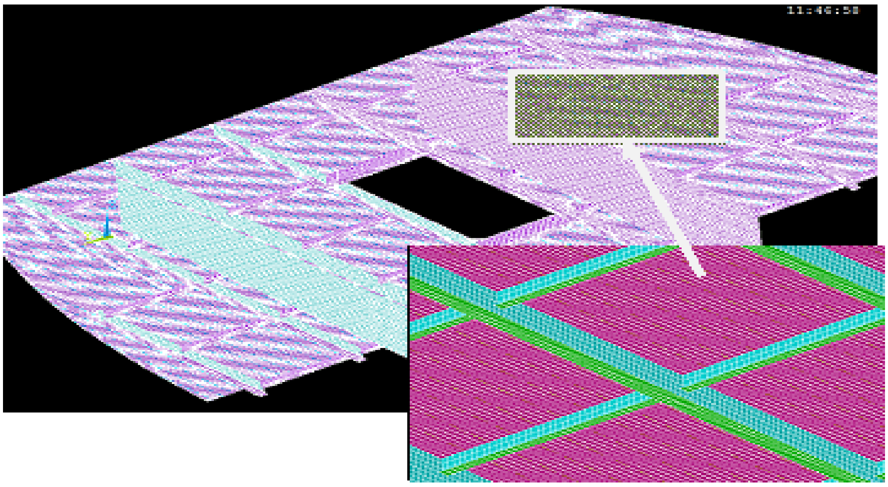

Fig. 3: Mesh Generation

The boundary conditions are assumed to suppress the displacements in x-, y- and z-direction at aft and fore boundaries of the model. A 0.02 MPa lateral pressure is applied on the deck.

The initial scantling is defined in Table 1. The Young's modulus E =2.060x$10^5$ MPa and the Poisson ratio is 0.3.

**Table 1: Initial Geometry**

| Element | Value (mm) |
|---|---|
| Longitudinal girders: flange width | 100 |
| Longitudinal girders: web height | 600 |
| Longitudinal girders: flange thickness | 10 |
| Longitudinal girders: web thickness | 5 |
| Transversal frames: flange width | 100 |
| Transversal frames: web height | 300 |
| Transversal frames: flange thickness | 10 |
| Transversal frames: web thickness | 5 |
| Deck thickness | 14 |
| Longitudinal wall thickness | 10 |
| Deck stiffener | Hp100x8 |
| Longitudinal wall stiffener | Hp160x8 |
| No. of stiffeners between girders | 9 |

The following, the design variables are considered:

- Plate thickness
- Longitudinal girders : web height and thickness, flange breath and thickness
- Transversal frames : web height and thickness, flange breath and thickness
- Longitudinal stiffeners profile : web height and thickness, flange breath and thickness
- Number of stiffeners between girders

The maximum and minimum dimensions allowed are presented in Table 2. The values of plate thicknesses and stiffeners profiles are taken from catalogues.

The volume of the structure is defined as the objective function to minimize. As a constraint, the maximum stress is imposed to be less than 235 MPa.

Some geometrical constraints are imposed:

- Web thickness of frames less than the double of the plate thickness

**Table 2: Design Variables Limits**

| | Min (mm) | Max (mm) |
|---|---|---|
| Long. girders: flange width | 50 | 500 |
| Long. girders: web height | 200 | 1000 |
| Long. girders: flange thickness | 5 | 40 |
| Long. girders: web thickness | 5 | 40 |
| Trans. frames: flange width | 50 | 500 |
| Trans. frames: web height | 200 | 1000 |
| Trans. frames: flange thickness | 5 | 40 |
| Trans. frames: web thickness | 5 | 40 |
| Deck thickness | 5 | 40 |
| Long. wall thickness | 5 | 40 |
| No. of stiffeners between girders | 5 | 15 |
| Deck stiffener | Hp80x6 | Hp430x20 |
| Long. wall stiffener | Hp80x6 | Hp430x20 |

- Web thickness of stiffeners less than the double of the plate thickness
- the plate thickness less than the double of web thickness of stiffeners
- Web height of the frames greater than the web height of stiffeners

## Results and Discussion

In this section, first, the obtained optimisation results are provided based on the optimisation workflow using AVEVA Marine, ANSYS Classic and modeFRONTIER as CAD software, FEA tool and optimiser respectively ( shown in Figs. 4-9 and Table 3). Then some results are given based on the optimisation workflow using FORAN, ANSYS Workbench and modeFRONTIER as CAD software, FEA tool and optimiser respectively (shown in Figs. 10-11).

From Figs. 4 and 5, we can see the variation of the objective function and maximum Von Mises stress. The optimum is reached on the 210th iteration. The minimum value of the weight is 83661.9 kg. The Von Mises stress at this iteration is 220.4 MPa. Fig. 6 shows the obtained FE result for the optimum solution.

For a comparison, additional to the initial design, the results of other iterations are plotted. On the iteration 179, we have the minimum value of the weight 79589.2 kg. This value is lower than optimum solution but even if the level of the maximum stress here is lower than the limit (226.2MPa) but one geometrical constraint (Web thickness of stiffeners less than the double of the plate thickness) is not respected, see Fig. 7. So, this solution is not feasible.

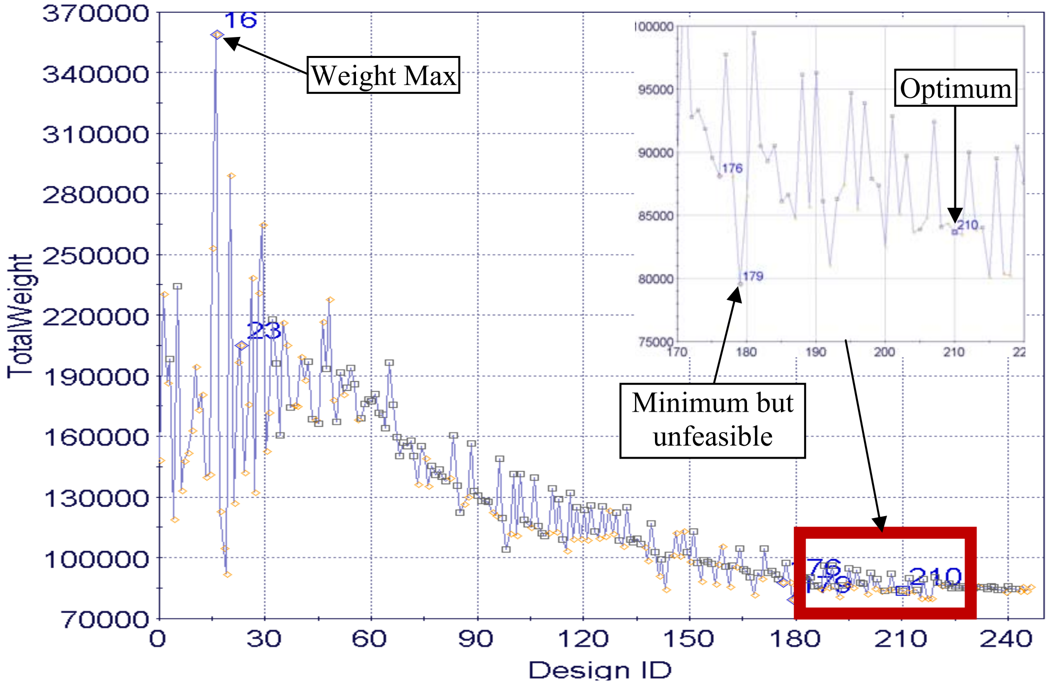

**Fig. 4: Total Weight Variation**

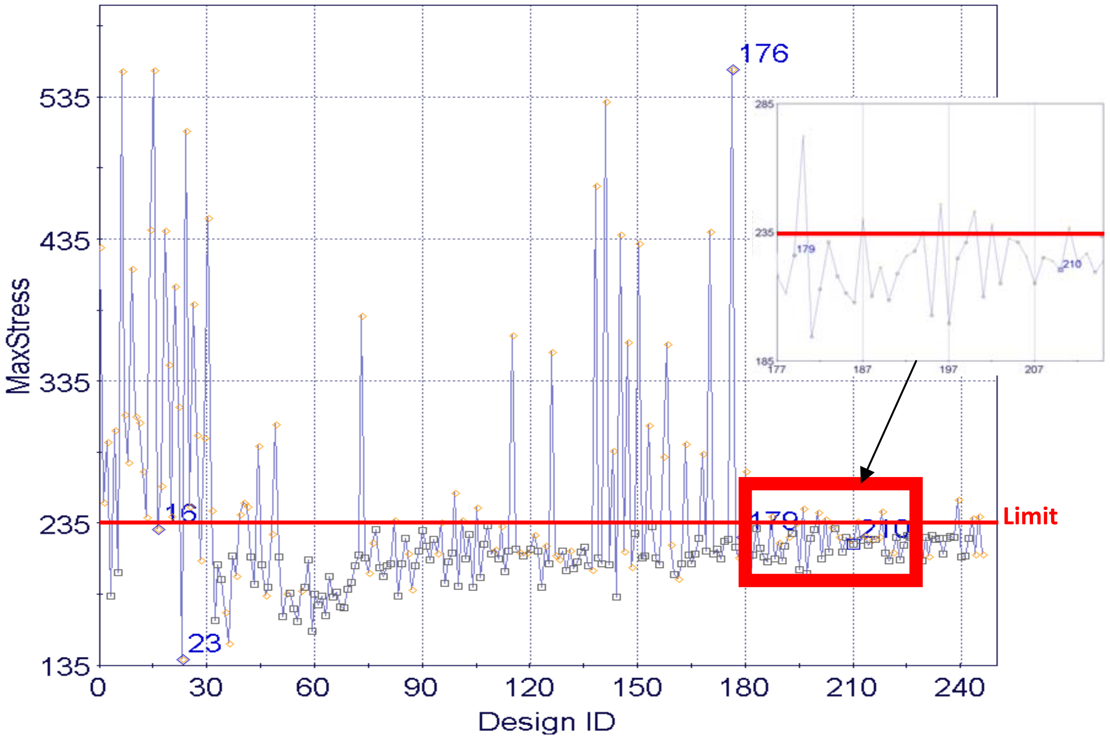

**Fig. 5: Maximum Stress Variation**

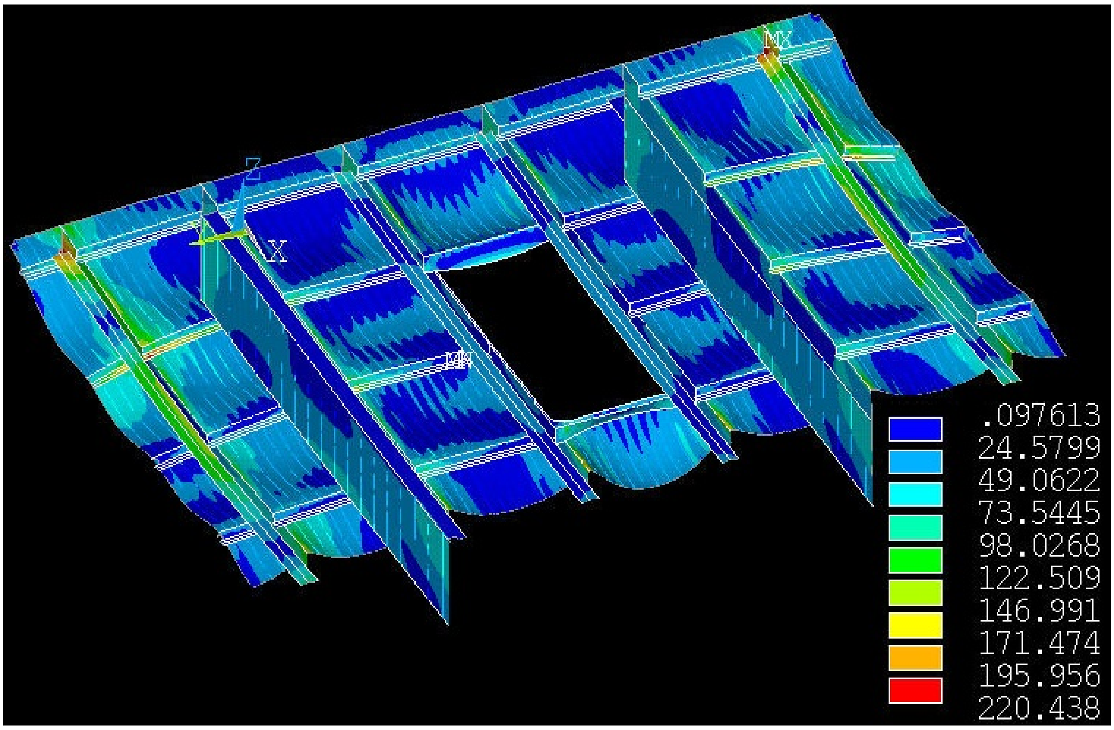

**Fig. 6: FE Results**

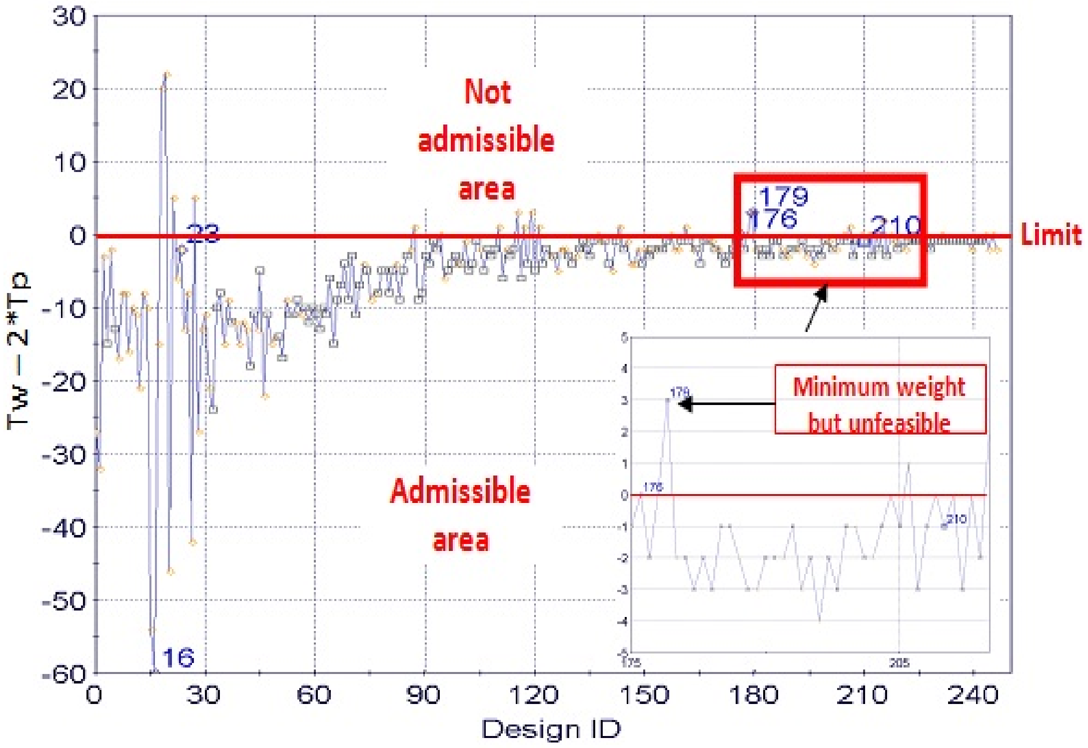

Fig. 7: Web Thickness of Stiffeners minus the double of the Plate Thickness

On the iteration 16, the weight is maximum. The plate thickness (39 mm), the number (14) and dimensions (hp430x20) of deck longitudinal stiffeners are maximum compared to the other iterations (Fig. 8 and 9).

The iterations 23 and 176 give the designs with the minimum and maximum level of stress.

In the Table 3, in addition to the initial design, we can see the values of the design variables on the iterations 16, 23, 176, 179 and the optimum 210.

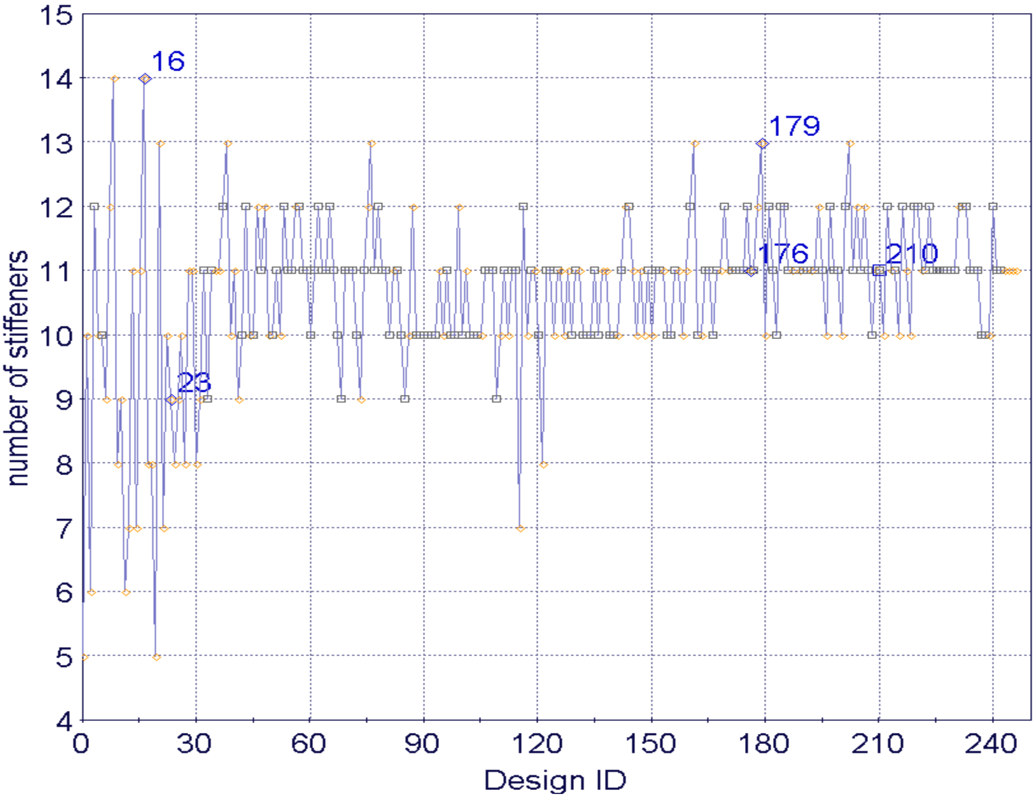

Fig. 8: Number of Stiffeners between Girders

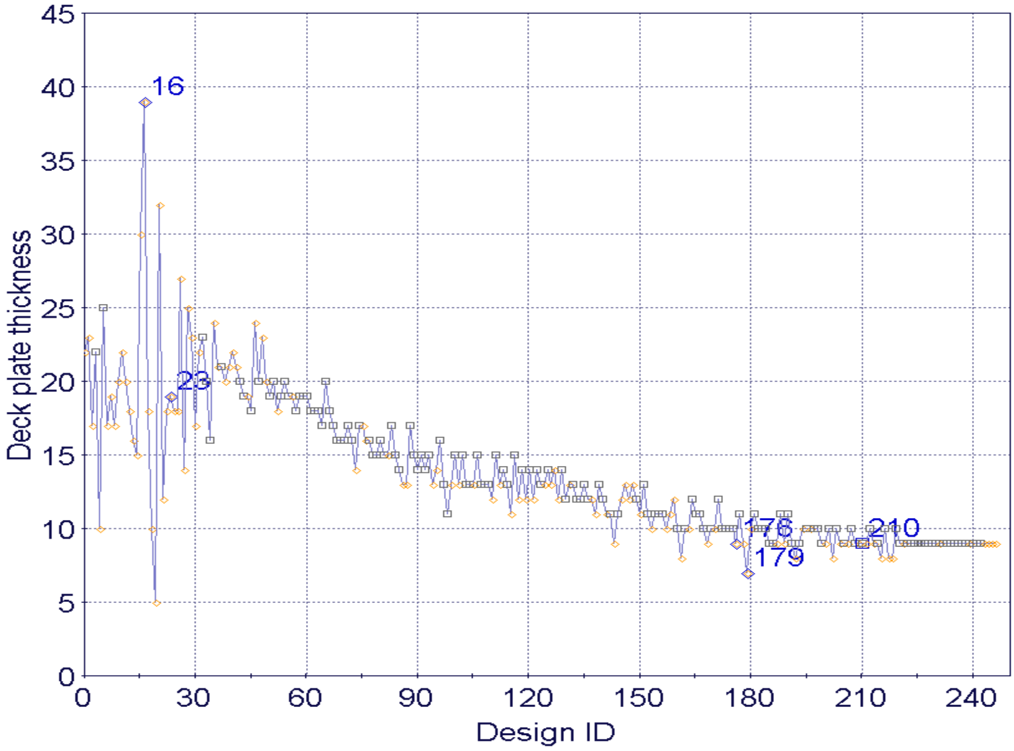

**Fig. 9: Deck Plate Thickness Variation**

**Table 3: Optimisation Results**

| Id | Initial Geometry | 16 | 23 | 176 | 179 | 210 |
|---|---|---|---|---|---|---|
| Deck stiffener web height | 180.0 | 430.0 | 320.0 | 80.0 | 80.0 | 80.0 |
| Deck stiffener web thickness | 11.5 | 20.0 | 13.0 | 7.0 | 6.0 | 6.0 |
| Deck plate thickness | 22.0 | 39.0 | 19.0 | 9.0 | 7.0 | 9.0 |
| Number of deck stiffeners between girders | 5.0 | 14.0 | 9.0 | 11.0 | 13.0 | 11.0 |
| Frame web height | 345.0 | 275.0 | 305.0 | 390.0 | 325.0 | 335.0 |
| Frame web thickness | 17.0 | 18.0 | 36.0 | 18.0 | 17.0 | 17.0 |
| Frame flange width | 375.0 | 165.0 | 275.0 | 225.0 | 210.0 | 225.0 |
| Frame flange thickness | 11.0 | 33.0 | 27.0 | 31.0 | 33.0 | 30.0 |
| Girder web height | 440.0 | 205.0 | 760.0 | 945.0 | 860.0 | 855.0 |
| Girder web thickness | 34.0 | 34.0 | 26.0 | 11.0 | 10.0 | 11.0 |
| Girder flange width | 255.0 | 125.0 | 445.0 | 495.0 | 500.0 | 480.0 |
| Girder flange thickness | 14.0 | 8.0 | 25.0 | 18.0 | 20.0 | 19.0 |
| long bulkhead stiffeners web height | 280.0 | 180.0 | 320.0 | 200.0 | 180.0 | 200.0 |
| long bulkhead stiffeners web thickness | 10.5 | 11.5 | 11.5 | 12.0 | 11.5 | 11.0 |
| long bulkhead plate thickness | 14.0 | 15.0 | 27.0 | 10.0 | 12.0 | 8.0 |
| Constraint : TW -2*TP = | -27.0 | -60.0 | -2.0 | 0.0 | 3.0 | -1.0 |
| Constraint : MaxStress | 430.1 | 231.4 | 140.0 | 555.2 | 226.2 | 220.4 |
| TotalWeight | 148808.3 | 359144.5 | 205599.6 | 88160.5 | 79589.2 | 83661.9 |

Fig. 10 is the obtained results based on the optimisation workflow using FORAN, ANSYS Workbench and modeFRONTIER as CAD software, FEA tool and optimiser respectively.

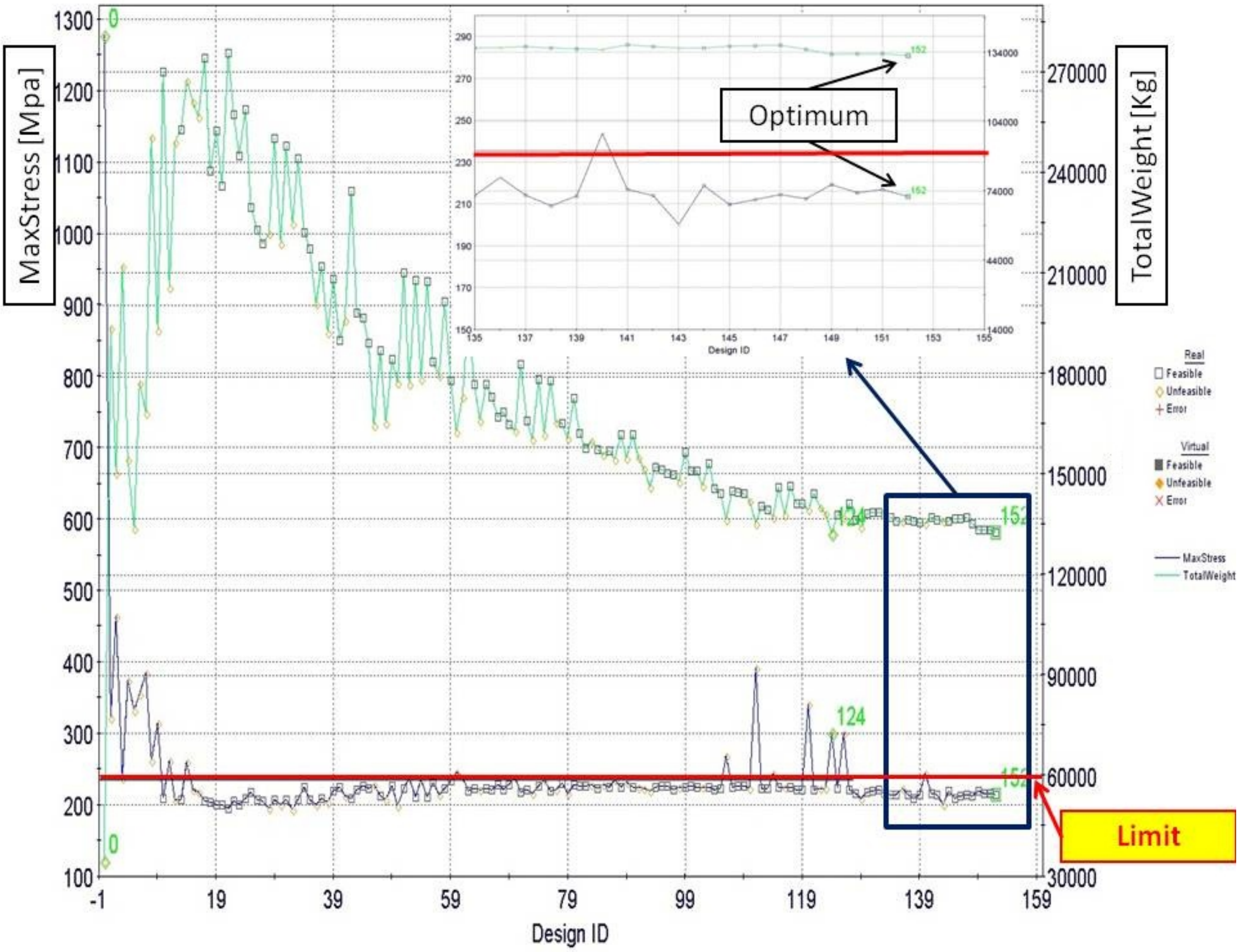

Fig. 10: Variations of Total Weight and Maximum Stress

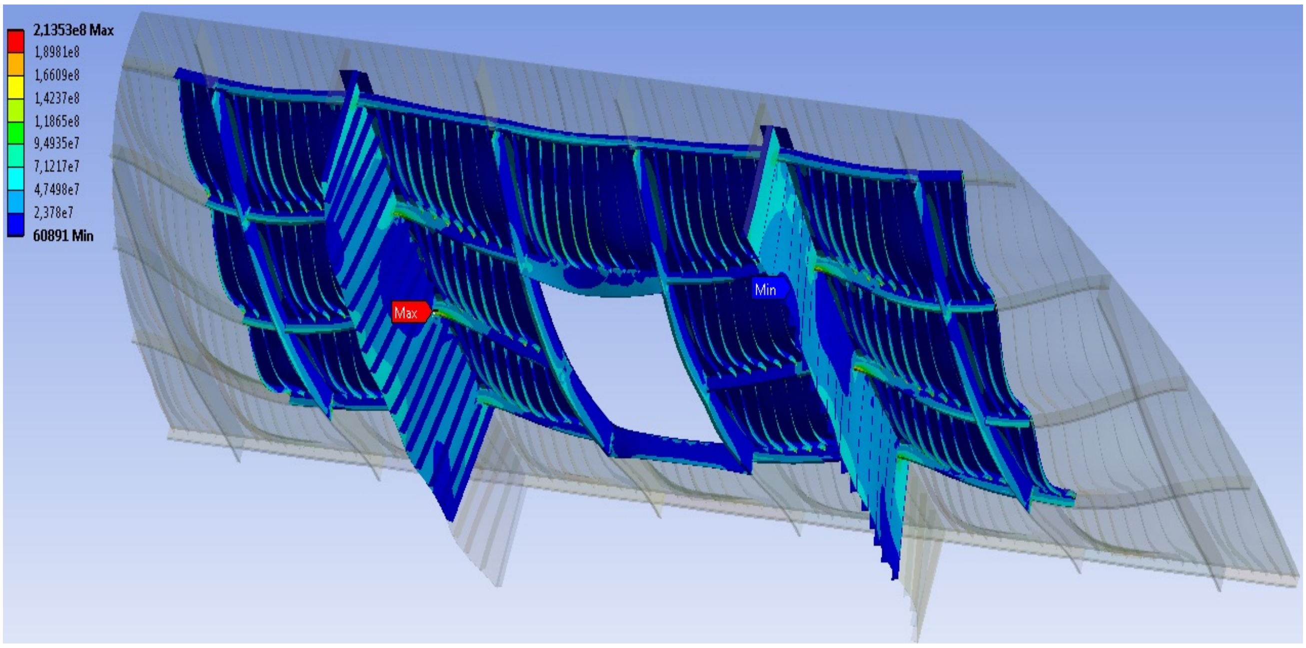

Fig. 11: FE Results

We can see that the optimum is reached after 151iterations. In other words, the optimum solution is achieved at the iteration 152 on which the minimum value of the total weight of the structure is 132477 Kg, and the maximum value of the Von Mises stress is 213.5 MPa. Fig. 11 shows the obtained FE result for the optimum solution.

## Conclusions

On the present work, the challenge was to develop an innovative structural optimization workflow. So, from a 3D CAD model, FEM model can be created automatically and the FEM results can be used by an optimization algorithm to evaluate an optimum solution.

Lots of efforts were done to perform a correct connection between the different modules included on the developed optimization workflow. The case study presented is a simple one. The goal is to test the optimization workflow.

A remaining work is to improve the optimization process by adding more structural constraints (fatigue, buckling, vibration...) and considering other or additional objective functions (minimum cost, maximum inertia, …) to get a real feasible optimum solution.

## References

Bohm M. (2010), “Interconnection of Rules based CAD Idealization for Ship Structures to ANSYS”, *ANSYS Conference & 28. CADFEM Users´ Meeting*, Aachen.

Doig R., Bohm M., Stammer J., Hernandez P., Griesch S., Kohn D. and Nilsson, P-O (2009), “Reducing Time and Effort for Structural Design and Assessment”, *Proceedings of the International Conference on Computer Applications in Shipbuilding*, p.39-42, Shanghai, China.

Doig R., Bohm M. (2010), “Simulation-Based Structural Design of Ships”, *Proceedings of 11th International Symposium on Practical Design of Ships and other Floating Structures*, Rio de Janeiro, Brasil.

Murty, Katta G. (1983), “Linear Programming”. *New York: John Wiley & Sons, Inc.*.