LARGE EDDY SIMULATIONS OF FLOW AROUND A CIRCULAR CYLINDER IN THE VICINITY OF A WALL

Mia Prsic [1], Muk Chen Ong [2], Bjørnar Pettersen [1], Dag Myrhaug [1]
[1] *Department of Marine Technology, Norwegian University of Science and Technology, NO-7491 Trondheim, Norway*
[2] *Norwegian Marine Technology Research Institute (MARINTEK), NO-7450 Trondheim, Norway*

Abstract: The flow around a circular cylinder in the vicinity of a rigid wall at Reynolds number 13100 is simulated using Large Eddy Simulations (LES) with Smagorinsky subgrid scale model. The simulations with gap-to-diameter ratios (G/D) of 0.2, 0.6 and 1 are carried out in order to investigate the modifications of the flow field and vortex shedding due to the presence of the wall. Influence of the incoming boundary layer profile is investigated through two simulations with the logarithmic boundary layer inlet profile of thicknesses 0.48D and 1.6 D and a simulation with a uniform inlet profile. The velocity field around and in the cylinder wake as well as the hydrodynamic values and the pressure distribution on the cylinder surface are used to understand the physics of the flow and to separate the influences of the wall proximity and the shear layers interaction.

Keywords: circular cylinder in the vicinity of a wall; Large Eddy Simulations; Smagorinsky turbulence model; gap-to-diameter ratio; boundary layer.

INTRODUCTION

Flow around a circular cylinder in the vicinity of a rigid wall is a topic of high interest in the marine technology environment. Free-spanning sub-sea pipelines, marine risers in the vicinity of the sea bed and circular elements of various bottom mounted marine structures are subjected to continuous strain due to the exposure to current and waves. To improve the safety of such structures, it is important to understand the flow around and the forces exerted on them.

In order to do so, Large Eddy Simulations (LES) are utilized to simulate the fully three-dimensional (3D) flow around a circular cylinder in the vicinity of a rigid wall. Simulations are made for the intermediate Reynolds number, Re = 13100 ($Re = U_\infty D/\upsilon$, where U_∞ is the free stream velocity, D is the cylinder diameter and υ is the kinematic viscosity of the fluid), corresponding to operational conditions of sub-sea pipelines. Several physical parameters are investigated: the influence of the gap-to-diameter ratio (G/D, where G is the distance between the cylinder and the wall), the influence of the boundary layer thickness (δ/D, where δ is the boundary layer thickness at the inlet) and the influence of the boundary layer profile.

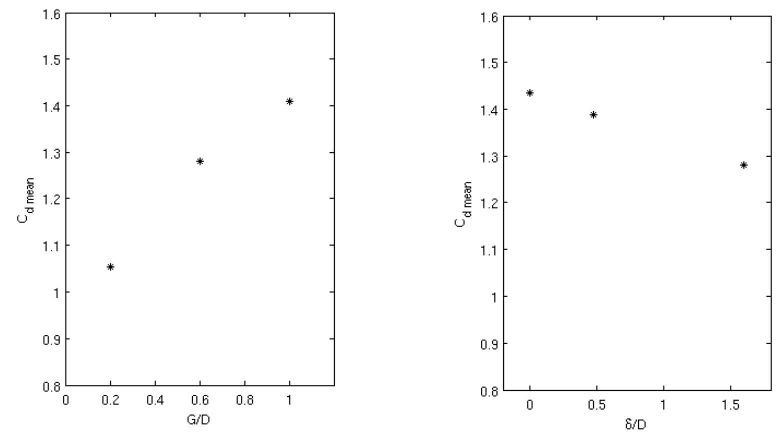
CODE VALIDATION: CYLINDER IN AN INFINITE FLUID AT Re = 13100

The code validation is performed by comparison of the LES results to the numerical and experimental results for a simpler and more thoroughly investigated case – flow around a circular cylinder in an infinite fluid subjected to the uniform flow with the same Re = 13100. The results are analysed through the standard hydrodynamic parameters and the details of the flow in the cylinder wake. They are further compared with the published numerical results of Breuer [2] and Tremblay et al. [10], with both the numerical and experimental results of Breuer [3] and Cantwell and Coles [4]. It was found that the present LES results agree well with these previous results.

RESULTS: CYLINDER IN THE VICINITY OF A WALL AT Re = 13100

The main purpose of the present paper is to investigate the flow around a circular cylinder in the vicinity of a wall. In order to simulate a natural flow profile near the sea bed, the logarithmic inlet profile is applied to the domain of 40D length, where the cylinder is placed at 10D from the inlet. Domain width of 4D is chosen due to its capability of capturing the 3D flow. Convergence studies are carried out in order to minimize the influence of the mesh element size and the time-step. The results are investigated through the time- and space-averaged drag and lift coefficients, the pressure distribution on the cylinder surface, and the details of the velocity fields in the cylinder wake.

The influence of several physical parameters is of interest in this paper. The effect of G/D is explored by choosing the values 0.2, 0.6 and 1. Figure 1 a presents $\overline{C}_d$ (the mean drag coefficient, averaged over time for the fully developed flow) versus G/D. It appears that the drag coefficient decreases as the cylinder approaches the wall. The results agree with the experimental data of Lei et al. [5] and Taniguchi and Miyakoshi [9] and to the numerical results of Ong et al. [7] and Liang and Cheng [6]. The influence of δ/D is investigated by choosing the values δ/D = 0, 0.48 and 1.6, where δ/D = 0 corresponds to a uniform inlet profile (see Figure 1 b). The results agree qualitatively with those presented by Lei et al. [5] and will in detail be compared with the published results of of Bearman ad Zdravkovich [1], Zdravkovich [11], and Sarkan and Sarkar [8]. The effects of G/D and δ/D will be discussed through the physical behaviour of the flow in the wake, and the forces and the pressure on the cylinder.



a) b)

Figure 1 a) $\overline{C}_d$ versus G/D for δ/D = 1.6; b) $\overline{C}_d$ versus δ/D for G/D = 0.6.

References

[1] Bearman, P. W., Zdravkovich, M. M., 1978. Flow around a circular cylinder near a plane boundary. Journal of Fluid Mechanics. 89, 33-47.

[2] Breuer, M., 1998. Large eddy simulation of the subcritical flow past a circular cylinder: numerical and modeling aspects. International Journal for Numerical Methods in Fluids, 28, 1280-1302.

[3] Breuer, M., 2000. A challenging case for large eddy simulation of high Reynolds number circular cylinder flow. International Journal of Heat and Fluid Flow, 21, 648-654.

[4] Cantwell, B., Coles, D., 1983. An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder. Journal of Fluid Mechanics, 136, 321-374.

[5] Lei, C., Cheng, L., 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, 80, 263-286.

[6] Liang, D., Cheng, L., 2005. Numerical modeling of flow and scour below a pipeline in currents; Part 1. Flow simulation. Coastal Engineering, 52, 25-42.

[7] Ong, M. C. ,Utnes, T., Holmedal, L. E., Myrhaug, D., Pettersen, B., 2012. Near-Bed Flow Mechanisms Around a Circular Marine Pipeline Close to a Flat Seabed in the Subcritical Flow Regime Using a k-ϵ Model. Journal of Offshore Mechanics and Arctic Engineering, v 134, n 2, p 021803.

[8] Sarkar, S., Sarkar, S., 2010. Vortex dynamics of a cylinder wake in proximity to a wall. Journal of Fluids and Structures, 26, 19-40.

[9] Taniguchi, S., Miyakoshi, K., 1990. Fluctuating fluid forces acting on a circular cylinder and interference with a plane wall. Experiments in Fluids, 9, 197-204.

[10] Tremblay, F., Manhart, M., Friedrich, R., 2000. DNS of flow around the circular cylinder at subcritical Reynolds number with Cartesian grids. Proceedings of the 8th European Turbulence Conference, EUROMECH, Barcelona, Spain. 659-662.

[11] Zdravkovich, M. M., 1985. Forces on a circular cylinder near a plane wall, Applied Ocean Research, Vol 7, No. 4, 197-201.