Numerical investigation of 3D flow around tandem cylinders

NTNU
Department of Marine Technology

Ragnhild Kalvig, NTNU
Supervisor: Bjørnar Pettersen, NTNU

Introduction

Circular cylinders are frequently encountered in the marine offshore industry. Examples are risers used for drilling and bringing hydrocarbons to the surface, the hull of a SPAR-platform, pipelines on the seabed, fish nets in fish farms, suction piles and the columns of a semi submersible. Examples of applications with two or more circular cylinders are multiple risers expanded from the side of a rig, the legs of an offshore platform and dual pipelines. In this work, flow around tandem cylinders in a free stream is investigated.

Objective

Flow around a single circular cylinder have been thoroughly investigated throughout the times. Less investigated is the flow around tandem cylinders. The downstream cylinder located in the wake of the upstream cylinder will see a very turbulent incoming flow which will induce large forces on this cylinder. It is very interesting to observe this flow and try to understand what is happening, to later be able to optimize design to avoid large movements on the downstream cylinder. The present work is based on the work by PhD candidate Mia Abrahamsen-Prsic who has done research on tandem cylinders in vicinity of a wall, (Abrahamsen-Prsic, 2015).

Numerical simulations give us the opportunity to investigate the details of the flow by numerically solving the Navier-Stokes equations. Not only on the cylinders themselves, but in every point in the computational domain. Areas such as the wake and boundary layer are especially interesting to look at. OpenFOAM is an open source software that is used for the numerical simulations in this work. The numerical model was given by Abrahamsen-Prsic. The computational grid is developed from the grid used in Abrahamsen-Prsics work, but extended to fit a free stream and not a wall.

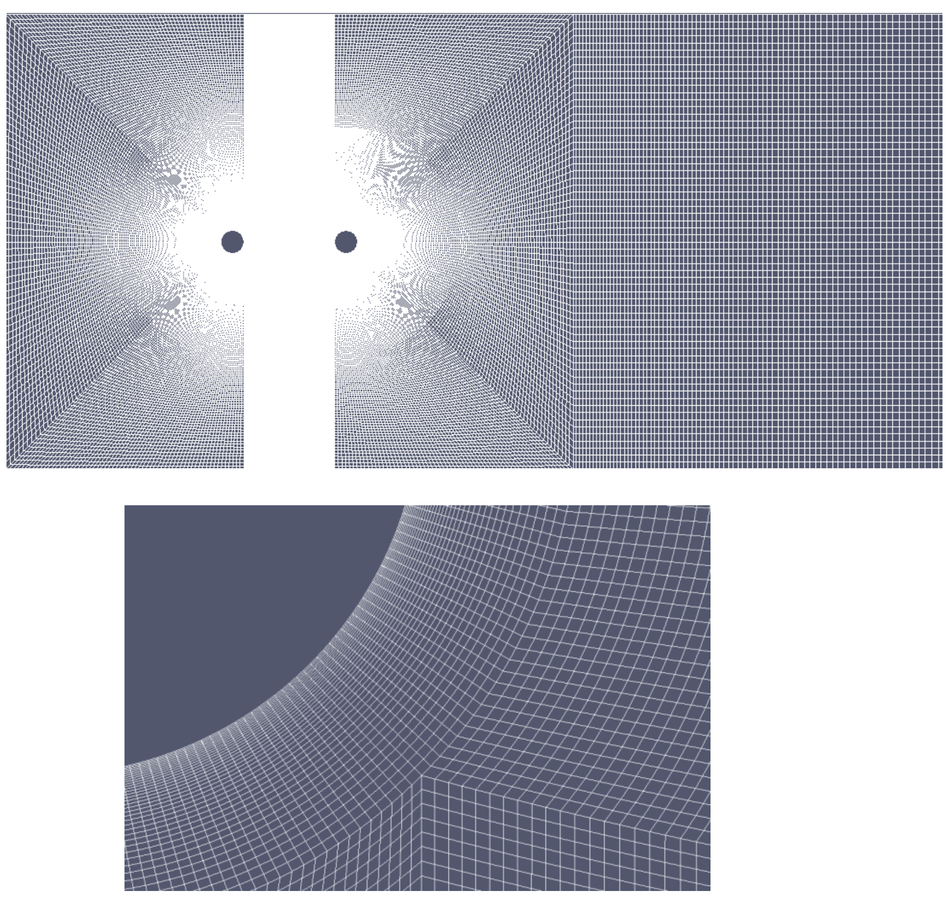

Figure 1. Computational grid with boundary layer details

Numerical model

Turbulence is a challenge in numerical simulations. It has to be modeled. In the present work a Large Eddy Simulation (LES) with a Smagorinsky subgrid scale model is used. In this turbulence model, only the smallest eddies has to be modeled by the subgrid scale model while the larger eddies are resolved. Some aspects of the simulation are:

- •Reynolds number, Re=13100 (subcritical flow regime)
- •Spacing between cylinders, S/D=5, D=diameter
- •Domain size: 45D x 20D x 4D (x, y, z)
- •Number of elements: 14.3 million
- •Number of elements in boundary layer: 40
- •Number of elements on the circumference: 254
- •The boundary conditions for the simulation are given in table 1. The sides of the domain have cyclic boundary conditions to simulate an infinite cylinder.

Velocity profiles were plotted at different locations with ticks at grid points to show that the number of elements in the boundary layer was sufficient.

Table 1. Boundary conditions

Patch	Velocity BC	Pressure BC
Top and Bottom	$\frac{\partial U}{\partial z}=0, \frac{\partial U}{\partial x}=0, v=0$	$\frac{\partial p}{\partial x_i}=0$
Inlet	$v=w=0, u=1.31$	$\frac{\partial p}{\partial x_i}=0$
Outlet	$\frac{\partial U}{\partial x_i}=0$	$p=0$
Cylinders	$U=0$	$\frac{\partial p}{\partial x_i}=0$

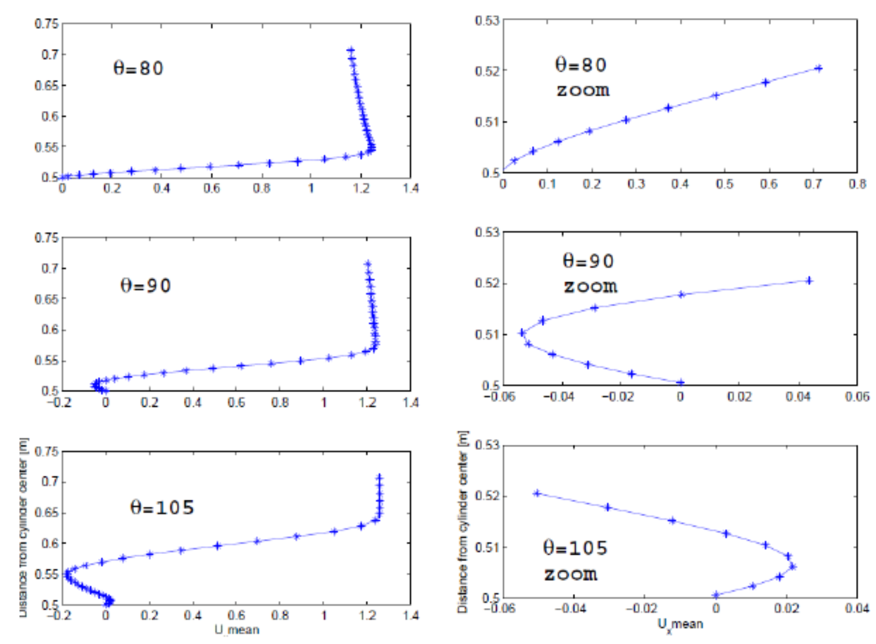

Figure 2. Velocity profiles with details of the near wall

Presentation & Evaluation of Results

The results are presented in form of vorticity distribution in the wake to show vortex formation and shedding, velocity distributions in the wake, pressure distributions over the cylinder circumference, drag- and lift coefficients, Strouhal number and surface streamlines to show separation. The vorticity distributions showed that vortices roll up and shed behind the upstream cylinder and impinges the downstream cylinder. This is called jump flow (Igarashi, 1981) and is due to the large spacings between the cylinders.

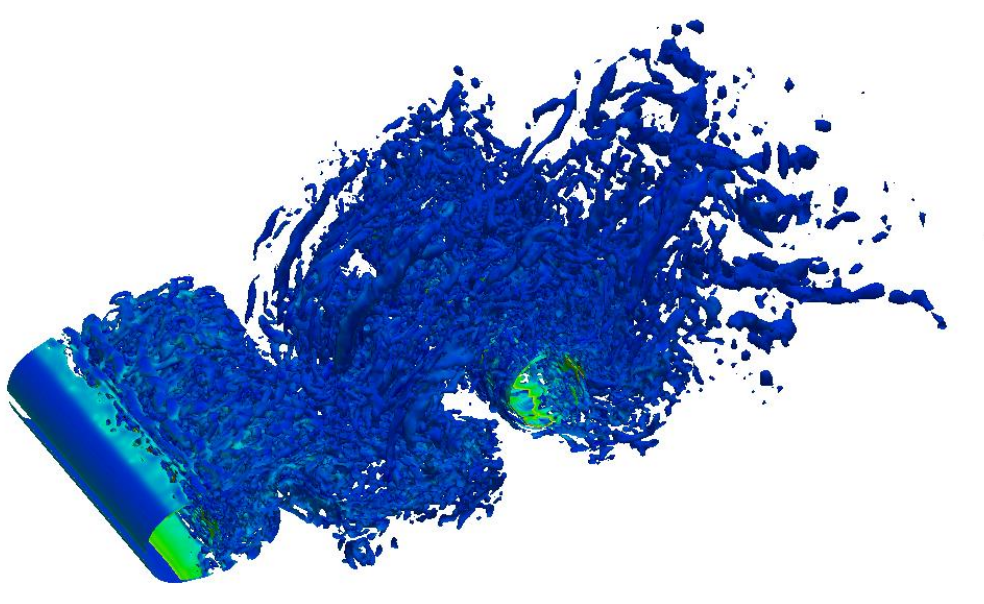

Figure 3 Vorticity contours visualized by the Q-criterion

Presentation & Evaluation of Results

The drag- and lift coefficients for the two cylinders are shown in figure 4.

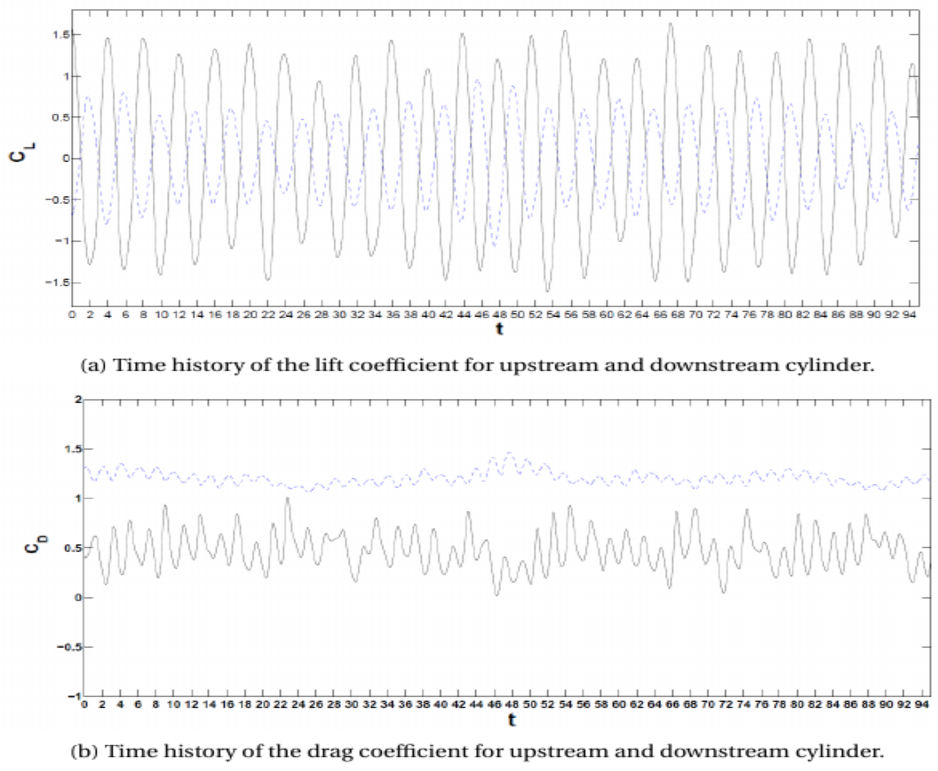

Figure 4. Drag- and lift coefficients for the cylinders. Solid line: Downstream cylinder, dashed line: upstream cylinder.

Figure 4a shows that the lift coefficient of the downstream cylinder is about twice as large as that of the upstream cylinder. This is due to the synchronous vortex impingement and formation of a new vortex on the downstream cylinder. The drag coefficient for the downstream cylinder shows the turbulent incoming flow by the large fluctuations and irregularities.

Figure 5 shows velocity streamlines on the surfaces of the cylinder. The upstream cylinder has a very even separation line while the downstream cylinder is characterized by very irregular flow.

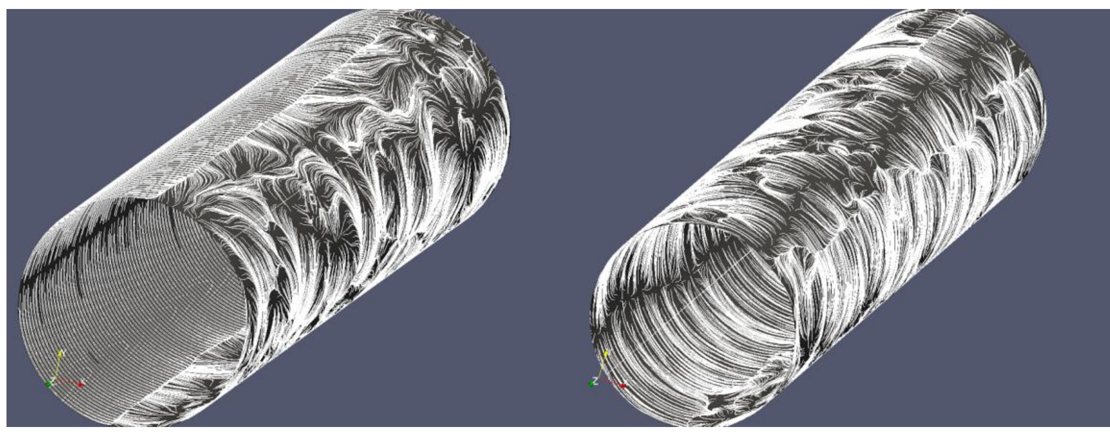

Figure 5. Velocity streamlines around the cylinders showing separation.

Conclusion

In general the results agreed well with results from previous research of tandem cylinders with similar flow conditions, both numerical and experimental. There were some discrepancies between experimental results and the numerical results, especially for the downstream cylinder. This could be due to the turbulence model not being able to pick up all the details of the turbulent flow around the downstream cylinder. As expected, the downstream cylinder was characterized by e very turbulent flow which led to large fluctuations of the results on the downstream cylinder.

References

Igarashi, T. (1981). Characteristics of the flow around two circular cylindersarranged in tandem: 1st report. Bulletin of JSME , 24(188):323-331.

Abrahamsen-Prsic, M., Ong, M.C., Pettersen, B., and Myrhaug, D. (2015). Large Eddy Simulations of flow around tandem cylinders in the vicinity of a wall.