# CFD VALIDATION OF A CONTROLLABLE PITCH MARINE PROPELLER USING TRULY AUTONOMOUS MESH GENERATION WITH ADAPTIVE MESH REFINEMENT

# Mathias Vångö<sup>1</sup> and Pietro Scienza<sup>1</sup>

<sup>1</sup>Convergent Science GmbH Hauptstrasse 10, 4040 Linz, Austria e-mail: mathias.vangoe@convergecfd.com, www.convergecfd.com

Key words: marine propeller, CFD, autonomous mesh, adaptive mesh refinement

**Abstract.** The design of propellers for maritime propulsion systems has a long history of using Computational Fluid Dynamics (CFD) as result of the constant desire to improve efficiency. The complex physics in addition to the motion of the propellers pose several challenges to CFD investigations, in particular with regards to mesh generation. In view of addressing these challenges, the present work proposes an alternative approach, which employs an autonomous mesh generation based on a modified Cartesian cut-cell methodology with Adaptive Mesh Refinement (AMR).

In this work, this approach is validated against the extensive open measurement data of the Potsdam Propeller Test Case (PPTC) from SVA Potsdam, which contains both open water tests as well as detailed transient velocity field measurements. Additionally, benefits of both steady-state and fully-moving transient approaches for propeller numerical analyses are discussed, together with a future outlook on cavitation phenomena within the presented framework.

# **1 INTRODUCTION**

More than ever, in recent times, the world is facing severe challenges involving greenhouse gases. The transportation sector, including shipping, is a major area where emissions must be reduced in upcoming years to comply with regulations. Thus, improving ship propulsion system efficiency, where the propeller is a key component, is of great importance. To that end, Computational Fluid Dynamics (CFD) is a valuable tool during the propeller design process, as it generates a vast amount of information which can be used for optimization purposes.

CFD has been extensively used for propeller applications. Steady-state open water simulations using a Multiple Reference Frame (MRF) approach have been studied by, *e.g.*, Gornicz et al. [1] using an unstructured mesh. Similarly, Sikirica et al. [2] also performed steady state open water simulations, taking it a step further and comparing various mesh approaches and turbulence models. Furthermore, cavitation is an important topic and has been studied by Lloyd et al. [3], while a blade optimization approach was presented by Papakonstatinou et al. [4]. Owen et al. [5] presented a framework for estimating propeller performance degradation due to blade roughness. It is evident that the meshing plays a crucial role and poses major challenges to the propeller design process. The mesh resolution, quality and even cell distribution all have major impact on CFD simulations in terms of result accuracy, simulation stability and runtime. To alleviate some of those issues, we propose a CFD model which employs a fully automated meshing strategy, using a modified Cartesian cut-cell approach with Adaptive Mesh Refinement (AMR). The methodology is evaluated by comparing steady-state open water thrust, torque and efficiency, as well as the transient velocity field to the in-situ Potsdam Propeller Test Case (PPTC) measurements carried out by Schiffbau-Versuchsanstalt Potsdam (SVA) [6, 7].

### 2 PPTC description

SVA carried out several investigations using the controllable pitch propeller VP1304, which included both open water tests as well as transient velocity field measurements using Laser Doppler Velocimetry (LDV) in a cavitation tunnel. VP1304 is a right-handed propeller with the diameter D = 0.25 m and consists of five blades. A complete list of the propeller data is given in [6].

From the open water tests, the full measured map of open water characteristics, i.e., thrust coefficient  $K_T$ , torque coefficient  $K_Q$  and propeller efficiency  $\eta_0$ , covering a wide range of advance coefficients J, was reported in [6] and serve as validation for our model. The open water tests were carried out with the fixed rotational speed of  $n = 15 s^{-1}$ , thus varying the advance coefficient only by means of altering the advance velocity. The dimensionless coefficients introduced above are defined as

$$J = \frac{V_A}{n D} \tag{1}$$

$$K_T = \frac{T}{\rho n^2 D^4} \tag{2}$$

$$K_Q = \frac{Q}{\rho n^2 D^5} \tag{3}$$

$$\eta_0 = \frac{J}{2\pi} \frac{K_T}{K_Q},\tag{4}$$

where  $V_A$ , T, and Q denote advance velocity, thrust, and torque respectively.

The LDV measurements reported in [7], where a vast amount of velocity measurements is available at various axial planes and radial positions, were used as validation for our model in a transient situation. Here, we compared the axial, tangential and radial velocity components reported at planes x = 0.1 and x = 0.2 D downstream of the propeller at the radial positions r/R = 0.7 and r/R = 1.0 as illustrated in Fig. 1. During this test, the operating condition was J = 1.253 and  $n = 23 s^{-1}$ .

# **3 COMPUTATIONAL MODEL**

In this work, CONVERGE [8] version 3.0.23 was used for all simulations. Both steady-state and transient simulations were carried out to make the necessary comparisons from the open water tests and LDV measurements. The approach is based on a (Unsteady) Reynolds-Averaged Navier-Stokes ((U)RANS) methodology using the well-known k- $\omega$  SST turbulence model as well



Figure 1: Illustration of the velocity measurement locations used as validation data for the simulation results in this work. Original figures from [7].

as the Pressure-Implicit with Splitting of Operators (PISO) algorithm to solve the pressure velocity coupling. Although CONVERGE has not previously been used for propeller simulations, it has been extensively used and validated in other application areas, such as internal combustion engines, gas turbines, as well as pumps and compressors [9, 10, 11]. A brief introduction to the numerical methodology is presented below, while a detailed description is given in [12].

## 3.1 Unsteady rotating geometry

In the transient simulation, the flow is governed by the incompressible URANS equations for a Newtonian fluid without body forces as

$$\nabla \cdot \bar{\mathbf{u}} = 0 \tag{5}$$

$$\rho \left[ \frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}} \bar{\mathbf{u}}) \right] = -\nabla \bar{p} + \nabla \cdot \tau_{ij}, \tag{6}$$

where  $\rho$  denotes density.  $\bar{\mathbf{u}}$  and  $\bar{p}$  represent the ensemble averaged velocity and pressure respectively arising from the Reynolds decomposition. Furthermore,  $\tau_{ij}$  represents the viscous stress tensor given as

$$\tau_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j},\tag{7}$$

where  $\mu$  is the dynamic viscosity and the last term is commonly referred to as the Reynolds stress tensor.

In our unsteady approach, the mesh is periodically regenerated during the simulation, which enables motion to be directly applied to the rotating boundaries. The meshing approach is further described in Section 3.3.

#### 3.2 Steady-state Multiple Reference Frame

In the steady-state approach, the Multiple Reference Frame (MRF) methodology was used to account for the propeller motion. In the MRF approach, the propeller remains stationary and the motion is accounted for by solving a portion of the simulation domain around the propeller in a rotating reference frame. This is a suitable approach for steady-state simulations as it makes the flow-field steady in the inertial reference frame and reduces significantly the required physical runtime. The governing equations are then augmented as

$$\nabla \cdot \bar{\mathbf{u}}_r = 0 \tag{8}$$

$$\rho \left[ \frac{\partial \bar{\mathbf{u}}_r}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}_r \bar{\mathbf{u}}_r) + 2 \left( \mathbf{\Omega} \times \bar{\mathbf{u}}_r \right) + \mathbf{\Omega} \times (\mathbf{\Omega} \times \mathbf{R}) \right] = -\nabla \bar{p} + \nabla \cdot \tau_{ij}, \tag{9}$$

within the rotating region, solving for the relative velocity  $\bar{\mathbf{u}}_r$ .  $\boldsymbol{\Omega}$  and  $\mathbf{R}$  represent the rotation vector and position vector respectively and are related to the velocity in the inertial reference frame as

$$\mathbf{u} = \mathbf{u}_r + \mathbf{\Omega} \times \mathbf{R}.\tag{10}$$

Also in our steady-state approach, the transient formulation of Eqs. (6) and (9) are solved using a pseudo-transient time marching approach, where the aim is to drive the transient terms to zero in a relaxed fashion by employing large pseudo-time steps.

### 3.3 Mesh

CONVERGE uses a modified Cartesian cut-cell meshing method, which allows the solver to retain the exact details of the geometry, without introducing any modification to the boundary shape. In this framework, the mesh generation process is fully automated and the grid is regenerated every time step, thus inherently allowing moving and deforming boundaries, which differs from commonly used techniques (*i.e.*, sliding mesh) in that the mesh elements remain stationary and undeformed.

Furthermore, in traditional CFD, a priori knowledge of local flow features is often required to achieve satisfactory results, which is especially challenging in complex scenarios, often leading to a sub-optimal use of the cell count. As the mesh is continuously updated in CONVERGE, it is allowed to be locally refined during runtime using AMR, making optimal use of the cell count. More precisely, AMR evaluates the magnitude of the sub-grid field  $\Phi'$  of user-specified variables (*e.g.*, velocity and temperature) to assess if mesh refinement should be applied, satisfying user defined criteria.  $\Phi'$  is defined as  $\Phi' = \Phi - \overline{\Phi}$ , where  $\Phi$  and  $\overline{\Phi}$  are the actual and resolved field, respectively. The sub-grid field is approximated as the second order derivative

$$\Phi' = -\alpha_{[k]} \frac{\partial^2 \bar{\Phi}}{\partial x_k \partial x_k},\tag{11}$$

arising from the infinite series expansion of  $\Phi'$ , where  $\alpha_k$  is  $(\Delta x_k)^2/24$  for rectangular shaped cells. Moreover, AMR can also act on boundaries based on a given non-dimensional wall distance  $(y^+)$  target. This results in an even  $y^+$  distribution on the wall boundaries and ensures that the mesh is adequately refined (*i.e.*,  $y^+$  is of appropriate magnitude to be compatible with the selected wall function).



Figure 2: Computational domains used for the open water simulations (top) and the transient one (bottom).

#### 4 SIMULATION SETUP

The simulation domains used for both the open water simulations as well as the transient one are shown in Fig. 2. In the open water simulations (upper figure), the outer domain was constructed as a cylinder with a diameter of 5 D and treated as a symmetry boundary, the inflow boundary being located 5 D upstream and the outflow boundary at 10 D downstream from the propeller. Here, the dimensions of the outer domain were chosen such that the boundaries had no influence on the near-propeller flow. Furthermore, the MRF region was also defined as a cylinder with the diameter 1.15 D and was extended from the base of the propeller hub to just beyond the hub tip. In the transient simulation, the actual geometry of the cavitation tunnel from the tests (lower figure) was instead used to define the outer domain.

For all simulations, a pressure boundary condition of 1 atm absolute static pressure was used at the outflow, while at the inflow, a fixed uniform velocity was imposed based on the advance coefficient.

Although the mesh is automatically generated, a few parameters must be specified as can be observed in Fig. 3. For both the open water and transient cases a base grid of  $\Delta x = 64 \, mm$  was used, denoting the largest cells observed in the domain. A relatively large cylindrical embedding with the size  $\Delta x = 8 \, mm$  was used around the propeller, whereas directly around it the mesh was further refined to  $\Delta x = 4 \, mm$ . For the propeller hub and blades, AMR based on  $y^+$  with a target value of 100 was used, resulting in a local maximum refinement close to the blade edges, with sizes as low as  $\Delta x = 0.5 \, mm$ . Additionally, velocity based AMR with the minimum



Figure 3: Snapshots of the used meshes for the open water simulations (left) and the transient one (right). The iso-surface in the right figure is showing Q-criterion = 5000.

size  $\Delta x = 2 \, mm$  was applied in all cases, with additional helicity based AMR with minimum  $\Delta x = 1 \, mm$  in the transient case. Here helicity, h, is defined as  $h = \boldsymbol{\omega} \cdot \mathbf{u}$ , where  $\boldsymbol{\omega}$  denotes the vorticity vector. This mesh strategy resulted in a final cell count of  $\approx 1.6$  M in the open water case, varying slightly for the various advance coefficients, while the transient case began with  $\approx 2$  M and continuously increased until a maximum of  $\approx 8$  M towards the end of the simulation. The AMR refinement can be appreciated in Fig. 3 - left for the open water case, where the mesh size vary on the blades due to the y+ constraint, and in Fig. 3 - right for the transient simulation, where the elements are concentrated in the wake region following the velocity and helicity variations.

## 5 RESULTS

The simulated open water characteristic map, compared with the measured one, is shown in Fig. 4. In general, it can be seen that a great agreement of thrust, torque, and efficiency was obtained for the complete map, capturing the peak efficiency at J = 1.4. Only the inflow boundary conditions were changed across all the operating conditions, without any other additional model calibration; this suggests that the chosen mesh strategy is working well for such application.

Figure 5 depicts the pressure- (upper) and  $y^+$ -contours (lower) on the propeller pressure-(left) and suction-side (right). On the pressure side, we see a rather low pressure towards the blade roots, which is increasing approaching the blade tips, especially towards the leading edges. On the suction side, we observe similarly low pressure at the blade roots, which increases towards the trailing edges. The lowest pressure is found at the leading edges on the suction side and would be the most likely location subject to cavitation. On both the pressure and suction side, we notice an overall rather uniform  $y^+$  distribution thanks to the boundary AMR. Evidentially, the  $y^+$  increases towards the blade tips, which is expected due to the surface velocity increasing



Figure 4: Open water characteristic map as obtained through simulations (solid red lines), compared with measurements (dashed black lines).

with increasing radius, deeming the additional mesh refinement necessary.

Figure 6 visualizes the velocity and pressure contours, together with an iso-surface representing the vortices, at a side view central plane. We can observe the expected formation of blade-tip vortices due to the pressure difference between the pressure and suction side of the blades, as well as the low-pressure hub vortex. The low-pressure region in the wake causes additional drag, and together with the vorticity reduces the overall propeller performance.

A quantitative comparison of the axial, tangential and radial velocity components is shown in Fig. 7 at the two planes x/D = 0.1 and x/D = 0.2, where the values are normalized by the advance velocity  $V_A$ , so that  $w_t = V_t/V_A$  and  $w_r = V_r/V_A$ . Generally, the trends from the measurements are well captured in the simulation, except for the velocity magnitudes at r/R = 1.0 which appears to be underpredicted, especially at the plane closer to the propeller. At this location, close to the blade tips, the velocity gradients are very strong and it is possible that further mesh refinement would improve this prediction.

# 6 CONCLUSION

In this work, we have introduced a CFD model using a truly autonomous mesh approach for propeller simulations. As the mesh is regenerated during the simulation, imposing boundary motion is relatively straightforward as the mesh elements themselves remain stationary. Furthermore, we have shown how mesh refinement can be dynamically and automatically applied with AMR based on either sub-grid scales of selected quantities or on boundaries based on the local  $y^+$ .

We validated the model against in-situ open water, as well as transient LDV measurements from the Potsdam Propeller Test Case by SVA Potsdam. More specifically, open water simula-



Figure 5: Propeller contours visualizing pressure- (top) and  $y^+$ -distribution (bottom) for the pressure- (left) and suction-side (right) respectively.

tions were carried out in a steady-state manner, where blade thrust, torque and efficiency were evaluated and compared to measurements over a wide range of advance coefficients. Moreover, a transient simulation was performed to validate the unsteady flow field by comparing to the LDV measurements.

In general, the simulations showed good agreements with the measurements. The open water characteristics were well predicted over the complete range of advance coefficients, with the only change to the case setup being the advance velocity thus highlighting the advantages of using boundary AMR. The transient simulation also showed a satisfactory agreement for the velocity components at all monitored locations as compared to measurements, except for some underprediction of the overall velocity magnitudes close to the blade tips, where large velocity gradients are present due to the formation of vortices.

Future work will incorporate cavitation modeling to the presented framework. Cavitation has a great negative impact on propeller performance and is a primary cause of propeller noise, and is thus highly important to accurately capture within the model.



Figure 6: Side view of a central plane visualizing velocity (left) and pressure (right) contours. The iso-surfaces are based on the Q-criterion to visualize vortex structures.



Figure 7: Comparison of velocity components at the two downstream planes x/D = 0.1 (upper) and x/D = 0.2 (lower) for two radial positions r/R = 0.7 (left) and r/R = 1.0 (right). The solid lines represent simulation results while the dashed lines visualize measurements.

## REFERENCES

- T. Gornicz and J. Kulczyk, "The Assessment of the Application of the CFD Package Open-FOAM to Simulating Flow around the Propeller," Marine Navigation and Safety of Sea Transportation: Maritime Transport & Shipping, vol. 247, 2013.
- [2] A. Sikirica, Z. Čarija, L. Kranjčević, and I. Lučin, "Grid type and turbulence model influence on propeller characteristics prediction," *Journal of Marine Science and Engineering*, vol. 7, no. 10, p. 374, 2019.
- [3] T. Lloyd, G. Vaz, D. Rijpkema, and B. Schuiling, "The Potsdam Propeller Test Case in oblique flow: prediction of propeller performance, cavitation patterns and pressure pulses," in Second International Workshop on Cavitating Propeller Performance, Austin, Texas, USA, 2015.
- [4] T. Papakonstantinou, G. Grigoropoulos, and G. Papadakis, "Marine propeller optimization using open-source CFD," in Sustainable Development and Innovations in Marine Technologies: Proceedings of the 18th International Congress of the Maritme Association of the Mediterranean (IMAM 2019), September 9-11, 2019, Varna, Bulgaria, p. 252, 2019.
- [5] D. Owen, Y. K. Demirel, E. Oguz, T. Tezdogan, and A. Incecik, "Investigating the effect of biofouling on propeller characteristics using CFD," *Ocean Engineering*, vol. 159, pp. 505– 516, 2018.
- [6] U. Barkmann, "Potsdam Propeller Test Case (PPTC) Open Water Tests with the Model Propeller VP1304." Report 3752, Schiffbau-Versuchsanstalt Potsdam, April 2011.
- [7] K.-P. Mach, "Potsdam Propeller Test Case (PPTC) LDV velocity measurements with the Model Propeller VP1304." Report 3754, Schiffbau-Versuchsanstalt Potsdam, April 2011.
- [8] K. J. Richards, P. K. Senecal, and E. Pomraning, "CONVERGE 3.0." Convergent Science, Madison, WI, 2022.
- [9] P. K. Senecal, E. Pomraning, K. J. Richards, and S. Som, "Grid-convergent spray models for internal combustion engine CFD simulations," in *Internal Combustion Engine Division Fall Technical Conference*, vol. 55096, pp. 697–710, American Society of Mechanical Engineers, 2012.
- [10] Y. Li, D. H. Rowinski, K. Bansal, and K. Rudra Reddy, "CFD modeling and performance evaluation of a centrifugal fan using a cut-cell method with automatic mesh generation and adaptive mesh refinement," 2018.
- [11] H. Omote, K. Hirota, T. Hotta, G. Kumar, and S. A. Drennan, "Combustion and Conjugate Heat Transfer CFD Simulations to Support Combustor Design," in *International Gas Turbines Conference*, Tokyo, Japan, 2015.
- [12] K. J. Richards, P. K. Senecal, and E. Pomraning, "CONVERGE 3.0 Manual." Convergent Science, Madison, WI, 2022.