Master Thesis in Marine Technology - 2013
Large Eddy Simulations of viscous flow around a circular cylinder

Sunniva S. Thingbø

sunnivt@stud.ntnu.no
Supervisor: Professor Bjørnar Pettersen

NTNU – Trondheim
Norwegian University of Science and Technology

Problem and scope of work

Flow around circular cylinders is an important topic in marine hydrodynamics. Many offshore structures are of cylindrical shape, such as risers and pipelines. The use of these structures have been increased, and they will be used also in the future. It is important to understand the flow field around these structures and the forces acting on them.

In this thesis, three-dimensional modelling of the flow around a circular cylinder is accomplished. Two cases are considered. The first (Case 1) i a cylinder in a steady, uniform current subjected to a Reynolds number of 3900. The second case (Case 2) is a cylinder in the vicinity of a rigid wall, also at Reynolds number 3900. For the second case, both a boundary layer velocity profile (Case 2a) and a uniform inlet velocity profile (Case 2b) is simulated. The gap-to-diameter ratio e/D is set to 0.2 for Case 2. Large eddy simulations (LES) with Smagorinsky subgrid scale model are applied to simulate the flow. LES have ability to resolve fine structures in the turbulent wake of the cylinder. A mesh convergence study is accomplished for Case 2a.

Introduction

Cylinder in steady current at Reynolds number of 3900

Simulations of flow around a circular cylinder are challenging, and thus verification of the CFD code is important. Therefore, the benchmark case of flow around a circular cylinder at Reynolds number equal to 3900 have been performed. This thesis focuses on three-dimensional modelling of flow around a circular cylinder in steady current. The details of the flow around the body and in the near wake are analysed.
The benchmark case of flow over a circular cylinder at Reynolds number equal to 3900 has been discussed by many over the years. The current research is mainly based on LES. Numerical studies are carried out by among others [Breuer, 1998], [Prsic et al., 2012], [Tremblay et al., 2000], [Parnaudeau et al., 2008], [Franke and Frank, 2002] and [Li, 2011].

Cylinder in the vicinity of a rigid wall at Reynolds number of 3900

Flow around a circular cylinder close to a rigid wall is a topic of high interest in the marine technology research community. Challenges which can be addressed to this problem are for example free-spanning subsea pipelines and marine risers in the vicinity of the seabed. The risers and pipelines are subjected to a continuous strain due to exposure of wind and waves. The safety for the structures can be improved if the flow around them and the forces acting on them are better understood.
When a pipeline is placed on an erodible seabed, scour may occur below the pipe due the flow. This may lead to suspended spans of the pipeline where the pipe is suspended above the seabed with a small gap, usually in the range from O(0.1D) to O(1D). It is therefore important to know what kind of changes that take place in the flow around the pipe and forces acting on it. [Sumer and Fredsøe, 1997]
Several experimental studies have been carried out to investigate the flow around pipelines close to the seabed in the subcritical flow regime, which ranges from 300 $Re < 3 \cdot 10^5$. Examples of experiments were a wind tunnel was used are [Lei et al., 1999] and [Zdravkovich, 1985]. Particle image velocimetry (PIV) are also used, e.g by [Oner et al., 2008], [Price et al., 2002] and [Wang and Tan, 2008].
Numerical studies have been carried out by among others [Ong et al., 2008], [Ong et al., 2010], [Sarkar and Sarkar, 2010], [Lei et al., 2000] and [Prsic et al.,]. None of the numerical studies or experiments are performed for Reynolds number 3900, but the flow regime is the same. LES is considered a promising tool based on the previously mentioned research.

References

[Breuer, 1998] Breuer, M. (1998). Large eddy simulation of the subcritical flow past a circular cylinder: numerical and modeling aspects. *International Journal for Numerical Methods in Fluid*, Vol. 28:pp. 1281–1302.

[Franke and Frank, 2002] Franke, J. and Frank, W. (2002). Large eddy simulation of the flow past a circular cylinder at ReD = 3900. *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 90:pp. 1191–1206.

[Lei et al., 2000] Lei, C., Cheng, L., Armfield, S. W., and Kavanagh, K. (2000). Vortex shedding suppression for flow over a circular cylinder near a plane boundary. *Ocean Engineering*, Vol. 27:pp. 1109–1127.

[Lei et al., 1999] Lei, C., Cheng, L., and Kavanagh, K. (1999). Re-examination of the effect of a plane boundary on force and vortex shedding of a circular cylinder. *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 80:pp. 263–286.

[Li, 2011] Li, Y. (2011). Large eddy simulation of flow around a cylinder at Re=3900 using a CFD code. *Applied Mechanics and Materials*, Vols. 94-96:pp. 1707–1710.

[Oner et al., 2008] Oner, A. A., Kirkgoz, M. S., and Akoz, M. S. (2008). Interaction of a current with a circular cylinder near a rigid bed. *Ocean Engineering*, Vol. 35:pp. 1492–1504.

[Ong et al., 2008] Ong, M. C., Utnes, T., Holmedal, L. E., Myrhaug, D., and Pettersen, B. (2008). Numerical simulation of flow around a marine pipeline close to the seabed. In *Proceedings of the 31st International Conference on Coastal Engineering.*

[Ong et al., 2010] Ong, M. C., Utnes, T., Holmedal, L. E., Myrhaug, D., and Pettersen, B. (2010). Numerical simulation of flow around a circular cylinder close to a flat seabed at high Reynolds numbers using a k-ϵ model. *Costal Engineering*, Vol. 57:pp. 931–947.

[Parnaudeau et al., 2008] Parnaudeau, P., Carlier, J., Heitz, D., and Lamballais, E. (2008). Experimental and numerical studies of the flow over a circular cylinder at Reynolds number 3900. *Physics of Fluid*, Vol. 20:085101.

[Price et al., 2002] Price, S. J., Sumner, D., Smith, J. G., Leong, K., and Paidoussis, M. P. (2002). Flow visualization around a circular cylinder near to a plane wall. *Journal of Fluids and Structures*, Vol. 16:pp. 175–191.

[Prsic et al., 2012] Prsic, M., Ong, M. C., Pettersen, B., and Myrhaug, D. (2012). Large Eddy Simulations of three dimensional flow around a pipeline in a uniform current. In *Proceedings of the 31st International Conference on Ocean, Offshore and Arctic Engineering.*

[Prsic et al.,] Prsic, M. A., Ong, M. C., Pettersen, B., and Myrhaug, D. Large eddy simulation of flow around a circular cylinder in the vicinity of a wall at Reynolds number of 13100. Not published.

[Sarkar and Sarkar, 2010] Sarkar, S. and Sarkar, S. (2010). Vortex dynamics of a cylinder wake in proximity to a wall. *Journal of Fluids and Structures*, Vol. 26:pp. 19–40.

[Sumer and Fredsøe, 1997] Sumer, B. M. and Fredsøe, J. (1997). *Hydrodynamics around Cylindrical Structures.* World Scientific Publishing Co. Pte. Ltd.

[Tremblay et al., 2000] Tremblay, F., Manhart, M., and Friedrich, R. (2000). LES of flow around a circular cylinder at a subcritical Reynolds Number with cartesian Grids. *Advances in LES of Complex Flows*, pages pp. 133–150.

[Wang and Tan, 2008] Wang, X. K. and Tan, S. K. (2008). Near-wake flow characteristics of a circular cylinder close to a wall. *Journal of Fluid and Structures*, Vol. 24:pp. 605–627.

[Zdravkovich, 1985] Zdravkovich, M. M. (1985). Forces on a circular cylinder near a plane wall. *Applied Ocean Research*, Vol. 7 (4):pp. 197–201.

Acknowledgements

This work is submitted to the Norwegian University of Science and Technology (NTNU) for partial fullfillment of the requirements for the degree og Master of Science in Marine Technology.

Numerical method

For the work, LES is chosen to simulate the flow. In a LES model, the larger scale turbulence is simulated directly and the subgrid scale turbulence is modelled. The subgrid scale turbulence is assumed not to be very case dependant, and is therefore modelled by a simple model: the Smagorinsky subgrid scale model.

The simulations are carried out using the software STAR-CCM+ and the supercomputer Vilje.

Case set-up

For all cases the Reynolds number, $Re = 3900$. The physical parameters are given in the table below. Structured mesh is applied for all cases.

The physical parameters for Case 1 and Case 2 are given:

Physical parameters for Case 1 and Case 2

U_∞ [m/s]	D [m]	ρ [kg/m^3]	μ [kg/ms]	Re [-]
0.39	0.1	1	10^{-5}	3900

Case 1

Case 1 consists of a cylinder in infinite fluid. The inflow velocity is steady and uniform. Figure 1 shows the computational domain in the XY-plane. The domain has an extension of $4 \cdot D$ in Z-direction. The type of boundary condition and values for the physical parameters which have been used are

- At the inlet: Velocity Inlet. Uniform constant inlet velocity: $U_\infty = 0.39$ m/s
- At the outlet: Pressure outlet. The pressure is set to zero.
- Top and bottom boundary: Symmetry plane. The shear stress is zero.
- The two sides which defines the spanwise extent: Periodic interface. The fluxes which cross one boundary are transformed and applied to the other.
- Cylinder wall: Wall. There are no velocityes through the wall and the tangential velocity is zero.

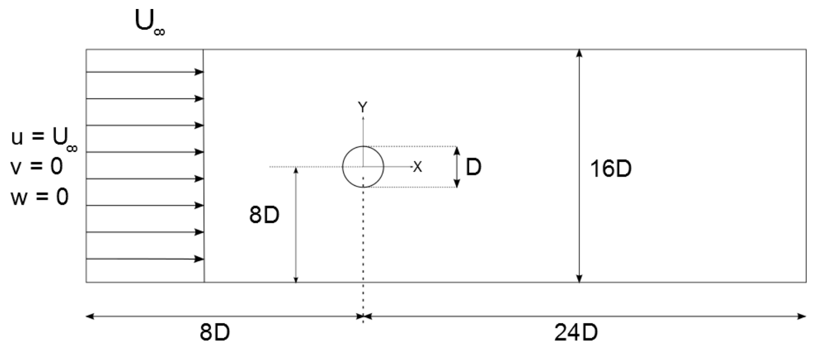


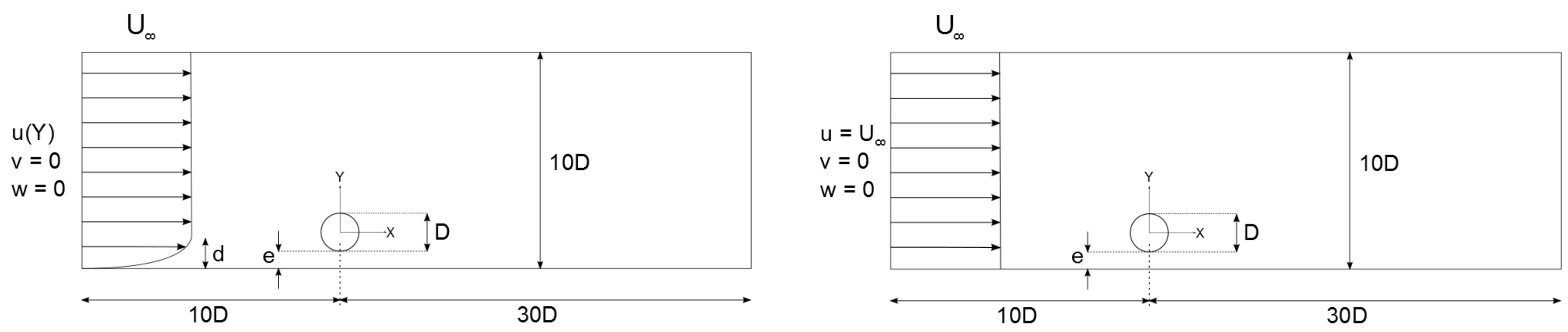
Figure 1: Set-up for Case 1

Case 2

For Case 2, the gap, e, between the cylinder wall and the rigid wall is equal to $0.2 \cdot D$. The set-up for Case 2 is illustrated in Figure 2. The type of boundary condition and values for the physical parameters which have been used are

- At the inlet: Velocity Inlet. $u(Y)$ given in below, $v = w = 0$
- At the outlet: Pressure outlet
- Top boundary: Symmetry plane
- Seabed: Wall
- The two sides which defines the spanwise extent: Periodic interface
- Cylinder wall: Wall

The velocity profile $u(y)$ for Case 2a is logarithmic and the boundary layer thickness δ is $0.48 \cdot D$. For Case 2b, $u(Y) = u = U_\infty$.



(a) Case 2a (b) Case 2b

Figure 2: Set-up for Case 2

Results and conclusion

Some of the simulations are still running on the supercomputer Vilje, and the postprocessing is not finished for the remaining. Their computational time is so far approximately 30 days. The results will be compared to [Prsic et al., 2012] and [Prsic et al.,]. The results will be given as space- and time-averaged velocity profiles for the instream and crossflow velocity profiles in the cylinder wake. Also the hydrodynamical parameters drag coefficient C_D, root mean square of the lift coefficient C_{Lrms}, pressure coefficient C_p and Strouhal number St will be calculated for each case.

The time averaged velocity contour lines for Case 1 and one of the meshes in Case 2a can be seen in Figure 3 and 4.

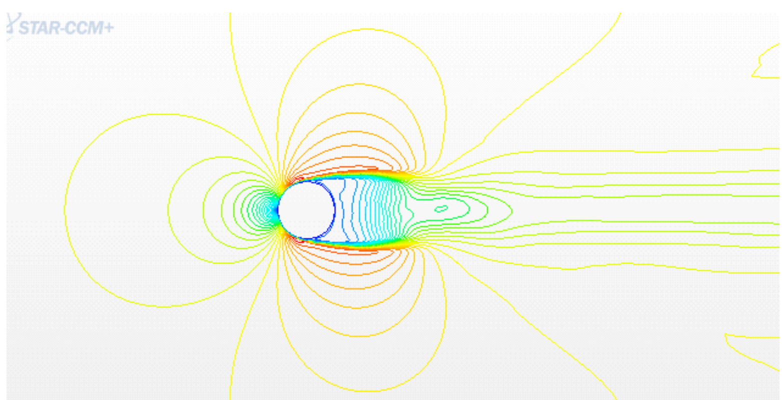


Figure 3: Time averaged velocity contour lines for Case 1 in the centre plane of the cylinder

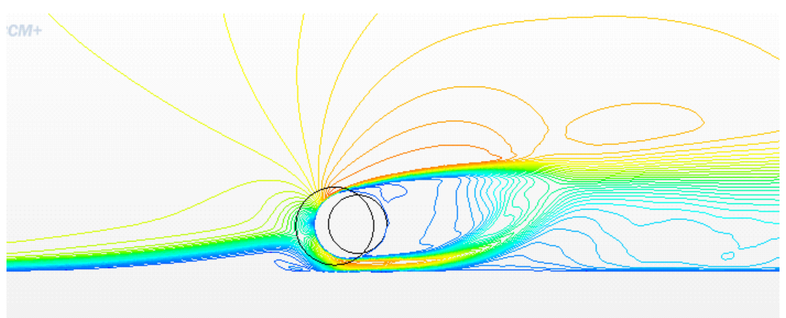


Figure 4: Time averaged velocity contour lines for Case 2 in the centre plane of the cylinder