International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 9, Issue 6 (December 2013), PP. 76-90

Towards a Ship Structural Optimisation Methodology at Early Design Stage

Abbas Bayatfar[1], Amirouche Amrane[2], Philippe Rigo[3]
[1]ANAST, University of Liege, Chemin des Chevreuils 1 (B52/3), 4000 Liège, Belgium
[2]ANAST, University of Liege, Chemin des Chevreuils 1 (B52/3), 4000 Liège, Belgium
[3]ANAST, University of Liege, Chemin des Chevreuils 1 (B52/3), 4000 Liège, Belgium

Abstract:- Ship structural optimisation with mathematical algorithms can be very helpful to find the best solution (minimum weight, minimum cost, maximum inertia, etc). Typically, finite element analysis (FEA) tools are used in ship structural assessment. But, to build FEM model from CAD one is not easy and needs a big amount of manual work. This paper presents an innovative optimisation workflow by which the following steps are automatically carried out, without any manual intervention. First, from 3D CAD model, the idealised CAD model is created by idealisation module taking into account FEM needs. Then, the idealised CAD model is transferred to a FEM tool. After that, the FEM model is meshed, loaded and solved. The obtained results (i.e. stress and weight) are transferred to optimiser tool. The optimiser evaluates the values of the objective function and the constraints previously defined and modifies the design variables (i.e. plate thickness and stiffener scantling) to create a new structural model, going to the next iteration of the loop. This process continues until the optimal solution is reached.

Keywords:- Ship structure; Optimisation methodology; FEM; CAD; BESST

I. INTRODUCTION

In shipbuilding industry, structural optimisation using mathematical algorithms is not yet largely implemented at the early design stage in an automatic process. This is while, ship structure optimisation with mathematical algorithms can be very helpful to find the best solution (minimum weight, minimum cost, etc). Typically, finite element analysis (FEA) tools are used in ship structural assessment. But, to build FEM model from CAD one is not easy. It needs a great amount of manual work (e.g. cleaning and simplifying the CAD geometry, defining missing data, etc) which may takes several weeks depends on the complexity of the model. Thus, to automatically perform ship structure optimisation, the idealised CAD model must be ready to use for FEM pre-processor. Also, a link must be created between the “CAD model” and the “FEM model” within the optimisation environment.

Taking look at literature, it can be found some contributions given to the research area mentioned above. For example, Birk [1] reported on the continuous development of an automated optimization procedure for the design of offshore structure hulls. Current results of the development of an efficient CAD-FEM interface for ship structures were presented by Doig et al. [2]. With the interface the direct extraction of FEM-friendly geometry is ensured, allowing drastically savings of assessment effort. Bohm et al. [3] described an interface of the ship construction CAD program AVEVA Marine and ANSYS. It idealises ship model data according to approval rules into an ANSYS geometry model. The study on how it is possible to use a 3D CAD tool at early design stages, to improve the overall design process, was presented by Alonso et al. [4]. It provides FORAN, a shipbuilding CAD/CAM system, with the necessary capabilities to ensure its efficient use at early design stages.
Following the above noted, the current study was undertaken to develop an innovative workflow towards ship structure optimisation loop at early design stage. The work was performed in the framework of the research activity carried out by the European Project BESST "Breakthrough in European Ship and Shipbuilding Technologies". The main focus of this paper is concerned with the development of an optimisation workflow supported by CAD/FEM integration, showing that works automatically without any manual intervention. There are two workflows provided in both which modeFRONTIER 4.4.2 is used as optimiser tool. In the first optimisation loop, AVEVA Marine 12.0.SP6.39 (as CAD software) is integrated with ANSYS Classic 14.0 as FEM software. And the second loop in which FORAN V70R1.0 and ANSYS Workbench 14.0 are used as CAD software and FEM software respectively.

In this regard, a typical deck structure (as an initial case study) was taken into consideration to evaluate the iterative process in both workflows. As it’s schematically shown in Fig. 1, the 3D CAD model is first transferred from the CAD software to the idealisation module. Then, the idealisation module generates a simplified geometry which belongs to the FEM needs. After that, the idealised CAD model is transferred to the FEM software to create meshed and loaded structural model. Finally, the FE analysis is done and the obtained results for the objective function and the constraints previously defined are transferred to the optimiser tool to be evaluated, in order to modify the design variables (plate thickness, stiffener dimensions, stiffener spacing, etc) and to create a new structural model. The optimisation iteration process will be continued until the convergence is attained.

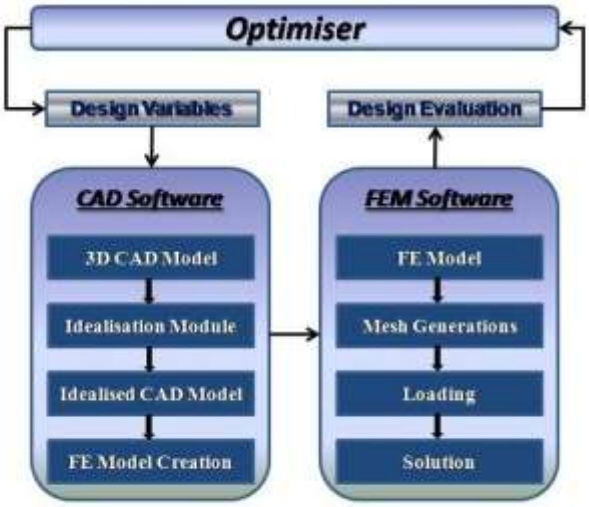
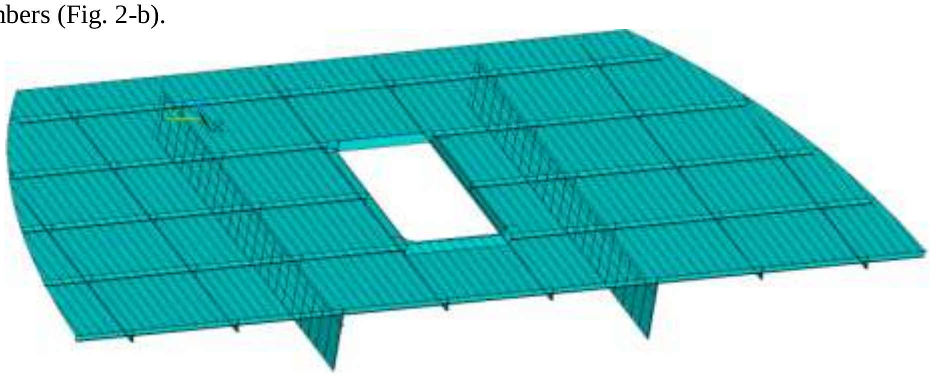


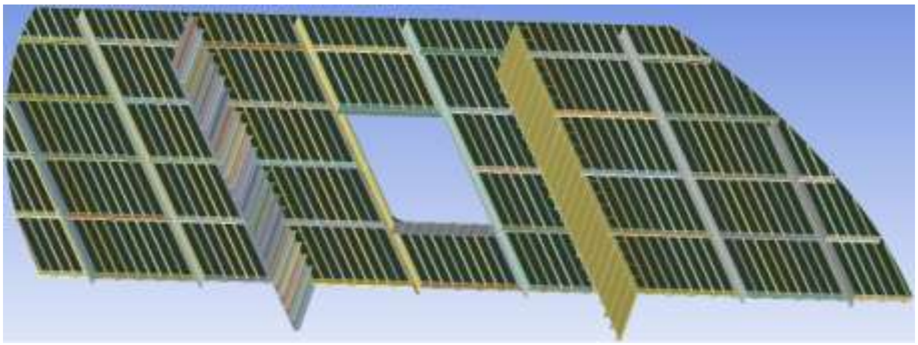
Fig. 1. Schematic of optimisation workflow

II. MODEL FOR ANALYSIS

The deck structure model was quite similarly created by CAD AVEVA Marine software [5] and CAD FORAN software [6]. The structure is constituted by deck plate, longitudinal girders, transversal frames, longitudinal stiffeners placed between girders, and two longitudinal walls along with its stiffeners. In AVEVA Marine model, the longitudinal stiffeners placed between girders and the stiffeners placed on two longitudinal walls were taken into consideration as beam members (Fig. 2-a) while those in FORAN model were considered as plate members (Fig. 2-b).



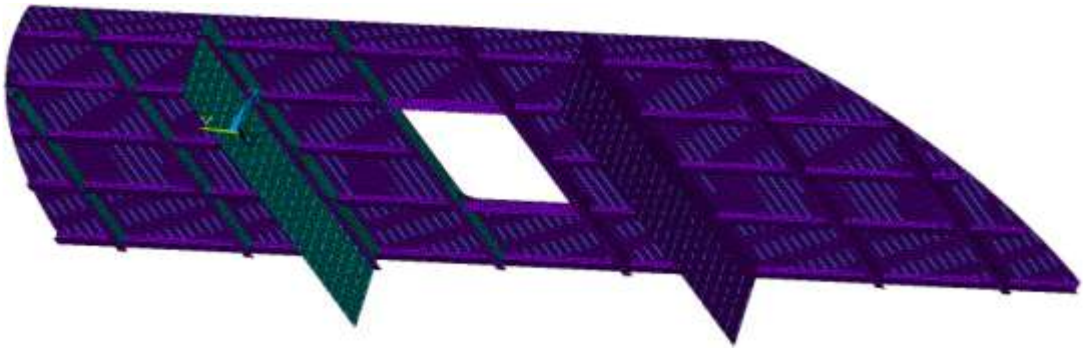
a) AVEVA Marine case study



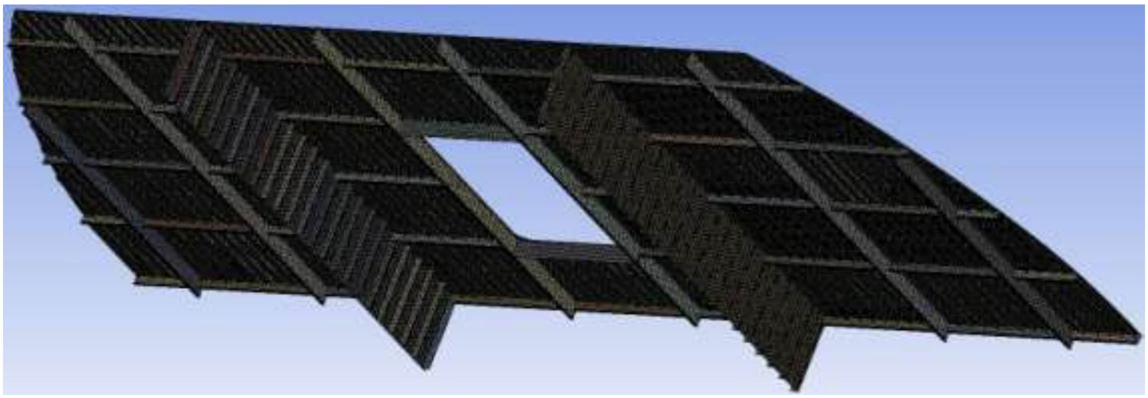
b) FORAN case study

Fig. 2. Deck structure model

Among the elements inside the library of ANSYS [7], SHELL63 and beam44 were selected in order to respectively discretise the plate and beam members (Fig. 3).



a) AVEVA Marine case study



b) FORAN case study

Fig. 3. Typical mesh generations

In AVEVA Marine model, the displacements in x-, y- and z- directions were suppressed at fore and aft sides, while all boundaries in FORAN model were restrained from displacements in x-, y- and z- directions. The FE analyses, in this study, were made based on a lateral pressure that acts on the deck plate (with plate side, not stiffener side), with the value of 0.02 MPa. In order to analyse the structural cases study in the optimisation loops, the maximum Von Mises stress value was taken into account from the inner part of the models (see Tables 6 and 8).

In the following, a summary of materials used in AVEVA Marine and FORAN cases study are given in Table 1.

Table 1 Summary of material properties used in cases study

Case study	***Young's modulus*** (E)	***Poisson ratio*** (υ)	***Yield strength*** (σ_Y)
	MPa	-	MPa
AVEVA Marine	206000	0.3	235
FORAN	200000	0.3	250

According to the initial scantlings provided for AVEVA Marine case study (Table 2),

Table 2
Initial scantling for AVEVA Marine case study

Member	***Design variable***	***Value (mm)***
Deck	Plate thickness	14
	Long. stiffener profile	HP100x8
	Numbers of stiffeners (between girders)	9
Transversal frame	Web height	300
	Web thickness	5
	Flange breadth	100
	Flange thickness	10

Hatch frame	Web height	600
	Web thickness	5
	Flange breadth	100
	Flange thickness	10
Longitudinal girder	Web height	600
	Web thickness	5
	Flange breadth	100
	Flange thickness	10
Longitudinal wall	Plate thickness	10
	Stiffener profile	HP160x8

and for FORAN case study (Table 3), the total structural weights are respectively 80649.92 kg and 74904 Kg.

Table 3 Initial scantling for FORAN case study

Member	*Design variable*	*Value (mm)*
Deck	Plate thickness	14
	Stiffener (between girders)	114x8
Transversal frame	Web height	300
	Web thickness	5
	Flange breadth	200
	Flange thickness	10
Hatch frame	Web height	600
	Web thickness	5
Longitudinal girder	Web height	600
	Web thickness	5
	Flange breadth	100
	Flange thickness	10
Longitudinal wall	Plate thickness	5
	Stiffener (placed on walls)	180x10

III. OPTIMISATION WORFLOW DESCRIPTION

III.1 AVEVA Marine based Workflow

Figure 4 presents the integration development of the optimisation workflow using AVEVA Marine 12.0.SP6.39, ANSYS Classic 14.0 and modeFRONTIER 4.4.2 [8] as CAD software, FEM software and optimiser tool respectively. The design variables used in the optimisation loop along with their lower and upper bounds are given in Table 4.

Table 4 Design variables limits for AVEVA case study

Member	*Design variable*	*Min (mm)*	*Max (mm)*
Deck	Plate thickness	5	40
	Long. stiffener profile	HP80x6	HP430x20
	Numbers of stiffeners (between girders)	5	15
Transversal frame	Web height	200	1000
	Web thickness	5	40
	Flange breadth	50	500
	Flange thickness	5	40
Longitudinal girder	Web height	200	1000
	Web thickness	5	40
	Flange breadth	50	500
	Flange thickness	5	40
Longitudinal wall	Plate thickness	5	40
	Stiffener profile	HP80x6	HP430x20

Also, the geometrical constraints imposed can be seen in Fig. 4 (see ellipse outline). Among which can be mentioned the following [9]:

- Web thickness of stiffeners to be less than the double of the deck plate thickness
- The deck plate thickness to be less than the double of web thickness of stiffeners
- Web height of frames to be greater than the web height of stiffeners

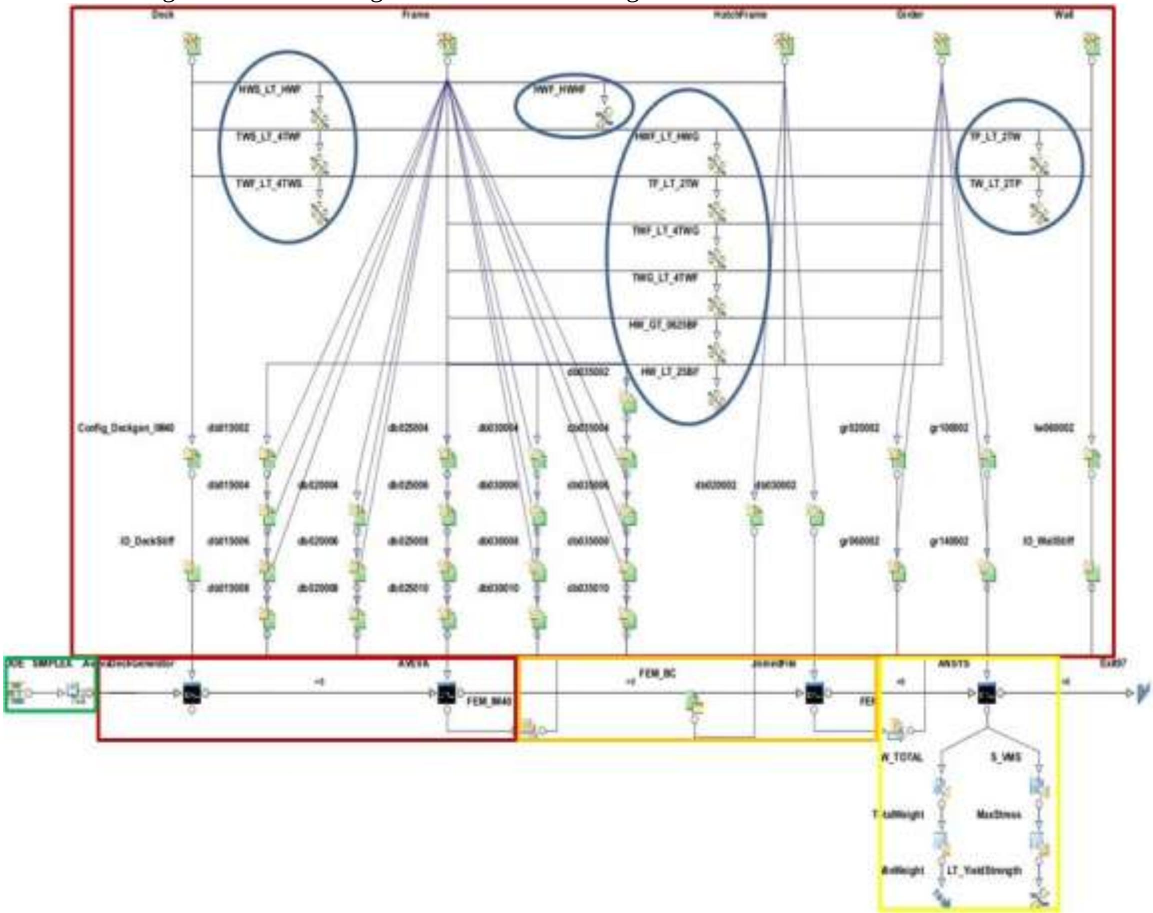


Fig. 4. AVEVA Marine based optimisation workflow

As it's shown above in red outline, AVEVA Marine is first lunched to create FEM model and to export it to ANSYS Classic input file (APDL file). Then, the automatic loading tool shown in orange outline combines the provided APDL file with the file included mesh generation, boundary and loading conditions, in order to be read by ANSYS Classic. After that, the FE analysis is done and the required results are provided in the result extraction module shown in yellow outline. In this module, the weight of the structure was defined as objective function to be minimised. And, as a structural constraint, maximum Von Mises stress was imposed to be less than the yield strength of the material. Finally, the obtained results for the objective function and the constraints previously defined are transferred to the optimiser tool (shown in green outline) to be evaluated, in order to modify the design variables (plate thickness, stiffener dimensions, stiffener spacing, etc) and to create a new structural model.

In this regard, from the library of algorithms included in modeFRONTIER 4.2.2, the design of experiments was taken as a constraint satisfaction problem (CSP) to find an assignment to each variable so that all geometrical constraints are satisfied. Also, SIMPLEX algorithm (used in mono-objective optimisation) was chosen to determine which designs need to be evaluated.

III.2 FORAN based Workflow

Figure 5 presents the integration development of the workflow using FORAN V70R1.0, ANSYS Workbench 14.0 and modeFRONTIER 4.4.2 as CAD software, FEM software and optimiser tool respectively. Here should be noted that the workflow provided in Fig. 5 is not a realistic optimisation, but it's more like a dimensioning task. This is because the design variables used in this loop could just be taken into consideration as below.

- Deck plate thickness
- Web thickness for stiffeners

- Web thickness and flange thickness for longitudinal girders
- Web thickness and flange thickness for transversal frames
- Wall plate thickness for longitudinal walls

The lower and upper bounds of the above-mentioned design variables were set between 5 (mm) and 40 (mm). From the Fig. 5, by ellipse outline, the geometrical constraints imposed can be seen, among which the following can be mentioned [9]:

- Web thickness of stiffeners to be less than the double of the deck plate thickness
- The deck plate thickness to be less than the double of web thickness of stiffeners

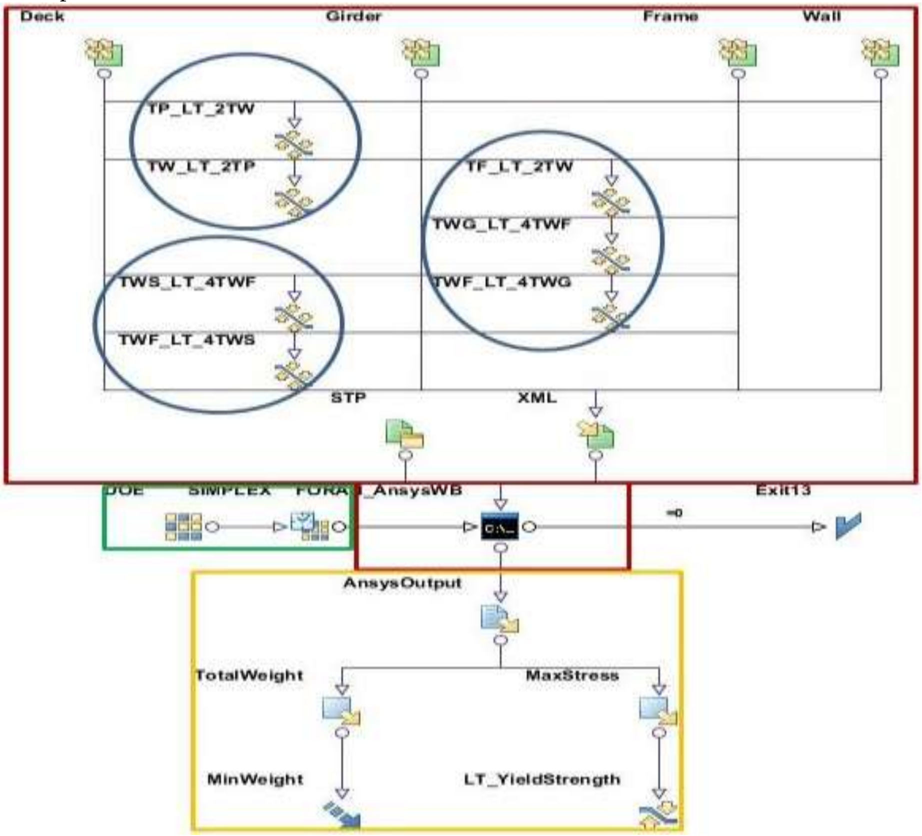


Fig. 5. FORAN based workflow

In the workflow shown above, in the red outline, the FORAN script tool reads both geometry file (STP file) and attribute file (XML file) provided by FORAN in order to create ANSYS Workbench model (WBPJ file). Then, in ANSYS Workbench environment, the required mesh, boundary and loading conditions are automatically applied. After that, the FE analysis is done and the required results are provided in the result extraction module shown in orange outline.

In this module, similar to AVEVA based optimisation workflow, the weight of the structure was defined as objective function to be minimised. And as a structural constraint, maximum Von Mises stress was imposed to be less than the yield strength of the material. Finally, the obtained results for the objective function and the constraints previously defined are transferred to the optimiser tool (shown in green outline) to be evaluated, in order to modify the design variables (i.e. thickness for the stiffeners, girders, frames and longitudinal walls) and to create a new structural model.

In this regard, from the library of algorithms included in modeFRONTIER 4.2.2, the optimization algorithm chosen was SIMPLEX which is used in mono-objective optimisation.

IV. RESULTS AND DISCUSSIONS

AVEVA Marine based optimisation workflow and FORAN based workflow were successfully validated and the obtained results are presented in this section. The communication between all integrated software and tools are fully in an automatic process, without any manual intervention on graphical user interface.

IV.1 AVEVA Marine Case Study

The convergence of the solution is obtained after 246 iterations. The total calculation time for one run, using the machine with Intel® Core ™ i7 CPU 860 @2.80 GHz and RAM 12.0 Go., is about one minute (the total run takes about 4 hours).

Figure 6 shows the convergence histories of the objective function (i.e. the total weight of the structure) and the structural constraint (i.e. the maximum Von Mises stress) by a multi-history chart. The optimum is reached after 209 iterations.

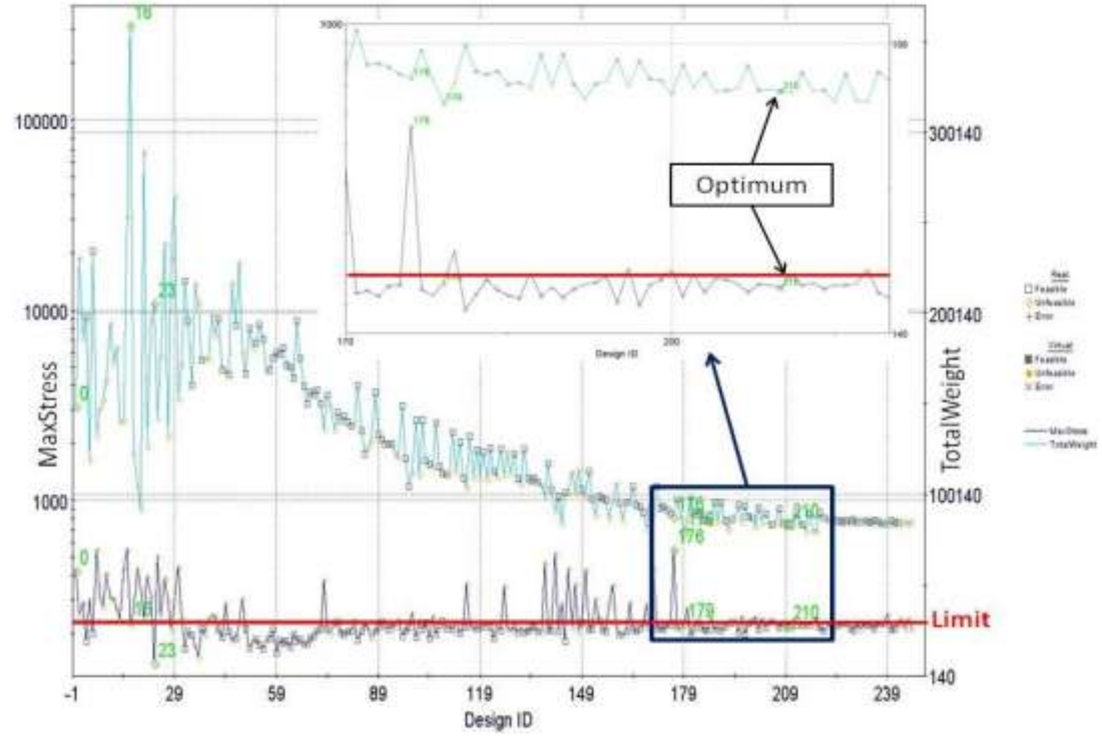
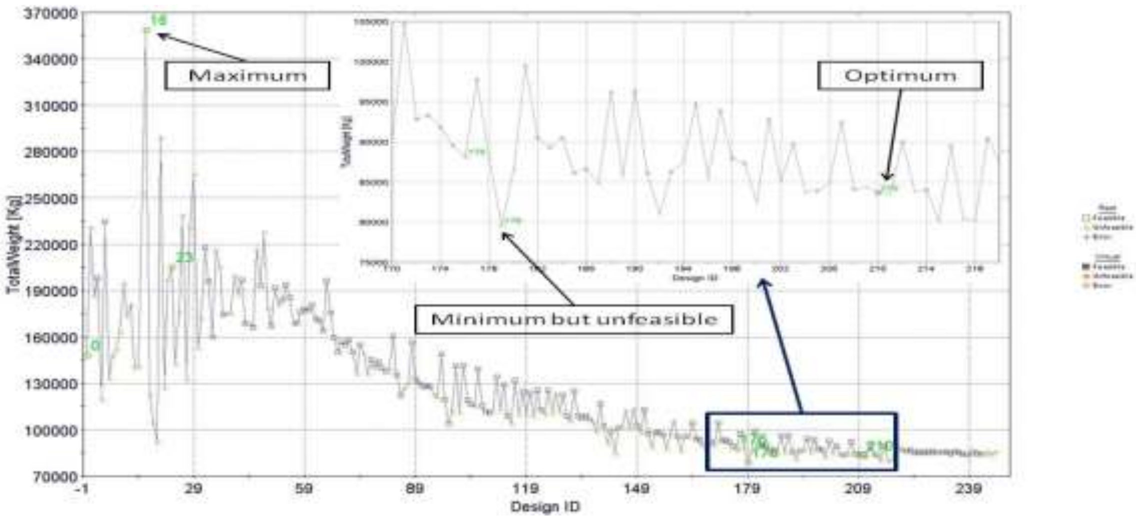
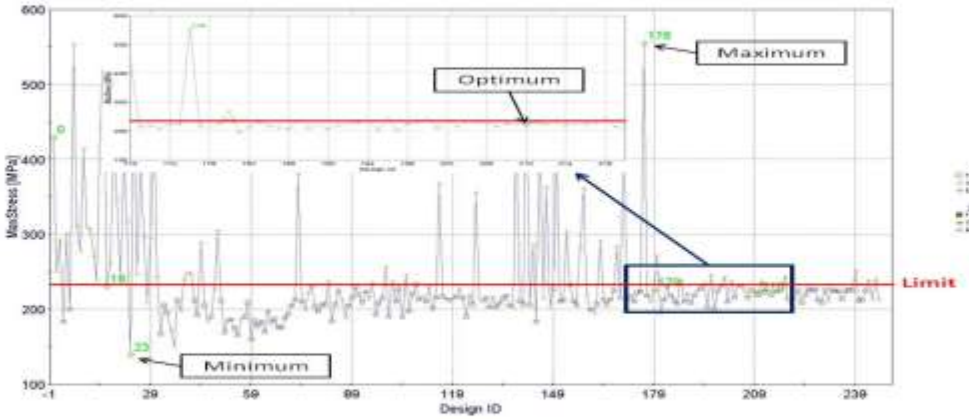


Fig. 6. Convergence histories of the objective function and the maximum Von Mises stress for AVEVA Marine case study

In other words, the optimum solution is achieved at the iteration 210 on which the total weight of the structure is 83661.9 Kg, and the maximum value of the Von Mises stress is 220.4 MPa. The total weight of the structure and the maximum value of the Von Mises stress respectively decrease up to 44% and 49%, compared with the original configuration. This can be seen in Fig. 7, and more clearly in Table 5 by which the optimisation results are given in detail for some iterations, i.e. 0, 16, 23, 176, 179 and 210.



a) For the objective function (i.e. the total weight of the structure)



b) For the structural constraint (i.e. the maximum value of the Von Mises stress)

Fig. 7. Convergence history for AVEVA Marine case study

Figure 7(a) reports the history plot of the total weight of the structure. As it can be seen, at the iteration 179, the total weight of the structure is 79589.2 Kg which is lower than the optimum solution (83661.9 Kg), and the maximum value of the Von Mises stress is 226.2 which is less than the limit shown in Fig. 7(b). However, this solution is unfeasible due to one geometrical constraint which is not respected. Figure 8 plots the history of this geometrical constraint (web thickness of frames to be less than the double of the deck plate thickness).

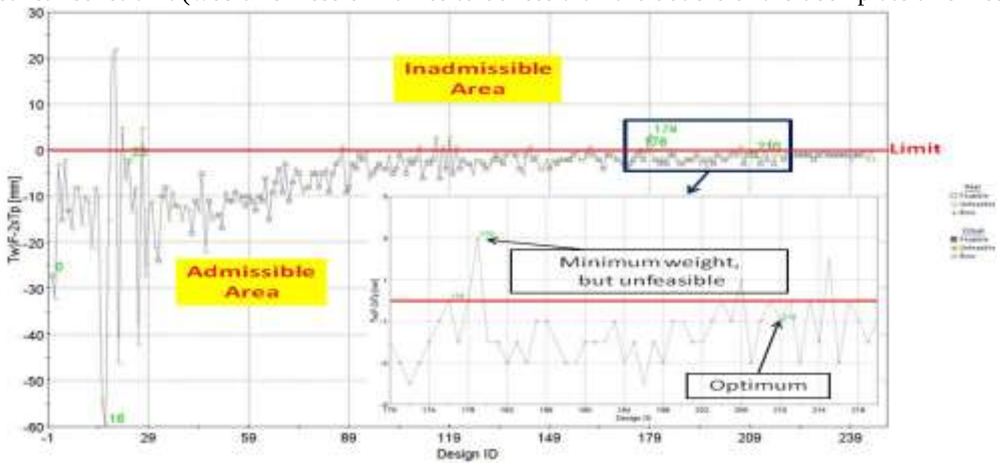


Fig. 8. Convergence history of the constraint 'web thickness of frames minus the double of the deck plate thickness'

In the following, Figs. 9, 10 and 11, respectively show the history plots of deck plate thickness (as design variable), number of stiffeners (as design variable) and one geometrical constraint (web height of frames to be less than the web height of girders).

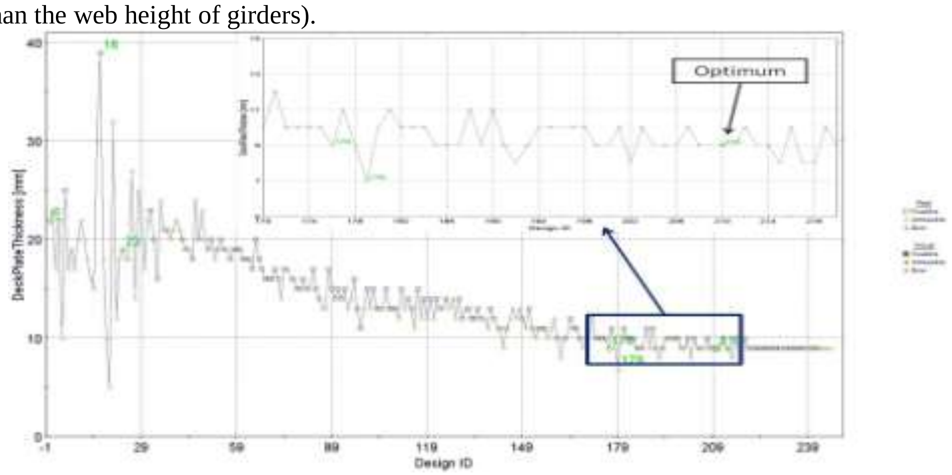


Fig. 9. Convergence history of the deck plate thickness for AVEVA Marine case study

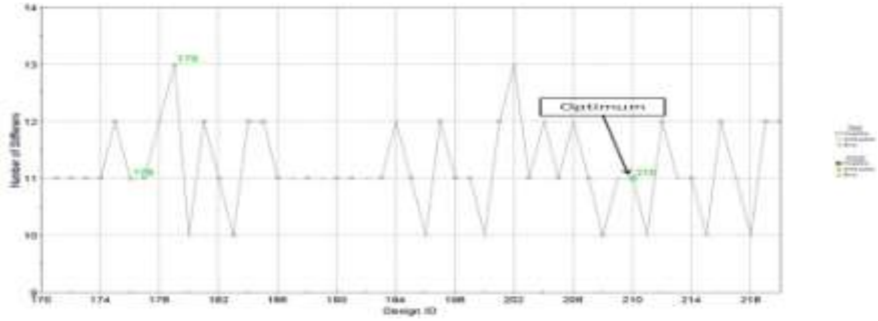


Fig. 10. Convergence history of the number of stiffeners placed between girders

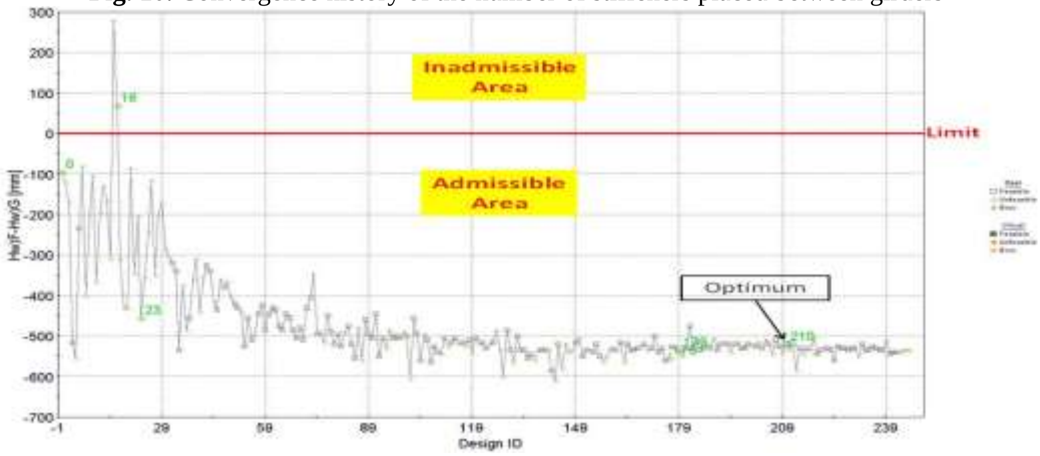


Fig. 11. Convergence history of the constraint 'web height of frames to be less than the web height of girders'

Also, in order to have a comparison, Table 5 gives some more details corresponds to the original configuration and the iterations below. The unit used for dimension, weight and stress are respectively mm, Kg and MPa.

- 16 (at which the total weight of the structure is in the highest level)
- 23 (at which the maximum value of the Von Mises stress is in the lowest level)
- 176 (at which the maximum value of the Von Mises stress is in the highest level)
- 179 (at which one geometrical constraint is not respected, although the total weight of the structure is lower than the optimum solution and the maximum value of the Von Mises stress is less than the limit)
- 210 (at which the optimum solution is reached)

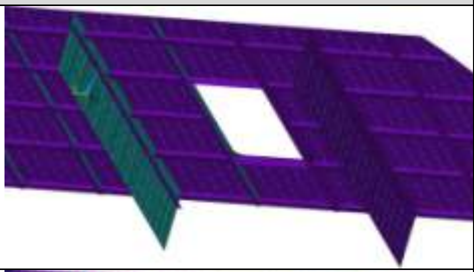
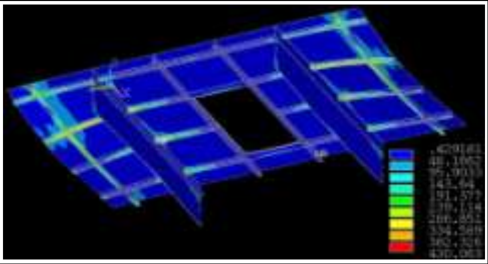
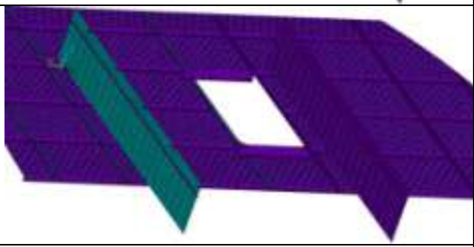
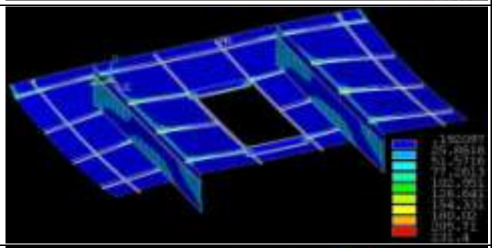
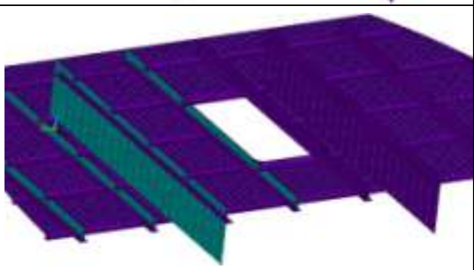
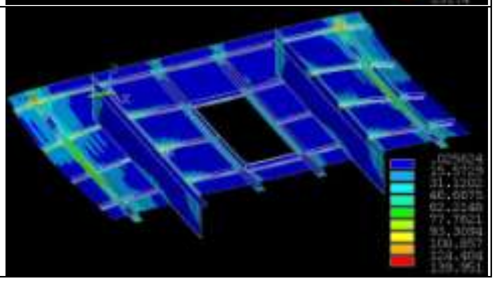
Table 5 Optimisation results in detail for AVEVA Marine case study

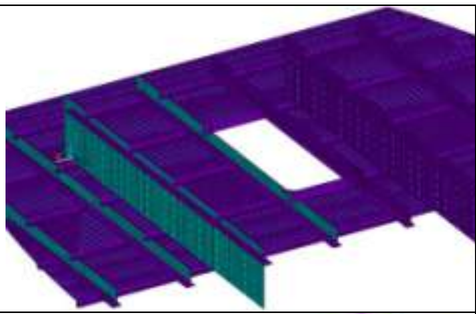
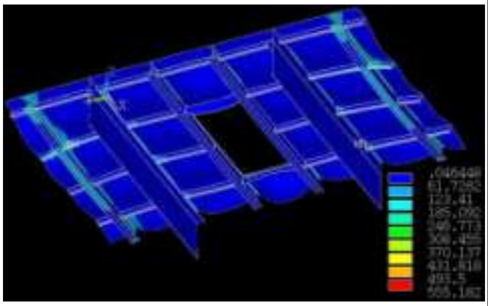
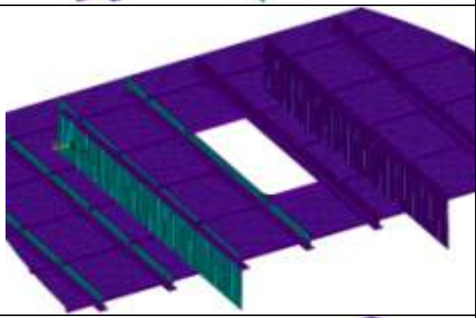
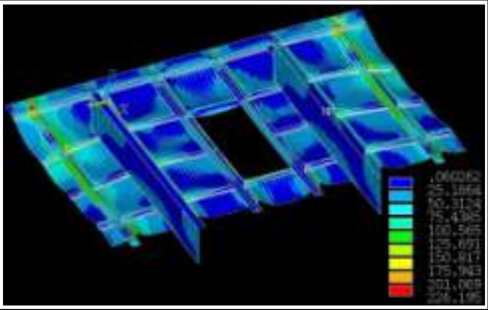
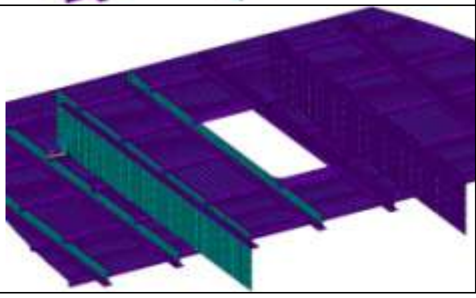
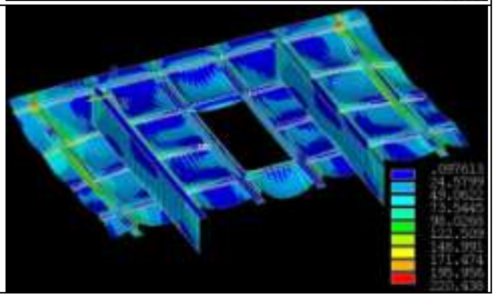
Id		***Original configuration***	***16***	***23***	***176***	***179***	***210***
Deck	Plate thickness	22	39	19	9	7	9
	Long. stiffener profile	HP80x11.5	HP430x20	HP320x13	HP80x7	HP80x6	HP80x6
	Numbers of stiffeners (between girders)	5	14	9	11	13	11
Transversal frame	Web height	345	275	305	390	325	335
	Web thickness	17	18	36	18	17	17
	Flange breadth	375	165	275	225	210	225
	Flange	11	33	27	31	33	30

	thickness						
Longitudinal girder	Web height	440	205	760	945	860	855
	Web thickness	34	34	26	11	10	11
	Flange breadth	255	125	445	495	500	480
	Flange thickness	14	8	25	18	20	19
Longitudinal wall	Plate thickness	14	15	27	10	12	8
	Stiffener profile	HP280x10.5	HP180x11.5	HP320x11.5	HP200x12	HP180x11.5	HP200x11
Geometrical constraint: TW)F-2xTp		-27	-60	-2	0	3	-1
Structural constraint: MaxStress		430.1	231.4	140	555.2	226.2	220.4
TotalWeight		148808.3	359144.5	205599.6	88160.5	79589.2	83661.9

The structural models correspond to the above-mentioned iterations along with its FE results are given in Table 6 (the unit taken is MPa).

Table 6
Structural models along with its FE results for some iterations for AVEVA Marine case study

Iteration	***Structural model***	***FE results***
0		
16		
23		

176		
179		
210 (Optimum)		

IV.2 FORAN Case Study

The convergence of the solution is obtained after 152 iterations. The total calculation time for one run, using the machine with Intel® Core ™ i7 CPU 860 @2.80 GHz and RAM 12.0 Go, is about 9 minutes (the total run takes about 21 hours).

Figure 12 shows the convergence histories of the objective function (i.e. the total weight of the structure) and the structural constraint (i.e. the maximum Von Mises stress) by a multi-history chart. The optimum is reached after 151iterations.

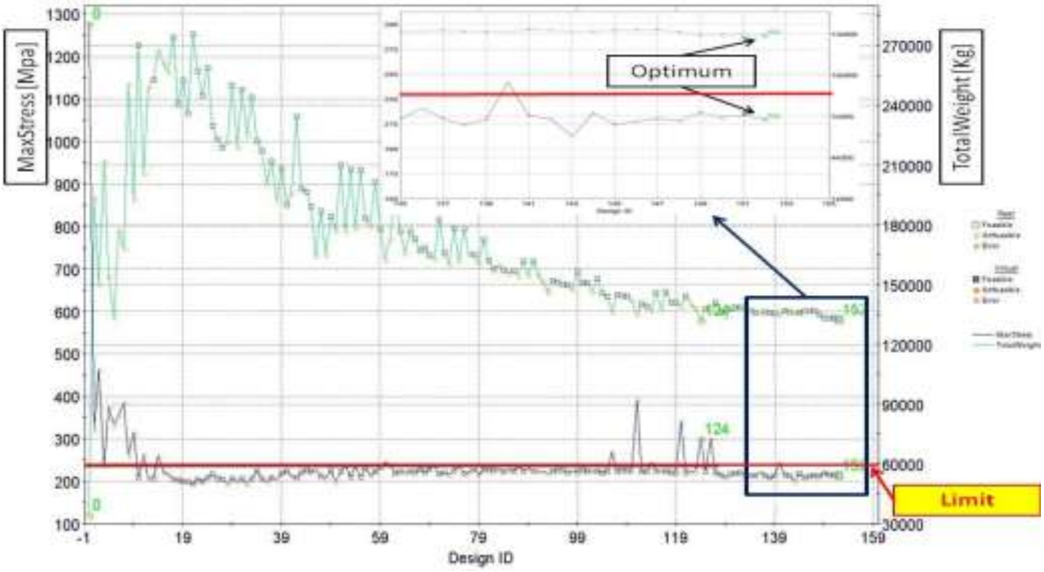
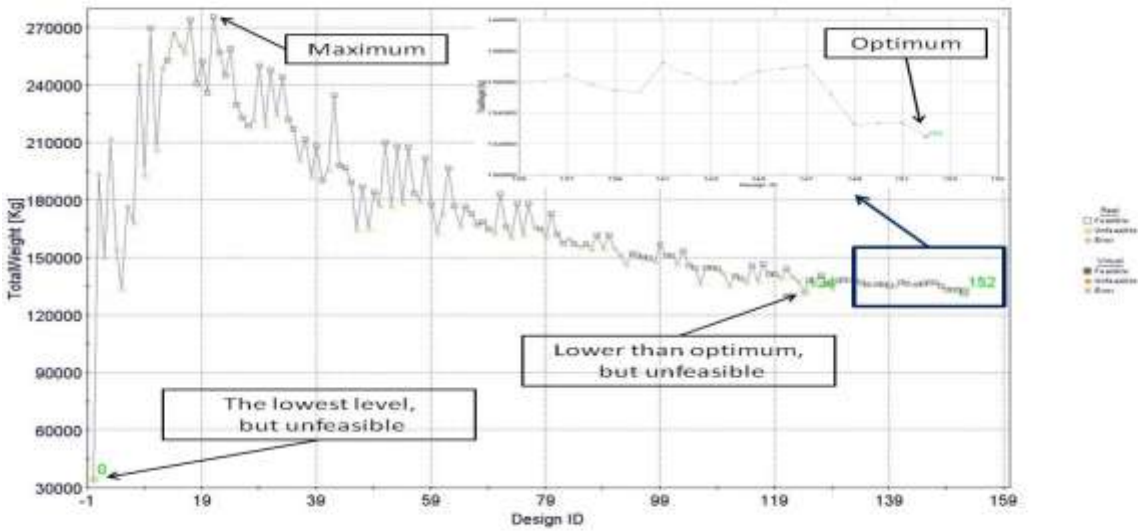


Fig. 12. Convergence histories of the objective function and the maximum Von Mises stress for FORAN case study

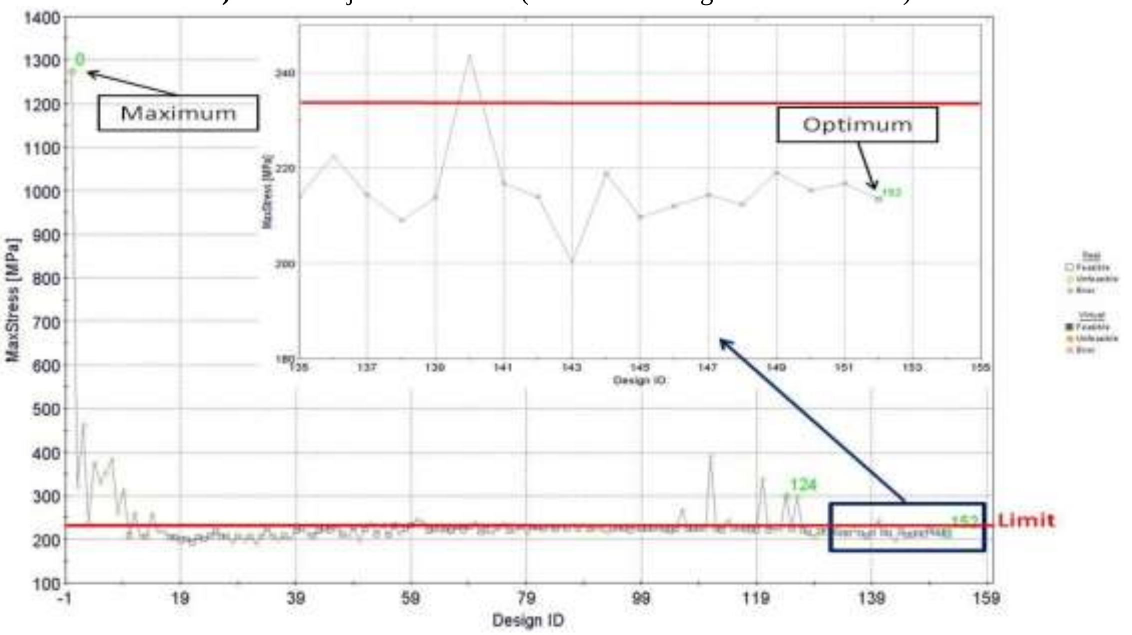
In other words, the optimum solution is achieved at the iteration 152 on which the total weight of the structure is 132477 Kg, and the maximum value of the Von Mises stress is 213.5 MPa. Compared with the original configuration, the total weight of the structure increases up to %74. This is while the maximum value of the Von Mises stress decreases up to 83% (from 1277.2 MPa to 213.5 MPa). This can be seen in Fig. 13, and more clearly in Table 7 by which the optimisation results are given in detail for some iterations, i.e. 0, 124 and 152.

Figure 13(a) reports the history plot of the total weight of the structure. As it can be seen, at the initial design (the iteration 0), the total weight of the structure is in the lowest level (34581.4 Kg). However, this solution is unfeasible due to the structural constraint which is not respected (at this iteration, the maximum value of the Von Mises stress is in the highest level, i.e. 1277.2 MPa). At the iteration 124, the total weight of the structure is 132280 Kg which is lower than the optimum solution (132477 Kg). However, this solution is unfeasible due to the structural constraint which is not respected (the maximum value of the Von Mises stress, at this iteration, is 299.7 MPa), and also due to the following (Figs. 14-16).

- The deck plate thickness exceeds the double of web thickness of stiffeners
- Web thickness of frames exceed four times of web thickness of stiffeners
- Web thickness of hatch frames exceed four times of web thickness of stiffeners



a) For the objective function (i.e. the total weight of the structure)



b) For the structural constraint (i.e. the maximum value of the Von Mises stress)

Fig. 13. Convergence history for FORAN case study

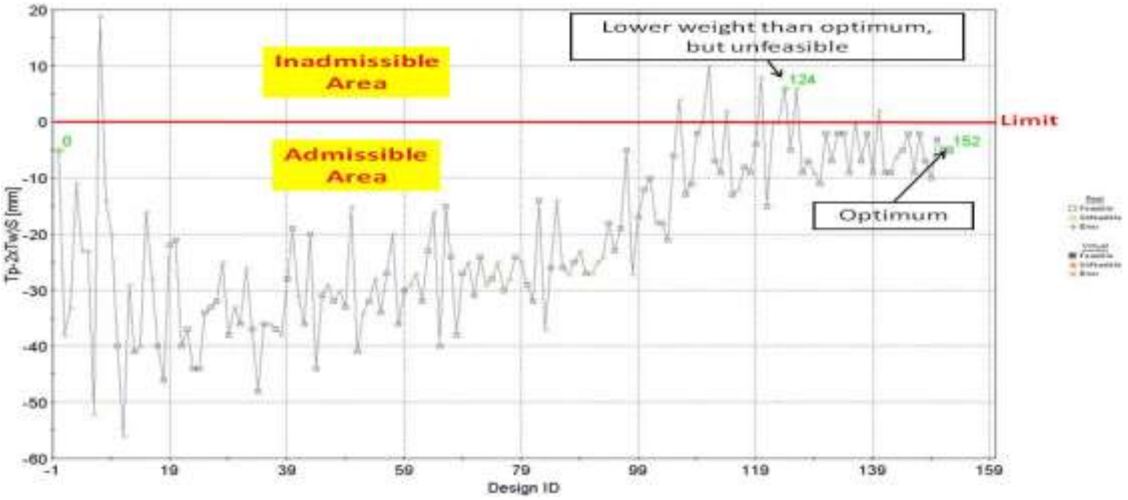


Fig. 14. Convergence history of the constraint ‘the deck plate thickness minus the double of the web thickness of stiffeners’

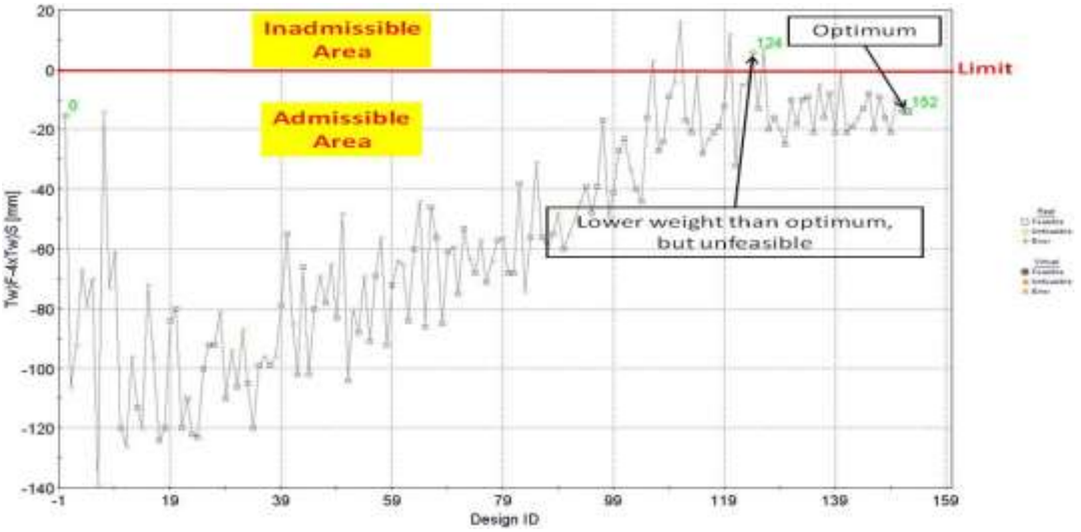


Fig. 15. Convergence history of the constraint ‘web thickness of frames minus four times of web thickness of stiffeners’

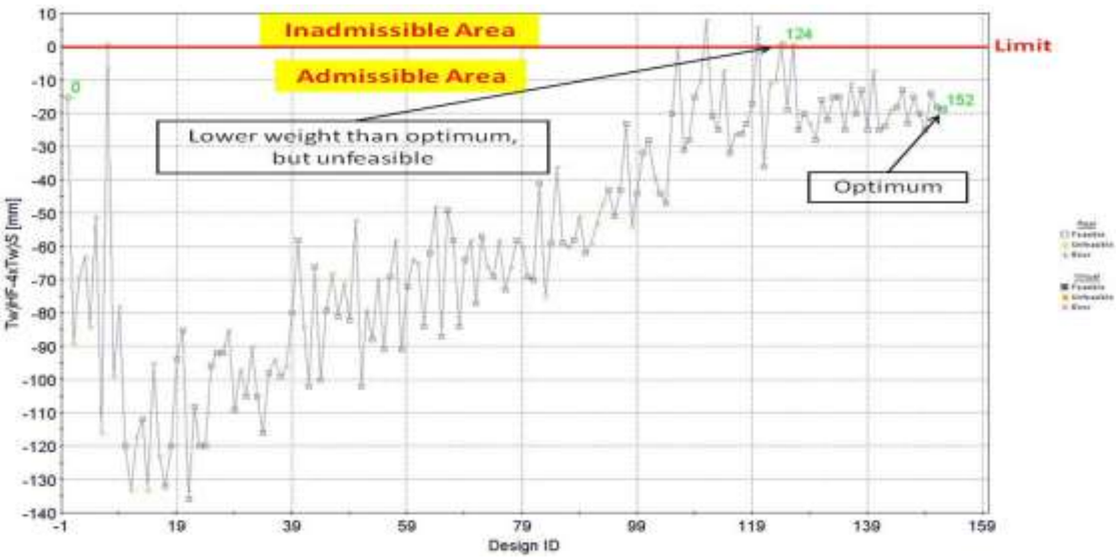


Fig. 16. Convergence history of the constraint ‘web thickness of hatch frames minus four times of web thickness of stiffeners’

In the following, Fig. 17 shows the history plot of deck plate thickness (as design variable).

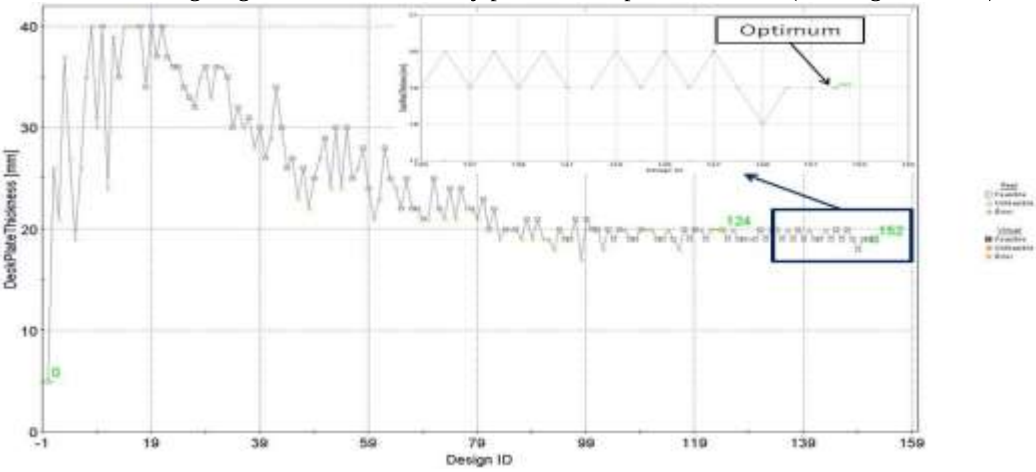


Fig. 17. Convergence history of the deck plate thickness for FORAN case study

Also, in order to have a comparison, Table 7 gives some more details corresponds to the initial design and the iterations below. The unit used for dimension, weight and stress are respectively mm, Kg and MPa.

- 124 (at which the total weight of the structure is lower than the optimum solution (132477 Kg). However, this solution is unfeasible due to some structural and geometrical constraints which are not respected)
- 152 (at which the optimum solution is reached)

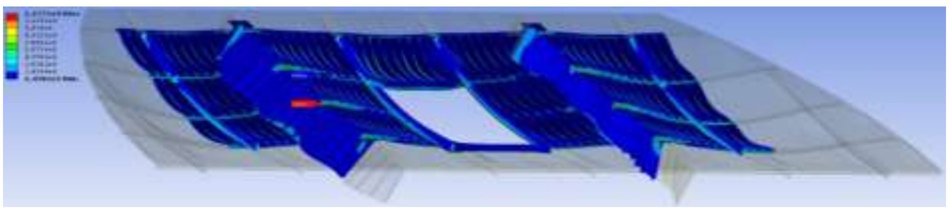
Table 7

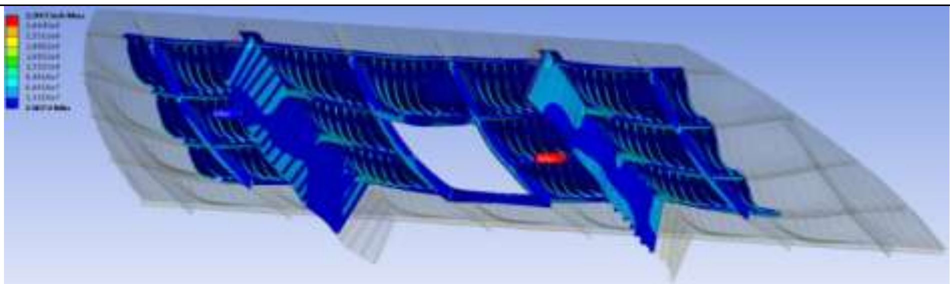
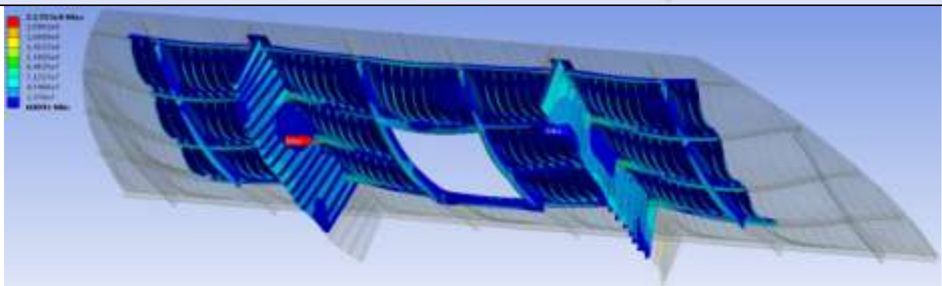
Optimisation results in detail for FORAN case study

Id		*Original configuration*	*124*	*152*
Deck	Plate thickness	5	20	19
	Stiffener web thickness	5	7	12
Transversal frame	Web thickness	5	34	34
	Web thickness (hatch frame)	5	29	29
	Flange thickness	5	38	40
Longitudinal girder	Web thickness	5	32	31
	Flange thickness	5	20	21
Longitudinal wall	Plate thickness	5	13	12
	Stiffener web thickness	5	22	22
Geometrical constraint: Tp-2xTW)S		-5	6	-5
Geometrical constraint: TW)F-4xTW)S		-15	6	-14
Geometrical constraint: TW)HF-4xTW)S		-15	1	-19
Structural constraint: MaxStress		1277.2	299.7	213.5
TotalWeight		34581.4	132280	132477

The structural models correspond to the above-mentioned iterations along with its FE results can be seen in Table 8 (the unit taken is Pa).

Table 8 Structural models along with its FE results for some iterations for FORAN case study

Iteration	*Structural model & FE results*
0	

124	
152 (optimum)	

V. CONCLUSIONS

The present work was performed in the framework of the research activity carried out by the European Project BESST "Breakthrough in European Ship and Shipbuilding Technologies". The challenge was the implementation of CAD and FEM software/tools in optimisation loops. Lots of efforts were put to manage correct connections and good data exchanges between different software/modules included in innovative structural optimisation workflows so that they successfully works in automatic processes without any manual intervention on graphical user interfaces. In this regard, a typical ship deck structure (as an initial case study) was taken into consideration to evaluate the iterative processes in the workflows.

The remaining study for the future is to work on a model respecting the structural necessities, in order to improve the optimisation processes by adding more structural constraints (buckling, fatigue, vibration, etc.) and considering additional objective functions (e.g. minimum cost, maximum inertia) to achieve a real feasible optimum solution.

ACKNOWLEDGMENT

The present work was performed in the framework of the research activity carried out by the European Project BESST "Breakthrough in European Ship and Shipbuilding Technologies". In this regard, the authors wish to acknowledge the support given by European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 233980 which has been led to the results presented in this paper.

REFERENCES

[1]. Birk L. Application of constrained multi-objective optimisation to the design of offshore structure hulls. Journal of Offshore Mechanics and Arctic Engineering 2008; 131 (No. 1).
[2]. Doig R., Bohm M. Simulation-based structural design of ships. In: Proceedings of the 11th International Symposium on Practical Design of Ships and other Floating Structures (PRADS), Rio de Janeiro, Brasil, 2010.
[3]. Bohm M. Interconnection of rules based CAD idealisation for ship structures to ANSYS. ANSYS Conference & 28. CADFEM Users´ Meeting, Aachen, Germany, 2010.
[4]. Alonso V, Gonzalez C, Perez R. Efficient use of 3D tools at early design stages. In: Proceedings of the 12th International Conference on Computer and IT Applications in the Maritime Industries (COMPIT), Cortona, Italy, 2013.
[5]. AVEVA Marine, Official homepage: http://www.aveva.com.
[6]. SENER Engineering Group, Official homepage of the FORAN system, Available at: http://www.foransystem.com.
[7]. ANSYS User's Manual (Version 14.0). Houston, Swanson Analysis Systems Inc. 2011.
[8]. ModeFRONTIER, Official homepage: http://www.esteco.com.
[9]. LBR5 User's Guide (version 5.8 a). Liege, University of Liege, 2005.