CFD and FEM model of an underwater vehicle propeller

Tadeusz Chruściel, MSc
Ewelina Ciba, MSc
Julita Dopke, MSc
Gdańsk University of Technology

ABSTRACT

Within the framework of the project for design and optimization of the Remotely Operated Vehicle (ROV), research on its propulsion has been carried out. The entire project was supported by CFD and FEM calculations taking into account the characteristics of the underwater vehicle. One of the tasks was to optimize the semi-open duct for horizontal propellers, which provided propulsion and controllability in horizontal plane. In order to create a measurable model of this task it was necessary to analyze numerical methodology of propeller design, along with the structure of a propellers with nozzles and contra-rotating propellers. It was confronted with theoretical solutions which included running of the analyzed propeller near an underwater vehicle. Also preliminary qualitative analyses of a simplified system with contra-rotating propellers and a semi-open duct were carried out. The obtained results enabled to make a decision about the ROVs duct form. The rapid prototyping SLS (Selective Laser Sintering) method was used to fabricate a physical model of the propeller. As a consequence of this, it was necessary to verify the FEM model of the propeller, which based on the load obtained from the CFD model. The article contains characteristics of the examined ROV, a theoretical basis of propeller design for the analyzed cases, and the results of CFD and FEM simulations.

Keywords: ROV, ROV propeller, semi-open duct for horizontal propellers, rapid prototyping propeller

Introduction

Evolution in the design of underwater vehicles of ROV type requires exploration of new solutions oriented on improving the quality of the offered products. One of them is a semi-open duct of horizontal thrusters that has been applied in a prototype of multitasking underwater robot developed within the framework of a research project conducted by the Department of Underwater Technology, Faculty of Ocean Engineering and Ship Technology, Gdansk University of Technology. One of the main tasks of the robot is to explore the seabed for geological purposes and evaluate the suitability of a land for the wind farm construction. This task implies the need for a high operating speed and high manoeuvrability when operating on a geological object. Providing both of these features while ensuring planned functionality require

the support for the design process throughout its range by numerical modelling in both CFD and FEM fields. One of tasks which have been implemented was to optimize the shape of semi-open ducts of horizontal thrusters (fig. 1).

The analysis of such an extensive issue started by checking the level of convergence between the prepared numerical models and the theoretical calculations for two cases: a free propeller and a propeller in the nozzle. In the next step the numerical model was extended by the disorder coming from the drive motor. The results obtained from the numerical analysis were compared with those recorded on the real object.

This enabled to refine faster the numerical model extended by the variable disorder derived from the semi-open ducts and the interference between the propellers.
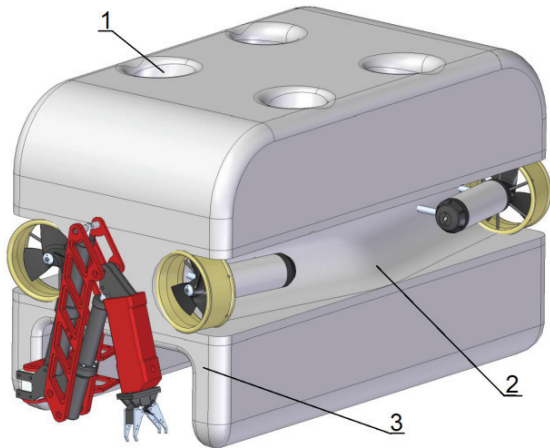
Fig. 1. ROV with semi-open duct for horizontal propellers
(1 - duct for vertical propeller; 2 - semi-open duct for horizontal
propeller; 3 - manipulating module)

The structure of the full model of the semi-open ducts for horizontal thrusters not only had to provide the information about the effectiveness of the solutions but also to make a basis for the analysis of the operational capacity of the ROV. Another aspect of the vehicle's propellers was the strength of the applied materials. Due to the fact that when designing the vehicle demonstrator a decision was made to use the technology of rapid manufacturing of prototype propellers, there was also a need to analyze their work using the FEM model. The loading force on the propeller was assessed using the pressure generated in the CFD analysis of the free propeller.

Theoretical overview of the issue

Due to the lack of publicly available theoretical solutions and guidelines for designing a system based on the semi-open ducts, a decision was made to build a numerical model that allows at least determining the quality of the proposed solution. At the beginning, the reference point of the thrust characteristics of the designed propeller working in the nozzle was adopted. These data were used to check the propeller in the numerical model in two basic operating states, i.e. as the free arrangement and in the nozzle. The characteristics of the propeller are discussed in Section "**Free propeller and nozzle**". Further steps of solving the problem included complementing the CFD model by disorders generated by the propeller and confronting it with the results of the tests performed on a real object. These considerations were supplemented by FEM model calculations, which are discussed in Section "**FEM model**".

Free propeller and nozzle

The design stage was based on the database of the functional components of a previously developed ROV bearing the name of Morświn. A group of the elements used to develop the numerical models and build the test bench in order to determine the suitability of design solutions,

included the propeller, the nozzle, and the drive motor. The nominal rotational speed of the propeller was 40 rps. The geometric parameters of the propeller which constituted the reference point are listed below, while the indicative geometric characteristics of the nozzle for this propeller are shown in Fig. 2

Geometric parameters of the propeller:

– propeller diameter: D = 0.22 m
– hub diameter: DH = 0.04 m
– number of wings: z = 3
– surface coefficient: AE/AO = 0.65
– pitch coefficient for 0.7R: P/D = 0.9845
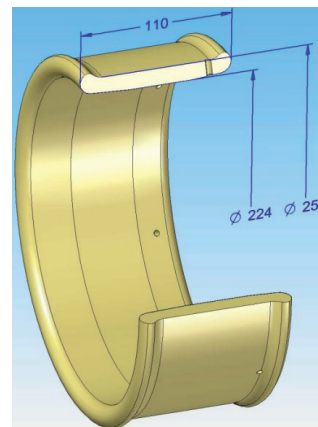– profiles type: NACA16

Fig. 2. The nozzle

According to the theory, the system being a combination of the propeller and the nozzle improves the thrust characteristics, as can be seen in Figs. 3 and 4.
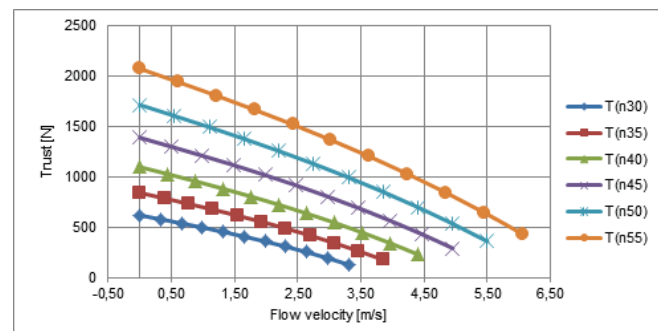

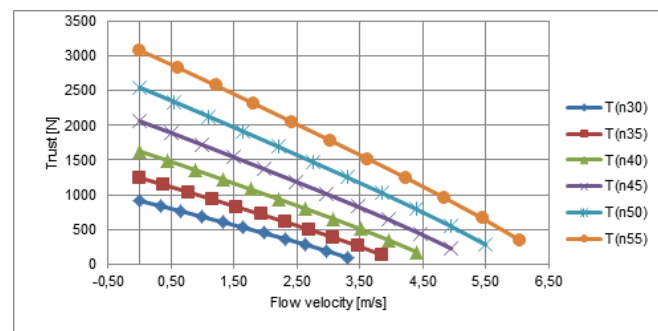*Fig. 3. Thrust of the free propeller*

Fig. 4. Thrust of the propeller with the nozzle

The above characteristics were determined during the design of the propeller for earlier project. In Section 3.1 they will be discussed in more detail with the reference to the results obtained from CFD calculations.

CFD model

While working on improving the flow model of the rotor, which in this case is the propeller of ROV, modelling making use of a frozen rotor rotary domain in quasi stationary systems has been applied. The use of this model made it possible to shorten the computation time, which was an important aspect in the context of time restrictions related to the implementation of the project. The expense for reducing the computing time demand was lower accuracy associated with model simplifications resulting from this method of modelling. The main disadvantage of this method of the calculation model included the lack of modelling the transition state, the transitions between the steady state and the turbulent state, and the state of merging of these two types of flows.

As already mentioned in the model with respect to the CFD, three models of varying degrees of sophistication were built. The domain for the first model calculations is shown in Fig. 5.
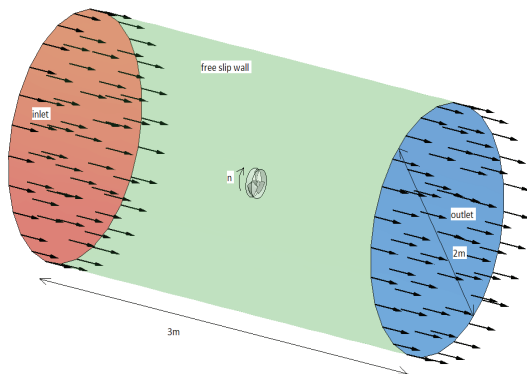

Fig. 5. Domain of the free propeller model

The propeller with a part of the rotating domain was placed in the middle of the span of the main domain. The results obtained using the above model are discussed in the initial part of Section 3.1.

The second model, whose results were compared with the solution of the theoretical model, was supplemented by a nozzle. This model uses the same size of the main domain, and the propeller together with the nozzle is placed in the middle of the span of the domain. In order to take into account the effect of the nozzle on the working surface of the propeller, the division between the main domain and the rotating domain is placed on the inner cylindrical surface of the nozzle. This is illustrated by Fig. 6, in which the division surface is highlighted in orange.

The third model, which was most complex in geometrical terms, was the model of the propeller in the nozzle complemented by a drive motor and the objects of the measuring devices mounted of a real bench.
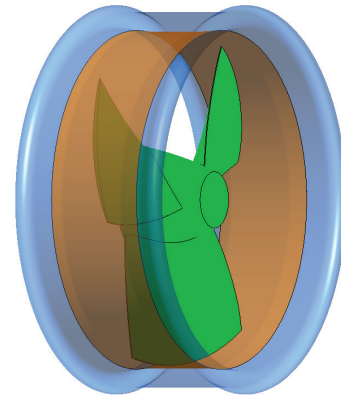

Fig. 6. Wall counter rotate velocity

The interface areas were introduced in this model in the same way as in the model based solely on the propeller and the nozzle. A detailed description of the method of measurement and the test bench is given in Section 2.4. The following is a sketch of the modelled system in a natural way simplified for the purposes of CFD.
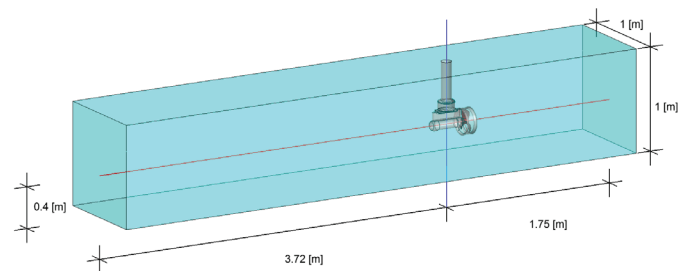

Fig. 7. Geometric parameters of CFD model No. 3

Dimensions of the domain cross section and the location of the thruster resulted from the dimensions of the circulating water channel which was available for the tests.

FEM model

The reason to create the FEM model for propeller loaded with pressure imported from the CFD model was, above all, a desire to see how large the strength reserve made by rapid prototyping item is. The rapid prototyping method that has been used to produce designed propellers was Selective Laser Sintering (SLS). It involves successive sintering of layers of powdered base metal. In the case of the tested propeller it was Alumide whose parameters are included in Table 1.

Table 1. Parameters of alumide

Parameter	Value
Density	1360 kg/m³
Young module	3800 MPa
Poisson number	0.4
Tensile strength	48 MPa

MOST WIEDZY   Downloaded from mostwiedzy.pl

The SLS method allows to produce a fully functional device from many different materials. In the literature there is no clear information concerning the real strength level of products made based on this kind of rapid prototyping. The only information that can be found refers to on simple material parameters of the base material. However, observing the structure of the material made using this technology a question arises what is the safe load that this propeller can carry and what is the level of internal stress in the structure. Then, assuming a safety factor of 2 we can decide, in the engineering way, whether the designed item can safely carry the load.

The model was built based on the load taken from the quasi static CFD model and complemented by the hydrostatic pressure acting on the object at a depth of 200 m. The dynamic load was assumed for the propeller rotating with the rotational speed of 55 rps in the flow with the linear speed of 3 m/s. In these conditions, the thrust force generated by the propeller was approximately equal to 1368 N. The type of the analysis was static analysis making use of the solid model and solid elements of solid187 type (10-node element with three degrees of freedom at each node UZ, UY, UZ). The main mesh with parts of 1.5 mm in size was locally concentrated to 0.5 mm within the passageway of the propeller blades to the hub (Fig. 8).
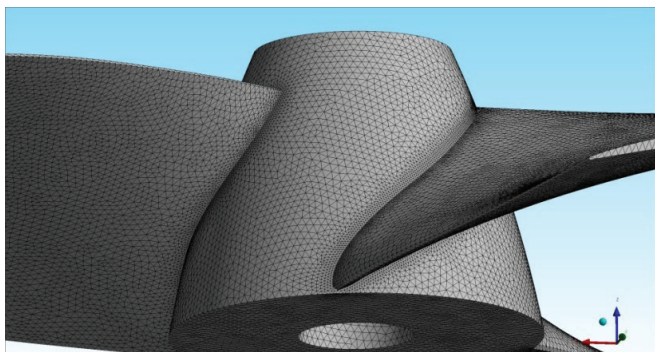

Fig. 8. Model mesh with local concentrations

Experimental model

The experimental model consisted of a drive unit equipped with a propeller in a nozzle, a dynamometer, and the circulating water channel in which the test was performed for flow rates of 0.0; 0.2; 0.5; 0.7 and 1.0 m/s. The tests were performed for work angles at 0; 22.5 and 45 deg, at two stages of propeller operation, i.e. the nominal operating condition (right rotation - push system) and the backward operating condition (left rotation - pull system). In the push system the water first passed through the drive motor, and then the disrupted flow reached the propeller. In the pulling system, the undisturbed stream of water first reached the propeller and then encountered the drive motor. The measurement system used in this test is shown in Fig. 9, while its orientation relative to the flow direction is given in Fig. 10.

Fig. 9. Experimental model

Fig. 10. Experimental model orientation

In order to minimize the impact of the test structure on the measured magnitude of the thrust force generated by the testing setup, the x-axis was bound with the drive motor, which allowed direct measurement of the thrust forces in tests with angles with respect to the axis of movement without the need to set the value of the components. Another procedure to remove forces generated at the test bench from the value measured for different flow velocities was calibration of the dynamometer each time the flow speed was stabilized. The calibration was performed before starting the motor, which allowed directly to record the change of the thrust as a function of propeller speed rotation.

Analysis of solutions

The main purpose of the construction of CFD models was to gain knowledge on how to model the discussed issue, and to confront the simulation results with the theoretical and experimental results. The applied computational method was verified by comparing with theoretical characteristics of the Series B-Wageningen. The level of coincidence of the compared values ranged from 1 to 5 %, and the variation of the characteristics of the determined parameters coincide with the theoretical values.

The promising results of the analysis gave rise to the application of this method for modelling the phenomenon of the real object. The following sections present the obtained results in relation to theoretical calculations and modelling studies.

Solution of theoretical, CFD, and experimental models

The discrepancy between the results of the theoretical calculations and numerical simulation for the free propeller did not exceed 5 %. Figure 11 shows sample comparison of the thrust generated on the propeller.
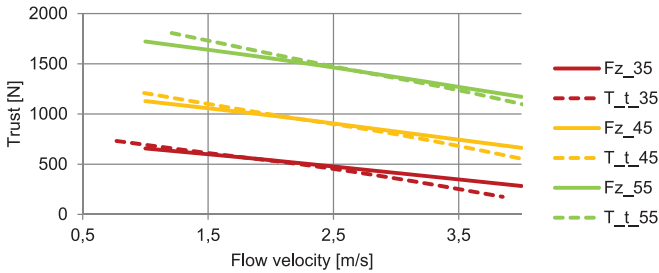

Fig. 11. Trust of the free propeller: T_t - theoretical solution; Fz - numerical solution

The above presented convergence level of the free propeller model was considered sufficient in the context of the task at hand. In the next step the CFD model was supplemented by the effect of the nozzle. Extensive simulation calculations allowed to obtain results revealing similar convergence to the free propeller model.
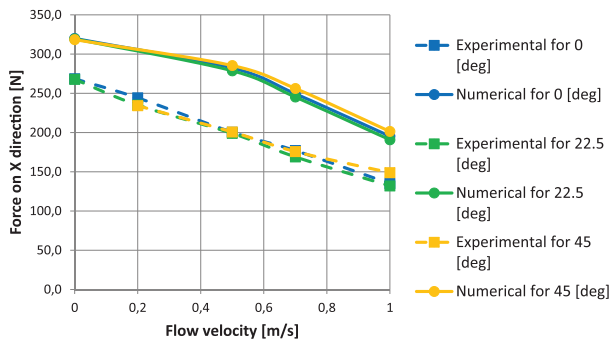

Fig. 12. Push configuration for 25 [rps]

At the same time, laboratory tests were performed for the configuration shown in section 2.4, along with simulations corresponding to flow conditions in the circulating water channel. The results of sample numerical solutions and experimental studies are summarised below in Figs. 12 and 13. They reveal that the method needs further development in order to obtain proper model of disturbances coming from the structure. In addition, the analysis of images of pressure fields behind the CFD model of the propeller has made it possible to observe swirls flowing down from propeller blades, what coincides with relevant information in the literature and testifies to the correctness of operation of the method.
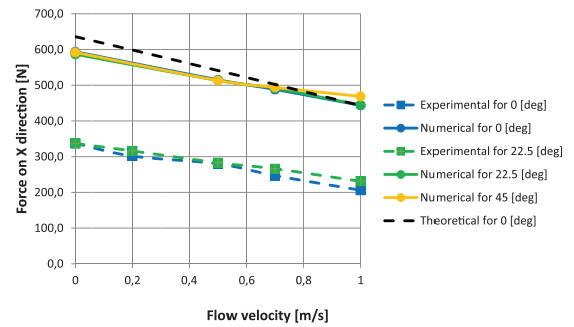

Fig. 13. Pull configuration for 25 [rps]

Solution of FEM model

The load imported from the CFD model for the push side and the pull side of the propeller blade does not exceed 4.5e5 Pa. The distribution of the load on the respective sides of the propeller's blades is given in Fig. 14.
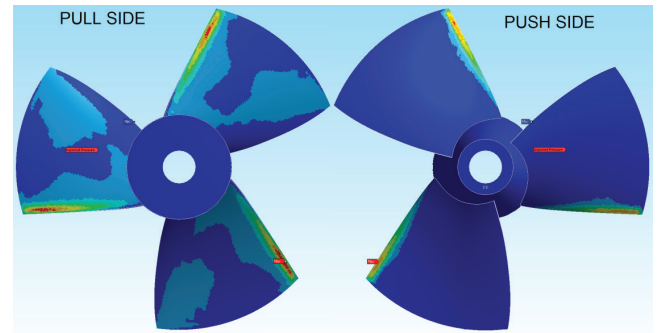

Fig. 14. Load from CFD model

The defined load supplemented by the hydrostatic pressure equal to 200 m H2O (this value reflects the maximum depth of operation of the vehicle) resulted in the maximum global strain equal to 0.0095 m.

As expected, the most strenuous region of the propeller proved to be the central parts of the propeller blades at the area of connection with the hub, which is presented in a picture of stresses reduced by the von-Mises hypothesis (Fig. 15). The values of these stresses ranged from 2.8e4 Pa to 8e7 Pa.
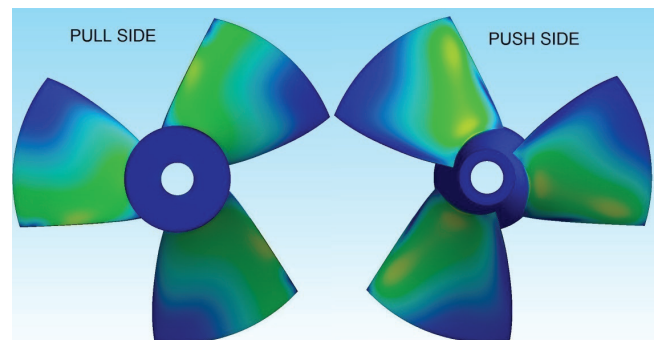

Fig. 15. Equivalent (von-Mises) stress

Conclusions

Referring to the results obtained from the FEM model and the assumed safety factor of 1.5, it was found that propeller made of alumide using the method of rapid prototyping (SLS) can transfer safely the dynamic load generated during its operation. In addition, a small deformation of the material should not significantly affect the efficiency of the propeller and change its hydrodynamic parameters.

In order to verify the performance of products manufactured by a rapid prototyping method, measures have been taken to develop a computational method based on FEM for structures made in the above process. Ultimately, this method is going to be verified in further experimental strength tests.

A conclusion from the experimental studies on the task in question performed from the point of view of hydrodynamics phenomena is that the proposed method of calculation needs further development, primarily in the field of modelling the interference of the nozzle and the propeller. However, the presently obtained results are satisfactory considering that further stages of the project will include assessing the ROV design solution, in which applying a semi-open ducts for horizontal thrusters will be based on the analysis of qualitative rather than quantitative analysis.

Acknowledgements

References

1. Bielewicz E., *Wytrzymałość materiałów*, Wydawnictwo Politechniki Gdańskiej, Gdańsk 1992.
2. Bielski J., *Inżynierskie zastosowania sytemu MES*, Wydawnictwo Politechniki Krakowskiej, Kraków 2013
3. Bielski J., *Wprowadzenie do inżynierskich zastosowań MES*, Wydawnictwo Politechniki Krakowskiej, Kraków 2010
4. Jarzyna H., Koronowicz T., Szantyr J., *Design of Marine Propellers, Selcted Problems*, PAN, Wrocław, Warszawa, Kraków 1996
5. Kobyliński L., *Śruby okrętowe*, Wydawnictwo Komunikacyjne, Warszawa 1955
6. Timoshenko S., Goodier J. N., *Teoria sprężystości*, Arkady, Warszawa 1962
7. Zagrajek T., Krzesiński G., Marek P., *Metoda elementów skończonych w mechanice konstrukcji*, Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa 2006
8. CFX-Solver Theory Guide for Ansys 14.5

CONTACT WITH THE AUTHORS

Tadeusz Chruściel, MSc
email: tadeusz.chrusciel@pg.gda.pl

Ewelina Ciba, MSc
email: eweciba@pg.gda.pl

Julita Dopke, MSc
email: dopke.julita@gmail.com

Gdańsk University of Technology
11/12 Narutowicza Street
80-233 Gdańsk,
Poland