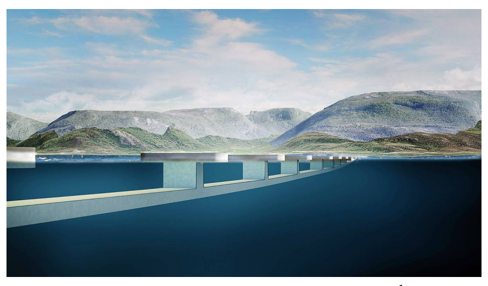
Unsteady RANS simulations of flows around rectangular cylinders with different aspect ratios at high Reynolds number

Sindre Misund Dahl
Norwegian University of Science and Technology

Supervisors: Professor Dag Myrhaug
Dr. Muk Chen Ong

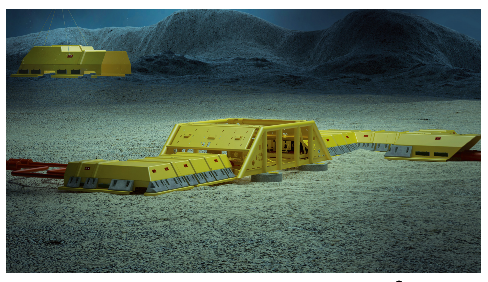
Motivation

- Flow around square and rectangular structural components are often seen in offshore and subsea engineering, e.g. flow around submerged floating tunnels, the columns of a tension-leg platform, remotely operated vehicles (ROV) and subsea modules.
- It is hard and expensive to achieve high Reynolds number flow conditions in laboratory testing. Computational Fluid Dynamics (CFD) is therefore an attractive alternative for engineering design.
- Flows around rectangular cylinders with various aspect ratios have been an important topic in the scientific community with realizations from both experiments and numerical simulations. Yet, not many numerical results are published at high Reynolds numbers.



Submerged floating tunnel[1]

Cross-section of submerged floating tunnel[2]



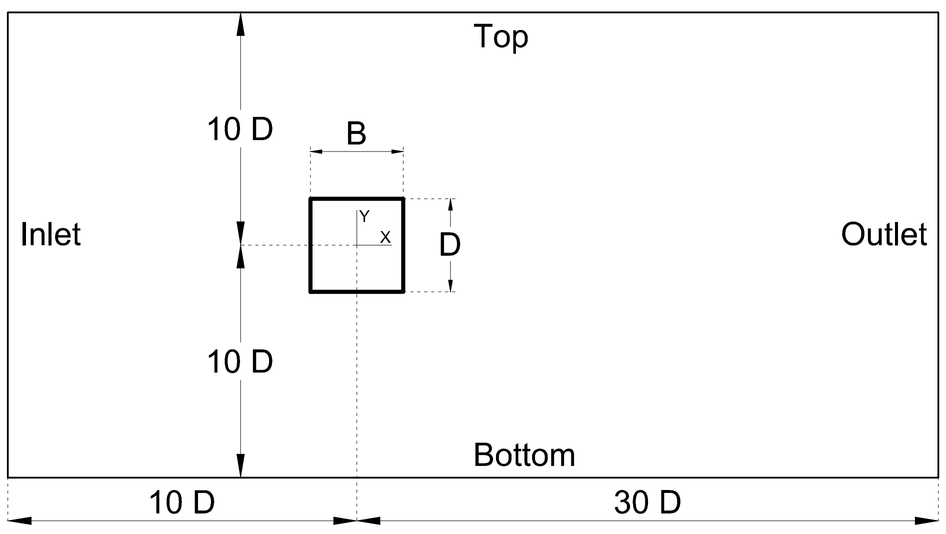
Subsea proctection covers[3]

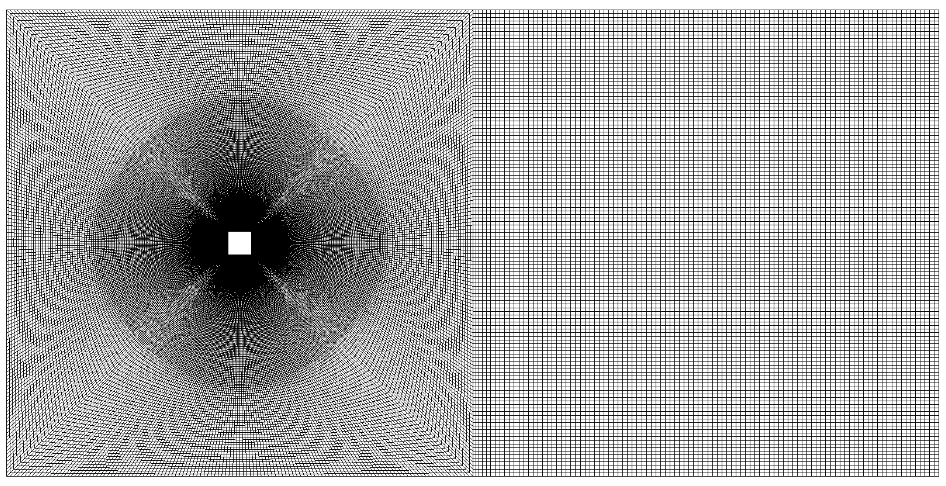
Objectives

- CFD is used to evaluate the hydrodynamic quantities, such as drag and lift forces, pressure distribution and flow structure around rectangular cylinders with different aspect ratios at high Reynolds number.
- The present numerical results are compared with available experimental results.

Modelling

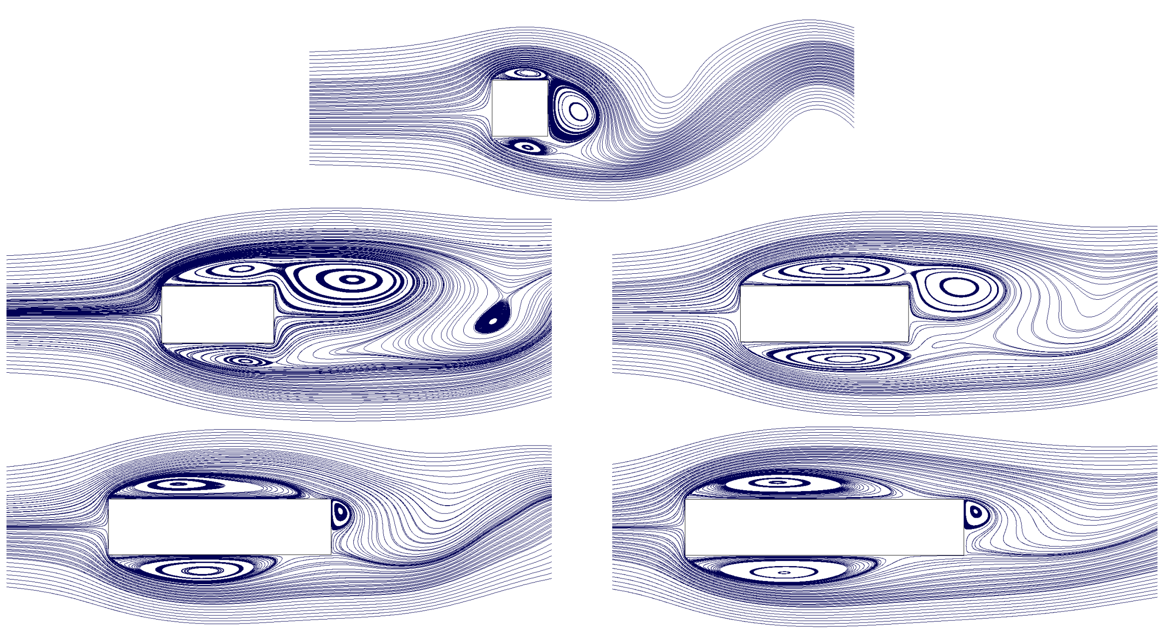
- Two-dimensional Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations are used together with the standard high Reynolds number $k - \varepsilon$ turbulence model.
- The open source CFD code *OpenFOAM* is adopted in this study.
- The geometric size of the computational domain and a typical mesh are shown below.



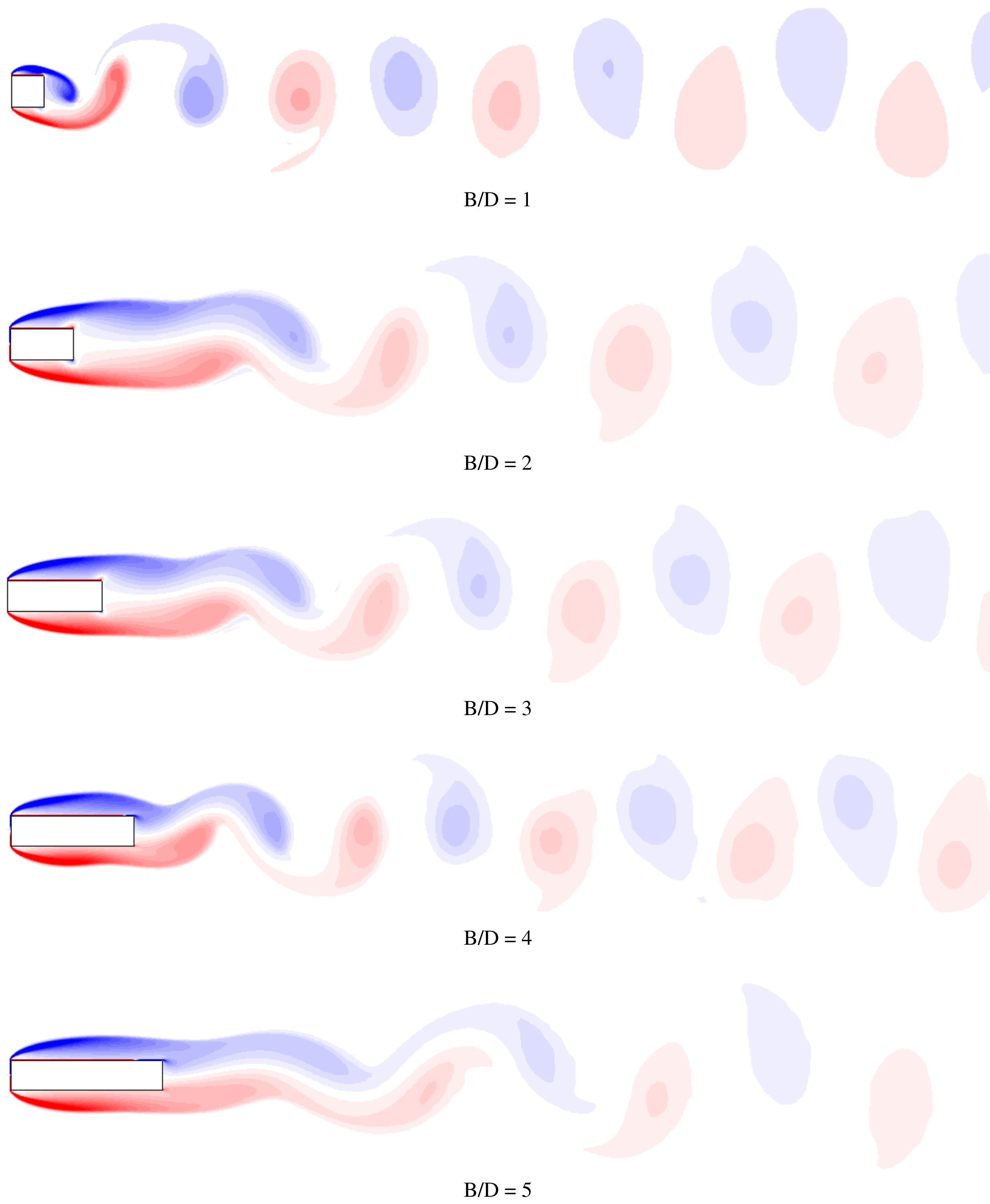


Results

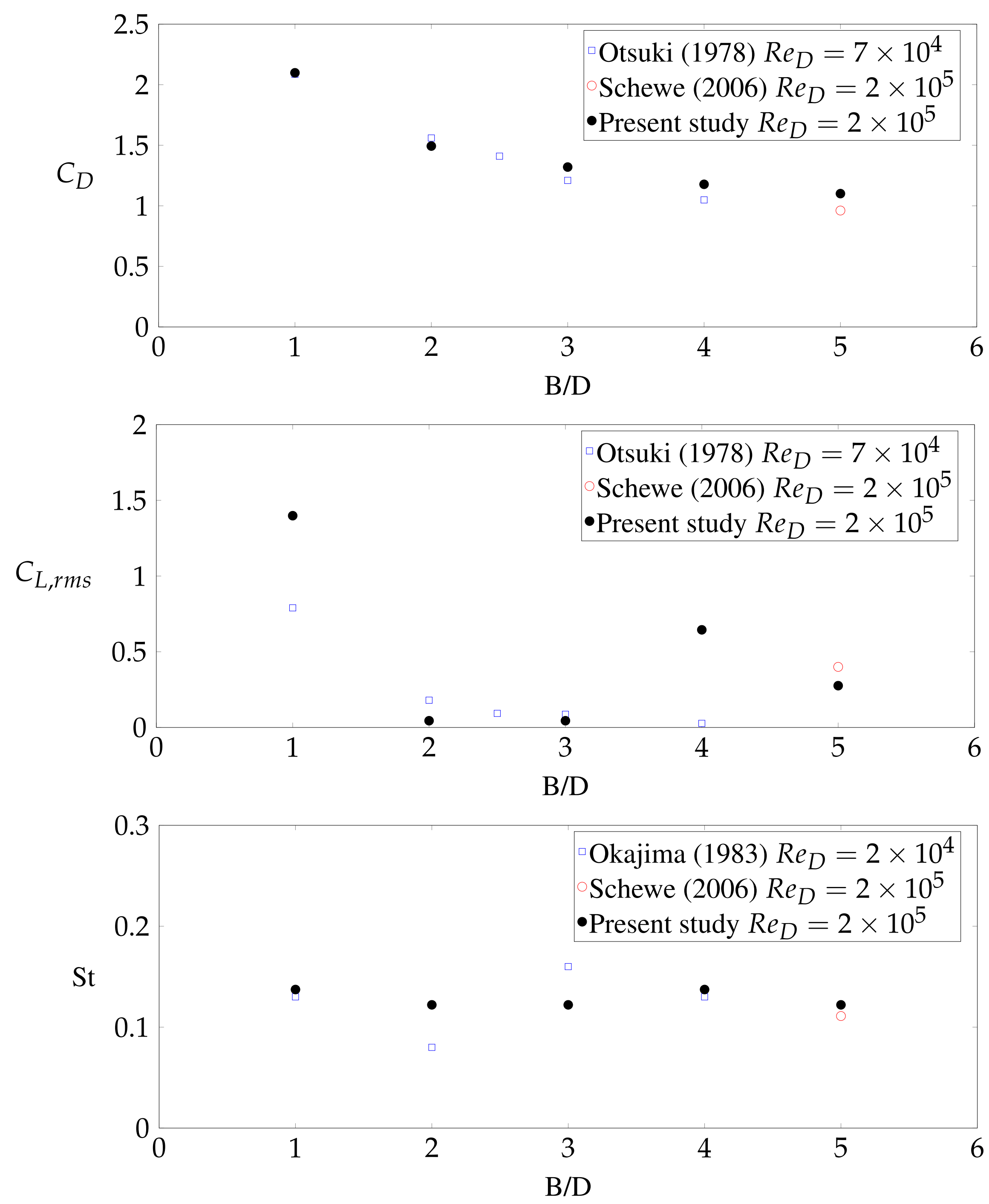
- All simulations have been performed at Reynolds number $Re_D = 2 \times 10^5$ ($Re_D = \frac{U_\infty D}{\nu}$, where U_∞ is the inlet velocity, D is the cylinder height and ν is the kinematic viscosity of the fluid)
- Grid and time step convergence studies have been performed.
- The mean drag coefficient ($\overline{C_D}$), the root-mean-square (rms) value of the lift coefficient ($C_{L,rms}$) and the Strouhal number (St) have been predicted, and the present predicted hydrodynamic quantities are in reasonably good agreement with the published experimental results.
- The flow structures (streamlines and vorticity contours) are also shown.



Streamlines from flow around rectangular cylinders with aspect ratios 1 to 5



Vorticity contours of the flows around rectangular cylinders with aspect ratios 1 to 5



The effect of different aspect ratio on $\overline{C_D}$, $C_{L,rms}$ and St obtained from the present study and published experimental results.

Summary

- It appears that the present CFD simulations are able to yield reasonably good agreement with the published experimental results.
- More experimental data are required for a further validation study.
- Overall, it appears that the results from the present study should be useful for engineering design purposes.

References

[1] Taken from http://www.tu.no/incoming/2012/11/22/1200011501.jpg/alternates/w1366f/1200011501.jpg
[2] Taken from http://www.vegvesen.no/Vegprosjekter/ferjefriE39/Illustrasjonar/_image/449053.png?_encoded=2f66666666666678302f35382f&_ts=13d5e5b2380
[3] Taken from http://www.highcomp.no/wp-content/uploads/2013/03/CAD_field.jpg
Okajima, A. (1983), *Journal of Wind Engineering* **17**, 1-19 (in Japanese).
Otsuki, Y., Washizu, K. & Ohya, A. (1978), *Proceedings of the Fifth Symposium on Wind Effects on Structures*, 169-176
Schewe, G. (2006), *Proceedings of the International Conference on Bridges*, Dubrovnik, Croatia, 351-358.