"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XX – 2017 – Issue 1
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

ASPECTS TO THE STUDY OF FLOW DYNAMICS AND CAVITATIONS ON DUCT – TYPE SHIP DEVICES

Cătălin FAITAR[1]
Iordan NOVAC[2]
[1] Eng., Constantza Maritime University
[2] Associate Professor Eng., Constantza Maritime University

***Abstract**: Contributions to the study of flow dynamics and cavitations on duct-type ship devices are engineering things studied today. Not only elements of the fluid mechanics, such as the fluid pressure, density, viscosity, speed and the hydrostatic equations are used for this study, but also the influence of the Reynolds and Mach numbers influencing the flow fluid are being studied. Some present studies regarding the current stage of the finite volumes method and nonlinear optimisation, including the mathematical foundations of numerical analysis of fluid dynamics best help for this. The CFX numerical analysis of models with and without wet duct and cavitation is being analysed by making a comparative study that includes the schemes mentioned above for speed, pressure and current lines.*

***Keywords:**Geometric modelling, CFD –Computational Fluid Dynamics, Elements of fluids mechanics, Ship propellers hydrodynamics, Cavitation, Numerical optimisation of WED geometry*

1. Elements for efficient ship propeller

Propulsion plant for surface vessels and submerged vehicles have taken after the standard impetus plant for a long time. Propellers must be composed in an approach to decrease noise and vibrations and consequently cavitation to the most minimal conceivable level accomplish propeller efficiency. Modifications made in the fundamental propeller geometries do not change the way we determine and analyse propeller performance.

1.1. Diameter of propeller

It was reasoned that the speed of the shaft and the propeller diameter are firmly related. A low shaft speed (for known diameter) is extremely gainful from an effectiveness perspective. In any case, it prompts to high shaft torque therefore large shafts and gearboxes. Thusly, while outlining the propeller an adjust must be found to guarantee its execution. For the most part, propulsive efficiency can be expanded by introducing propellers with large diameters. Be that as it may, the propeller diameter of the vessel is constrained by the draught of the vessel. Advancement of propeller configuration is done to meet the adjust.

1.2.Speed of propeller

Choice of RPM of the vessel assumes a critical part in the propeller outline. The rotational speed decided for the vessel must be unique must be different from the resonant frequencies of the shaft, hull and other propulsion machinery. It is often seen that low RPM design increases the propulsive efficiency by 10 to 15 percent.

1.3. Blades number of propeller

The number of blades picked affects the level of precarious powers following up on them. Considering the efficiency perspective, optimum open water efficiency increases with increase in the number of blades.

1.4. Hydrodynamic outline of blade

Blade outline assumes a vital part in propulsive effectiveness. Research and trials utilising propellers with different sharp edge regions have demonstrated that productivity increments by diminishing theblade area. This is because the frictional drag increments with expanded sharp edge zone. In any case, it ought to be remembered that the quality of the propeller can't be bargained by diminishing the area to a substantial degree.

1.5. Angle of attack

The plan of angle of attack of the propeller and its comparing chamber relies on upon the outline lift which must be dictated by the maritime planner. If a larger angle of attack is chosen then the section designed would be less powerless to weight side cavitation and more defenceeless to suction side cavitation. The invert additionally remains constant if the angle of attack is diminished.

1.6. Pitch/Diameter ratio

To accomplish the best propulsive efficiency for a given propeller diameter, an ideal Pitch/Diameter proportion is to be discovered, which again relates to a specific design rate of revolution. Aside from the traditional changes in the parameters of the propeller in order to achieve higher efficiency, there are sure outside parts which help in enhancing the flow around the propeller, henceforth, enhancing propulsive efficiency.

1.7. Stern tunnels

These devices help in reducing the wake peak effect for vessels having V-shaped sterns, thus reducing the effect of vibration.

1.8. Schneekluth ducts

This kind of devices divert the flow to the upper bit of the propeller circle in this way evening out the impact of the wake and enhancing body efficiency. These are additionally fit for accelerating the flowby method for the aerofoil

DOI: 10.21279/1454-864X-17-I1-035

shape cross area. This is finished by making a low-pressure surface before the channel which is accomplished through its design**.**

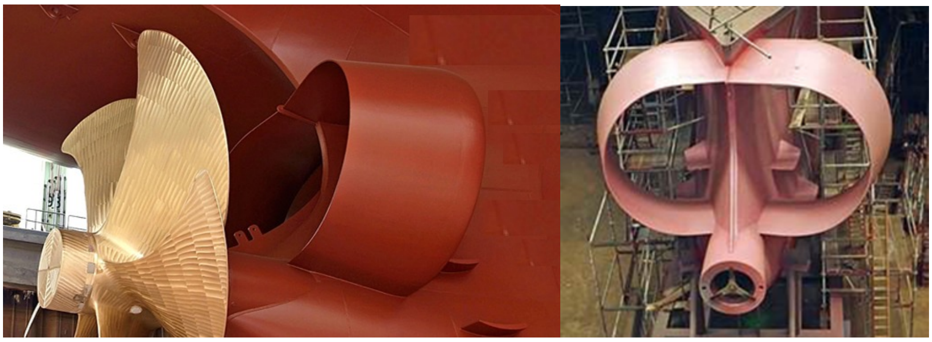


Figure 1. Schneekluth ducts

2. Various kind of duct design ideas

There are different kind of energy saving devices for vessels. Thedevice before the propeller is responding with the last phases of the development of the limit layer around the stern of the vessel, and the devices at the propeller station and behind propeller are working inside both the body wake field and the slipstream of the propeller, yet they are all endeavoring to recoup the energy loss so that expansion propulsive efficiency. Numerous energy-saving devices have a nearby connection vessel with the propeller, so firstly we need to think the effectiveness and loss of propeller, what characteristic misfortunes are happen and how to recuperate them if conceivable. Wake equalising ducts were developed by Schneekluth in the year of 1986. A few thoughts are conceived and surveyed quickly beneath for more efficient Duct design, and connected to the design.

2.1.Eccentric propeller duct

This thought is to put a symmetrical pipe not parallel to the pole line or separated from shaft line. This thought can be valuable to control the flow into specific ranges of the propeller circle. Furthermore, it might be relied upon to impact hub wake top and to increment powerful wake an incentive by an increment of effective wake part like asymmetric aft hull form.

2.2.Asymmetric profile angle of upper and lower duct

This thought can be viewed as valuable to control the flow into a favoured heading and to lessen the resistance caused by a lower duct. Another valuable impact may quicken the hub flownon-uniformly so that outflow could become homogeneous. This last capacity could be useful to impact the wake peak in the 12'o clock position positively.

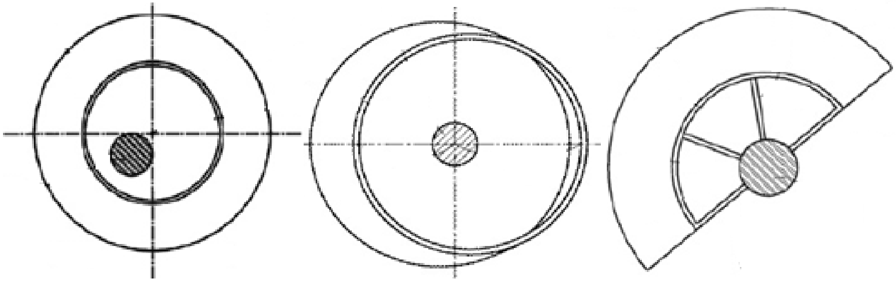


Figure 2. Eccentric duct, tilting duct, inclined half circular duct

2.3.Tilting duct

The tilting duct is near the deviated duct. Channel shape is symmetric, however, gives an angle of inclination for installation. It could however likely less demanding been made.

2.4.Inclined half circular duct

If there should be an occurrence of a correct turning propeller, port side up and coming flow could be recoursed against propeller rotational direction and star-board side up and coming stream not to be disturbed. So this thought might guarantee to expand the propulsive effectiveness.

3.Mathematical modelling of fluid flow alongside the hull, with conventional duct and with Schneekluth duct

3.1.Model description

Mathematical modelling of fluid flow alongside the ship structure, first, with conventional duct and, second, with Schneekluth duct, in the presence or absence of cavitation includes numerical

DOI: 10.21279/1454-864X-17-I1-035

simulation, with cavitation modelling, of the fluid flow alongside the hull. Once the numerical model has been defined, both as far as CAD-geometrical generation of a port vessel stern and border conditions of the 'negative' of CAD geometry are concerned, which represent, in fact, the seawater flow, further on the study goes to the CFX numerical analysis of models, with and without WED duct, in the absences of cavitation.

3.2.Model vessel with conventional duct

The in-house vessel and a stern duct are selected as the benchmark geometry for the present paper. Table 1shows the major dimensions of the hull and propeller in full scale. In Table 1, the following notations are used; Lpp is the vessel length between perpendiculars, B is the vessel width, d is the vessel draught, C_B is the block coefficient, D_p is the propeller diameter, H/D_p is the pitch ratio of the propeller, a_E is the expanded area ratio of the propeller and Z is the number of blades in propeller. Figure 3 includes the duct design parameters: DT.E. is the trailing edge diameter of the duct, L_d is the chord length of the wing section of the duct and β is the opening angle of the duct. The duct is configured in such a way that its vertical and lateral centres are identical to shaft centerline of the ship structure. See thatthe strut of the duct is not reproduced in the present geometry.

Table 1. Dimensions of hull and propeller

Hull		Propeller	
L_{pp}[m]	217	D_p [m]	7.1
B [m]	32.26	H/D_p	0.8
d [m]	12.2	a_E	0.5
C_B	0.84	Z	4

3.3.Model simulation with conventional duct

The model was simulated with CFD technique. All the computational outcomes demonstrate fantastic consent to the test information confirm by the way that the correlation blunders in self-impetus coefficients are under 1%, and in this manner, the central accuracy of the present CFD technique is guaranteed.

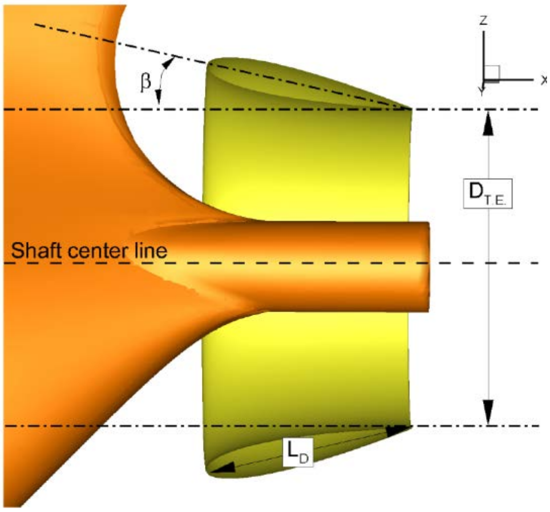


Figure 3. Duct design parameters

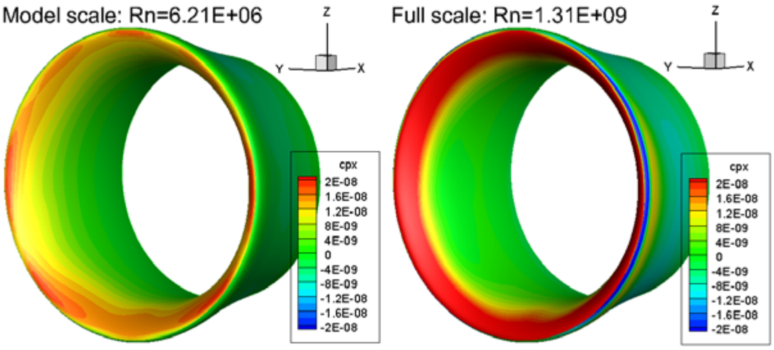


Figure 4. 3D perspective of the axial component of pressure resistance from resistance simulation

DOI: 10.21279/1454-864X-17-I1-035

According to Figure 4 the expansion of the pressure resistance in full-scale than in model scale is started from the internal driving edge of the duct. These outcomes infer that the expansionin resistance because of the channel gets to be distinctly clear in full-scale and is covered at the season of model scale experiment/simulation.

Figure 5 introduces the wake proportion (E_i) evaluated by different criteria, e.g. three kinds of empirical methods and full scale CFD. The general pattern in 1-wt is similar between model and full scale, i.e. the duct improves 1-wt among all hulls by 2% to 3% in contrast with bare hulls.

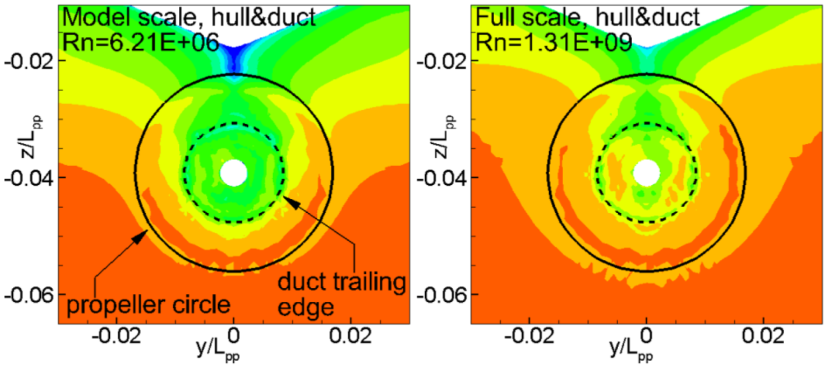
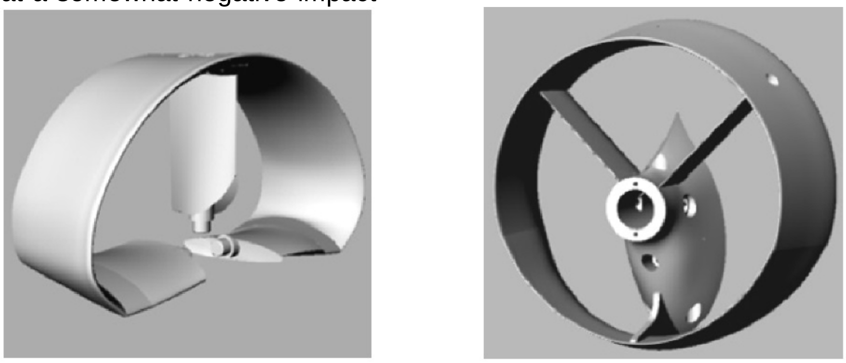


Figure 5. Wake ratio for three different hulls, without (left) and with (right) duct

3.4. Model simulation with Schneekluth duct

Numerical simulations of an ordinary vessel and those of a WED duct vessel are compared, in completely a similar flow conditions and meeting conditions for solutions. Every simulation benefits from direct conclusions, beginning with analysis of distribution of pressures in various default control plans and finishing with an analysis of Oz axle component of fluid-structure interaction force, which demonstrates an impetus change for propulsion improvement of 4,64 %, consistent with less "eager" analyses on the examinations available. We analyze CFX numerical investigation of models with and without WED duct, in the presence of cavitation, which also brings into the review the impact of cavitation. It has been found that a somewhat negative impact on all parts of propeller action, by the by even the advantage coming about because of WED fitting reductions from 4,64 % to 3,19%, reliable this time with the most critical investigations available. Another hypothetically critical result is that the WED duct has no impact at all on propeller cavitation, a result which, while it might, truth be told, seem overwhelming, illuminates the debate on the presence of little WED impacts on cavitations. The examinations displayed in the review have basically not uncovered this angle without question. The drawn conclusions are really relevant to the investigation of efficiency, the setting of these devices, but also of the impact the devices have on cavitations, and such impact is being described in this paper.



(a) (b)

Figure 6. Unconventional half circular duct & conventional circular duct

To start with the unconventional half circular duct that consists of 3 basic parts, in particular, a large upper semi-circle part, an almost flat part and centre fin part(Figure 6a).In complete it covers the upper half round range of the propeller plane, being associated with the centre point with its flat

DOI: 10.21279/1454-864X-17-I1-035

parts at port and starboard side. Particularly the sharp edge between wing part and duct ring was made round to smooth move and half circular duct was designed for the axial flow to be homogenised. The objectives for this duct design are a compensation of an upper/lower-asymmetry the uncovered structure wake at the propeller disk and making a preswirl flow, in advance, against propeller rotational course. The asymmetry in wake dissemination can barely be repaid by the propeller outline as it were. The single blade thrust will be expanded at the 12'o clock position and the upper propeller disk will thus be highly loaded. Typically, it would be more effective to make a propeller disk loading more uniformed.The CFD estimations were performed to discover the impacts of wings and duct. CFD results show that the stagnation pressure line almost coincides with the main edge (Figure 7).

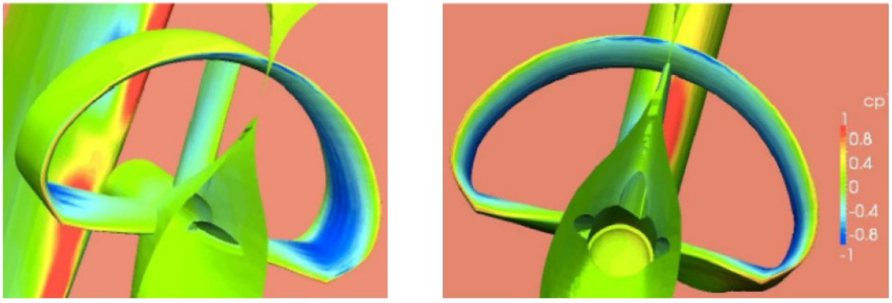


Figure 7. Analysis for unconventional half circular duct and rudder.

Figure 8 gives shapes for the total axial speed at the duct outlet plane. Here the total axial component is characterised as the axial component obtained by enacted body force propeller design. Inside the duct the increasing speed impact is noticeable but the level is not fundamentally reduced at the outlet, which means the inflow at the entrance of duct goes through the duct to the trailing edge well without separation. This seemed, by all accounts, to be an urgent detail point for the best possible outline of duct sections. Moreover an integration of surface forces acting on the duct gave a support of the propeller thrust by around 1.3%.

Second, the conventional circular duct has been examined similarly. The objective of the conventional circular duct (Figure 6b) was to remunerate basically a solid non-consistency in the spiral bearing which is a trademark showing up by and large in the estimations of the uncovered body wake for full vessel. To arrive at accelerated flow at the duct outlet, the areas were cambered to the inside, which amount of camber was adjusted to give a reasonable accelerated flow at the duct outlet. Also, a channel was intended to evade assessed division impact.

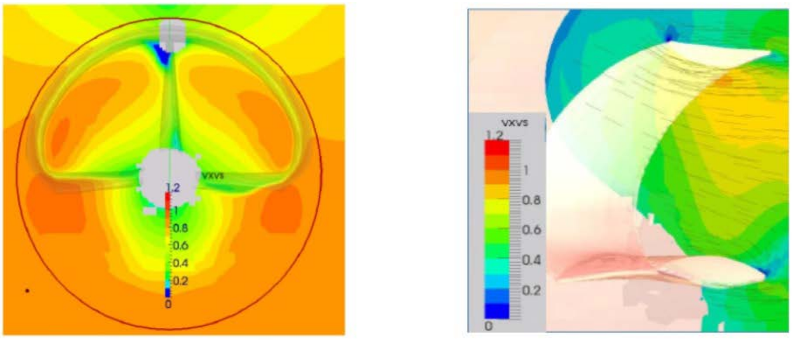


Figure 8. Results of axial flow by contour plot

Conclusions

Resistance and self-propulsion simulations are carried out in the model and full scale, and the computational outcomes uncover taking after material science. The self-propulsion simulations clearly present the trend of scale effect in1-t, 1-wt and resultant ηH. The trend of Δ in 1-t is opposite between model and full scale among three hulls. This is caused by the changing in the flow direction to the duct between model and full scale which yields difference in thrust originated from the duct. On the other hand the trend of Δ in 1-wt is similar between model and full scale among three hulls. Energy saving devices have been investigated

DOI: 10.21279/1454-864X-17-I1-035

meant to support the propeller performance, especially on vessels with large block coefficients. General choices of energy saving devices are ducts and stators in front of the propeller and rudder-fins and rudder bulb mounted downstream of the propeller. If there should arise an occurrence of various utilisation of the energy saving devices, compatibility investigation should be additionally carried out, because there does not remain potentially recoverable energy loss due to duplication of saving function as much as their own self-resistances.Measuring the recovery potential in terms of power reduction at given vessel speed our intention was to reach the 5% gain with the best duct arrangement. For reviewing the process of design and model testing, following basic steps were preceded on the way to the optimum solution:

- the unconventional half circular duct concept with horizontal connections to the propeller hub was selected from a variety of initial ideas;
- conventional circular pre-swirl duct was studied in parallel;
- cavitation tests and hull pressure measurement for the design propeller with and without the best duct configuration of the unconventional half circular duct were performed.

Bibliography

[1] Hyun-Joon Shin, Jong-Seung Lee, Kang-Hoon Lee, Myung-Ryun Han, Eui-Beom Hur, Sung-Chul Shin,*Numerical and experimental investigation of conventional and un-conventional preswirl duct for VLCC*, International Journal of Naval Architecture and Ocean Engineering. 2013. Sep, 5(3): 414-430 DOI: http://dx.doi.org/10.3744/JNAOE.2013.5.3.414

[2] Nobuaki Sakamoto, Yasutaka Kawanami, Munehiko Hinatsu, Shotaro Uto,*Viscous CFD Analysis of Stern Duct Installed on Panamax Bulk Carrier in Model and Full Scale*, National Maritime Research Institute, Conference Paper, May 2014

[3] George Adrian Laurențiu Martinaș,*Contributions to the study of flow dynamics and cavitation on duct-type ship devicesr*,Maritime University Of Constanta, 2015

[4] Hollenbach U., Reinholz O., *Hydrodynamic trends in optimizing propulsion, 2nd Int. Symp. Marine propulsors (smp'11)*, Hamburg, 2011

[5] Kawakita C., Kawabuchi M., *Prediction of full scale hydrodynamic performance for seiunmaru 1st. by CFD* , Jasnaoe Annual Spring Meeting, 2011

[6] http://www.kpubs.org/article/articleMain.kpubs?articleANo=E1JSE6_2013_v5n3_414&viewType=article

[7] http://www.marineinsight.com/naval-architecture/10-factors-considered-efficient-ship-propeller-design

DOI: 10.21279/1454-864X-17-I1-035