Numerical Simulations of Flow Around a Simplified Hull Form

Øyvind A. Hagen Emblemsvåg

Introduction

The role of Computational Fluid Dynamics (CFD) is becoming more important in both research and commercial activities. In the industry, CFD is used for a wide variety of hydrodynamic applications, from estimation of ship resistance to detailed analysis of flow around various specialized structures.

This master thesis is designed to be an introduction to the world of CFD in hydrodynamics. It consists of a thorough introduction to the background theory of turbulent flows, a practical introduction to the computational methods used, and results of simulations of flow around simple geometries using different turbulence modelling techniques. The geometry was originally though of as a simplified ship hull.

The geometry used was a rectangular box, in both a fully submerged case and a floating case. Highly separated flow will occur in both cases. The results of interest are the flow field in general and the forces on the body. Both Reynolds Averaged Navier-Stokes (RANS) methods and Large Eddy Simulations (LES) have been used. The objective has been to compare different models and their applicability. The open source CFD package OpenFOAM has been used for all simulations.

Geometry and Fluid Domain

Two flow cases have been investigated, the floating box and the fully submerged double-body box. The fluid domains can be seen below. All cases are 3D, with Reynolds number of 26,400. The aspect ratio of the body is 5:1. Number of cells range from about 3-15 million. Grids for the RANS cases were made through iteration by the author, while the grid for the LES case was made according to Arslan et al. (2011). LES requires much finer grids than RANS.

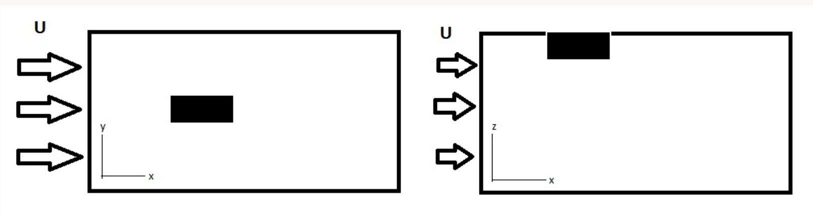

Figure: Floating body case domain

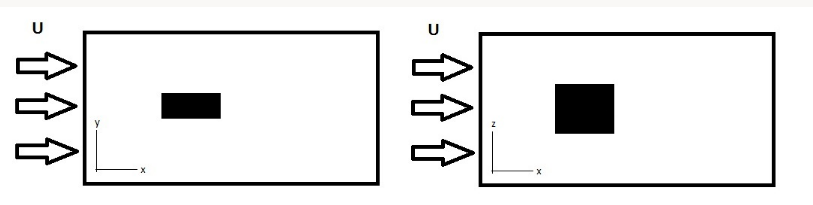

Figure: Double-body case domain

Turbulence Modelling Methods

The RANS approach is based on decomposing the flow variables into a mean and a fluctuating part. The fluctuating part is then modelled by a turbulence model, and the mean flow Navier-stokes equations are solved. Two RANS turbulence models were used, the k-omega SST model of Menter (1993) (equation 1 and 2), and the realizable k-epsilon model of Shih et al. (1995). Both these models are improved versions of the basic models and are reported to be applicable for separated flows.

$$\frac{Dk}{Dt} = \tau_{ij}\frac{1}{\rho}\frac{\partial u_i}{\partial x_j} - \beta^* \omega k + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_k \nu_t)\frac{\partial k}{\partial x_j}\right] \quad (1)$$

$$\frac{D\omega}{Dt} = \frac{\gamma}{\rho \nu_t}\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu+\sigma_\omega \nu_t)\frac{\partial \omega}{\partial x_j}\right] + 2(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \quad (2)$$

The LES model used was the Smagorinsky subgrid-scale model. The basic approach of LES is to filter the equations of flow in space, so that only motions larger than a certain length scale are resolved.

Results

The figures below show some of the results so far. All results are from RANS double-body cases using realizable k-epsilon. Graphics are from paraview.

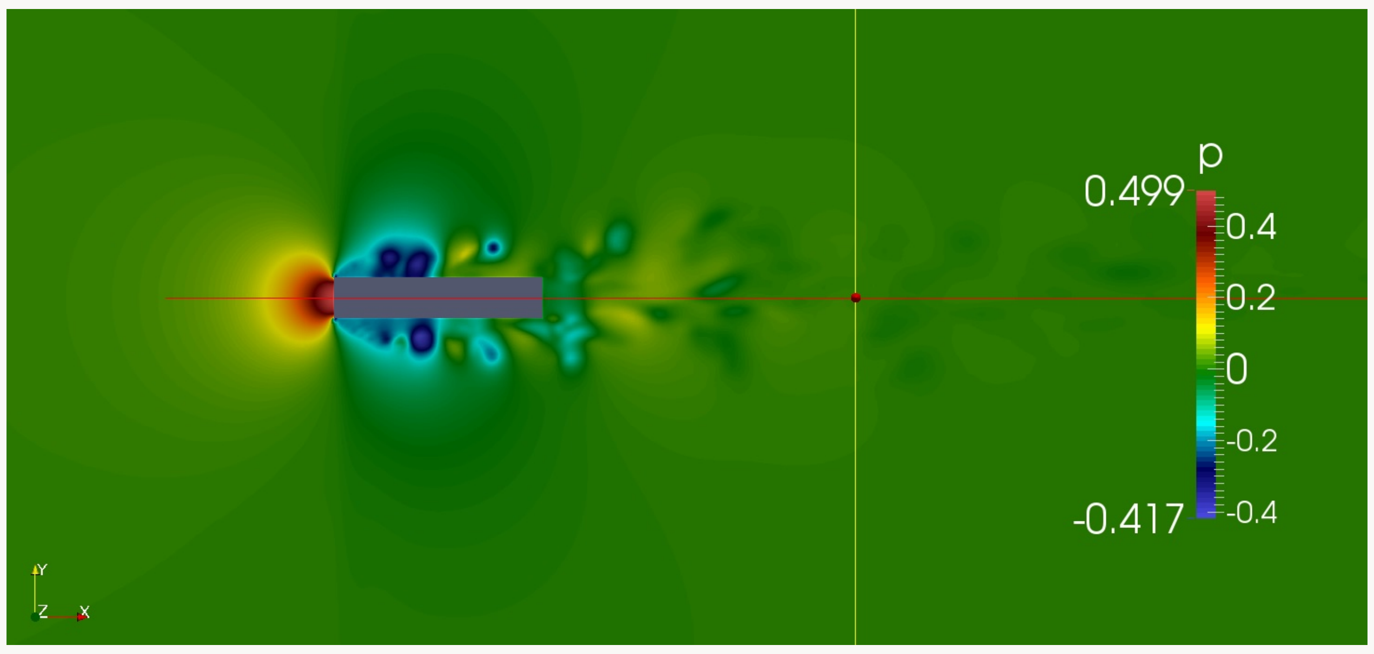

Figure: Plot of relative pressure showing vortex shedding

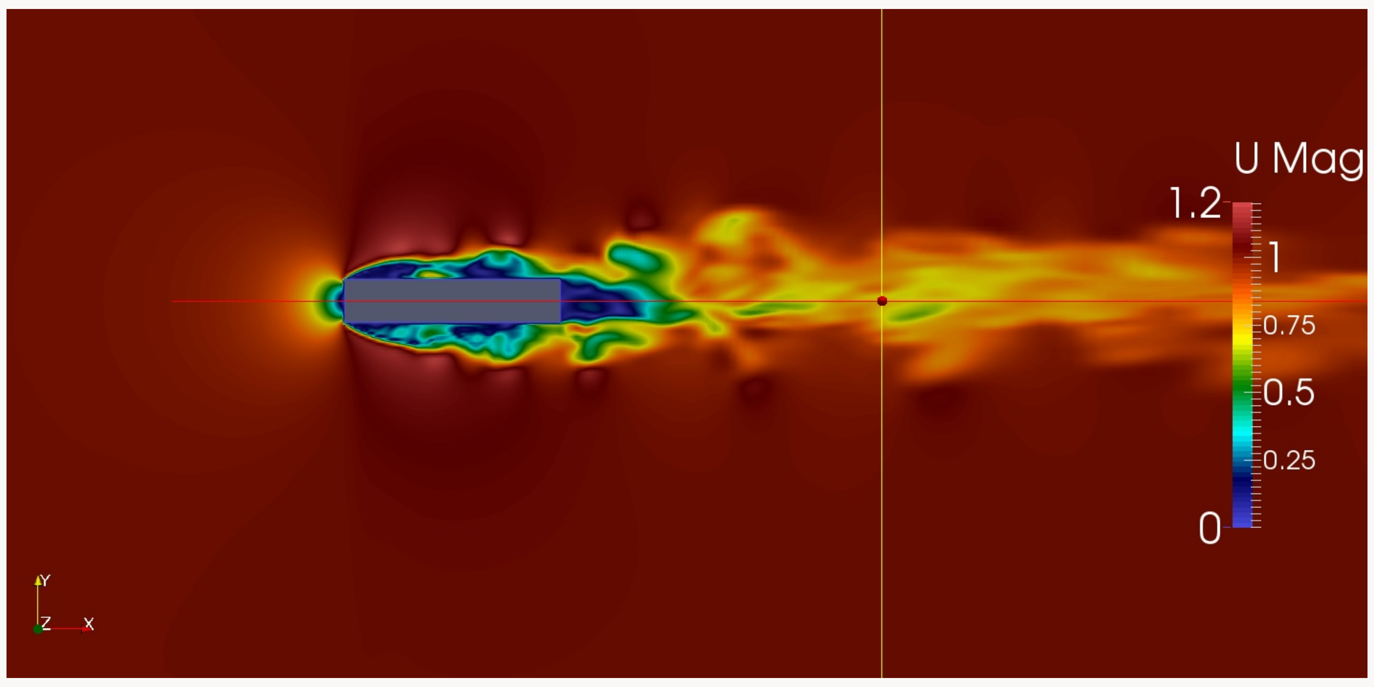

Figure: Plot of velocity showing separation and reattachment, and general flow instability

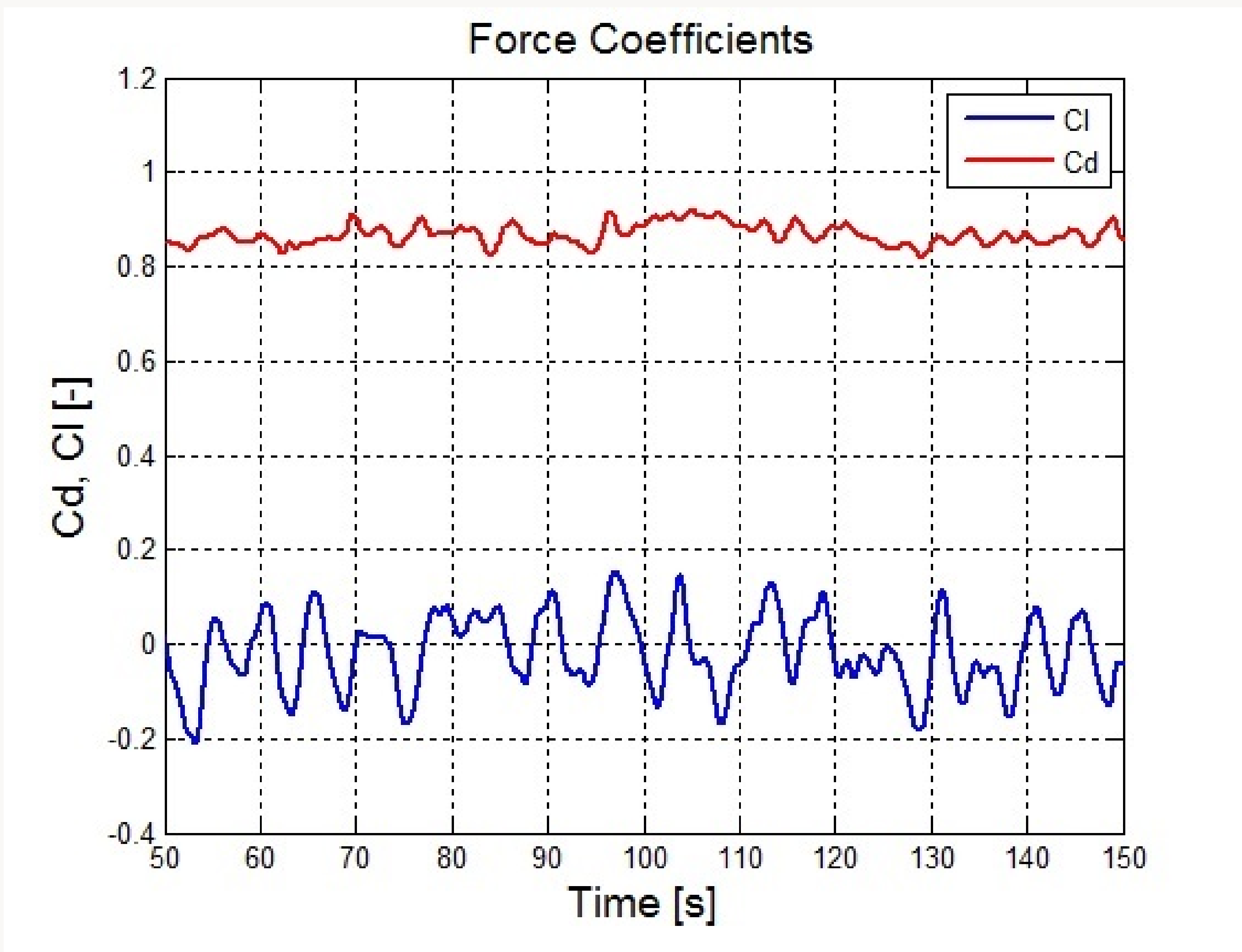

Figure: Plot of lift and drag coefficients showing non-harmonic oscillations in a turbulent flow environment

Discussion

The results show that the flow field is highly irregular and that both vortex shedding and reattachment is present. A preliminary result of the comparison between RANS and LES is that the LES results are even more irregular, resulting in lower maximum values of lift coefficients. This result is also found by Rodi (1997). Drag coefficients and vortex shedding frequencies are quite close for all cases.

Regarding turbulence models, it was found that the k-omega SST model did not perform well when using wall functions. This is why the realizable k-epsilon model was the main focus in the RANS simulations. The Smagorinsky SGS LES model was found to perform well for this flow. No other LES model has been applied due to time limitations.

Conclusions

It is too early to tell the exact conclusions form the simulations. However, it has been seen that the RANS approach can appropriately model the flow in question. Vortex shedding has been captured by both RANS and LES methods. Reasonable force coefficient results have been acquired so far, although there is little data to compare with for validation. LES takes a lot more time than RANS simulations, so use of LES should be compensated by more accurate results if it is to be used.

References

Tufan Arslan, Bjornar Pettersen, and Helge I Andersson. Calculations of the flow around rectangular shaped floating structures. In *ICWE 2011, 13th International Conference on Wind Engineering, Amsterdam*. Norwegian University of Science and Technology, 2011.

Florian R. Menter. Zonal two equation kappa-omega turbulence models for aerodynamic flows - nasa-tm-111629. In *24th Fluid Dynamics Conference, Orlando, Florida*. Sponsoring Organization: NASA Ames Research Center, 1993.

W. Rodi. Comparison of les and rans calculations of the flow around bluff bodies. *J. Wind Eng. Ind. Aerodyn.*, 71: 55–75, 1997. ISSN 0167-6105.

Tsan-Hsing Shih, William W. Liou, Aamir Shabbir, Zhigang Yang, and Jiang Zhu. A new k-epsilon eddy viscosity model for high reynolds number turbulent flows. *Computers and Fluids*, 24(3):227–238, 1995. ISSN 00457930. doi: 10.1016/0045-7930(94)00032-T.

Advisor: Håvard Holm

NTNU

NTNU