Multi-Objective Design Optimization of a Rudder, using Automated CAE Model Set-Up with ANSA Pre-Processor

George Korbetis, BETA CAE Systems S.A., Thessaloniki/Greece, ansa@beta-cae.gr
Dimitris Georgoulas, BETA CAE Systems S.A., Thessaloniki/Greece, ansa@beta-cae.gr

Abstract

Components of great importance for the ship safety are subjected to extensive analyses to ensure minimum failure risk and high performance. In such cases, the design process involves Finite Element Method simulations with which the model behavior for several loading conditions is evaluated. Analysts need to provide feedback to the designers regarding the simulation results and propose design changes towards the achievement of the required product's performance characteristics. Following that, the designers have to propagate the proposed changes by updating the products geometry. The automated definition of the CAE simulation model becomes essential during the above design loop, because a great amount of engineering hours are saved. Through an automated process, the analyst is able to produce CAE models for various disciplines and load cases, which are updated for each design change of the product. This paper presents the implementation of an automated process for the definition of a CAE model of a spade rudder with rudder trunk and the strength analysis of it. The automated CAE model set-up capability of ANSA pre-processor is used to define CAE models for static, contact and CFD analyses. ANSA, and μETA post-processor, which is exploited for the extraction of responses from the solver results, are coupled to the modeFRONTIER optimizer for the identification of the best rudder shape.

1. Introduction

The use of the Finite Elements Analysis becomes increasingly popular in marine design, since it can give us valuable information on the strength, hydrodynamics, and model behavior of complicated parts and assemblies. However it is always time consuming to set up and standardize a simulation process within the design process. The pre- and post- processors of BETA CAE Systems, ANSA and μETA, provide sophisticated tools in order to facilitate the FEA process. Time consuming tasks like meshing, model simplification and mesh quality improvement have been replaced by automated processes which save a lot of engineer working hours. Additionally, such tools guide the engineer to set up the model in the most proper way and help him to overcome any human errors.

This paper demonstrates the suggested process of the definition of a strength analysis and the application of the multi objective optimization during the design process on a ship's rudder. A simplified CFD (computational fluid dynamics) analysis calculates the maximum force applied on the rudder's surface. The calculated force is taken as the loading condition for the rudder's structural analysis. The static analysis is set up using automated processes like the results mapping, the contact pair definition and the batch meshing which are provided by ANSA. Model shape parameterization is performed by the ANSA Morphing Tool and the Optimization Task and different design variables are defined to control the model. To investigate the model behavior, the structural analysis is coupled with an optimization process. The first step of this process identifies the critical variables of the increase of the model's strength, while a second step performs a multi-objective optimization to find the best combination of the input variables.

2. Problem definition

The case study model is a spade rudder with rudder trunk of a Handysize bulk carrier. The ship's length is 169 m and its maximum speed 15 knots. When the ship is fully loaded the whole rudder and skeg are submerged beneath the sea. The rudder cross section is NACA 0015 and its main dimensions are x_1=3.56 m, x_2=5.05 m, b=6.2 and A=26.7 m^2, Fig.1. The rudder and stock material is steel with Young's Modulus 210 GPa, Yield stress 235 MPa and density 7.86·10^{-6} Kg/mm^2.
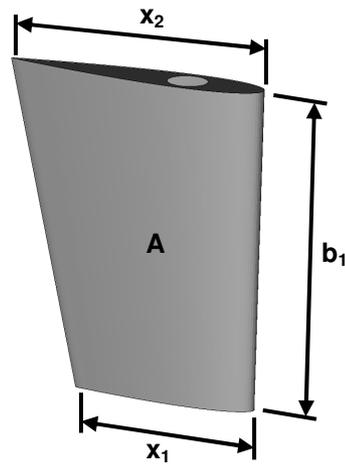
Fig.1: Rudder sketch

The stock is supported by a THORDON elastomeric bearing of type SLX assembled on the rudder truck. It is connected to the rudder's body by a cone coupling which is welded on the body. The stock uppermost part is clamped to a rotary valve which drives the rudder. The elastomeric bearing's Young's Modulus is 440 MPa, its maximum working contact pressure 10-12 MPa.

The main force that strains the rudder is produced by the water flow around it. The maximum force appears at the vessel's full speed, when the rudder turns to the maximum angle of $35°$. The weight and buoyancy are considerably small in relation to the water flow force; therefore they are not taken into account. To evaluate the force that is distributed to the whole surface of the rudder, a CFD analysis is performed. The results from the CFD analysis are used as boundary conditions to the structural analysis of the rudder assembly.

3. Model set-up

Two different FE models should be prepared for the structural and CFD analyses. Both models are created by the initial geometry, however, different representations are defined for each model. Each representation contains only the needed parts and entities for each analysis and the proper meshing parameters, quality criteria, boundary conditions and auxiliary geometry (i.e. fluid domain). The geometrical model is shown in Fig.2.
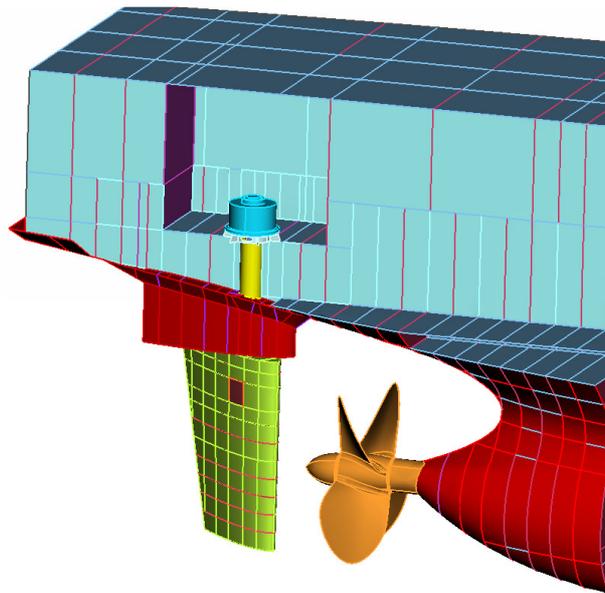
Fig.2: Geometrical model

3.1. CFD analysis

The first load case is a CFD analysis performed in order to evaluate the pressure on the rudder body. The first estimation of the pressure is done using a simplified model of the rudder. The effect of hull and propeller on the water flow is not taken into account for this case study and the flow is considered steady. For the definition of the CFD model, the outer skin of the rudder is isolated from the geometrical model and assembled to the computational domain which represents an open ocean area.

3.1.1. Meshing definition

The rudder's surface is meshed with a variable size triangular mesh. The boundary layer, generated on the rudder and the fluid surface separation, consists of five layers of prisms, generated in absolute mode with a growth factor of 1.2 and first height of 5 mm. A tetra mesh was generated on the fluid domain with element length varying from 0.05 to 2 m. In the fluid domain and more specifically on the area around the rudder, a fine mesh is required to increase the result's accuracy. In this case a special entity of ANSA, the Size Box, is defined in this area which constrains the element length of the fluid domain that resides inside the Size Box volume at 0.1 m. The final model consists of approximately 3,000,000 elements, Fig.3. The mesh is applied automatically on the model and fluid domain through the ANSA Batch Meshing Tool.
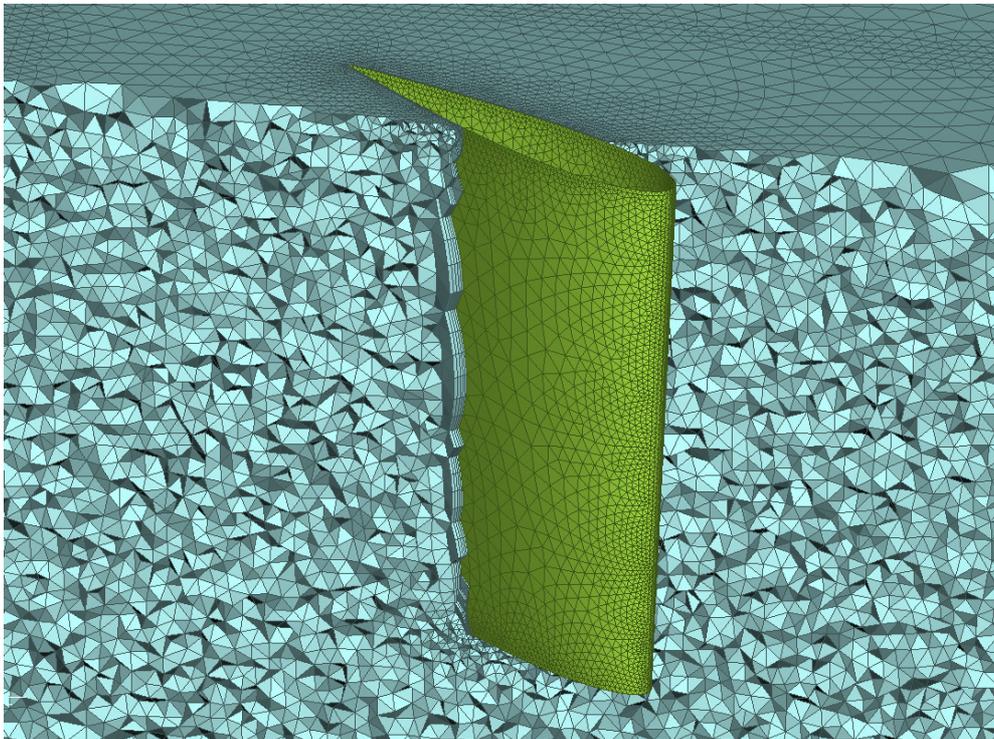
Fig.3: CFD mesh

3.1.2. Results in META

For the CFD analysis, ANSYS FLUENT v13.0 is used. After about 1000 iterations conversion is achieved. The static pressure is exported in ABAQUS format in order to feed the static analysis that will take place later on.

The results of the solution can be viewed and evaluated through μETA (ver. 6.7.2), which supports the read in of numerous CFD solvers results. As expected, the flow separates early on the rudder's surface, since the angle of attack is considerably high. μETA offers many ways to view CFD results such as streamlines, vectors and contours as shown in Figs.4 to 7 and is capable of calculating many factors and values like the Cd and Cl factors and the Yplus values.
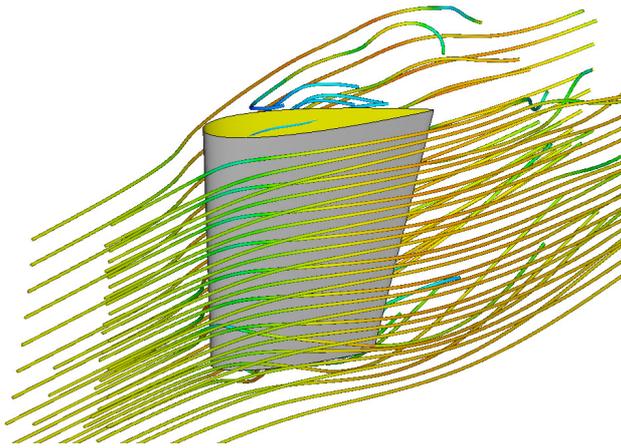
Fig.4: Velocity in streamlines
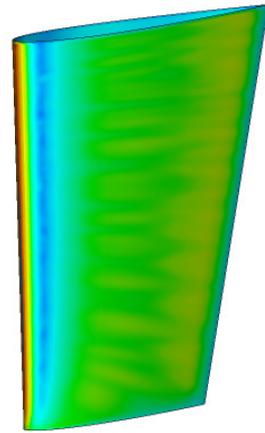
Fig.5: Yplus in contour plot
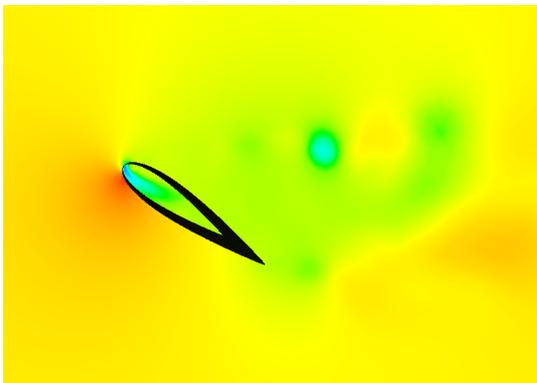
Fig.6: Pressure in contour plot
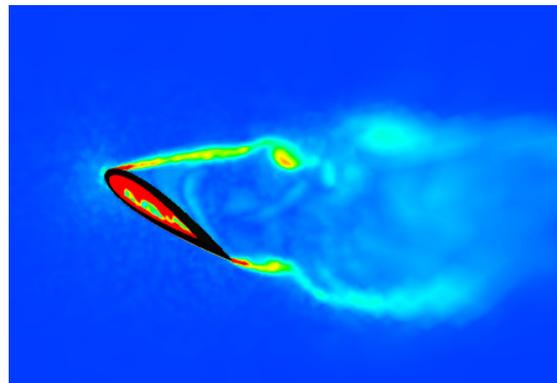
Fig.7: Vorticity in contour plot

3.2. The static analysis

The second analysis calculates the strength of the rudder assembly and the contact pressure between the bearing and the stock. The analysis is static, nonlinear and it is set up for the Abaqus 6.10. The representation of the geometrical model that participates in the analysis is shown in Fig.8.
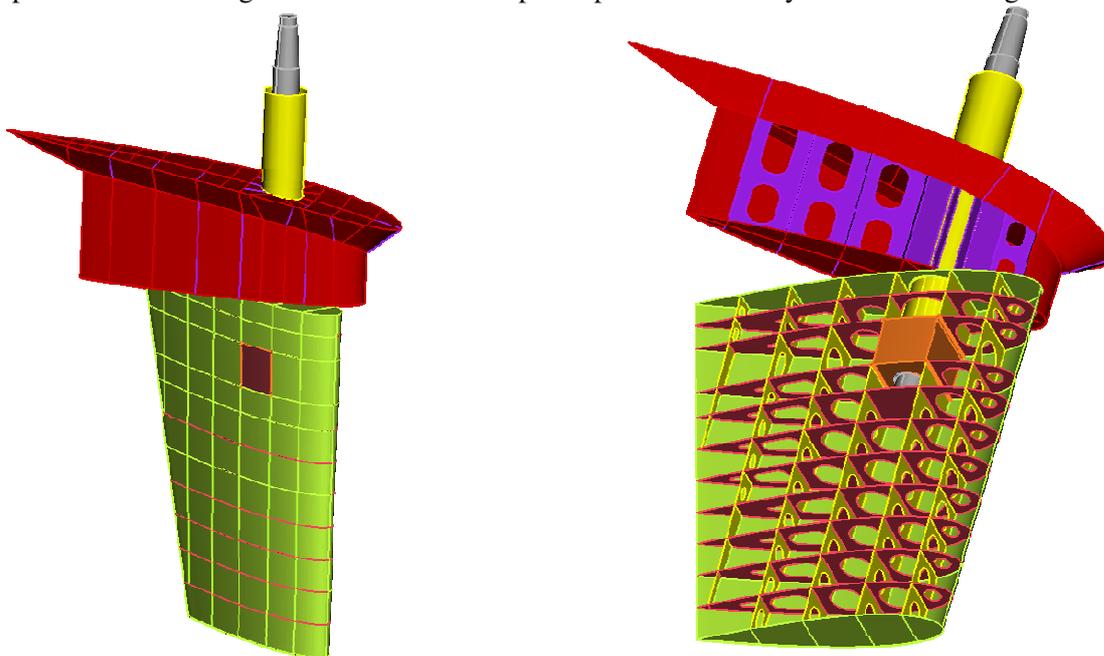
Fig.8: Structural model

3.2.1. Meshing definition

The rudder assembly is meshed with 1st order shell elements of mean element length 50 mm. Special treatment is prescribed for holes where on these perimeters one zone of elements is applied. The quality criteria that are used are listed on Table I. For the meshing process the ANSA Batch Meshing Tool is used. In this tool the meshing parameters and quality criteria are prescribed for a meshing scenario which is applied on the thin wall parts of the rudder and skeg. One more meshing scenario is defined for the cone coupling. In that case three layers of PENTAS are created on the part's conical surface to ensure successful coupling with the stock, since penetration problems and difficulty to converge may occur during solution. Above the layers, first-order solid elements (TETRAS) are prescribed for the rest of the solid part. The Batch Meshing Tool runs and applies meshing and quality improvement on all selected parts, according to the prescribed meshing scenarios without the need of any user interaction. The meshed parts are shown in Figs.9 to 11.

Table I: Rudder meshing specifications

Quality criteria	Shells	Solids	Calculation
Aspect ratio	3		Abaqus
Skewness	0.8	0.6	Abaqus
Warping	10		Ideas
Min. Length	15	15	
Max. Length	70	70	
Min. Angle Quads	45		Abaqus
Max. Angle Quads	135		Abaqus
Min. Angle Trias	30		Abaqus
Max. Angle Trias	120		Abaqus
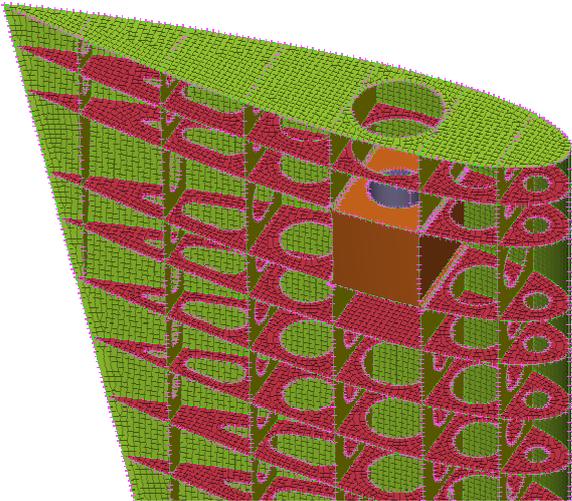
Fig.9: Rudder body mesh
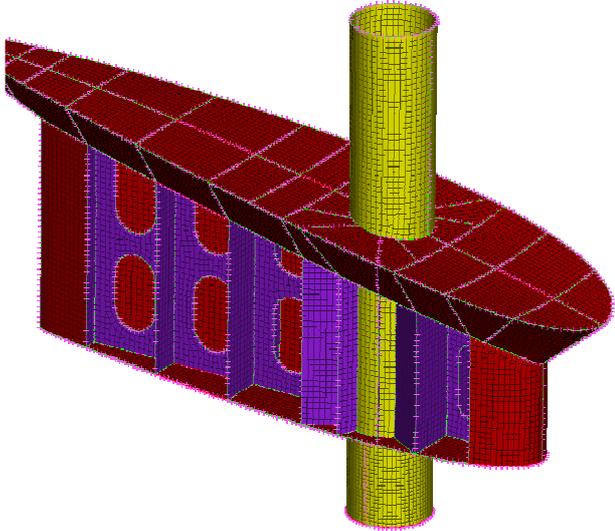
Fig.10: Rudder skeg mesh

The rudder stock and the bearing are parts of great importance for this analysis and their results need to be accurate. Thus, thin layers of HEXA elements are applied on the stock perimeter to ensure accuracy of the desired results. In addition, the whole stock model is meshed with HEXA elements. The HEXA meshing process is a semi-automatic process in ANSA since special entities, the Hexa Boxes, are created around the stock model. Each box represents a meshing domain, in which meshing parameters, such as node number, spacing and number of layers are defined. After the definition of the Hexa Boxes the mesh is created automatically. Three layers of HEXA elements are applied on the stock / bearing contact area with element length of 4.5 mm, while the rest of the part is meshed with 30 mm HEXA elements, Fig.12. The bearing is also meshed with HEXA elements. Since its geometry is very simple there is no need to use the Hexa Boxes, Fig.13.
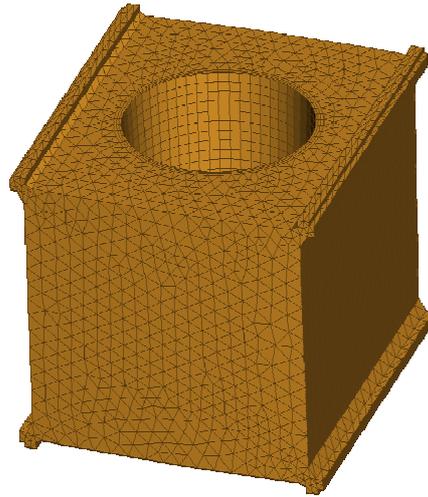
Fig.11: Cone coupling mesh
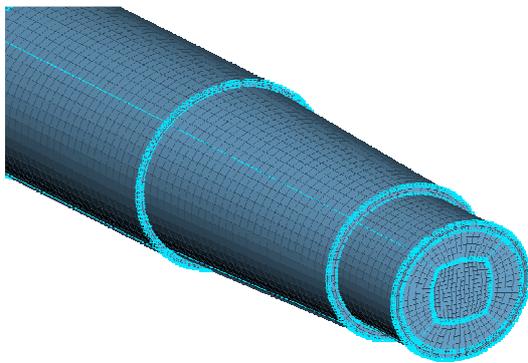
Fig.12: The stock mesh
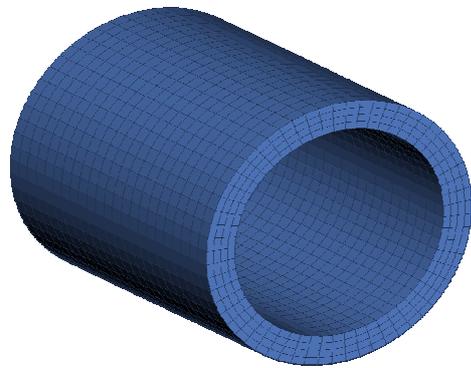
Fig.13: The bearing mesh

3.2.2. Defining boundary conditions

The structural analysis is limited to the rudder assembly, so the rest of the ship is considered rigid. Thus, boundary conditions are applied on the upper nodes of the rudder skeg which constrain the displacement and rotation in all degrees of freedom, Fig.14.
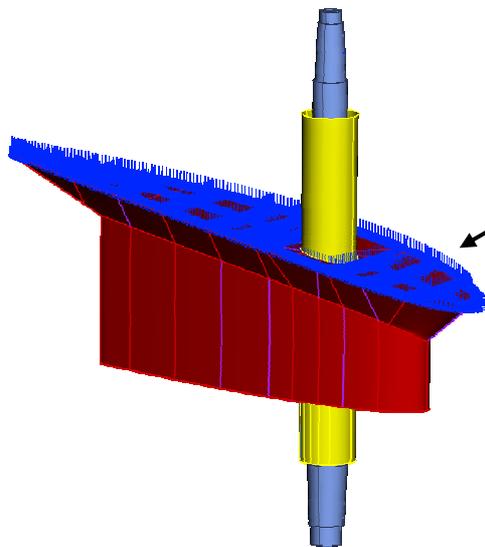
Fig.14: Boundary constraints of skeg
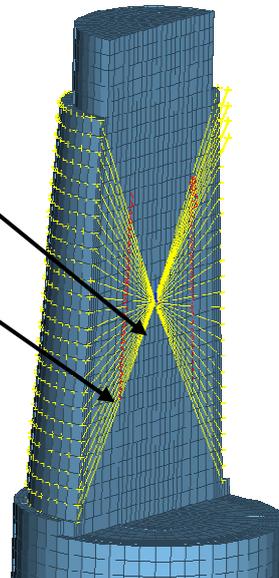
Fig.15: Boundary constraints of stock

The rotary valve is also considered as rigid body. The latter constrains the displacement of the stock at all axes and the rotation of the Z axis. To represent this constraint, a BOUNDARY entity of Abaqus is created in the center of the coupling which is connected to the stock coupling surface using rigid MPC elements, Fig.15.

3.2.3. Automatic definition of contacts

The rudder stock is connected to the rudder by the cone coupling and supported on the rudder trunk by the elastomeric bearing. These connections are simulated by CONTACT PAIR entities according to Abaqus, in order to evaluate the pressure that occurs on the contact surfaces during loading. A special tool of ANSA for this purpose identifies the mating surfaces of the candidate pairs so that the user can decide whether to accept them and define the type of the coupling. Two contact pairs represent the coupling of the stock with the cone flange, Fig.16. The type of the above pairs is defined as TIE according to Abaqus, since no movement is expected in this area. The contact pair between the stock and the bearing is defined as CONTACT, according to Abaqus, since the connection is not fixed and small sliding is allowed, Fig.17. In this case, the actual area of contact and the distribution of the pressure depend on the load case and the rigidity of the stock, trunk and bearing. A friction model is applied with friction coefficient of 0.2.
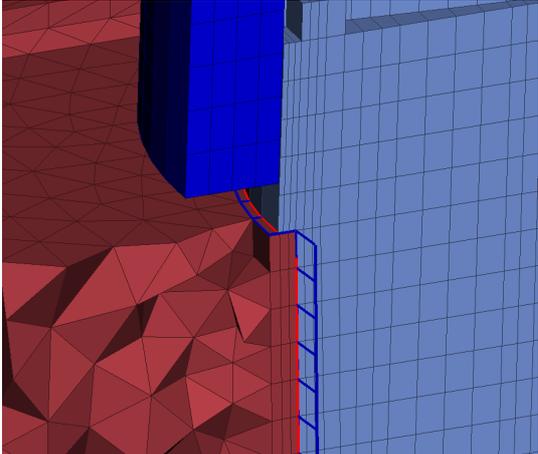
Fig.16: Contact between cone coupling and stock
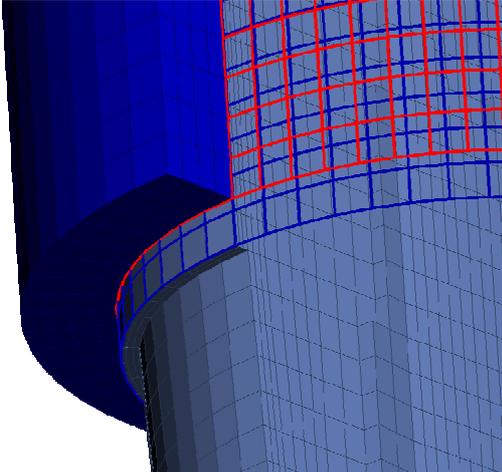
Fig.17: Contact between bearing and stock
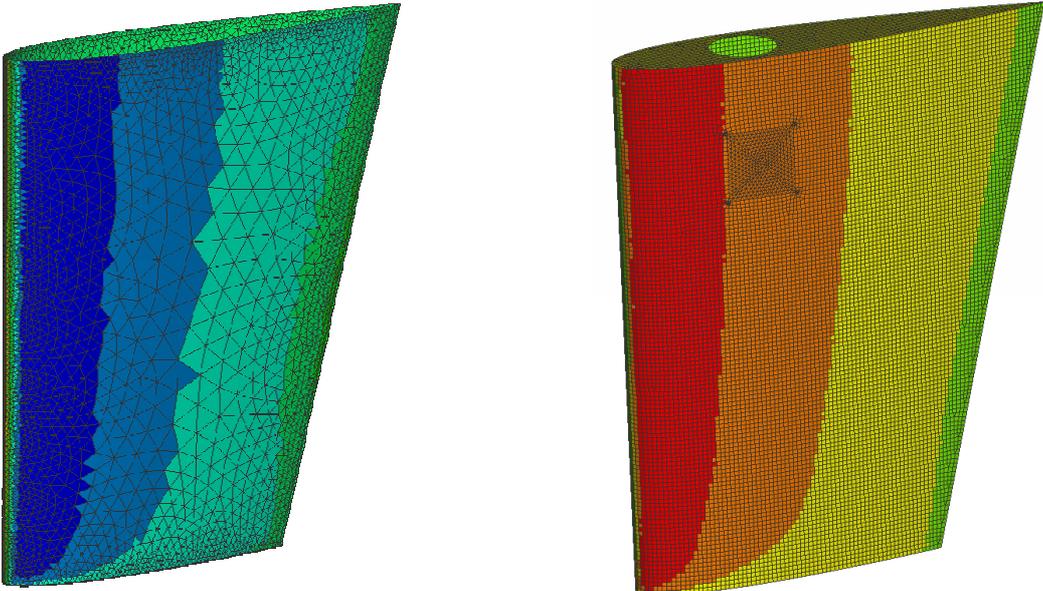
Fig.18: Mapping pressure from CFD to static models

3.2.4. Mapping the CFD results

The pressure applied on the rudder's surface, which is calculated by the CFD analysis, is considered as the loading condition for the static analysis. Therefore, the pressure results are exported from Fluent to an Abaqus format file containing pressure values for each element. However, the mesh of the rudder for the CFD analysis is not compatible with the one for the static analysis. In this case, the pressure derived from the CFD analysis is mapped to the static analysis mesh by regenerating equivalent pressure entities to the target elements. This is achieved with the use of the ANSA Results Mapper, which is able to import results of any type from numerous solvers, Fig.18. To complete the FE model that will be exported to Abaqus, the proper materials, shell and solid properties and header are defined. Output requests are defined for strain, stress and contact pressure.

4. Optimization problem

An initial run of the FE Model reveals the parts and areas where maximum stress and pressure appear. The μETA post-processor is used to identify the critical areas and create reports and statistics for all parts. As shown in the following figures the maximum value for the Von Misses stress occurs on the rudder trunk near to the supporting webs. The value 215 MPa which is close to the Yield stress makes the increase of the truck's thickness essential. Contact pressure results are read in μETA for the bearing - stock coupling. The maximum pressures appear close to the bearing's lower edge, Fig.20. The maximum pressure value is 9 MPa which is acceptable. However, more uniform pressure distribution is desired. Furthermore, the deflection of the rudder is measured at the lower nodes of the model which is 32.4 mm.
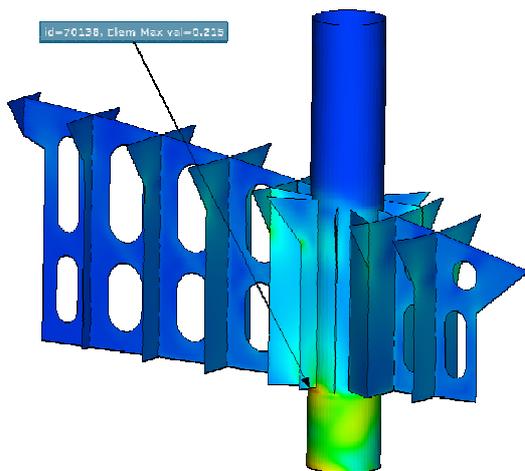
Fig.19: Von Misses stress at rudder trunk
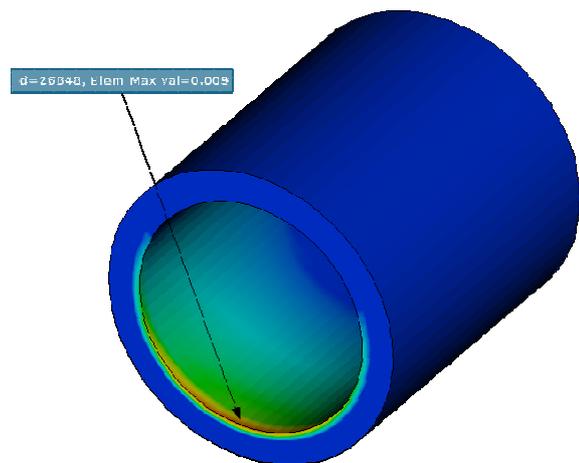
Fig.20: Contact pressure at bearing
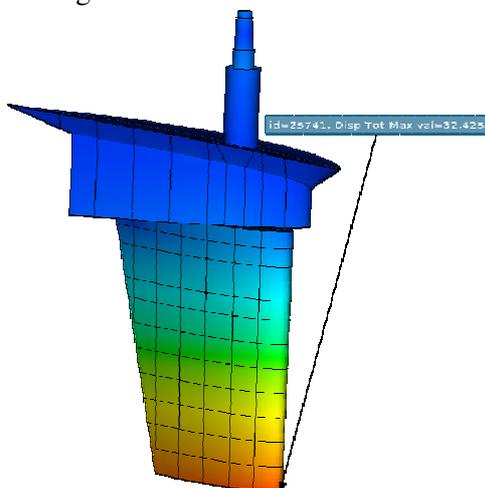
Fig.21: Deflection at rudder body
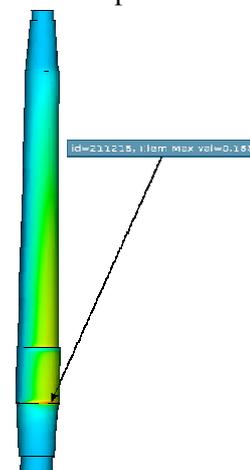
Fig.22: Von Misses Stress at stock

In this case study the model's behavior can be improved by modifying certain parameters that correspond to the model's shape and shell thickness. The target of the optimization is to minimize the contact pressure on the bearing and the deflection of the rudder body. The maximum stress and the total rudder mass should be kept within an acceptable range. The model's improvement is performed in two steps. First, a statistical analysis indicates which of the variables have big influence on the model's behavior and which would be the appropriate design space. Furthermore, a multi objective optimization searches for the optimum solution.

4.1. Defining morphing parameters

The shaping of the geometrical or FE model is achieved by the ANSA Morphing Tool. This tool provides several ways of morphing while ensuring smooth and controllable results. Special entities, the Morphing Boxes, are created around the area of the model to be modified. As the shape of the Morphing Boxes can be modified in several ways, the model surrounded by the Boxes follows the modification. Thus, the shaping takes place. Morphing should be performed parametrically in order to be driven by the optimization process. In this case Morphing Parameters which control the Morphing Boxes and are connected to the design variables of the optimization problem are defined. Several Morphing Parameters which control modifications of the model shape will be defined, Table II. Their bounds are prescribed according to the minimum / maximum permissible values that derive from technical specifications, model geometry and manufacturability.

Table II: The Morphing Parameters

Design Variable	Minimum	Initial	Maximum
Bearing Outer Radius	302	350	350
Bearing Inner Radius	265	275	275
Bearing Length	-300	0	300
Stock Radius	235	235	260
Holes Radius	-50	0	34
Vertical Web Move 1	-290	0	45
Vertical Web Move 2	-100	0	200
Trunk Support Webs	10	18	25
Trunk	10	20	25
Rudder Blade Skin	10	15	20
Rudder Horizontal Webs	10	15	20
Rudder Vertical Webs	10	15	20

For the modification of the stock geometry Cylindrical Morphing Boxes are used which are ideal for axi-symmetric models and the relative Morphing Parameters which control stock diameter between rotary valve coupling and bearing, Fig.23. The same group of Cylindrical Morphing Boxes also controls the thickness and length of the bearing using respective Morphing Parameters.
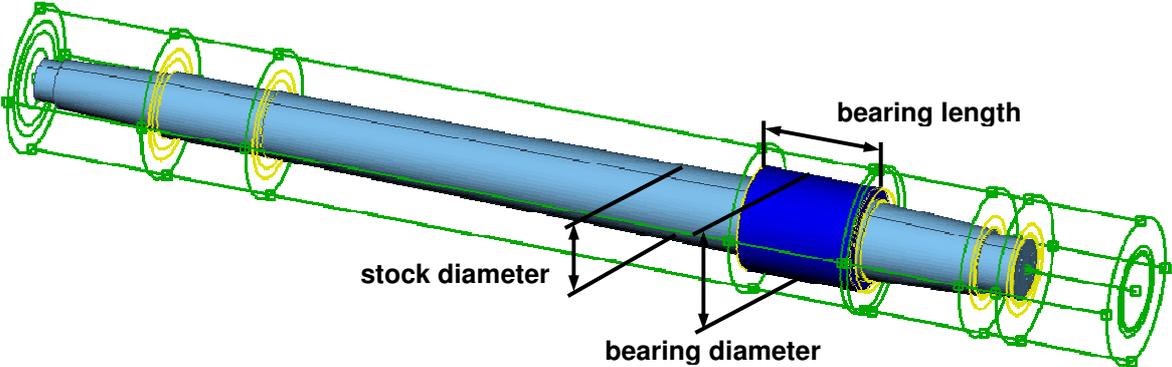
Fig.23: Cylindrical Morphing Boxes on the stock

Two groups of Hexahedral Morphing Boxes and relative Morphing Parameters are defined to control the position of the rudder's vertical webs, Fig.24. The webs slide along the rudder's outer skin without affecting its shape. Since the elements of the horizontal webs are squeezed during the above movement, special care should be taken to the horizontal webs holes in order to avoid distortion. ANSA Nested Elements are applied on the holes to constrain the holes' shape. Now, any movement of the vertical webs will lead to a "rigid" movement of the holes without any distortion of their shape and diameter.

Nested Elements
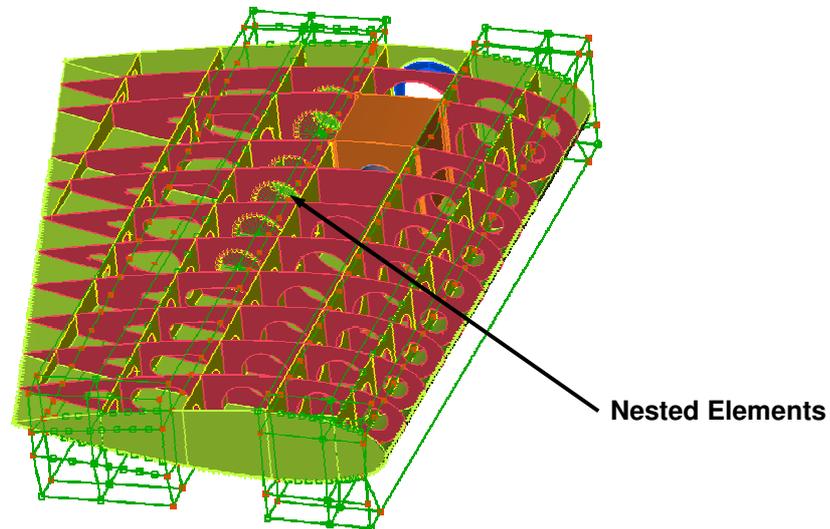
Fig.24: Hexahedral Morphing Boxes on the rudder body

A last Morphing Parameter is defined for the modification of the holes diameter on the horizontal and vertical webs. The parameter applies a relative modification to all webs holes. This is achieved by Direct Morphing capability of the Morphing Tool, which enables the shaping without the definition of Morphing Boxes and can be controlled by morphing parameters as well.

4.2. Defining ANSA parameters

The shell thickness of several parts of the rudder assembly is set as design variables. These are the parts of the vertical and horizontal webs, the trunk, the trunk supporting webs and the rudder skin. The shell thickness values are extracted to design variables through the ANSA Parameters which can control parametrically any entity of the FE model.

4.3. Set-up the Optimization Task

The Optimization Task is a powerful ANSA tool, which facilitates the set-up of an optimization problem, Fig.25. Here, the engineer can define the sequence of every action that will be executed in each optimization loop. The design variables and their properties and bounds are defined in this Task and connected with the Morphing and ANSA parameters that where defined previously. Furthermore, complicated actions like the mesh improvement or reporting can be defined in this Task using the ANSA scripting language.

In this case study three actions are added to the Optimization Task to ensure the model's validity. After morphing, some elements are squeezed, which leads to poor mesh quality. A scripting command is added after the morphing sequence which identifies areas of poor quality elements and regenerates the mesh. The second scripting command added is a check for penetration among the parts which may occur after changing their thickness. Finally, one more scripting command is added which calculates the model's mass and exports it to an ascii file. The mass value will be used as objective parameter for the optimization problem.
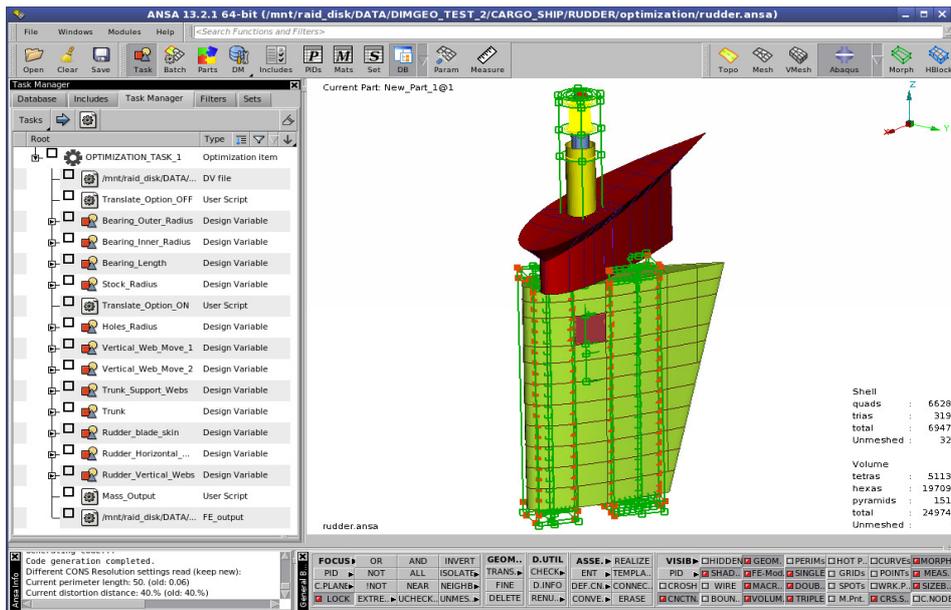
Fig.25: Optimization task

4.4. Extracting responses from META

The objective parameters and constraints of the optimization problem are extracted from the results of the structural simulation. To achieve this, a baseline run of the static analysis is performed and the results are read in μETA. Through the Annotation tool of μETA, the maximum value of the contact pressure on the bearing and the maximum stress and deflection on the rudder assembly are identified. The OptimizerSetup tool of μETA is used to export the identified values into an ascii file of a special format, Fig.26. The file is read from modeFRONTIER, www.modefrontier.com, and the values are set as responses. The above sequence, of extracting the responses, is recorded automatically in a session file in order to be reproduced in every optimization loop.
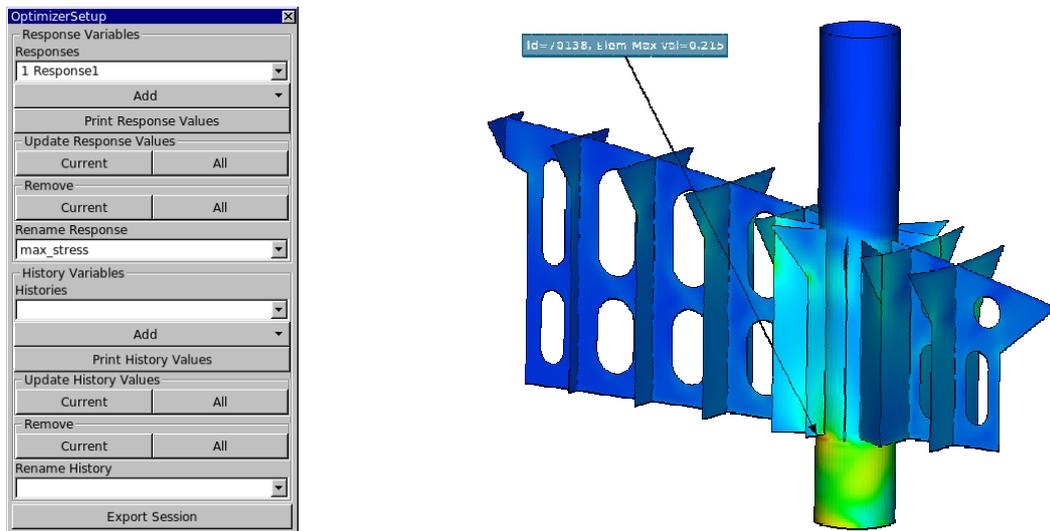
Fig.26: OptimizerSetup Tool

4.5. Coupling ANSA / META to modeFRONTIER

For this optimization problem the modeFRONTIER 4.4.1 is used which provides dedicated "nodes" for ANSA and μETA. Thus, the coupling of the pre- and post- processing sequence is done without the need of any scripting or customization. As the ANSA node is defined, the input variables are automatically defined corresponding to the Design Variables that have been set in the ANSA

Optimization Task. One more "node" is set for the Abaqus solver which is used in this analysis. This "node" receives the FE model that ANSA exports in every optimization loop as input. Finally, the μETA "node" is defined which reproduces the μETA sequence of extracting responses and feeds the constraints and objective parameters. In this case study the maximum contact pressure of bearing surface and rudder assembly mass are set as objective parameters while the maximum stress and deflection are set as constraints, Fig.27.
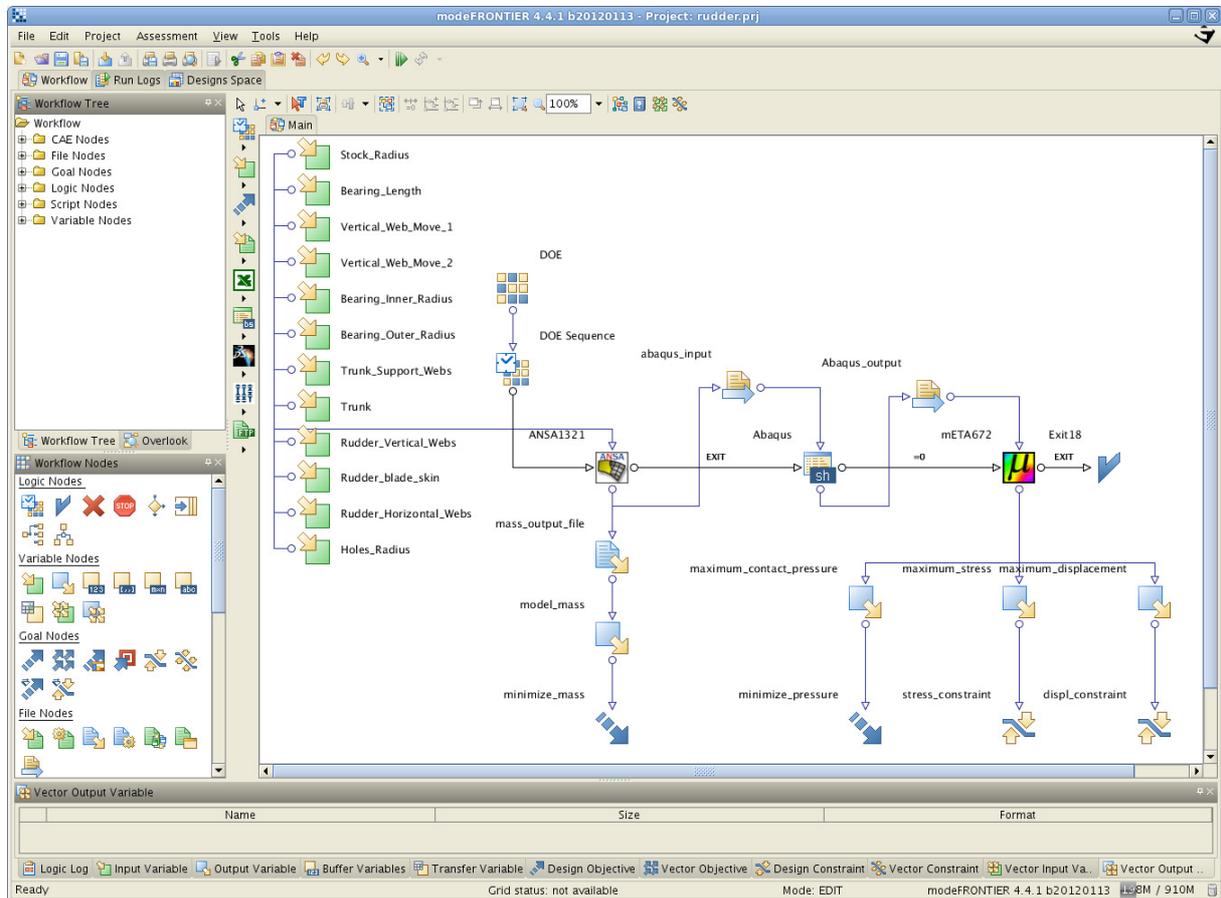
Fig.27: modeFRONTIER setup

5. Statistical analysis

The time needed to solve the optimization problem depends on the number of design variables, the number of iterations and the size of the model that has to be solved. Selecting a large number of design variables and a wide design space may lead to accurate results, but it would be a very time consuming process. On the contrary, if only a few design variables are selected, the process will be fast but some important solutions may be missed. In order to overcome this problem, a statistical analysis for correlation is performed prior to the optimization. All the candidate design variables participate to this analysis. A DOE Reduced Factorial algorithm is set in modeFRONTIER which creates 30 experiments in order to perform the statistical analysis.

The correlation between the design variables and the responses gave a valuable outcome for the setup of the optimization problem. As seen in Fig.28 the design variables Holes Radius, Vertical Web Move 1, Vertical Web Move 2 and Rudder Blade Skin do not contribute significantly to the objective parameters change. So they are excluded from the optimization problem set up. Furthermore, the design variables Rudder Vertical Web and Rudder Horizontal Web have positive correlation with the minimize_mass objective parameter. Thus, the upper bound of both design variables can be lowered from 20 to 15. The actions above will reduce the design space and therefore the calculation time without any compromise to the optimization problem's accuracy.

Fig.28: Statistical analysis results

6. Performing optimization

After the final adjustment of the design variables, the optimization problem is set up. The Multi Objective Genetic Algorithm (MOGA II) is selected and the process runs for 360 iterations. The feasible iterations are shown in the contact pressure vs. mass diagram, Fig.29, in grey color while unfeasible ones in white. The bubbles diameter represents the maximum stress. The Pareto Frontier appears which shows the best solutions for this case study. The optimum solution reduces contact pressure for 12.5% and the mass for 7.98% while the maximum stress and the displacement are kept below the acceptable values, Table III. The design variables values for the optimum solution are shown on Table IV.
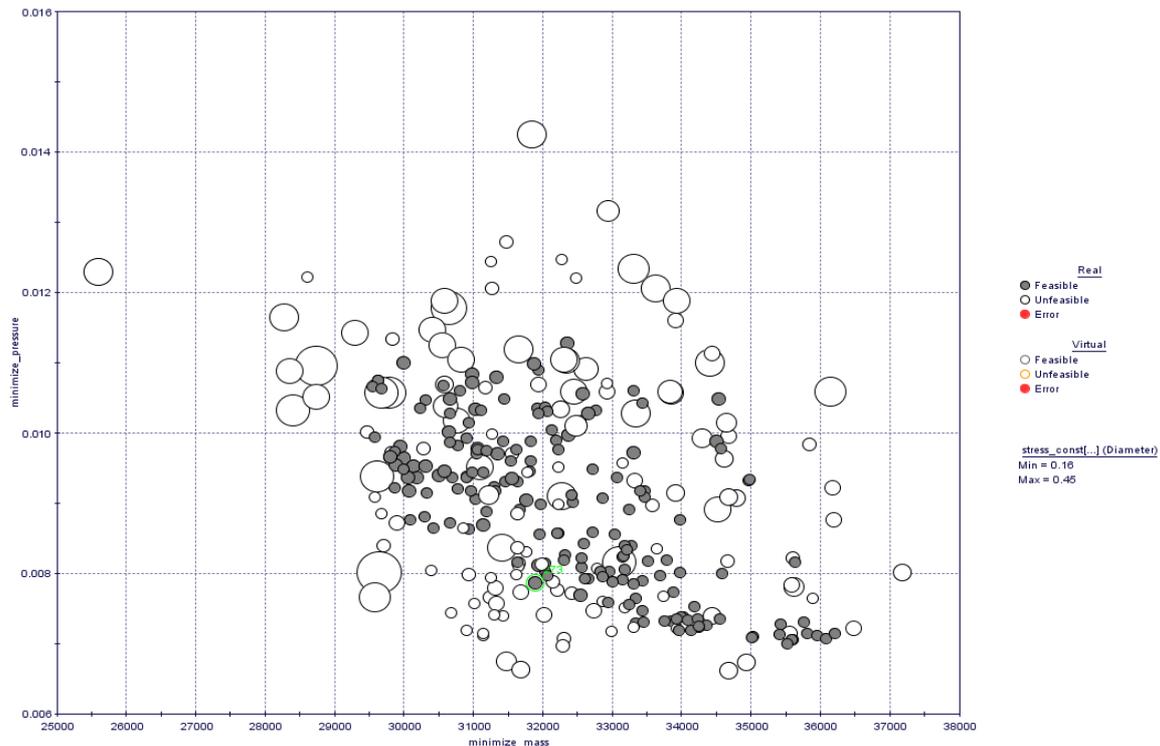
Fig.29: The contact pressure vs. mass diagram

Table III: Design variable values

Design Variable	Initial	Optimum
Bearing Outer Radius	350	338
Bearing Inner Radius	275	268
Bearing Length	0	300
Stock Radius	235	260
Trunk Support Webs	18	10
Trunk	20	21
Rudder Horizontal Webs	15	14
Rudder Vertical Webs	15	15

Table IV: Objective functions and constraints

Objectives / Constraints	Initial	Optimum	Percentage reduction
Contact pressure [MPa]	9	7.868	12.5
Mass [Kg]	34654.2	31888.4	7.98
Maximum Stress [MPa]	215	183	14.88
Displacement [mm]	32.4	31.9	1.54

7. Conclusions

In this case study a spade rudder with rudder trunk was subjected to a design optimization process in order to improve its strength and behavior. The results of the analysis led to significant changes to its shape and shell thickness. To define the whole optimization process, several analyses are needed, as well as the use of many programs. Furthermore, many actions are needed to setup the FE models for the analyses to be performed. Thus, the use of tools that automate the simulation process and facilitate the analysis set up turns to be a matter of great importance.

ANSA and μETA pre- and post- processors have been used successfully for the definition of CFD and structural analysis. The use of sophisticated tools like Batch Meshing, Results Mapper and Contact Wizard of ANSA facilitates and automates the definition of a ready to run FE model. Furthermore, the Morphing Tool and the functionality of ANSA and μETA for the optimization set up offers the engineer a great tool for the geometrical or FE model parameterization and optimization.

References

GIASSI, A.; MAISONNEUVE, J.; BENNIS, F.; (2003), *Multidisciplinary optimization and robust design approaches applied to the concurrent design,* COMPIT, Hamburg

GL (2011), "*Rudder and Manoeuvring Arrangement" and "Steel Plates, Strips, Sections and Bars"*, Rules and Guidelines of Germanischer Lloyd, Hamburg

RAWSON, K.J.; TUPPER, E.C. (1997), *Basic ship theory*, Butterworth & Heinemann, pp.555-563