**MARITIME UNIVERSITY OF CONSTANTA**
**FACULTY OF NAVAL ELECTROMECHANICS**
**DEPARTMENT OF MECHANICAL ENGINEERING**

# DOCTORATE DISSERTATION

## CONTRIBUTIONS TO THE STUDY OF FLOW DYNAMICS AND CAVITATION ON DUCT-TYPE SHIP DEVICES

## - ABSTRACT -

**SCIENTIFIC SUPERVISOR**
**Prof. Univ. Dr. Ing. Nicolae BUZBUCHI**

**DOCTORAT CANDIDATE**
**Ing. George Adrian Laurențiu Martinaș**

**CONSTANŢA**

**2015**

# SUMMARY OF THE PhD THESIS

With a highly complex, interdisciplinary characteristic, the paper, as a whole, represents a benchmark in the study of flow dynamics and cavitation on duct-type ship equipment.

Since the reduction of fuel consumption over the last decades has represented a major strategy plan within the commercial maritime transport and ship operators industry, since fuel price on markets with small fluctuations has shown a rising trend, and there emerged numerous national and international legislative elements, forcing the industry to resort to various solutions to reduce emissions of SOx, NOx and $CO_2$, operators have put pressure on ship designers to develop project solutions in accordance with requirements of legislation.

In view of the above-mentioned terms, and with a desire for exploitation vessels to become more efficient, there have been developed simpler solutions for fitting specific devices towards the stern area of a ship. All these solutions and devices are known as Energy Saving Devices (ESD) or occasionally as retrofit technologies. Being classified as devices which reduce advancement resistance of a ship and devices which increase the efficiency of a ship propulsion system, the latter group can be split into devices that can be fitted in front of the propeller, devices that can be fitted right next to the propeller and devices which can be fitted behind the propeller.

Among these, there is one device, WED (Wake Equalising Duct), which makes the subject of the present study, which is expected to increase the efficiency of the propeller by 6-7 %. Although the study of a ship hydrodynamic performance can be split into three major categories: ’Advancement resistance and propulsion’, ’Behaviour at sea’, and ’Manoeuvrability’, efficiency analysis can be classified in three distinct approaches: **The empirical approach:** due to the high costs incurred for construction of models, this type of designing is considered, nowadays, out of date. **Experimental approach,** be it with scale models or samples at sea: despite the fact that procedures used to predict resistance at full scale using models are well accepted by the scientific community, in reality validation is difficult to put into practice and hard to attain. **Numerical approach** entails the use of CFD (Computational fluid dynamics).This way of designing has become, in modern times, essential for any design process. Using CFD models, numerical models can by analysed at real scale, for each part of interest of the hull: bow, stern or the whole ship, but they are really dependent on the software used, on the designers’ experience and on the computational power at hand. Numerical models, the denser finite volume networks they possess, the more chances they have to generate results accuracy, which increases in arithmetic progression, whereas with 3D models, the required computational power increases in geometric progression. Traditionally, if a new type of ship hull was developed, then the design flow had to be: designing-model execution-testing-redesigning-...-designing-model execution-final model testing, thus requiring an average of 10 models to be tested. In modern era, costs associated with such approach are prohibitory, that is why, making use of geometric generation of the hull on CAD systems, with the help of CFD, hundreds of options can be numerically tested, prior to the test involving scale model, which makes viable the shapes optimization process as well, as we shall see with one of the coming chapters of this study. As a result of the increase of the impact of numerical simulations from 1980 up to present, orders towards scale models testing pools have massively dropped, and have been redirected to testing leading to fine adjustment of CFD numerical tests.

This study, with its objectives, will follow the numerical approach by simulative presence of a WED duct in terms of flow dynamics and cavitation.

Keywords:

Geometric modelling, CFD –Computational Fluid Dynamics, Elements of fluids mechanics, Ship propellers hydrodynamics, Cavitation, Numerical optimization of WED geometry

The content of the doctorate dissertation is the following

In the fallowing lines the approached elements will be briefly presented.

The study consists of 8 chapters, figures, tables, equations and mathematical relationships, over 70 biographical titles and over 220 pages, statistical figures which reflect the magnitude of the study. The chapters are presented in logical sequence, its purpose being to facilitate comprehension both of the theoretical basis, and especially of the original aspect of the thesis.

In order to follow these objectives, the chapters are arranged in logical sequence, while the mathematical apparatus is presented in such manner to facilitate understanding of theoretical bases of the study. In such, the beginning chapter (Chapter 1), entitled ’Necessity and appropriateness of the proposed topic and objectives’ argues the actuality and the importance of the topic, and it is an introductory chapter defining the motivation for research as well as presenting a review of the current stage, which indicates an ample bibliographic study of the research in the field. A review of the controversial opinions is presented, a fact that turns the topic into a current one and highlights, at the same time, the importance of the thesis. As everybody knows, commercial naval fleet operators are constantly under the pressure of regulatory bodies to reduce fuel consumption, thus simple solutions for fitting specific devices onto the bow area of exploitation vessels have been developed, so that their efficiency, expressed in the above-mentioned terms, to be increased. All these solutions and devices are known as Energy Saving Devices (ESD), sometimes also known as retrofit technologies. The way they are promoted as efficiency increasing by the shipbuilders, for example when applied to WED duct (Wake Equalising Duct, which represents the subject of the present study), they are expected to contribute to the increase of propulsion improvement by 6-7 %, nevertheless any shipbuilder, when required to write down such figures, in contracts, will be reserved when making such guarantees. Often times, these figures are indicated in a general way, without defining their computational and measurement conditions, therefore operators' doubt about them can thus be understood. Furthermore, the size of the benefits in the area of 5-7 % is placed right in the uncertainty zone as far as measurements are concerned, while the large Reynolds numbers deter the use of models for testing purposes, as we will see in one of the chapters of this study.

As far as the objectives of the study are concerned, they are structured as follows: as main explicit objective, the impact of the presence of a WED duct in terms of flow dynamics and cavitation on propulsion of ships, including:

- Geometric modelling of a ship hull, the stern part of a port container with and without the WED duct, using Computer Aided Design Techniques (CAD),
- Numerical modelling of flow alongside the hull, with and without a WED duct, in the circumstances of cavitation absence – comparative study;
- Numerical modelling of flow alongside the hull, with and without WED duct, in circumstances of cavitation presence – comparative study;
- Optimization of WED duct geometry using advanced techniques such as ANSYS Design Explorer

'Elements of fluids mechanics' is the title of Chapter 2, which together with Chapter 3, entitled 'Current stage of finite volumes method and nonlinear optimization', offers an overview of the main theoretical aspects underpinning the study. This way, once the main concepts on fluid dynamics are determined, the influence of Reynolds and Mach numbers on determining fluid flow is explained. Further on, the mathematical foundations of numerical analysis of fluid dynamics (CFD –Computational Fluid Dynamics) can be found. There comes the description of the principles of Optimization, with Optimization methods, Sub problem Approximation and first order Optimization, and the end of the chapter presents the parametric design and DOE theory (Design of Experiments).

”Ship propellers hydrodynamics”, is the title of Chapter 4, where it gets addressed. Theoretical elements of ship propellers hydrodynamics and presentation of the rotating disk method are discussed in detail in the first part of this chapter. The discussion on cavitation on ship propellers is the final topic concluding this chapter, dedicated to the theoretical part.

Chapter 5 is entitled’ Mathematical modelling of fluid flow alongside the hull, with and without Schneekluth duct, in the presence or absence of cavitation” and it includes numerical simulation, with and without cavitation modelling, of the fluid flow alongside the hull, with and without Schneekluth duct and comparative study. Once the numerical model has been defined, both as far as CAD-geometrical generation of a port container 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 – comparative study. One by one, numerical simulations of a normal ship and those of a WED duct ship are compared, in absolutely the same flow conditions and convergence conditions for solutions. Each subchapter benefits from direct conclusions, starting with analysis of distribution of pressures in various default control schemes and ending with analysis of Oz axle component of fluid-structure interaction force, which shows a propulsion improvement of 4,64 %, consistent with less ’enthusiastic’ analyses on the market. The next subchapter discusses CFX numerical analysis of models with and without WED duct, this time in the presence of cavitation- comparative study, which additionally brings into the study the effect of cavitation. It has been found that it has a rather negative effect on all aspects of propeller activity, nevertheless even the benefit resulting from WED fitting decreases from 4,64 % to 3,19%, consistent this time with the most pessimistic analyses on the market. Another theoretically significant result is that the WED duct has no influence at all on propeller cavitation, result which, while it may, in fact, appear daunting, clarifies the dispute on existence of small WED influences on cavitations. The analyses presented in the study have simply not revealed this aspect beyond any doubt.

The drawn conclusions are really relevant for the study of efficiency, setting of these devices, but also of the impact the devices have on cavitations, and such impact is being described in this chapter.

Chapter 6, entitled “Calculation results of numerical optimization of WED geometry at the entrance into the propeller area”, based on the idea that, due to the geometrical diversity of hulls there isn’t a WED duct to universally match all types of ships, presents the optimization process necessary to opt for the most viable solution. The geometrical differences bring about different flow conditions in the stern area, which are also dependent on the specific cruise speed of a ship, so that each single WED duct needs to be 'tailored' so that it better matches the actual situation. A poorly designed WED duct not only does not improve the vessel performances, but it also actually prejudices them. The process through which the optimum geometry of the WED duct can be found is the computer assisted optimization process. As we know, a successful project is the result of often divergent 'negotiations' between various objectives and design parameters. In that sense, 'exploration' of a given project becomes necessary, and this cannot be conducted with only one single version of the project. Several 'design points' or versions of the project, which were obtained by variation of its parameters, are necessary (design or input data) in order to gather sufficient information to quantify the impact of each 'input' parameter (geometric, base material resistance parameters, loadings, supports, etc) on 'output parameters' (which can be equivalent tensions, fluid speed rates, pressures on specific area, etc). Linking project input varying parameters and output project performance quantifying parameters will be achieved by Experiment Design Method (DOE-Design Of Experiments) combined with response surface methodology. Firstly, based on the number of project input parameters, the design points will be established by way of CCD-Central Composite Design. This way, the design space will be populated with a sufficient number of design points, so that reliable response surfaces can be drawn through them. The design space is given by the maximum and minimum values each input parameter ranges between.

Chapter 7, 'Qualitative validation of models', mentions the impossibility to draw conclusions from WED testing of Froude scale models, due to the Reynolds numbers effects and separation levels on power lines. Scale models deduced results were compared to those obtained and presented by other researchers. In this context, the author examined the WED effects on a chemical tanker ship. The response surface graphics and the sensitivity charts, which show which way and how much the optimization parameters impact on the output parameters, in our case the pressure building up inside the WED duct and fluid speed through the entrance area into the propeller field, are the most important among the practical results of the study.

Research results conducted under the PhD thesis have been disseminated during scientific communication sessions, during the 'LNG MASTERPLAN' European project and on occasion of published articles including the results of the research, within various scientific papers, including BDI magazines.