CFD ACCURACY STUDY OF FLOW AROUND AN AZIPULL

{ SOLVEIG MASDAL HOVDEN }
solveimh@stud.ntnu.no
DEPARTMENT OF MARINE TECHNOLOGY

NTNU – Trondheim
Norwegian University of Science and Technology

INTRODUCTION

Rolls-Royce Marine Propulsion in Ulsteinvik has developed a fully automated open water simulation tool called Propulsion Open Water Simulations (POWS). This system can be used for any propulsion and thruster system. This enables propeller designers without any prerequisite skills in RANS based CFD methods to run advanced simulations. The designer provides geometry file, propeller type and other data for the simulations and gets a report back with the results. The computation is performed on the local High-Performance Computer Cluster.

The simulations in POWS are done using the commercial license based software ANSYS Fluent as solver. This is an expensive solution and it is proposed to change the solver to the open source code OpenFOAM. This master thesis will consider the possibility of using OpenFOAM in the fully automated system POWS. With a new solver, new mesh requirements are needed to ensure the mesh is robust to avoid divergence, and efficient to make the calculations as little computational demanding as possible. Also, the numerical schemes used are investigated, here the goal is to find a stable, accurate and inexpensive set of numerical schemes. As the solver is made for an automated system the stability requirement is of great importance. The turbulence model is decided based on common practice and the parameters included are found using simplified calculations. The results are validated using POWS and experimental results.

OBJECTIVES

The main objectives of this master thesis are to

1. investigate the possibility of using OpenFOAM as a solver in POWS,
2. generate a mesh that is suitable for OpenFOAM and
3. optimize the solver settings to ensure robustness and manageable computational effort.

METHOD

When changing the solver, the mesh requirements of the new solver must be met, a new automated meshing script is developed to accomplish this. The mesh is made using BOXERMesh, it is made for an Azipull100 and validated with Azipull120 and Azipull150. Only Azipull120 is presented in this poster. The goal for the mesh is to make an inexpencive mesh which gives stable solutions for all propellers.

Different numerical schemes are compared to see how the results are affected. The stability of the solver is the main criteria for the schemes, accuracy is also considered. Different gradient and divergence schemes are tested in addition with limiters for the surface normal gradient scheme. The effect of under-relaxation is also investigated to find a balance between the stability and efficiency. To validate the new solver and mesh, the results are compared with results obtained using ANSYS Fluent and experimental results.

OpenFOAM is a solver without any graphical user interface, the simulations are set up using a folder system with three folders, the *0* folder which includes the initial conditions, the *constant* folder which includes the transport and turbulence properties, the mesh and the moving reference frame properties. The last folder is the *systems* folder which includes information about the solver and schemes. When investigating different meshes, schemes and solver parameters several simulations where run. To do this in an efficient and systematic manner a script run in the Linux terminal is used to generate the mesh, set up the simulation, run it and provide visualization of the results.

The effect of different turbulence models is not investigated in this thesis. The $k-\omega$ SST turbulence model is used, the values for k and ω are calculated based on simplified equations. The turbulence model is chosen based on common practice in Rolls-Royce Marine and the results from my project thesis. The steady state simulations are performed using the SIMPLE algorithm. A transient simulation using the PIMPLE algorithm is also set up, different solver options are considered to ensure a stable and accurate simulation.

SUPERVISORS

Sverre Steen, NTNU

Jonas Eriksson, Rolls-Royce Marine

RESULTS

The final mesh is showed in Figure 1 and Figure 2. Som information about the mesh is included in Table 1. The mesh is a Cartesian octree mesh with a bodyfitted prismatic layer around the propeller. The smalles cells are located at the leading edge of the propeller where it is level 11 (for one level a cell is divided into 8 cells). Around the propeller there is a volume refinement which is at level 5, the same level as the thruster body. Edge refinement is used to capture critical regions.

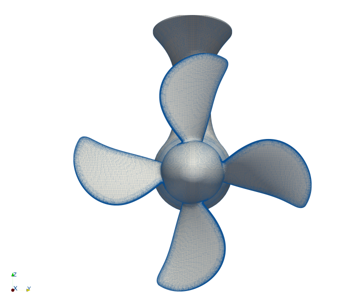

Figure 1: Mesh from the front.

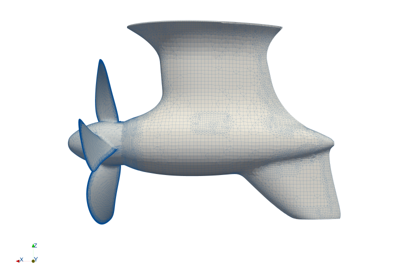

Figure 2: Mesh from the side.

Table 1: Mesh statistics for the Azipull120 mesh.

Number of cells	10.8M
Maximum mesh non-orthogonality	85.9
Average mesh non-orthogonality	16.1
Max aspect ratio	150

The scheme and solver test where performed on a mesh with 30% less cells, this is done to save computation time and to test the stability on a more demanding mesh. The under-relaxation is also higher than for the final runs. For the surface normal gradient and the Laplacian scheme the corrected option without any limiting proved to be most stable on the cases run. For the divergence, *linearUpwindV* is used for the velocity and *upwind* for the turbulence parameters. It is more important for the turbulence parameters to be bounded than accurate.

Low under-relaxation generaly increases stability and numbers of iteration before the convergence criterias are met. In this case the numbers of iterations did not increase severly and the solution became more stable, the under relaxation used for the validation is 0.3 for the pressure field and 0.5 for the velocity, k and ω equations.

The open water diagram of the Azipull120 is showed in Figure 3. The numbers on the axis are removed to not violate the non-disclosure agreement. The results calculated are compared with results from POWS and experimental results from HSVA and MARINTEK. The results are in good agreement with the results from POWS. The results are closest to the HSVA results compared with MARINTEK. The simulation converged for all advance numbers tested for Azipull120.

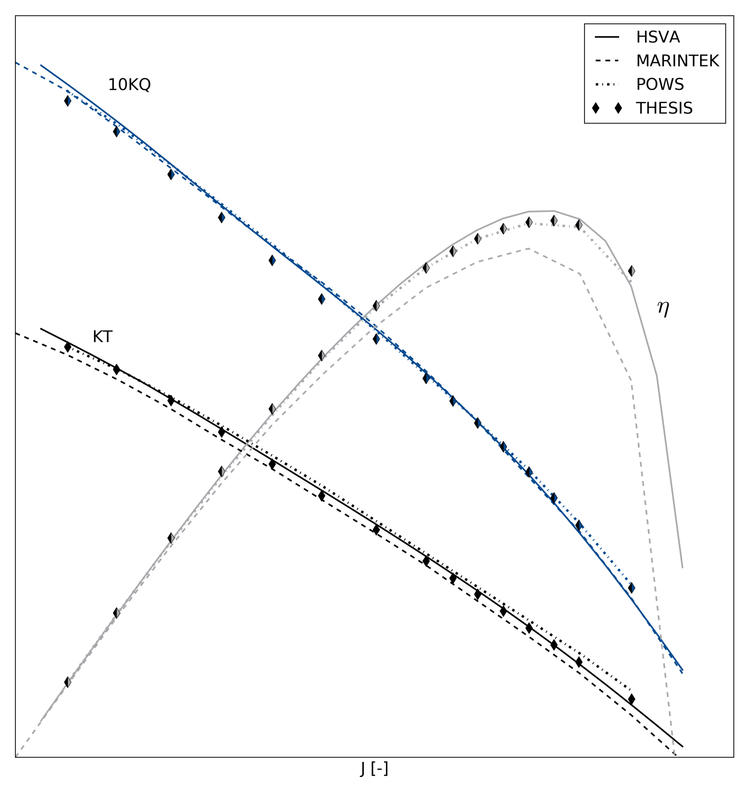

Figure 3: Open water diagram for the Azipull120.

Contour plots of the pressure is showed in Figure 4 and Figure 5. Figure 4 shows high pressure at the hub, leading edge and the neck of the thruster body as expected.

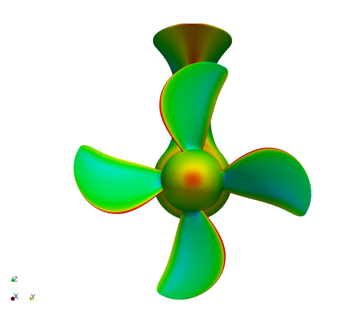

Figure 4: Pressure contours from the front.

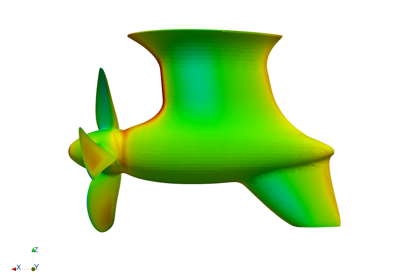

Figure 5: Pressure contours from the side.

CONCLUSION

The mesh created is computationally expensive. Meshes with around 30 % less cells have proven to converge on each propeller, but not the same mesh on both validation propellers. The coarser meshes does not capture the geometry accurately, the leading edge gets saw shaped as the meshing code have problems with detecting the edge. This problem is reduced by reducing the cell size in this region.

The maximum non-orthogonality is high, 85.9. Cells with non-orthogonality over 70 degrees are considered high. To account for the high non-orthogonality the surface normal gradient and Laplacian schemes are corrected and the pressure correction equation is solved three times. The mesh non-orthogonality can be an important parameter when considering stability. The maximum value is only in one cell, this makes it dangerous to use alone as a decision criterion. The average non-orthogonality and the maximum aspect ratio are acceptable.

The results from the scheme investigations are expected results and in agreement with common practice. The stability is the most heavily weighted criteria as the systems shows problems with convergence. The accuracy is also considered, but not as heavily.

OpenFOAM is possible to use as solver in POWS. The results for steady state simulations are within a tolerable accuracy. The results are close to those obtained by POWS. It is difficult to compare with the model tests as they differ. The thrust coefficient KT is between the two model tests and slightly under the POWS results. The torque coefficient KQ seems to be slightly underpredicted compared with the other results. The difference looks bigger than it really is as the plotted values are 10 times the calculated value. The propeller efficiency η is in good agreement with POWS, slightly under the HSVA results and above the MARNTEK results.

It is time consuming to set up OpenFOAM as a solver as all information must be given by the user. Since there is no graphical user interface this requires understanding of the solution process. To generate each simulation, an automated system is required to work in a rational manner.

OpenFOAM seem to require higher mesh quality than ANSYS Fluent based on the limited investigations done in this thesis. This is possibly a challenge when making an automated system. The robustness is important to avoid excessive manual work and delays in results. It also important to always get reliable results.

The quality of the grid generated by BOXERMesh is not completely satisfactory with the setup used in this thesis. One problem is the CAD files imported, the quality of these may differ, but the main problem is the leading edges on the propeller. These edges easily gets rough due to cells collapsing. This can also lead to collapsing of the prismatic boundary layer. In an ANSYS mesh with similar total number of cells the edges are better captured. A finer grid is required in BOXERMesh to get equally smooth edges. It has to be noted that the grid is made by an inexperienced user and can probably be improved.

The high mesh requirements from OpenFOAM and the low-quality grids generated by BOXERMesh combined is a challenge. It is possible to overcome this, but a higher number of cells than for the current POWS system seems unavoidable. It is possible to run simulations with around the same amount of cells, but for a general automated robust system for several propellers this is difficult.

FUTURE RESEARCH

In this thesis two thrusters are investigated, POWS is generated to work for more thruster types. To implement more thruster types a more general meshing code is required. Ducts for instance must be included. The system for setting up the cases also gets more complex as more options are required.

Transient simulations exist as an option in POWS. Transient simulations are running, but the results are not ready at the present date. Stability is a challenge when attempting inexpensive transient simulations. The mesh quality togheter with reasonable timesteps have proven to be a challenge.

REFERENCES

[1] J. Blazek. *Computational Fluid Dynamics: Principles and Applications*. Elsevier Science, 2001.

[2] Hrvoje Jasak. Error analysis and estimation for the finite volume method with applications to fluid flows, 1996.