First International Symposium on Marine Propulsors
smp'09, Trondheim, Norway, June 2009

Numerical Prediction of Vortex Generated by Hydrofoil

Paweł Flaszyński[1], Jan Szantyr[1], Paweł Dymarski[2], Marek Kraskowski[2]

[1]Gdańsk University of Technology, Mechanical Faculty, Gdańsk, Poland
[2]Ship Design and Research Centre – CTO S.A., Gdańsk, Poland

ABSTRACT

The purpose of the project described in this paper is to develop and validate a Computational Fluid Dynamics method especially suited for prediction of concentrated vortices behind marine propeller blades. Further objective is to use this method for prediction of the tip vortex cavitation, which is a source of intensive hydro-acoustic signal and should be avoided as far as possible during propeller design.

Numerical prediction of the vortex generated by the propeller-like hydrofoil is presented in the paper. Simulations have been performed with Fluent 6.3. The influence of the different turbulence models on the streamwise vortex intensity and location is analysed.

Numerical results are compared with experimental data obtained in the cavitation tunnel of Ship Design and Research Centre CTO in Gdansk. Measurements enable to make the comparison of the global forces acting on the foil and comparison of the two velocity components (LDA) at the selected sections downstream of the foil.

Keywords

Tip vortex, hydrofoil, vortex structure, LDA measurements

1 INTRODUCTION

The CFD methods became a very useful tool during design process in a wide range of applications. However, the effective application of CFD tools to the hydrodynamic performance prediction requires a basic study and validation. Vortex generated on the propeller blade tip is a classical vortical structure. It is known that the proper quantitative prediction of the vortex structure, its intensity and dissipation process is still a challenging task. The prediction of voriticity evolution is important for the cavitation and hydro-acoustics problems. Such effects are frequently taken into account during design process, so the tools development enabling, not only qualitative, but also quantitative vortex prediction is the present-day problem.

2 HYDROFOIL GEOMETRY AND EXPERIMENTAL SETUP

The presented investigation has been performed for the marine propeller blade designed and manufactured (CNC machining) in Ship Design and Research Centre – CTO S.A. The span of hydrofoil is 357 mm, the mean chord is 302 mm (Figure 1). The tests have been carried out in CTO's Cavitation Tunnel (Figure 2). Size of the test section is: 0.8m x 0.8m x 3.0m. Velocity measurements were performed with the use of 2D Laser-Doppler Anemometer (Figure 3). The two components of velocity were measured: streamwise (X direction) and spanwise (Z direction). The measurements have been carried out in three planes located 10 mm, 70 mm and 330 mm downstream from the blade tip. The area of measurements was a rectangle 62 mm x 41 mm, the size of mesh was 1mm x 1mm.

Figure 1. Hydrofoil

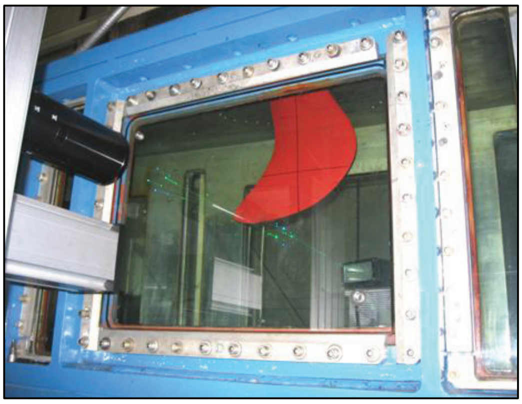

Figure 2. Hydrofoil in CTO's Cavitation Tunnel

Figure 3 View on test section and LDA equipment

3 NUMERICAL MODEL DESCRIPTION

The computational mesh was generated with ICEM CFD Hexa. The block-structured mesh consists of hexahedral cells. Number of control volumes of the initial grid was close to 1.2 million. Then, it was adapted after first simulations and it finally contains 1,7 mln cells. The mesh was refined in the tip vortex according the local streamwise vorticity. For the investigations, the low Reynolds turbulence models were taken into account, so the mesh is refined close to wall in order to keep y^+ less than 2. The mesh structure is shown in Figure 4. In Figure 5, the adapted mesh in tip vortex region is presented.

The propeller blade is inclined to the main flow direction in order to have 2.5º angle of attack.

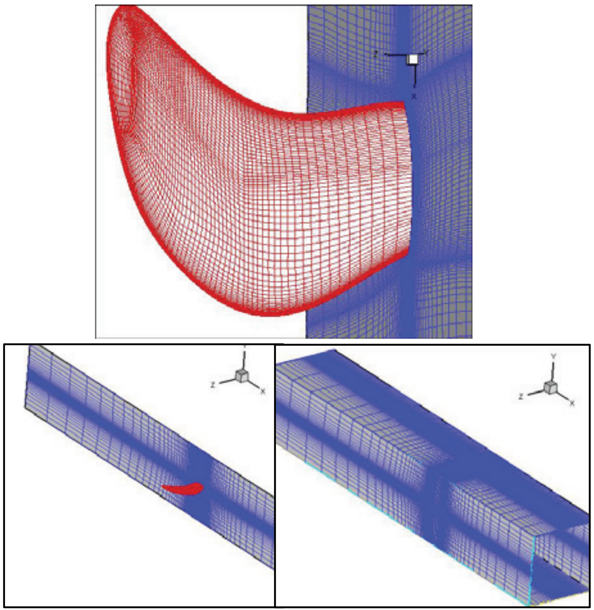

Figure 4. Blade (upper) and outer surfaces mesh (lower)

Numerical simulations have been done with Fluent 6.3. MUSCL (Monotone Upstream-Centered Schemes for Conservation Laws) scheme was applied. Below, the results for Spalart-Allmaras and k-omega SST turbulence models are presented. The selected models are well know and used in wide range of applications. Spalart-Allmaras approach is widely used in external aerodynamics. It is known that results obtained on basis of this model enable relatively good prediction of the boundary layer development and forces acting on the walls. The disadvantage of this model is poor ability of vortex evolution prediction. From numerical point of view, it converges faster than k-omega SST approach, what can be considered as an advantage in the design process.

Velocity 4 m/s at the inlet plane was set according to experimental data. Boundary conditions for turbulence models were not available. Turbulence intensity 1% and eddy viscosity ratio 10 were applied. At the upper, lower and side surfaces symmetry conditions were applied. The boundary layers on the sidewalls does not affect on the vortex meaningfully.

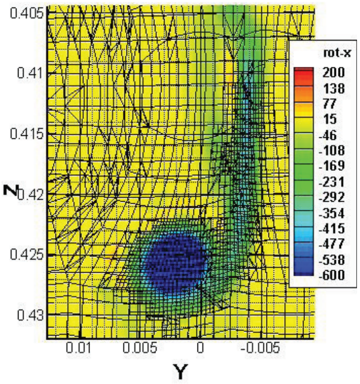

Figure 5. Adapted mesh (streamwise vorticity in background)

3 FLOW VELOCITY COMPONENTS – CFD vs. EXP

Based on measurements, the two components of velocity are available streamwise (X direction) and spanwise (Z direction) at three sections downstream to blade, shown in Figure 6. In such case, direct comparison of tip vortex intensity (streamwise vorticity) is not possible.

In Figures 7-9, the component of velocity in X direction is shown, at sections 10 mm (1), 70 mm (2) and 330 mm (3) downstream to blade, respectively. In Figures 10-12, the component of velocity in Z direction is presented in the same control planes.

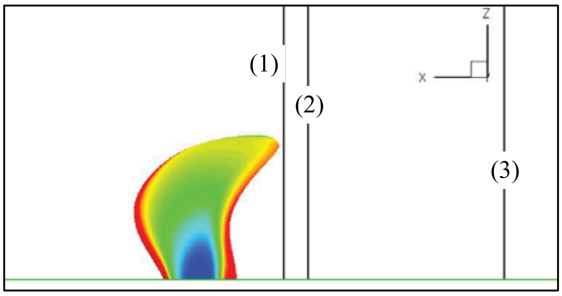

Figure 6. Blade (pressure contour) and control planes location

The best agreement of numerical prediction and experimental data is obtained at the section close to blade tip. Further from the blade, the differences are higher. One can see that results for k-omega SST model are closer to the experimental data.

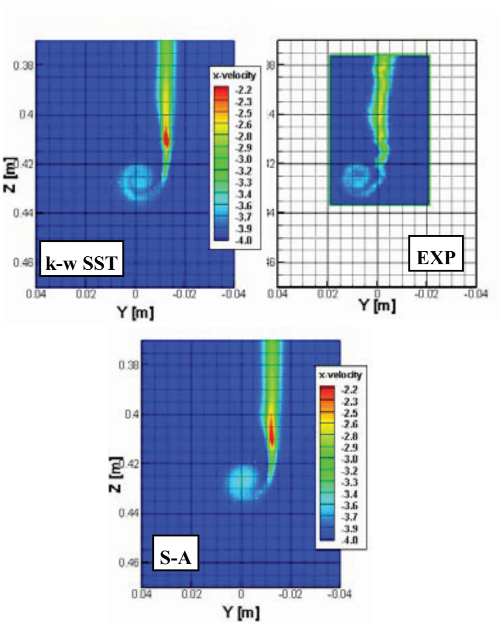

Figure 7. Velocity component in X direction (10 mm downstream to blade tip)

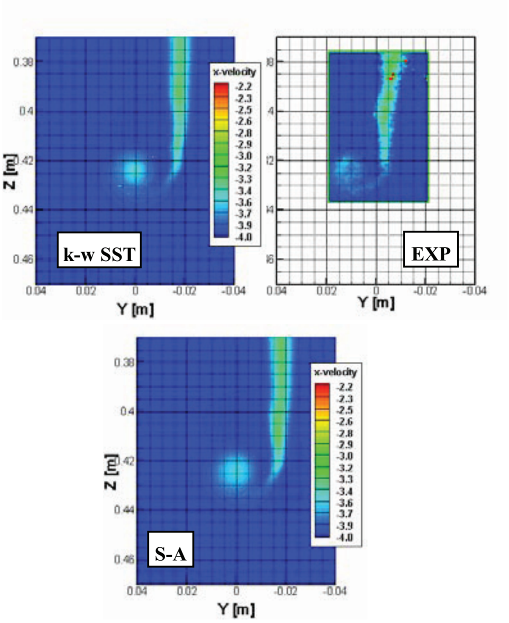

Figure 8. Velocity component in X direction (70 mm downstream to blade tip)

Spalart-Allmaras approach does not allow to obtain the proper vortex structure even close to the blade. It means that the differences appears already at the vortex origin.

At the first control plane, the size of the vortex is very similar for both models and compares well with the measured one. At the next planes, numerical prediction indicates existence of vortex with larger radius than experimental one. In case of Spalart-Allmaras model, the vortex dissipates much faster and at the section 330 mm downstream to the blade high differences in size and intensity are observed.

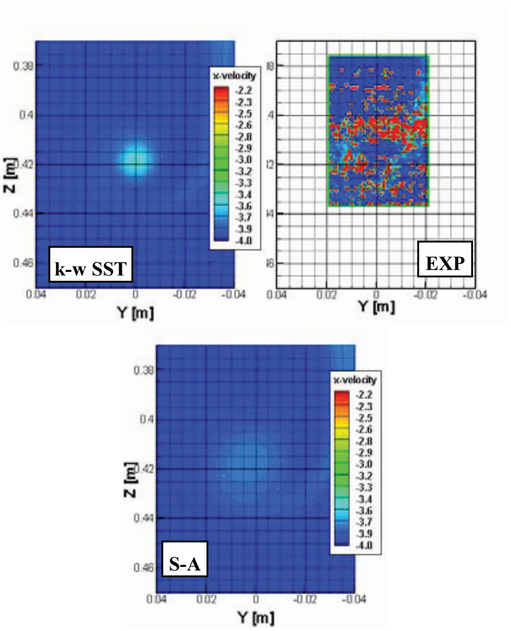

Figure 9. Velocity component in X direction (330 mm downstream to blade tip)

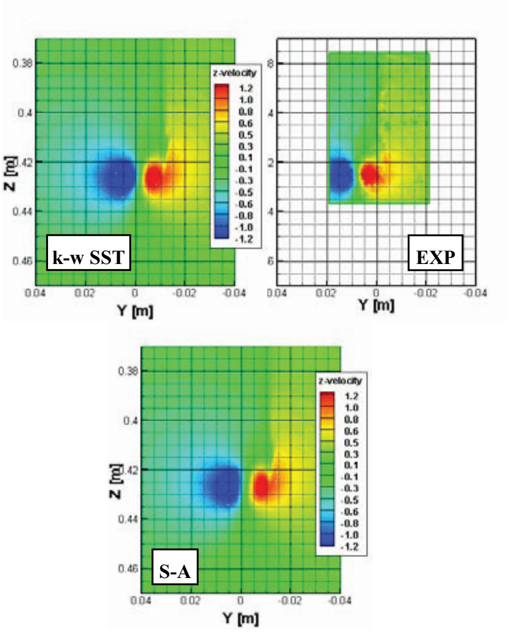

Figure 10. Velocity component in Z direction (10 mm downstream to blade tip)

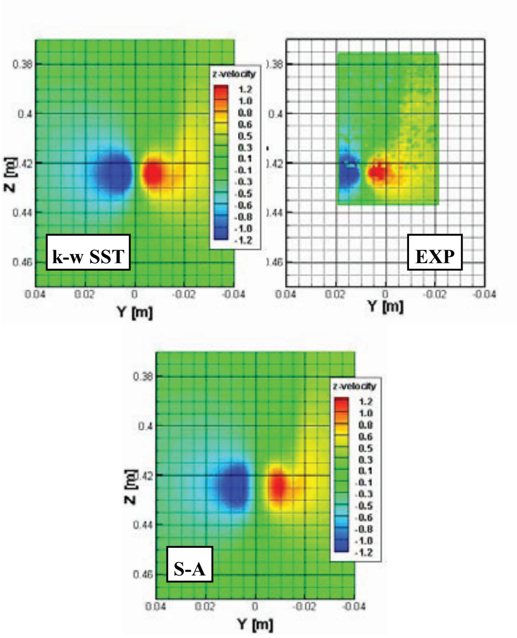

Figure 11. Velocity component in Z direction (70 mm downstream to blade tip)

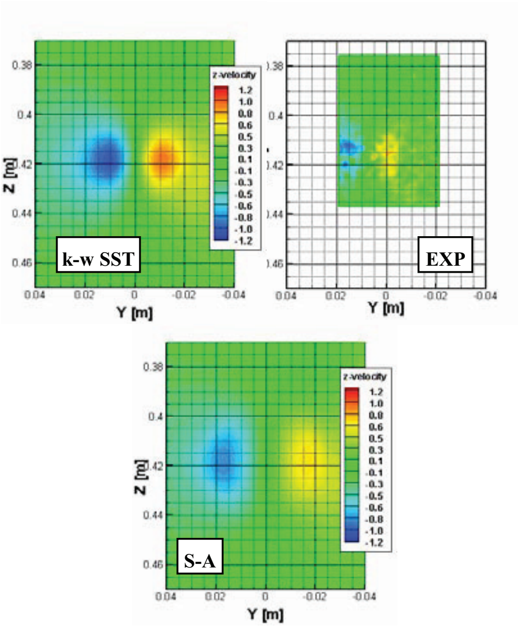

Figure 12. Velocity component in Z direction (330 mm downstream to blade tip)

4 STREAMWISE VORTICITY

The most valuable measure of the vortex intensity is vorticity magnitude. In the analysed case, the streamwise vorticity is a dominant part of the vorticity vector and it can be considered as a representative for the tip vortex. As it was mentioned above, there is no available data for vorticity comparison between numerical analysis and experimental data. Nevertheles, the compared velocity components indicate better prediction of the tip vortex by k-omega SST model. Based on such conclusion, one can compare streamwise vorticity obtained for both turbulence models and present relative differences.

In Figure 13-15, contours of vorticity component in X direction are shown at the three control sections. Vorticity distribution at the control planes confirms conclusions arising from the velocity component comparison. It is clearly shown that the vortex dissipates much faster in Spalart-Allmaras case than in k-omega SST one.

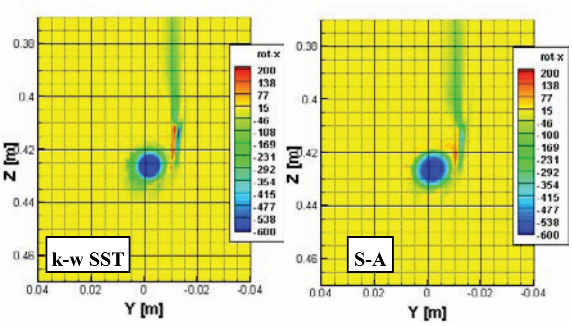

Figure 13. Streamwise vorticity (10 mm downstream to blade tip)

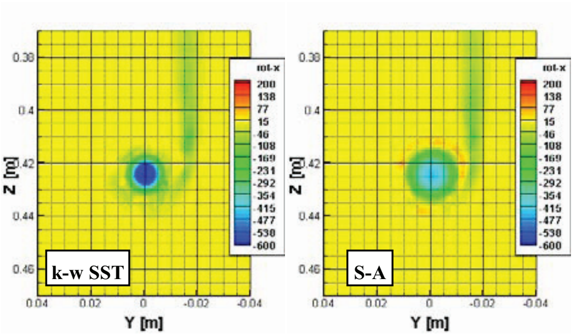

Figure 14. Streamwise vorticity (70 mm downstream to blade tip)

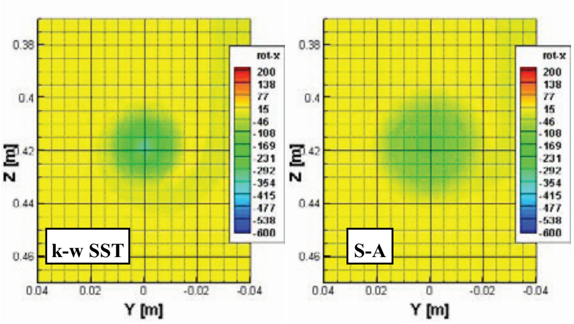

Figure 15. Streamwise vorticity (330 mm downstream to blade tip)

5 SUMMARY AND CONCLUSIONS

Numerical and experimental investigation of the tip vortex evolution was presented. Numerical simulations have been performed with Fluent 6.3 and two turbulence models. The results are compared the velocity components in streamwise and spanwise direction measured with the 2D Laser-Doppler Anemometer. It was shown that better prediction of the vortex intensity was captured by k-omega SST model. Spalart-Allmaras turbulence model does not allow to obtain the proper vortex structure even close to the blade. It means that the differences appears already at the vortex origin. Both models are based on the assumption of the isotropic turbulence, so in the area where the vortex originates, they do not predict flow structure properly. Nevertheless, it seems that k-omega SST model can be applied for the design analysis quite effectively. In such approach local mesh refinement according to local flow parameters distribution enables of obtaining satisfactory results in a relatively short time.

6 ACKNOWLEDGMENTS

The research presented in this paper was supported by Polish Ministry o Science and Higher Education, Grant No. N504 033 32/2951

REFERENCES

Wilcox D.C. (2003) ‘Turbulence Modeling for CFD’ 2nd Edition. DCW Industries

Ferziger J.H., Peric M. (1999) ‘Computational Methods for Fluid Dynamics’, 2nd ed., Springer

Fluent 6.3 User’s Guide (2006)

Spalart, P. R., Allmaras, S. R., (1992) ‘A One-Equation Turbulence Model for Aerodynamic Flows’ AIAA Paper 92-0439