

Numerical simulation of flow around two side-by-side square cylinders with horizontal offsets mounted on the seafloor

Yukun DAI, Norwegian University of Science and Technology
Supervisors: Pro.Dag Myrhaug, Dr.Lars Erik Holmedal and Dr.Hong Wang

Introduction

With increasing humans demand of energy and resources for economy, oil and gas in land can not satisfy the need of industrial development. People start to explore the resources in the deep ocean. For this purpose, Countless complex offshore structures have been constructed in past several decades, such as oil drilling platforms and wind turbines. Most of these offshore structures have direct link to the foundation on sea bed, thus when consider the stability of the structures, the effect of sea bed should be accounted. A big number of models for flow around foundation on sea bed is discussed in the literature. However, most of them consider smooth sea bed and perfect turbulent boundary layer. This assumption does not account sea bed roughness due to the existence of sands and dunes, because complex terrain of sea bed alters the inflow and indirectly change the hydrodynamic force on the structure. A comprehensive understanding of above interactions between sea bed and flow is vital for a reliable understanding of the problem of flow around structures.

A common simplified model of sea bed roughness is cube. Cube is one of the simplest geometries that has bluff corner, while bluff corner is the common feature of most seabed's geometry and is the major part that alter the flow. Compared with cylinder, the flow passing cube separates at fixed positions on the corner, and not periodic but steady vortices and recirculations are produced after that. Sea bed roughness is also continuous, different roughness can be simulated by multiple cubes with different interval. For example, a smooth sea bed is similar to cubes with infinite interval; highly rough sea bed is modelled by multiple cubes with small interval. Therefore, multiple cubes is a reasonable simplified model for sea bed roughness.

Besides, sea bed roughness is tiny structure, boundary layer on the sea bed is generally comparable to the structures in height. Thus Reynold number calculated by the dimension of the roughness is very low, a low-Re model should be applied, which contrast to most of the simulations about flow around cube that use high-Re model.

Objectives

This study focus on the interaction between two cubes and conducts a series of numerical simulations in 2D. Due to the lack of high performance computer, Simulation with LES or DNS is not affordable, even 2D simulation is pretty hard to achieve. Since the models are simple geometry, RANS method can produce result accurate enough, thus RANS method with 2D model is applied in this study. the behavior of flow passing single square cylinder was studied first, then its result would be compared with flow passing double square cylinders. The results of different Interval between square cylinders would also be compared.

Methodology

To simulate the turbulent flow, RANS equation with $k - \omega$ turbulence model is applied, the basic equations for this model are

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = 2\nu_T S_{ij} \frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \beta_k k \omega$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = 2\nu_T S_{ij} \frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] - \beta_\omega \omega^2$$

Low Reynold number flow with $Re = 4000$ is set on the inlet, the length of domain is large enough to avoid boundary effect on the simulation, as shown in figure 1.

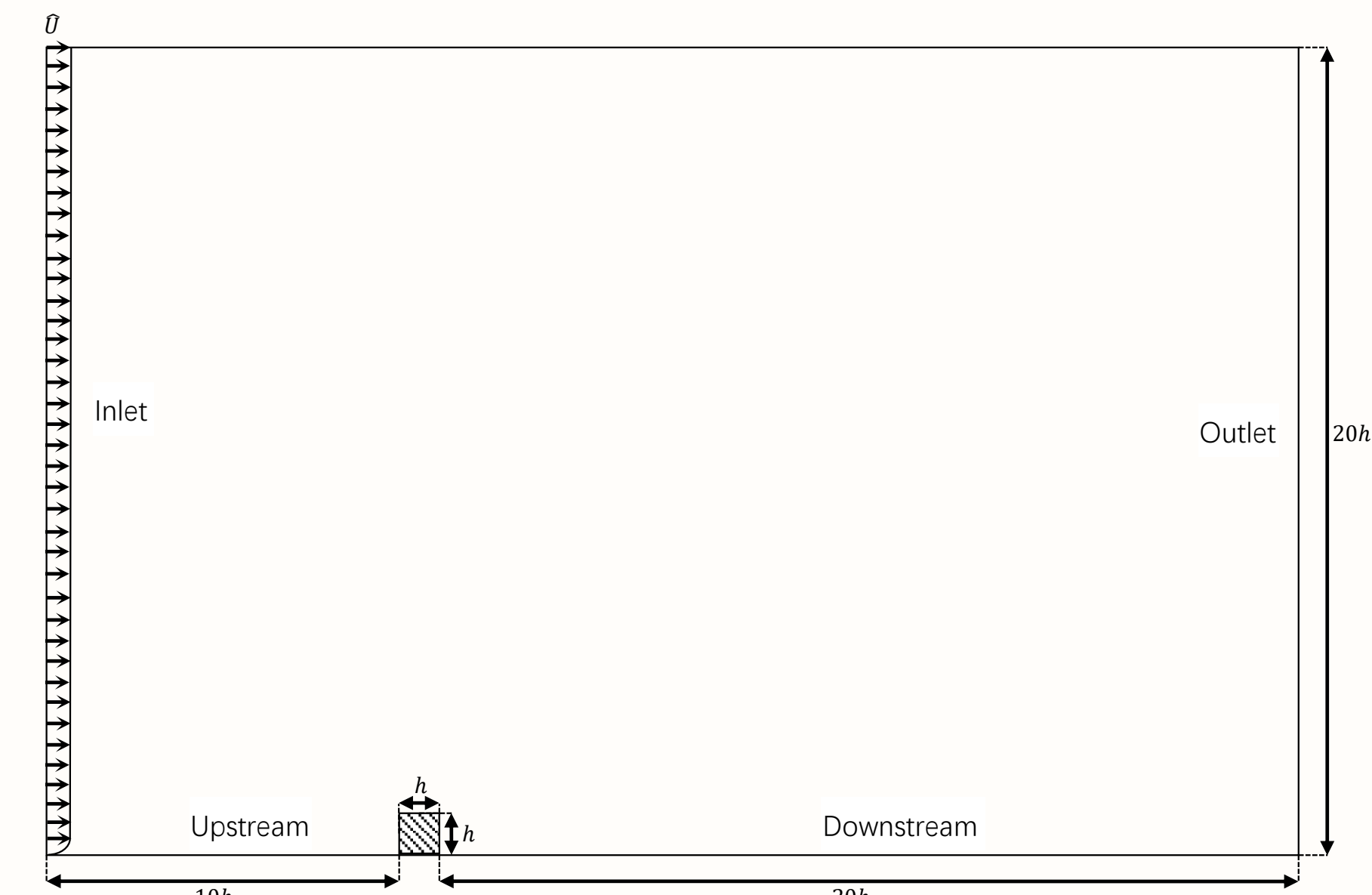


Figure 1: Set of domain

CFD software OpenFOAM is used to solve the equations. OpenFOAM is a open source software that suitable for various type of CFD problem. The solver can be developed by the user himself and it is free to change the source code. OpenFOAM is based on finite volume method, and is capable to simulate potential flow and turbulent flow. It has advantages of high efficiency and freedom in programming.

Results

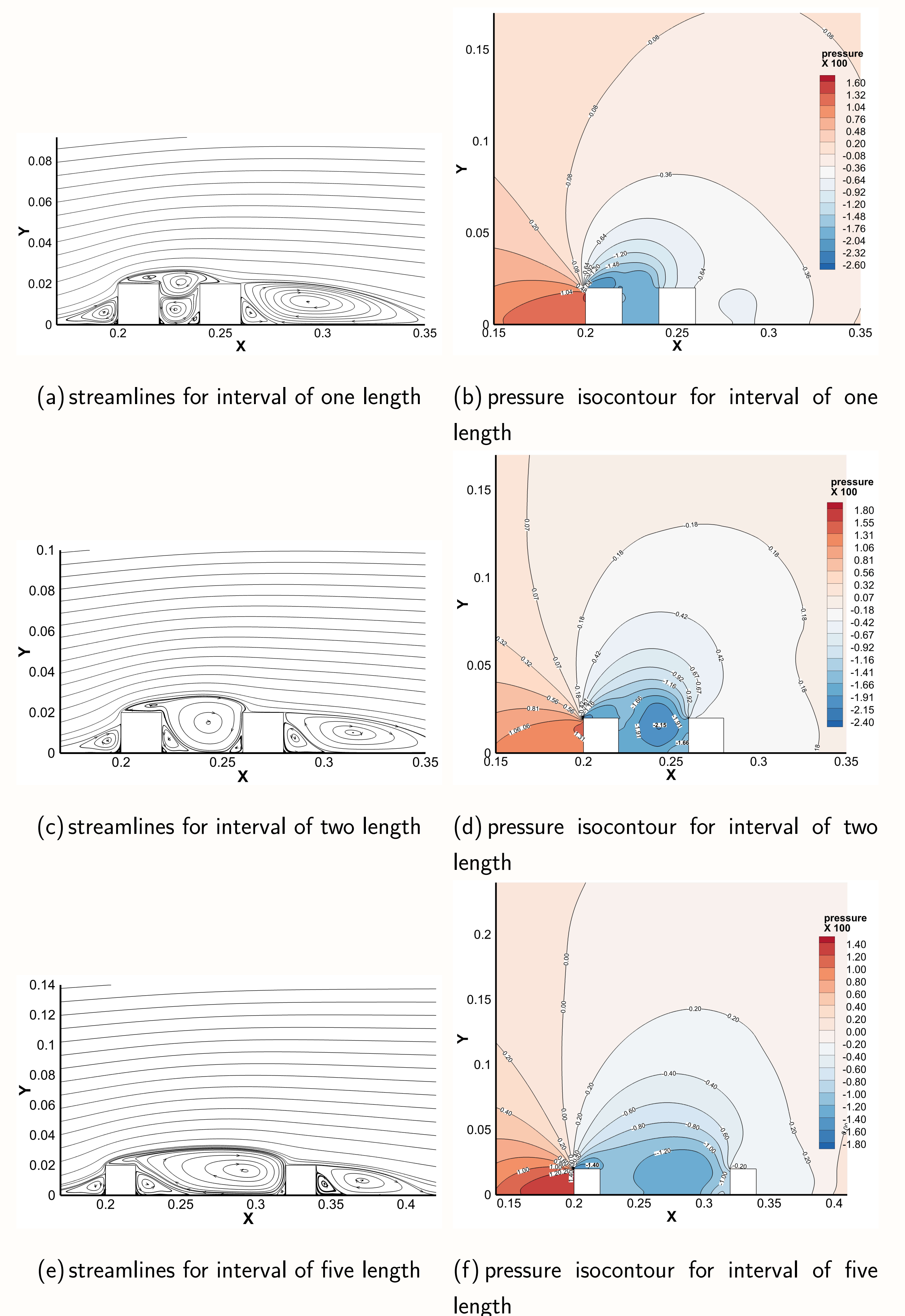


Figure 2: streamlines and pressure plot of averaged velocity