

# Jan Kulczyk

Wroclaw University of Technology, Institute of Machines Design and Operation, Poland

**Abstract:** During the design process of the ship, a lot of attention is given to tune-up parameters of the propeller, which is one of the most important part of the propulsion system. Designing new generation, more effective and efficient propellers require knowledge of hydrodynamic phenomena occurring in the propeller area. For this reason, a lot of time and funds is spent on the propeller research and development. Usually these are experimental tests in cavitation tunnels and towing tanks. However, more and more research is carried out by CFD simulation. OpenFOAM software is open source CFD package, which becomes popular alternative to commercial codes. This paper attempts to answer the question: if OpenFOAM is suitable for such specific applications like accurate simulation of flow around the propeller? The paper presents the results of a CFD simulation of marine propeller created with OpenFOAM software. The obtained results were compared with the of the commercial CFD codes simulations and the experimental research.

Key words: Ship, CFD, OpenFOAM, simulations.

# 1. Introduction

Designing propulsion system for new ship is responsible and important task. To do this properly it necessary to obtain knowledge about propellers characteristics. This is why so many different computational and experimental methods are now available. The experimental methods seem to be the most reliable however they require a high technology facility. Now a days they are being displaced by computer simulation methods. The research has confirmed [1] that CFD calculations can be successfully applied to simulate complex flow (with separations, reattachments, huge pressure gradient, etc.) around rotating propeller. Now on market there are couple commercial CFD solvers (codes), designed for calculating complex flows, including propellers slip stream. Beside huge costs, their main impediment is closed code, which prevents advance user introducing modifications into program. In response to the need of fully customizable CFD solver, there have been started open-source project called OpenFOAM. It was not designed for solving the problems of ship hydrodynamics, however it could be successfully used to determine propeller open water characteristic. What will be shown in next sections.

# 2. Object of Study

In research there was used a 5-blade propeller from "Potsdam propeller Test Case" program [2]. Fig. 1 shows the propeller geometry. Main reason of choosing this propeller was availability of experimental results. Main parameters of the construction are shown in Table 1. Calculations were performed for constant propeller revolution n=10 [rps] and variable inflow velocity  $V_A = 0.4-4$  [m/s].



Fig. 1 Propeller VP1304 - Potsdam propeller Test Case.

**Corresponding author:** Jan Kulczyk, research fields: maritime transport.

-	-		
Parameter name	Symbol	Value	Unit
Diameter	D	0.250	[m]
Pitch ratio (r/R=0.7)	P <sub>0.7</sub> /D	1.635	[-]
Aspect ratio	$A_E / A_0$	0.77896	[-]
Hub ratio	$d_{H}/D$	0.300	[-]
Chord length (r/R=0.7)	<b>C</b> 0,7	0.10417	[m]
Number of blades	Z	5	[-]

Table 1 Propeller main parameters.

# 3. CFD Models

#### 3.1 Computational Domain

The Propeller geometry was divided into 5 identical regions along symmetry axis, with respect to propeller's periodicity. Thus, only one blade was modelled. Cyclic boundary walls were used to simulate periodicity of the flow. Outer boundary of computational domain was shaped in cylinder-like form. The main dimensions of domain were adjusted to propeller diameter (D) and they were defined as following: 1.5 D in upstream direction, 3.5 D in downstream direction, outer surface diameter - 2.5D. The geometry of the computational domain is presented in Figs. 2 and 3.

Propeller geometry was simplified by closing the

gaps between blades and a hub and between hub and shaft. This simplification have negligible influence on propeller open water characteristic. Both simplifications are shown in Fig. 4. The computational domains were discretized in program ICEM-CFD. Unstructured tetrahedral elements were chosen as a numerical mesh type. Advance size function was used to guarantee proper element size on high curvature blade surfaces. Geometry periodicity was taken into consideration during mesh generation process.

The OpenFOAM CFD toolbox contains its own grid generators. Authors of paper have not successfully built propellers mesh with provided tools. The main problem was unacceptable mesh quality. Mesh of the whole propeller (without periodicity), generated with OpenFOAM tools, was presented in Ref. [3].

The toolbox provides as well pack of numerical grid converters from external to OpenFOAM format. Program "fluent3DMeshToFoam" was used to import generated mesh into OpenFOAM environment.

Result dependence from number of discrete elements were tested on two different meshes. Coarse meshes were about 900 thousand elements and detailed mesh



Fig. 3 Computational domain - blade view.

Fig. 4 Mesh on blade and hub. Closed gaps between: shaft and propeller, shaft and hub.

were about 2 millions of elements. Both of the meshes were generated with caution to size of first layer elements. Non dimensional height of the first layer elements was set up in recommended range y+ = 30 -300 [4]. Numerical simulation of both size meshes gave similar quality of results. No significant differences were noted.

## 3.2 Numerical Methods

Simulations were calculated with CFD toolbox OpenFOAM in version 2.1.1. Reference values were obtained from Ansys Fluent 13.0 system. The Ansys Fluent was validated many times and it was proven to be reliable tool in ship hydrodynamics applications [5]. In both calculation systems it was used the same numerical meshes.

3.2.1 Ansys Fluent

In Ansys Fluent system it was used pressure based, steady state calculation model. The propeller motion was simulated with rotating reference frame method. It was used SIMPLE solution algorithm and second order gradient schemes. Flow turbulence was calculated with "k-epsilon realizable" model. All parameters and numerical model details can be found in (Ansys 2010). List of defined boundary conditions used in Ansys Fluent is shown in Table 2.

### 3.2.2 OpenFOAM

In OpenFOAM CFD toolbox there was used two

 Table 3 OpenFOAM boundary condition types.

solution methods. Most of the calculation were made with "MRFSimple" solver. This is steady state, incompressible, multi reference frame solver, based on SIMPLE algorithm. Additionally to validate numerical simulation, for advance coefficient J=0.64, flow was also calculated with "PimpleDyMFoam". This is much more complicated incompressible, transient solver, based on PISO-SIMPLE merged algorithm. Additionally, this solver can operate with dynamic, moving meshes.

Flow turbulence was calculated with "k-omega SST" model.

It was used "linear" gradient schemes, and limited second order divergence schemes. Laplacian terms were treated with second order conservative schemes.

Computations periodicity was simulated with "Arbitrary Mesh Interface (AMI)", which is numerical method designed to operate with non-conformal patches. All detailed information about model and numerical methods can be found in official documentation [6, 7]. Boundary conditions defined in this case were listed in Table 3.

Table 2 Boundary conditions type	es.
----------------------------------	-----

Patch name	Boundary condition	
INLET	velocity inlet	
OUTLET	outflow	
BLADE+HUB	wall, rotating with reference frame	
OUTER	slip wall	
CYCLIC	periodic	

Patch name	U	р	nut	k	omega
INLET	fixed	zero	calculated	Fixe	fixed
	Value	Gradient		Value	Value
OUT-	inlet	fixed	calculated	inlet	inlet
LET	Outlet	Value		Outlet	Outlet
BLADE, HUB	fixed Value	zero Gradient	nutk Wall	kqR Wall	omega Wall
neb	Value		Function	Function	Function
OUTER	slip	slip	slip	slip	slip
CYCLIC	cyclic AMI	cyclic AMI	cyclic AMI	cyclic AMI	cyclic AMI

## 4. Results

The "MRFSimple" solver from OpenFOAM CFD toolbox appeared to be reliable tool in calculation flow around rotating propeller. The solver was stable. Forces and moments stabilized after about 3000 iterations, which is comparable to commercial computational systems. Exemplary stabilization process of moment acting on a

blade is shown in Fig. 5. Calculation residuals decreased to acceptable level – Fig. 6.

The "PimpleDyMFoam" solver generated similar results. Difference in values of forces acting on blade, calculated with "MRFSimple" and "PimpleDyMFoam" was about 2%. Only the time of calculation in case of transient solver was much more longer (more than 10 times).



Fig. 5 Stabilization of torque acting on a blade during solution process, J=0.64.



Fig. 6 Calculation residuals history during solution process.

Propeller thrust coefficient characteristics is shown in Fig. 7. Agreement between simulation and experiment results is very good. The average relative difference in results is 5%. However for large values of advance coefficients (J>1,5), when the thrust force is relatively small, the difference increase up to 58%. But this fact has no significant impact on shape of whole KT(J) characteristic. On figure 7 there are also presented results from Ansys Fluent system. Their agreement to experimental result is very good as well.

The torque coefficient agreement between CFD and experimental results is very impressive. The comparison of the results is shown on figure 8. Average relative difference in this case is 4% and there is no increase in difference for large advance coefficient values. Fig. 8 shows the torque coefficient characteristic calculated with Ansys Fluent. These results are good as others.



Fig. 7 Thrust coefficient characteristics.



Fig. 8 Torque coefficient characteristics.



Fig. 9 Pressure coefficient distribution on propeller blades (suction side) - Ansys Fluent.

Figs. 9 and 10 show the contours of pressure coefficients distribution on suction side. Figs. 11 and 12 present values of pressure coefficient in function of X coordinate. Both sides (suction and pressure) have

Fig. 10 Pressure coefficient distribution on propeller blades (suction side) - OpenFOAM.

similar distribution for OpenFOAM and Ansys Fluent solver. The agreement is satisfactory. Relative differences are 9% for maximal and minimal values of calculated pressure coefficient.



Fig. 11 Pressure coefficient distribution on propeller blade in function X position - Ansys Fluent.



Fig. 12 Pressure coefficient distribution on propeller blade in function X position – OpenFOAM.

## **5.** Conclusions

Presented research confirms that OpenFOAM CFD toolbox can be successfully applied to evaluate propeller characteristic. The agreement between simulation and experimental results is very good. Flow and pressure fields calculated with OpenFOAM are not significantly different than those obtained from other commercial computational systems, like Ansys. OpenFOAM can be reasonable alternative for those who need flexible and not expensive CFD software.

# Acknowledgments

Calculations have been carried out in Wroclaw Centre for Networking and Supercomputing (http://www.wcss.pl), grant No 223.

#### References

- [1] Carlton J, 2007, Marine Propellers and Propulsion, Elsevier.
- [2] Klasson O.K. & Huuva T, 2011, Potsdam Propeller Test Case (PPTC), Second International Symposium on Marine Propulsors, Germany.
- [3] Wilcox D. C., 2002, Turbulence Modeling for CFD, DCW Industries inc.
- [4] Jiyuan Tu & Guan Heng Yeoch & Chaoqun Liu, 2008, Computational Fluid Dynamics a Practical Approach, Elsevier.
- [5] Ansys, 2010, ANSYS FLUENT User's Guide release 13, ANSYS, Inc.
- [6] Barkmann U. H., 2011, Open Water Tests with the Model Propeller VP1304, Schiffbau-Versuchsanstalt Potsdam GmbH, http://www.sva-potsdam.de/pptc.html.
- [7] OpenFOAM, 2012, User Guide, OpenFOAM Foundation, http://www.openfoam.org/.