TRANSNAV
http://www.transnav.eu
the International Journal
on Marine Navigation
and Safety of Sea Transportation
Volume 9
Number 2
June 2015

DOI: 10.12716/1001.09.02.12

Numeric Wake Equalizing Duct Geometry Optimization for a Given Ship

G. Martinas & O.S. Cupsa
Maritime University of Constanta, Romania

ABSTRACT: The reduction of fuel cost has always been one of the key strategic business goals for ship owners and operators. In the current climate of high oil prices, the reduction of fuel costs becomes essential; and furthermore a variety of recent legislations require owners and operators to move towards the reduction of emissions from ships of SOx, NOx and CO.
Hence the pressure on designers to achieve both reduced fuel costs and reduced emissions by optimising the hull and propeller has never been higher. In parallel to the performance improvement of new built vessels, there has been great interest in the potential to enhance the performance of existing vessels through retrofit of devices to the hull. In any case for instance the WED device must be customized to fit to the afterbody of the ship in terms of performing its supposed function. The Designer is therefore placed in the front of multiple geometric solutions from between he has to make a choice. This paper is intended to help the Designers to have a rational choosing approach by involving the numeric optimization of the geometry of the WED in order to select the best fitted WED to perform the best in order to achieve some predefined parameters. In this paperwork a given geometry of a WED device is taken and via Design Optimization the geometry of the duct was refined so that better results are achieved with a smaller and more compact WED. In doing so, the Designer is assisted by numeric optimization methods to choose from only three final candidates instead of several thousands in order to provide the best fitted WED geometry for a given ship afterbody.

1 INTRODUCTION

The reduction of fuel cost has always been one of the key strategic business goals for ship owners and operators. In the current climate of high oil prices, the reduction of fuel costs becomes essential; and furthermore a variety of recent legislations require owners and operators to move towards the reduction of emissions from ships of SOx, NOx and CO.

Hence the pressure on designers to achieve both reduced fuel costs and reduced emissions by optimising the hull and propeller has never been higher. In parallel to the performance improvement of new built vessels, there has been great interest in the potential to enhance the performance of existing vessels through retrofit of devices to the hull. A wide range of concepts has been proposed, many of which involve modification or control of the flow in the vicinity of the propeller. The interest in these devices arises with increasing oil price. These devices are commonly called “energy saving devices (ESD)” and sometimes “retrofitting technologies” although many can be considered for new designs as well.

The savings look very attractive to ship operators, for instance a saving of 6-7% from installation of a wake equalising duct (Schneekluth, 1986, Schneekluth

(WED) and Bertram, 1998) or 7-9% from a combination of wake equalising duct and pre-swirl fins (Mewis, 2008, Mewis, 2009). In general, the negative aspects of the devices include the considerable cost of installation, and also the reported reluctance of manufacturers to guarantee the claimed savings.

The claims may not give details as to the conditions under which the savings have been achieved and/or how the savings have been calculated and/or measured. Furthermore, the magnitude of the savings may well be within the range of uncertainties and measurement errors on the full-scale vessel. Consequently, cautious operators may well be skeptical about the validity of the figures being presented to the market, and it is absolutely reasonable and necessary for a buyer to verify independently the amount of savings before any investment on an ESD or ESDs.

In any case for instance the WED device must be customized to fit to the afterbody of the ship in terms of performing its supposed function. The Designer is therefore placed in the front of multiple geometric solutions from between he has to make a choice. This paper is intended to help the Designers to have a rational choosing approach by involving the numeric optimization of the geometry of the WED in order to select the best fitted WED to perform the best in order to achieve some predefined parameters.

2 CAD AND FINITE VOLUME ANALISYS (FVA) MODEL OF THE SHIP

The goal of this paper is to calculate via software Ansys 13™ the best geometric solution for a WED device. It is known that a poor design of the WED is not only improving the overall efficiency of the vessel but may have an adverse impact failing to achieve its purpose.

The model has as departure point a real portcontainer as seen below, with the following parameters (Fig. 1):

- Length L- [m]- 173
- Breadth B- [m]- 25
- Draught T -[m] -9.50
- Diameter D- [m]- 5
- Number of blades Z - 6
- Propeller RPM-120
- Average Speed-16 knots (7 m/s)

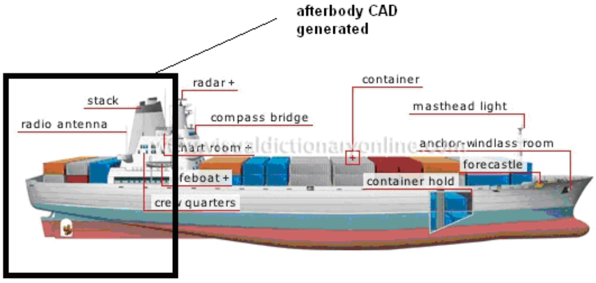

Figure 1. Port-Container

In order to have a starting point for the simulation, first of all the afterbody was firstly CAD generated with the WED device attached, and all the parameters for fluid flow were calculated accordingly. From that point, the Ansys Design Explorer was involved in order to optimize the geometry (Fig. 2).

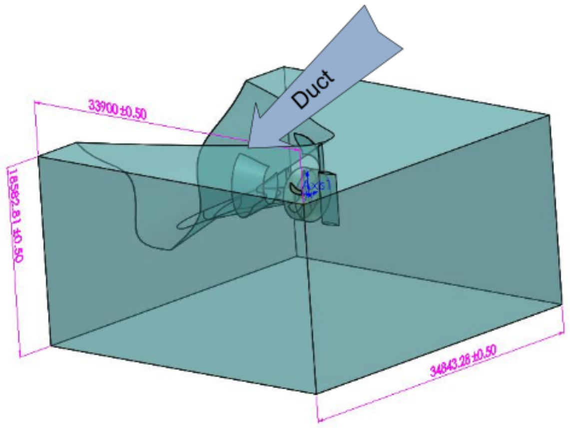

Figure 2. CAD geometry with WED

In order to provide more details on the geometry of starting model of the WED device and the optimization input parameters, the below figure is shown, with dimensions in [mm] (Fig. 3):

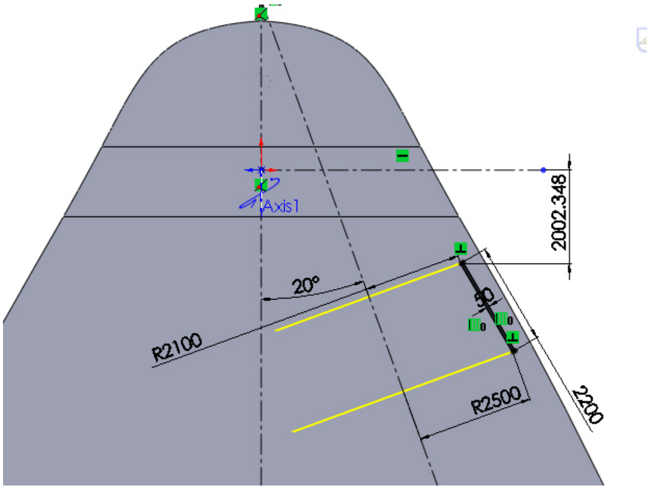

Figure 3. WED device geometry

5 input geometric parameters were defined as follows (Table 1):

Table 1. Input geometric parameters

	Name	Type	Lower limit	Upper limit
P1	Angle Minimize	Continuos	14 [grade]	22 [grade]
P2	Duct Length Minimize	Continuos	1980 [mm]	2420 [mm]
P3	Small radius Minimize	Continuos	1600 [mm]	2250 [mm]
P4	Bigger cone radius Minimize	Continuos	2250 [mm]	2750 [mm]
P5	Distance from the propeller Minimize	Continuos	1800 [mm]	2200 [mm]

The fluid domain was divided in two: the fluid domain which is surrounding the afterbody having the relative velocity on Oz axis of 7 m/s and the Propeller fluid domain with CFX option of "frozen Rotor" where the fluid is moving circularly around OZ axis with 120 RPM. In between these two domains interfaces were established. The other boundary conditions were inlet, outlet and openings as shown below (Fig.4):

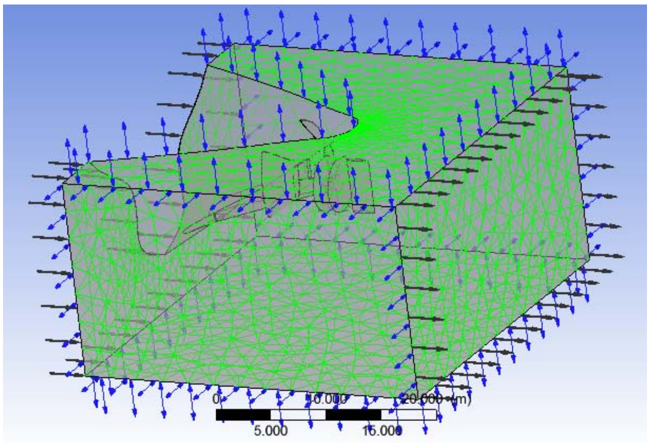

Figure 4.Boudary Conditions

In order to make clear some important surfaces, three control planes were defined as follows (Fig. 5):

- Control plane number 1 (P1) placed at 1200 mm above the propeller axis and coplanar with the two WED devices axis;
- Control plane number 2 (P2) which is including the propeller axis;
- Control plane number 3 (P3) placed at 1500 mm away from the propeller domain;
- Target Plane which is in fact one of the propeller domain interfaces as below:

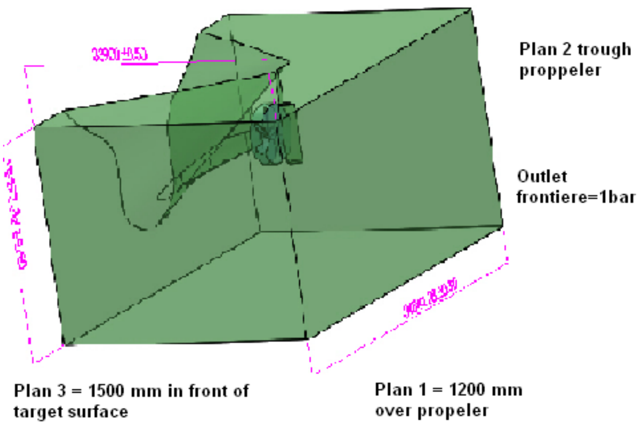

Figure 5.Control Planes

Taking into account the above defined control planes as output parameters needing to be optimized, were defined as being the **average fluid velocity** passing through the Target Plane (suspected to improve the propeller efficiency-the bigger the better) and the **average pressure** on the inside of the WED device (suspected to increase the drag-the smaller the better) (Table 2):

Table 2. Output parameters

ID	Parameter Name	Starting Value	Unit
P6	Velocity Target (maximize)	14.769	m s^-1
P7	Pressure Duct (minimize)	89874	Pa

3 CFA SIMULATION AND OPTIMIZATION RESULTS

After reaching the convergence of the given starting models and going through Design Explorer Module, 27 design points (Table 3) were calculated in order to define the response surfaces of the project:

Table 3. Design Points

Name	P1 - Angle (degree)	P2 – Duct Lenght (mm)	P3 – Small Radius (mm)	P4 – Bigger Cone Radius (mm)	P5 – Distance Propeller (mm)
1	18	2200	1925	2500	2000
2	14	2200	1925	2500	2000
3	22	2200	1925	2500	2000
4	18	1980	1925	2500	2000
5	18	2420	1925	2500	2000
6	18	2200	1600	2500	2000
7	18	2200	2250	2500	2000
8	18	2200	1925	2250	2000
9	18	2200	1925	2750	2000
10	18	2200	1925	2500	1800
11	18	2200	1925	2500	2200
12	16.867	2137.7	1832.9	2429.2	2056.7
13	19.133	2137.7	1832.9	2429.2	1943.3
14	16.867	2262.3	1832.9	2429.2	1943.3
15	19.133	2262.3	1832.9	2429.2	2056.7
16	16.867	2137.7	2017.1	2429.2	1943.3
17	19.133	2137.7	2017.1	2429.2	2056.7
18	16.867	2262.3	2017.1	2429.2	2056.7
19	19.133	2262.3	2017.1	2429.2	1943.3
20	16.867	2137.7	1832.9	2570.8	1943.3
21	19.133	2137.7	1832.9	2570.8	2056.7
22	16.867	2262.3	1832.9	2570.8	2056.7
23	19.133	2262.3	1832.9	2570.8	1943.3
24	16.867	2137.7	2017.1	2570.8	2056.7
25	19.133	2137.7	2017.1	2570.8	1943.3
26	16.867	2262.3	2017.1	2570.8	1943.3
27	19.133	2262.3	2017.1	2570.8	2056.7

Taking each and every design points to be calculated via CFA (Table 4), the output parameters to be optimized are shown below:

Table 4. Output calculated parameters for each design points

Name	P6 – Velocity Target (m s^-1)	P7 – Pressure Duct (Pa)
1	14.518	1.0123E+05
2	14.75	99999
3	14.514	1.0294E+05
4	14.553	1.02E+05
5	14.482	1.0077E+05
6	14.527	1.033E+05
7	14.541	97174
8	14.561	97907
9	14.581	1.0307E+05
10	14.556	1.0094E+05
11	14.571	1.0135E+05
12	14.65	1.015E+05
13	14.596	1.0155E+05
14	14.574	1.008E+05
15	14.543	1.0136E+05
16	14.6	98976
17	14.539	99788
18	14.563	98563
19	14.576	99455
20	14.595	1.0255E+05
21	14.553	1.0314E+05
22	14.547	1.0218E+05
23	14.514	1.0297E+05
24	14.588	98469
25	14.577	1.0143E+05
26	14.57	1.0046E+05
27	14.558	1.0152E+05

The software is automatically selecting the maximum and minimum calculated values of output parameters as below (Table 5):

Table 5. Minimum and maximum output parameters

	Minimum	Maximum
P6 - VelocityTarget (m s^-1)	14.478	14.784
P7 - PressureDuct (Pa)	67648	1.4648E+05

By judging the above minimum and maximum value of the pressure inside the duct, the difference is 20 folds which is clearly making a difference between a good and a poor design.

By setting the goals of maximization or minimization defined above for all the parameters, at the end of the optimization process three best candidates will be generated (Table 6):

Table 6.The three best candidates

	Candidate Point 1	Candidate Point 2	Candidate Point 3
P1 - Angle (degree)	★★★ 14.164≈ 14 grade	★★★ 14.644	★★ 15.284
P2 - Duct Length (mm)	★★ 2049≈ 1050	★★★ 1997.4	★★★ 1988.8
P3 – Small Radius (mm)	★ 2081.8≈ 2080	★★★ 2242.3	★ 2028.3
P4 – Bigger Cone (mm) Radius	★★ 2330.3≈ 2330	★★★ 2282.3	★★ 2295.1
P5 – Distance From (mm) The propeller	★★ 2159.4≈ 2160	▬ 2005.4	★★ 2154.7
P6 – Velocity Target (m s^-1)	★★★ 14.78	★★ 14.756	★ 14.702
P7 – Pressure Duct (Pa)	★ 98351	★ 94591	▬ 1.0633E+05

Here once again the Designer is called to make a decision, we choose the first candidate having three stars to the most of the parameters.

In order to have an overall idea on the influence of each and any input parameter has on the output parameter, the sensitivity charts and response surfaces are the best aids for judgment. Since the possible combinations are many, only few response surfaces are given below.

Duct Pressure-Duct Angle and Duct Length Response Surface (Fig.6)

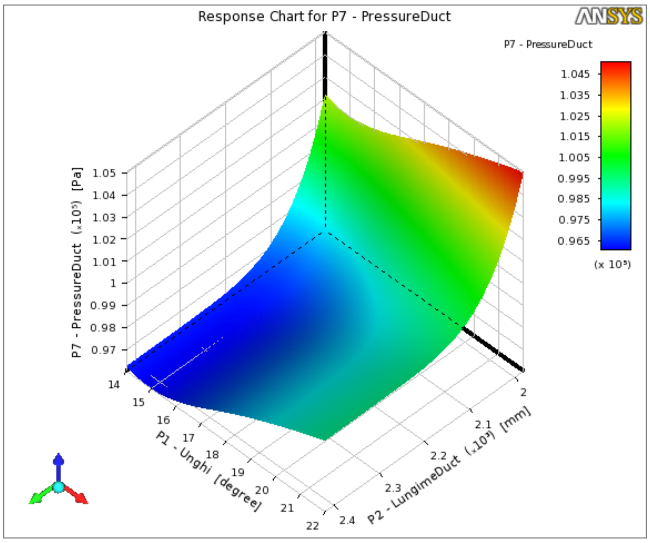

Figure 6. Duct Pressure-Duct Angle and Duct Length Response Surface

From the above chart one may imply that the duct Angle has a bigger influence than the Length of the duct over the Pressure inside the duct, which is somehow in line with a common sense hypothesis. The smaller the angle is and the bigger the length is then the smaller the inside pressure will be.

Duct Pressure-Duct Angle and Duct Small Radius Response Surface (Fig.7)

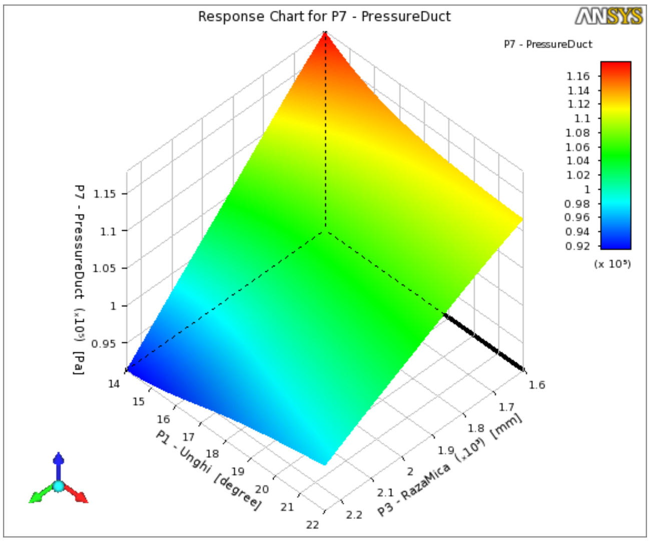

Figure 7. Duct Pressure-Duct Angle and Duct Small Radius Response surface

The pressure inside the duct will be smaller as the angle is smaller and the small radius will be smaller.

Velocity over the Target Surface-Duct Angle and Duct Length Response Surface (Fig.8)

The smaller the Duct angle (near 14 degrees) and the Length is, the bigger the velocity of the fluid on the target surface will be.

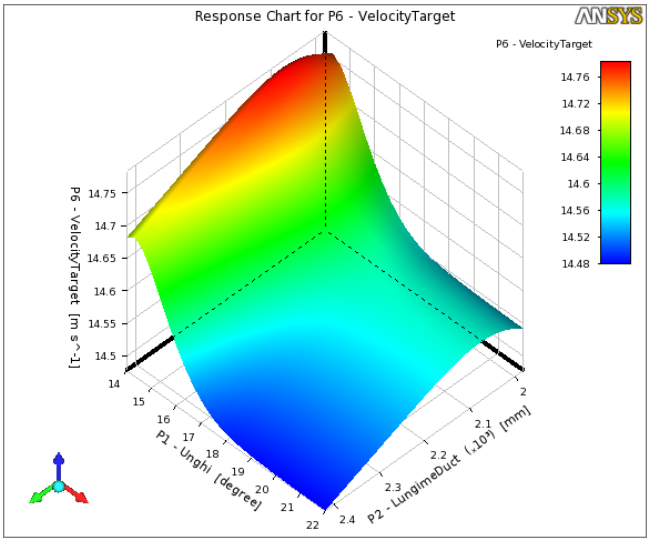

Figure 8. Duct Pressure-Duct Angle and Duct Small Radius Response surface

In order to place all the influences on a single chart (Fig. 9), the sensitivity chart is used as below:

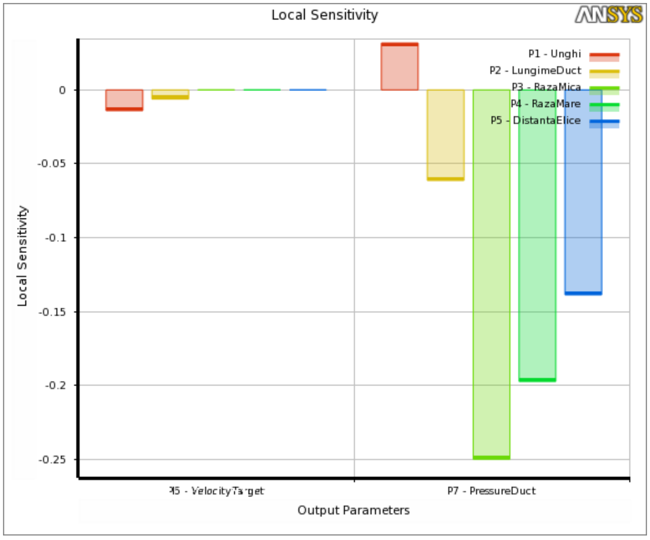

Figure 9. Sensitivity chart

Finally if one wants to visualize the effect of the optimized input parameters, we can recalculate the model with these new optimized parameters for geometry resulting for the output parameters the following figures:

The influence of optimized parameters on the velocity through the target surface (Fig. 10-11)

By analyzing the figures below, one may see that the shape of the velocity fields on the upper zone of the propeller is more extended, so that the optimized version is “pushing” more fluid on this zone.

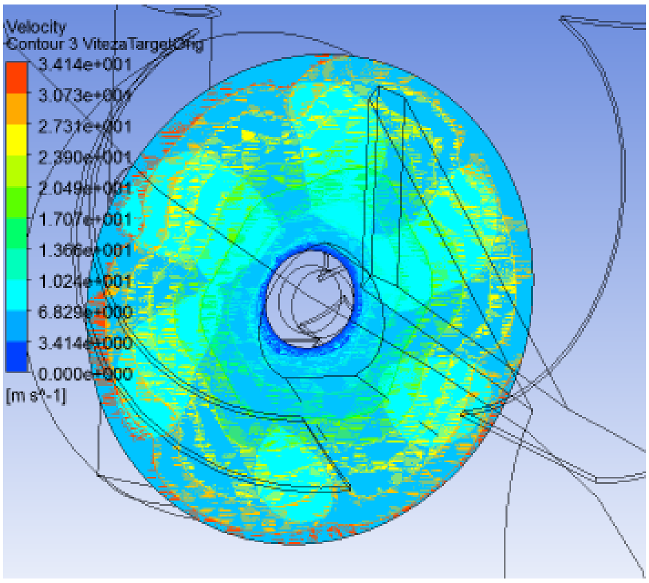

Figure 10. The starting model velocities

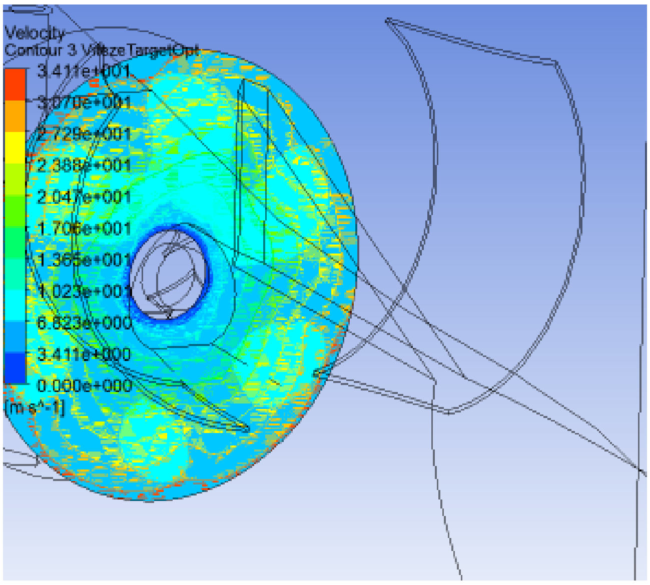

Figure 11. The optimized model velocities

The influence of optimized parameters on the pressure inside the duct (Fig. 12-13)

The starting model has a bigger (red colored) pressure fields on the inside the duct whereas the optimised version has smaller and placed at the outlet zone of the duct pressure fields, which is an indication that the inside pressure and the drag is then diminished for the optimized version.

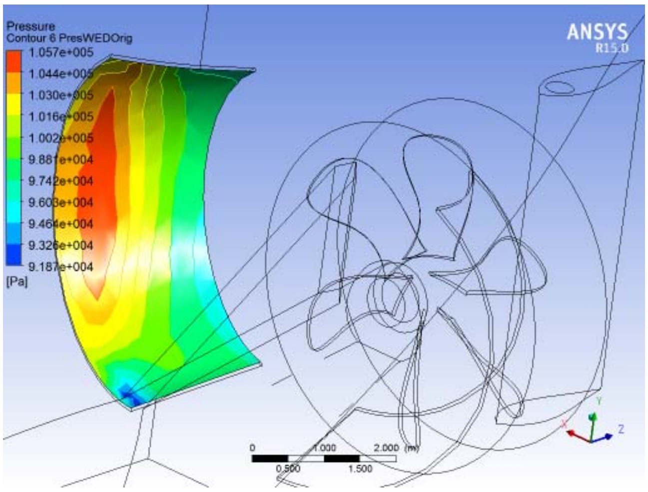

Figure 12. The starting model duct pressures

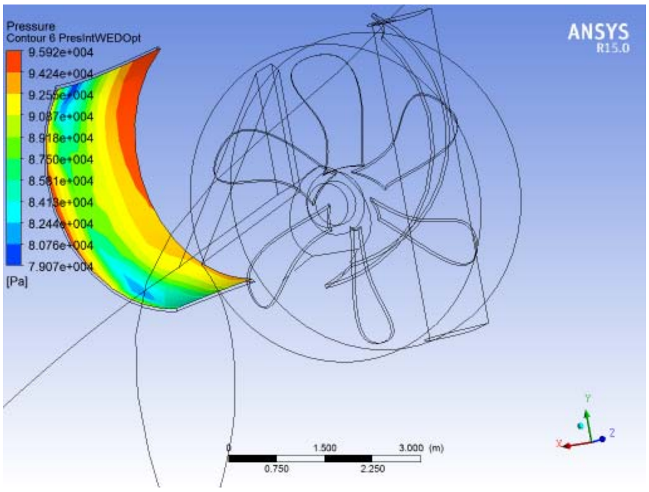

Figure 13. The model duct pressures

4 CONCLUSIONS

The wake equalizing duct (WED) is one of the most commonly used energy saving devices for improving the propulsion performance of a ship; and reducing the propeller-excited vibrations and viscous resistance forces.

In this paperwork a given geometry of a WED device is taken and via Design Optimization the geometry of the duct was refined so that better results are achieved with a smaller and more compact WED. In doing so, the Designer is assisted by numeric optimization methods to choose from only three final candidates instead of several thousands in order to provide the best fitted WED geometry for a given ship afterbody.

REFERENCES

H. Schneekluth and V. Bertram, *Ship Design for Efficiency and Economy*, Butterworth-Heinemann, Oxford, 1998.

E. Korkut, *A case study for the effect of a flow improvement device (a partial wake equalizing duct) on ship powering characteristics*, Ocean Engineering, 2005.

F. Celik, *A numerical study for effectiveness of a wake equalizing duct*, in Ocean Engineering, 2007.

J. Friesch, C. Johannsen, *Propulsion optimization tests at high Reynolds numbers*, SNAME Transactions 102, 1994, pp. 1–21.

H.J. Heinke, K. Hellwig-Rieck, *Investigation of Scale Effects on Ships with a Wake Equilizing Duct or with Vortex Generator Fins*", Second International Symposium on Marine Propulsors, Hamburg, Germany, 2011

J. Carlton, *Marine Propellers and Propulsion*, Butterworth-Heinemann, Oxford, 2007.

ITTC 1999, *Final report of the specialist committee on unconventional propulsors*, 22nd International Towing Tank Conference, Seoul, Korea and Sanghai, China, 1999.

A.Y. Odabasi, P.A. Fitzsimmons, *Alternative methods for wake quality assessment*, Int. Shipbuilding Prog., 25(282), 1978.

E. Huse, *Effect of Afterbody Forms and Afterbody Fins on the Wake Distribution of Single Screw Ship*", NSFI Report No.R31-74,1974