

## Numerical Simulation of Boundary Layer Flow Around Simplified Subsea Structures

Dennis Alexander Nymo

Marine Technology Submission date: June 2015 Supervisor: Dag Myrhaug, IMT Co-supervisor: Muk Chen Ong, IMT

Norwegian University of Science and Technology Department of Marine Technology

#### MASTER THESIS IN MARINE TECHNOLOGY

#### SPRING 2015

#### FOR

#### STUD. TECHN. DENNIS ALEXANDER NYMO

# NUMERICAL SIMULATION OF BOUNDARY LAYER FLOW AROUND SIMPLIFIED SUBSEA STRUCTURES

Subsea structures on the seabed are often subject to high Reynolds number flows ( $\text{Re} > 10^4$ ). These high Reynolds number flow conditions are hard and expensive to achieve in an experimental setup, which requires appropriate experimental facilities, minimizing human and instrument errors during measuring hydrodynamic quantities etc. Therefore an attractive alternative is to use Computational Fluid Dynamics (CFD) to provide the essential hydrodynamic quantities (such as drag and lift coefficients, as well as pressure distribution around the subsea structures) for engineering design. Turbulence modelling will be employed in this study. The effects of different structural geometries (such as, rectangular and square shapes, as well as the shape of subsea protection cover), and boundary layer thickness to the height of the structure will be investigated. The flow structure will also be studied. Predictions of hydrodynamic quantities will be compared with published experimental data.

The student shall:

- 1. Give a background of CFD and flow around bluff bodies
- 2. Describe the computational tools
- 3. Give a background description of turbulence modelling
- 4. Perform numerical simulation of boundary layer flows around simplified subsea structures
- 5. Present and discuss the results

The work scope may prove to be larger than initially anticipated. Subject to approval from the supervisor, topics may be deleted from the list above or reduced in extent.

In the thesis the candidate shall present his personal contribution to the resolution of problem within the scope of the thesis work.

Theories and conclusions should be based on mathematical derivations and/or logic reasoning identifying the various steps in the deduction.

The candidate should utilize the existing possibilities for obtaining relevant literature.

The thesis should be organized in a rational manner to give a clear exposition of results, assessments, and conclusions. The text should be brief and to the point, with a clear language. Telegraphic language should be avoided.



The thesis shall contain the following elements: A text defining the scope, preface, list of contents, summary, main body of thesis, conclusions with recommendations for further work, list of symbols and acronyms, reference and (optional) appendices. All figures, tables and equations shall be numerated.

The supervisor may require that the candidate, in an early stage of the work, present a written plan for the completion of the work. The plan should include a budget for the use of computer and laboratory resources that will be charged to the department. Overruns shall be reported to the supervisor.

The original contribution of the candidate and material taken from other sources shall be clearly defined. Work from other sources shall be properly referenced using an acknowledged referencing system.

The thesis shall be submitted in three copies:

- Signed by the candidate
- The text defining the scope included
- In bound volume(s)
- Drawings and/or computer prints which cannot be bound should be organized in a separate folder.
- Advisors: Dr. Muk Chen Ong, Marintek Professor Dag Myrhaug Leiv Apelund, Subsea 7

Deadline: 10.06.2015

Golacy

Dag Myrhaug Supervisor

"I am an old man now, and when I die and go to Heaven there are two matters on which I hope for enlightenment. One is quantum electrodynamics, and the other is the turbulent motion of fluids. And about the former I am really rather optimistic"

Quote by Lamb, H. from 1932 (Goldstein, 1969)

# Preface

This Master Thesis ends my five-year Master of Technology education in Marine Technology at the Norwegian University of Science and Technology (NTNU) in Trondheim. The subject was proposed by Subsea 7 in collaboration with the Department of Marine Technology.

I would like to thank my supervisor Professor Dag Myrhaug at the Department of Marine Technology for his patience and guidance. A special thanks to my second supervisor Muk Chen Ong at MARINTEK for sharing knowledge of Computational Fluid Dynamics (CFD) and support throughout the semester. I am also grateful for the technical support and interesting discussions regarding glass reinforced plastic (GRP) covers with Leiv Aspelund from Subsea 7.

Trondheim, June 10, 2015

Dennis Alexander Nymo

## Abstract

Computational Fluid Dynamics (CFD) have been applied to find essential hydrodynamic quantities of simple structures on the seabed subjected to boundary layer flow from ocean currents. This is relevant for subsea structures where the stability might be an issue such as glass reinforced plastic (GRP) covers used to protect subsea equipment. They protect the equipment from trawlers and falling objects.

Reynolds numbers based on the structures height h was  $3.41 \times 10^4 < Re_h < 1.19 \times 10^5$  for all simulations. This was the same Re as in published experiments used for validation of the drag coefficients  $C_D$  obtained from numerical simulations.

The length of the structure was normal to the flow direction. The structures length to height ratio was large, hence two-dimensional numerical simulations may be assumed to give accurate results at the middle cross section along the structures length. Numerical simulations of boundary layer flow over geometries with the shape of a square, rectangle and simplified cross sections of GRP covers have been performed. The lift coefficient  $C_L$ and  $C_D$  were obtained for each geometry. Non-dimensional distance from fluid separation of the structures upstream corner to the reattachment point downstream of the flat bottom surface  $x_R/h$  was calculated. Streamlines, pressure and velocity distribution were also presented from the numerical simulations.

Results of  $C_D$  for various values of boundary layer thickness to height ratio  $\delta/h$  and width to height ratio b/h from present study have been compared to published experimental data. The hydrodynamic quantities obtained from numerical simulations confirmed that the numerical simulations gave similar results as previous experimental studies. Hence it may be assumed that the same method would be able to predict physically sound values of  $C_D$  and  $C_L$  for geometrically simplified GRP covers. Local velocity properties for square cross section with  $\delta/h = 0.73$  from present study was also validated against published experiment. The resulting trend from the present study seem to agree with the experiment.

Two-dimensional Reynolds-Averaged Navier-Stokes (RANS) equations with a standard  $k-\epsilon$  turbulence model was applied. The commercial CFD software ANSYS CFX was used for all simulations, in combination with ANSYS ICEM for making a structured mesh and TecPlot360 for post-processing. Mesh convergence studies have been performed for all cases which showed satisfactory mesh independence for global hydrodynamic quantities. This indicates that the mesh used was of good quality.

Transient (Unsteady RANS) and steady-state (RANS) simulations have been carried out.

The steady-state runs did not have fluctuating values of  $C_D$  and  $C_L$  in any simulation, indicating that two-dimensional boundary layer flow over simple structures with sharp corners may be treated as a stationary problem. Transient numerical simulations of square geometry at  $\delta/h = 1.70$  have been performed. The simulation confirmed that the flow configuration can be treated as time independent. This is due to negligible fluctuations of global quantities such as  $C_D$  and  $C_L$ .

Results from numerical simulations show excellent agreement of  $C_D$  when compared to experiments for  $\delta/h \ge 1.70$ , hence further study of various width to height ratio b/h when  $\delta/h = 1.70$  was performed. The resulting value of  $C_D$  from the numerical simulations indicated a slight over prediction when  $\delta/h \le 0.73$ .

As  $\delta/h$  increased between  $0.73 \leq \delta/h \leq 2.55$ , the values of  $C_D$ ,  $C_L$  and  $x_R/h$  was found to decrease. For values of b/h between  $1 \leq b/h \leq 5$ , a decrease in  $C_D$  and  $x_R/h$  were observed while  $C_L$  increased. The hydrodynamic coefficient  $C_L$  was based on h, hence the increase in  $C_L$  was mainly due to the larger area on top of the structure as b/h increased. Larger areas where the integrated low pressure could act, resulted in higher total lift force.

Due to the similarity in results from square and rectangular cross sections of structures at  $\delta/h = 1.7$  compared to physical experiments, hydrodynamic quantities were calculated for simplified GRP covers at the same boundary layer thickness to height ratio. The hydrodynamic quantities obtained for the GRP covers was close to the same values from numerical simulations of square and rectangular structures. Two GRP covers were applied in the numerical simulations, GRP 1 and GRP 2. Resulting force coefficients for GRP 1 from the simulations were  $C_D = 0.799$  and  $C_L = 0.763$ . GRP 2 had  $C_D = 0.612$  and  $C_L = 1.982$ .

Reattaching flow on the structures was found to be present for rectangular shapes when b/h = 5 while no reattachment was found for b/h = 3. This is physically correct according to published experiments. Reattachment was also present for GRP 2, however not visible for GRP 1. GRP 2 is 2.4 times wider than GRP 1.

A study of the effect of including the mudmat for GRP 1 was conducted and the influence on the flow structure was minimal. Numerical simulations of the same geometry with and without a mudmat were conducted. The resulting values of  $C_D$ ,  $C_L$  and  $x_R/h$  showed less than 1.0% relative change between the simulations, hence the mudmat may be neglected.

It appears that the present CFD simulations are able to predict hydrodynamic quantities reasonably well for boundary layer flow over simple two-dimensional geometries. The method may be used to predict forces on GRP covers. This is relevant for the stability of the covers on the seabed, and the required weight of the structure.

## Sammendrag

Numeriske strømningsberegninger eller "Computational Fluid Dynamics" (CFD) har blitt brukt for å finne essensielle hydrodynamiske aspekter ved grensesjiktstrømning over forenklede geometrier på havbunnen utsatt for havstrømninger. Dette er relevant for undervannsstrukturer hvor forflytning grunnet hydrodynamiske krefter kan være et problem som f.eks. beskyttelsesstrukturer i komposittmateriale (glass reinforced plastic (GRP) covers). Disse beskyttelsesstrukturene brukes over kritisk undervannsutstyr og beskytter mot trålere og fallende gjenstander.

Reynolds tall basert på geometriens høyde h var satt til å være  $3.41 \times 10^4 < Re_h < 1.19 \times 10^5$  i alle numeriske simuleringer. Verdien av Reynolds tall ble bestemt basert på tilgjengelig forsøk med samme verdier for Re. Forsøket ble brukt til å sammenligne verdier for  $C_D$  fra de numeriske simuleringene og forsøket. Lengden av geometrien var vinkelrett i forhold til strømningsretningen. Geometriens lengde-høydeforhold kan regnes som stor, noe som medfører at todimensjonal simulering kan tenkes å gi riktige resultater ved et tverrsnitt midt langs lengden av geometrien. Geometriene som er implementert i simuleringene er kvadrat, rektangel og et forenklet tverrsnitt av beskyttelsesstruktur i komposittmateriale for undervannsutstyr. Dragkoeffisient  $C_D$ , løftkoeffisient  $C_L$ , strømlinje-, trykk- og hastighetsfordeling ble regnet ut. Dimensjonsløs lengde til punktet hvor strømningen treffer havbunnen bak strukturen etter å blitt separert fra strukturens fremste hjørne  $x_R/h$  ble også beregnet.

Den hydrodynamiske koeffisienten  $C_D$  ble utregnet for ulike verdier av grensesjikttykkelsehøydeforhold  $\delta/h$  og bredde-høydeforhold b/h og resultatet har blitt sammenlignet med et fysisk forsøk som er publisert. De hydrodynamiske koeffisientene utregnet fra numeriske simuleringer stemmer overens med tidligere forsøk. Det var derfor antatt at samme framgangsmåte som var brukt på kvadratiske og rektangulære geometrier kunne brukes for å få verdier av  $C_D$  og  $C_L$  til beskyttelsesstrukturer som var fysisk fornuftige. Hastighetsprofiler for kvadratisk tverrsnitt ved  $\delta/h = 0.73$  fra de numeriske simuleringene ble sammenlignet med tilsvarende vardier fra forsøk. Hastighetsprofilene ser ut til å stemme overens med tilsvarende profiler fra forsøk.

Tidsgjennomsnitt av Navier-Stokes (Reynolds-Averaged Navier-stokes) ligninger i to dimensjoner med standard  $k - \epsilon$  turbulens modell ble brukt i samtlige analyser. Tidsgjennomsnitt av Navier-Stokes ligninger blir også kaldt RANS-ligninger. ANSYS CFX er et kommersielt dataprogram, og det er brukt for å gjøre samtlige numeriske strømningsberegner. ANSYS ICEM er brukt for å lage et strukturert rutenett (mesh) og TecPlot360 er brukt til å prosessere resultatene fra numeriske strømningsberegninger (post-processing). Konvergensstudie for rutenett er gjennomført ved å øke antall elementer nær geometrien og ellers i domenet for å se hvordan det påvirker  $C_D$ ,  $C_L$  og  $x_R/h$ . Disse verdiene forandret seg i liten grad da antall elementer ble økt, noe som indikerer at løsningen er lite avhengig av rutenett. Det valgte oppsettet av rutenett kan derfor regnes for å være av god kvalitet.

Simuleringer som er tidsavhengige og tidsuavhengige har blitt gjennomført. De stasjonære løsningene hadde ingen fluktuerende verdier for  $C_D$  eller  $C_L$ , noe som tyder på at grensesjiktstrømning over enkle todimensjonale geometrier med skarpe hjørner kan løses uavhengig av tid. Tidsavhengige analyser har blitt gjennomført for kvadratisk tverrsnitt med  $\delta/h = 1.70$ . Analysen bekreftet også at strømningen kan ses på som tidsuavhengig.

Det ble funnet at  $C_D$  stemte godt overens med fysiske forsøk når  $\delta/h \geq 1.70$ . Verdien av  $C_D$  fra de numeriske simuleringene viste tegn til konservativ løsning for  $\delta/h \leq 0.73$ . Flere simuleringer med ulike verdier av b/h er gjennomført for  $\delta/h = 1.70$ , siden denne grensesjikttykkelsen viste seg å stemme godt overens med fysisk forsøk.

En nedgang av verdier for  $C_D$ ,  $C_L$  og  $x_R/h$  ble påvist for økende verdi av  $\delta/h$  ved  $0.73 \leq \delta/h \leq 2.55$ . Da b/h økte for  $1 \leq b/h \leq 5$ , viste det seg en nedgang for  $C_D$  og  $x_R/h$  mens  $C_L$  hadde en økning. Den hydrodynamiske koeffisienten  $C_L$  var basert på h. Dette medførte at  $C_L$  økte hovedsakelig som resultat av at arealet på toppen av geometrien ble større når b/h økte. Større areal hvor det lave trykket blir integrert over fører til høyere total løftkraft.

Resultatene fra beregning av grensesjiktstrømning over beskyttelsesstrukturene var mellom verdiene som ble funnet for firkantede tverrsnitt. De hydrodynamiske koeffisientene for beskyttelsesstrukturen kalt GRP 1 ble funnet til å være  $C_D = 0.799$  og  $C_L = 0.763$ . Den andre beskyttelsesstrukturen ble referert til som GRP 2 og analysene gav  $C_D = 0.612$ og  $C_L = 1.982$  for denne geometrien.

En studie av effekten til en støttestruktur (mudmat) som skal hindre at undervannsutstyr skal synke ned i havbunnen ble gjennomført. Denne strukturen ble inkludert i geometrien for GRP 1 og resultatet ble sammenlignet med analysene uten støttestrukturen. Effekten av denne strukturen hadde på hvilken som helst verdi av  $C_D$ ,  $C_L$  eller  $x_R/h$  en maksimal relativ forskjell på under 1.0%. Det ble derfor konkludert med at denne strukturen kan neglisjeres.

Strømningen som separeres ved fremre hjørne av den rektangulære geometrien treffer strukturen igjen når b/h = 5. Dette er ikke tilfelle når b/h = 3 eller mindre. Dette fenomenet stemte overens med tidligere forsøk som er publisert. Den separerte strømningen traff også toppen av GRP 2, noe som ikke var tilfelle for GRP 1. GRP 2 er 2.4 ganger bredere enn GRP 1.

Det fremstår som om metoden for numeriske simuleringer av grenesjiktstrømning er i stand til å produsere resultater som stemmer overens med virkeligheten for firkantede geometrier på havbunnen med ulike verdier av  $\delta/h$  og b/h. Metoden kan derfor anses som pålitelig å bruke for å beregne hydrodynamiske krefter involvert på beskyttelsesstrukturer. Dette er relevant for å undersøke hvor mye vekt som trengs for at strukturen ikke skal flyttes av havstrømninger.

# Contents

1	Intr	roduction
	1.1	Background
	1.2	Definition of terms
	1.3	Published work
	1.4	Outline of the thesis
<b>2</b>	The	ory
	2.1	Flow Characteristics
	2.2	Turbulence         8
	2.3	Turbulent Boundary layer
	2.4	Conservation of mass and momentum
	2.5	Forces acting on the body 13
3	Con	nputational Fluid Dynamics 15
	3.1	Introduction to CFD
	3.2	ANSYS CFX 10
	3.3	Mesh
	3.4	Turbulence model
	3.5	Courant-Friedrichs-Lewy number
	3.6	Post-processing
<b>4</b>	Nur	nerical set-up 21
	4.1	Domain
	4.2	Mesh set-up
	4.3	Boundary Conditions
	4.4	Numerical Flow Characteristics
<b>5</b>	Squ	are and rectangular structures 33
	5.1	Mesh Convergence
	5.2	Results and discussion
		5.2.1 Force coefficients
		5.2.2 Pressure
		5.2.3 Velocity $\ldots \ldots 40$
		5.2.4 Streamlines 4
6	Sim	plified GRP Cover 45
	6.1	Simplified Subsea Structures
	6.2	Mesh Convergence

	6.3	Result 6.3.1 6.3.2	s and discussion . Force Coefficients Streamlines	· · ·	  	· · · ·	  		  		  		  		  			  		• • •	49 49 50
7	Conclusion 5							53													
8	Recommendations for Further Work 55								55												
Re	References 5'									57											

# List of Tables

4.1	Skewness mesh metrics spectrum adapted from Bakker $(2006)$	23
5.1	Hydrodynamic quantities from grid convergence study for square and rect- angular cross section at given values of $\delta/h$ and $b/h$	34
5.2	Hydrodynamic quantities from given values of height between the flat bot-	
	tom surface and the upper boundary when $\delta/h = 0.73$ and $b/h = 1$	35
5.3	Hydrodynamic quantities for square cross section compared to equivalent	25
5.4	Hydrodynamic quantities for square and rectangular cross section with given value of $b/h$ compared to equivalent values from experiments (Fujimoto	55
	et al., 1975)	36
6.1	Hydrodynamical quantities from grid convergence study for cross sections of simplified GRP covers	48
6.2	Hydrodynamical quantities from grid convergence study for cross sections	
	of GRP 1 with mudmat	48
6.3	Hydrodynamic quantities for GRP covers	49

# List of Figures

1.1	GRP covers at the Aasgard field from Reinertsen (2015)	1
1.2	Set-up for two dimensional square cross section with the definition of $n, b, d$	ર
13	Set-up for two dimensional cross section of a simplified GRP cover with	0
1.0	the definition of $h_{c}$ $b_{c}$ $U_{co}$ , $\delta$ and the coordinate system used	3
1.4	Set-up for two dimensional cross section of a simplified GRP cover includ-	0
	ing mudmat with the definition of $h, b, U_{\infty}, \delta$ and the coordinate system	
	used	4
2.1	Velocity as function of time at a point in turbulent flow adapted from	
	Versteeg and Malalasekera (2007)	9
2.2	Developing boundary layer on a horizontal plate modified from Frei (2013)	10
2.3	Non-dimensional velocity distribution near a smooth wall adapted from	10
	Versteeg and Malalasekera $(2007)$	12
3.1	Setup in ANSYS Workbench and the connection between the software	
	packages	16
3.2	Structured and unstructured quadrilateral grid modified from Yunus and	
	Cimbala (2006) $\ldots$	18
4.1	Computational domain used for square and rectangular structures as well	
	as cross section of simplified GRP covers	22
4.2	The block-topology used for the whole domain of square cross section	24
4.3	The block-topology used close to the square cross section	24
4.4	The block-topology used close to GRP 1	25
4.5	The block-topology used close to GRP 1 with mudmat	25
4.6	Mesh close to a wall with illustration of $h_p$	26
4.7	Global mesh for square cross section at $\delta/h = 1.70$	27
4.8	Local mesh for square cross section at $\delta/h = 1.70$	27
4.9	Local mesh for CPP 1	21
4.10	Local mesh for GRP 1 with mudmat	$\frac{20}{28}$
4 12	Local mesh for GRP 2	$\frac{20}{28}$
4.13	Boundary conditions applied	$\frac{20}{29}$
4.14	Comparison of numerical boundary layer in present study and experimental	_0
	boundar layer from Fujimoto et al. (1975) at the location of the upstream	
	facing side of the structure, without the structure itself being present	30

5.1	Drag coefficient for square cross section at given $\delta/h$ from present study and experiments (Eujimete et al. 1075)	37
59	Drag coefficient for square and rectangular cross section at $\delta/h = 1.70$ and	51
0.2	big coefficient for square and rectangular cross section at $\theta/n = 1.70$ and given value of $h/h$ from present study and experiments (Fujimete et al.	
	given value of $n/n$ from present study and experiments (Fujimoto et al., 1075)	27
59	1975)	- 37 - 20
0.0 E 4	Dressure coefficient for square cross section at given value of $0/n$	30
3.4	Pressure contour plot of square and rectangular structure when $\delta/\hbar = 1.7$	20
	for given values of $b/n$	39
<b>5.5</b>	Time average of norizontal velocity along vertical lines at given location	
	along the horizontal axis from present numerical simulations of square cross	40
- 0	section compared to experiments (Liu et al., 2008)	40
5.0	Time average of horizontal velocity contour plot of square cross section at	4.1
	$\delta/h = 0.73 \dots \dots$	41
5.7	Streamlines contours over square cross sections for given value of $\delta/h$	42
5.8	Streamline contours over square and rectangular cross section when $\delta/h =$	
	1.70 for given value of $b/h$	43
6.1	Detailed three-dimensional drawing of GRP covers used in present study	46
6.2	Detailed cross section including dimensions of GRP 1	46
6.3	Detailed cross section including dimensions of GRP 2	46
6.4	Geometry and dimensions for simplified version of GRP 1	47
6.5	Geometry and dimensions for simplified version of GRP 1 with mudmats	47
6.6	Geometry and dimensions for simplified version of GRP 2	47
6.7	Drag and lift coefficient for square, rectangle and GRP covers at given	
	value of $b/h$ from present study and experiments (Fujimoto et al., 1975).	50
6.8	Streamlines contours over GRP covers	51

# List of Abbrevations

CFD:	Computational Fluid Dynamics
DES:	Direct Numerical Simulation
FVM:	Finite Volume Method
GRP:	Glass Reinforced Plastic
LES:	Large Eddy Simulation
RANS:	Reynolds-Averaged Simulation
URANS:	Unsteady Reynolds-Averaged Simulation
RMS:	Root Mean Square

# List of Symbols

<i>A</i> :	$[m^2]$	Area projected on a plane normal to the direction of flow
$A^*$ :	[—]	Constant in equation for law of the wake
<i>b</i> :	[m]	Width of structure
<i>c</i> :	[m/s]	Speed of sound
C:	[_]	Courant number
$C_1$ :	[_]	Constant in the model equations for $\epsilon$
$C_2$ :	[_]	Constant in the model equations for $\epsilon$
$\tilde{C_D}$ :	[_]	Drag coefficient
$C_L^L$ :	[_]	Lift coefficient
$C_{max}$ :	[_]	Maximum Courant number
$C_{nrms}$ :	[_]	Root Mean Square of pressure coefficient
$C_{\mu}$ :	[—]	Turbulent-viscosity constant in the $k - \epsilon$ turbulence
٣		model
$C_p$ :	[—]	Pressure coefficient
$D_e$ :	[m]	Hydraulic diameter
E:	[—]	Log-law constant
$F_D$ :	[N]	Total drag force over the structure
$F_L$ :	[N]	Total lift force over the structure
<i>h</i> :	[m]	Height of structure
$h_p$ :	[m]	Hight of the closest mesh element to the wall
$\dot{H}$ :	[-]	Shape factor of the boundary layer
k:	$[m^2/s^2]$	Turbulent kinetic energy
<i>l</i> :	[m]	Length of the structure
L:	[m]	Characteristic length
Ma:	[-]	Mach number
<i>p</i> :	[Pa]	Pressure
$p_0$ :	[Pa]	inlet pressure $\delta$ from the wall
$p_{stat}$ :	[Pa]	Static pressure
Re:	[-]	Reynolds number
$Re_{crit}$ :	[-]	Critical Reynolds number
$Re_{De}$ :	[-]	Reynolds number based on hydraulic diameter
$Re_h$ :	[—]	Reynolds number based on the structures height
$Re_x$ :	[—]	Reynolds number based on length traveled along hori-
		zontal flat plate

$Re_{\theta}$ :	[—]	Reynolds number base on momentum thickness
S:	[m]	Path along the surface of the square structure
t:	[s]	Time
u:	[m/s]	Horizontal velocity component
u':	[m/s]	Horizontal component of turbulent fluctuating velocity
$u_i$ :	[m/s]	Cartesian component of the velocity vector
$u_i$ :	[m/s]	Cartesian component of the velocity
$u_n$ :	[m/s]	Velocity normal to a plane close to a boundary
$u^+$ :	[—]	Dimensionless velocity
$\overline{u_i'u_i'}$ :	$[m^2/s^2]$	Reynolds stress component
$u_{\tau}$ :	[m/s]	Friction velocity
U:	[m/s]	Horizontal mean velocity component
$U_{\infty}$ :	[m/s]	Free stream velocity far from the structure and the flat
	L / J	bottom surface
x:	[m]	Distance traveled by the fluid along a flat plate
$x_B$ :	[m]	Horizontal distance from origo too reattachment point
		of the fluid downstream of the structure
$x_i$ :	[m]	Cartesian coordinates
$x_i$ :	[m]	Cartesian coordinates
$X_1$ :	[m]	Horizontal axis in coordinate system
$X_2$ :	[m]	Vertical axis in coordinate system
y:	[m]	Distance from the wall
$y = y^+$	[_]	Dimensionless distance from the wall
$\frac{v}{\overline{y}^+}$ :	[_]	Mean dimensionless height of first cell close to a wall
$\delta$	[m]	Boundary layer thickness
$\delta *$	[m]	Displacement thickness
$\delta_{ij}$ :	[_]	Kronecker delta
$\Delta t$ :	[s]	Time step
$\Delta x$ :	[m]	Stream-wise length of an element in the domain
$\epsilon$ :	$[m^2/s^3]$	Rate of dissipation of turbulent kinetic energy
$\kappa$ :	[_]	von Kármán constant
$\ell$ :	[m]	Turbulent length scale
$\sigma_k$ :	[_]	Turbulent Prandtl number for kinetic energy
$\sigma_{\epsilon}$ :	[_]	Turbulent Prandtl number for dissipation
ρ:	$[kq/m^3]$	Density of fluid
$\tau_{m}$ :	[Pa]	Surface shear stress
$\mu$ :	$[kq/m \cdot s]$	Dynamic viscosity of fluid
$\mu_t$ :	$[kq/m \cdot s]$	Turbulent viscosity of fluid
$\nu$ :	$[m^2/s]$	Kinematic viscosity of fluid
$\theta$ :	[m]	Momentum thickness
$\theta_n$ :	[m]	Angle that the outer normal of dA on the structure
		makes with the positive flow direction

# Chapter 1

# Introduction

## 1.1 Background

The amount of subsea installations have increased in the last decades on the Norwegian Continental Shelf. Some parts of the subsea structures are critical to protect from dropped objects and fishing gear which may damage the equipment. Examples of such structures are pipeline connection points, spools and umbilical control connections close to the subsea templates. Glass-reinforced plastic (GRP) covers are often used as protection over these structures. GRP is the material of choice for these covers as it is competitive compared to steel in terms of corrosion resistance, fabrication and installation weight. The weight of a GRP cover is one third of an equivalent cover in steel, hence more likely to move due to hydrodynamic forces. GRP covers which is going to be used to protect flowlines, Mono Ethylene Glycol (MEG) lines, power cables and fiber optic cables between the subsea gas compression station and template at the Aasgard field. GRP covers are also going to protecting subsea structures such as pipeline end manifolds (PLEM) as seen in figure 1.1.



Figure 1.1: GRP covers at the Aasgard field from Reinertsen (2015)

The installation loads and force from objects which may hit the covers are part of the

design criteria for GRP Covers. Current and wave loads are relevant in terms of the stability of the covers. Velocities from ocean current results in boundary layer flow parallel to the seabed of oil and gas fields. Aastad Hansten is a deep water gas field where wave interaction with the seabed can be assumed to be non existing. Hence modeling of the ocean current is relevant towards obtaining the forces on the covers that can be used for calculating the required weight of the covers.

The velocities from extreme currents at the seabed result in high Reynols number Re and turbulent flow which induce a complicated vortex system at the bluff or blunt body of a GRP Cover. The flow structure is depending on Re, incident boundary layer thickness over the height of the body  $\delta/h$  and the geometry of the structure. Analytical solutions for these hydrodynamic problems are not yet feasible, hence numerical simulations or experiments are necessary to find the forces involved.

Computational Fluid Dynamics (CFD) is an attractive alternative to experiments for engineering design due to the expense of achieving high Reynolds number flow conditions in laboratory testing. Recent advances in computing power have made CFD more widely used in the industry and for research. Reynolds-averaged Navier-Stokes (RANS) simulation is still the preferred method compared to Large Eddy Simulation (LES) or Direct Numerical Simulation (DES) in the industry. This is due to the efficient use of computational cost with reasonable accuracy for RANS simulations.

In the present thesis, boundary layer flow over simplified subsea structures such as GRP covers and the hydrodynamic coefficients are going to be studied utilizing numerical methods. RANS approach will be applied to solve the flow field. A validation study of boundary layer flow over structures with square and rectangular cross section are going to be performed and hydrodynamic coefficients will be compared to published experimental data.

## **1.2** Definition of terms

A two-dimensional square cross section of a long structure is illustrated in figure 1.2. The coordinate system used is  $X_1$  and  $X_2$ . The horizontal axis is  $X_1$  and the vertical axis is  $X_2$ . The center of the coordinate system is upstream where the flat bottom surface and the wall of the structure meet. The height of the structure is h, the width is b and the length is l. A path following the surface of the square structure is defined as S, starting in origo of the coordinate system and ending where the back of the structure and the flat bottom surface meet. The definition of S is seen in figure 1.2. The path S is used when extracting values of pressure along the wall for a square structure with various values of  $\delta/h$ . The free stream velocity in the horizontal direction outside the boundary layer is  $U_{\infty}$ . The free stream velocity, hence the notation  $\infty$  is used. The boundary layer thickness  $\delta$  is defined as the length from the wall to where the velocity in x-direction reach 99% of  $U_{\infty}$  (Yunus & Cimbala, 2006). The square cross section is subjected to boundary layer flow as illustrated in figure 1.2, where the profile used is slightly different than the one shown.



Figure 1.2: Set-up for two dimensional square cross section with the definition of h, b, path S,  $U_{\infty}$ ,  $\delta$  and the coordinate system used

Similar setup is used for the analysis of boundary layer flow over a simplified geometry of a GRP cover as seen in figure 1.3. Figure 1.4 illustrates a GRP cover including mudmat.



Figure 1.3: Set-up for two dimensional cross section of a simplified GRP cover with the definition of  $h, b, U_{\infty}, \delta$  and the coordinate system used



Figure 1.4: Set-up for two dimensional cross section of a simplified GRP cover including mudmat with the definition of h, b,  $U_{\infty}$ ,  $\delta$  and the coordinate system used

These are the geometries analyzed in the present study. Numerical simulations for  $\delta/h = 0.73$ , 1.70 and 2.55 as well as width to height ratios b/h = 1, 3 and 5 are performed. Two simplified versions of GRP covers with b/h = 1.60 and b/h = 3.87 are also simulated. They are referred to as GRP 1 and GRP 2 respectively. A study of the effect of including mudmat is performed for GRP 1.

### 1.3 Published work

Boundary layer flow over obstacles mounted on flat surfaces have been extensively investigated through experiments and numerical simulations due to its importance to various industries. It has been adapted for analyzing the flow and its effect over parts in gas turbines, heat exchangers, buildings as well as structures on the seabed. The topic is relevant for marine subsea engineering and fluid engineering.

There are numerous published results from experiments and numerical simulations regarding two-dimensional cross sections of long structures (rib) with different b/h ratio inside channel flow. Few publications are found regarding high Reynolds number  $Re_h$  boundary layer flow based on h over single square cross section where the roof of the channel is far from the structure. There are no published numerical simulations known to the author of two-dimensional boundary layer flow over single square and rectangular cross sections where the top boundary is not influencing the result at  $3.41 \times 10^4 < Re_h < 1.19 \times 10^5$ . No published literature is found for boundary layer flow over subsea protection covers.

Fujimoto et al. (1975) measured the pressure distribution on two-dimensional square structures immersed in the turbulent boundary layer flow inside wind tunnel experiments with various values of  $U_{\infty}$ ,  $\delta/h$  and b/h. The values of Reynolds numbers were  $3.41 \times 10^4 < Re_h < 1.19 \times 10^5$ . The structures were long surface mounted obstacles that were subjected to flow normal to their length. Two-dimensional characteristics of the flow was obtained

close to the center of the structures length. Fujimoto et al. (1975) found that in the range of  $\delta/h \geq 0.73$  and  $b/h \leq 6$ , the pressure drag of the structure can be expressed by a logarithmic empirical formula. The results showed that by increasing  $\delta/h$ , the drag coefficient  $C_D$  decreased. He also found that  $C_D$  decreased when b/h increased up to b/h = 5, where the change in drag was small with increasing b/h. Surface-flow visualization revealed that the reattachment of flow occurred when  $b/h \geq 4$ .

Good and Joubert (1968) conducted wind tunnel experiments for high Reynolds number boundary layer flow over two-dimensional vertical plates (fence). He proved that  $C_D$ varies logarithmically from  $\delta/h \geq 1.20$  for high Reynolds number boundary layer flow over the structures. He also found that when  $\delta/h < 1.20$ ,  $C_D$  increase more rapidly as  $\delta/h$  decrease and the drag coefficient do not vary logarithmically.

Keshmiri (2012) performed a numerical sensitivity analysis of two- and three-dimensional square cross sections of structures in channel flow. RANS equations and Lien–Chen–Leschziner  $k - \epsilon$  turbulence model were used and the flow conditions with  $Re_{De} = 3.0 \times 10^4$ . The Reynolds number was based on the hydraulic diameter  $D_e$  of the channel. The results for the three-dimensional channel were in good agreement with experimental data, and a two-dimensional channel could be used to represent the centerline of a three-dimensional channel with satisfactory accuracy.

Martinuzzi and Tropea (1993) studied the flow around surface mounted, prismatic obstacles with square cross sections in experimental channel flow at  $Re_{De} = 8.0 \times 10^4$  and  $1.15 \times 10^5$ . The effect of length over height l/h on the flow patterns was investigated. The results show that the middle region can be considered to be fully two-dimensional for length-to-height ratio l/h greater than 10, and that the wake is two-dimensional at l/h greater than 6. They also found that the recovery length of the shear layer in the wake of the obstacle is shorter for cases which are three-dimensional compared to twodimensional.

Ryu et al. (2007) investigated the characteristics of a turbulent flow in channels with two-dimensional surface mounted structures at  $Re_{De} = 2.0 \times 10^4$ . The RANS method and the  $k-\omega$  turbulence model were applied. Span-wise structures of various geometries were analyzed. They were square, triangular, and semicircular shapes as well as a wavy wall (sinusoidal function). The square shaped structures were reported to exert the most resistance among the four shapes considered while the wavy wall offers the least. The Reynolds-averaged numerical results are compared to experiments and found to describe the essential features of the flow over a surface with two-dimensional structures.

Castro (1984) conducted wind tunnel experiments on flow past a surface mounted, twodimensional structure with rectangle cross sections. He found that increasing  $\delta/h$  would lead to reduction of the reattachment length of the separated shear layer.

Liu et al. (2008) measured statistical turbulence properties of turbulent flow over a twodimensional structure which was mounted on a wind tunnel wall. The value of  $Re_h = 1.32 \times 10^4$  was used. Boundary layer profile is found at the upstream facing side of the structure without the structure being present. The value of  $\delta/h = 0.75$  is found at this location. The velocity fluctuations and wall-pressure fluctuations were experimentally measured and the dominating frequency was found to be high. One of the motions

which induced these fluctuations was caused by a recirculation zone in the wake of the structure. The fluid reattaches on the flat bottom surface downstream of the structure after the separation from the obstacle. The reattachment point was found to moved back and fourth with the frequency of the fluctuating velocity and pressure. The reattachment zone was a 1.2*h*-long region centered  $x_R/h = 10.75$  downstream of the structure. The non-dimensional value  $x_R/h = 10.75$  is defined as horizontal length the water particle travels from it separates on the upstream corner of the structure to the reattachment on the flat bottom surface downstream of the structure. He also found that the Root Mean Square (RMS) value of the pressure coefficient  $C_{p,rms} = 0.04$  on the top of the structure.

### **1.4** Outline of the thesis

Chapter 2 gives a review of the flow characteristics, turbulent boundary layer in which the structures are immersed, and the forces acting on the bluff bodies.

Chaper 3 describes CFD, the computational tools and turbulence modeling.

Chapter 4 explains the domain, mesh, boundary conditions and numerical boundary layer used.

Chapter 5 provides the results and discussion of numerical simulation of boundary layer flow around square and rectangular structures on a flat bottom surface.

Chapter 6 presents the results and discussion of numerical simulation of boundary layer flow around simplified GRP covers on a flat bottom surface.

Chapter 7 gives the conclusion of the master thesis.

Chapter 8 presents recommendations for further work.

## Chapter 2

## Theory

This chapter gives a understanding of the basic theory which is relevant when studying a structure submerged in a turbulent boundary layer. The difference between a surface mounted obstacle and a free span cylinder is acknowledged.

## 2.1 Flow Characteristics

There are major differences between a long surface mounted structure with two-dimensional flow properties such as a GRP cover, and a cylinder which is not attached to a wall such as a free span pipeline or riser. The most important thing is that there is no von Kármán vortex street where repeating pattern of swirling vortices are shed from the cylinder with respect to time. Instead there is a large dominating vortex in the wake of the structure and no large scale vortex shedding. This makes the long structure a much more time independent phenomenon compared to a cylinder which is not mounted on a wall.

The flow properties of boundary layer flow over a surface mounted structure is dependent on multiple parameters such as  $\delta/h$ , free stream turbulence intensity, boundary layer profile and geometry of the structure (Castro, 1984; Adams & Johnston, 1988). Another important parameter for the flow is the non-dimensional Reynolds number Re which is the ratio between inertia forces and viscous forces defined as

$$Re = \frac{Inertiaforces}{Viscousforces} = \frac{\rho U^2 L^2}{\mu U L} = \frac{\rho U L}{\mu}$$
(2.1)

where  $\rho$  is the density of fluid, U is the time average of horizontal fluid velocity component,  $\mu$  is the dynamic viscosity and L is the characteristic length scale (Yunus & Cimbala, 2006).

Examples of characteristic lengths are h, the distance traveled by the fluid along a flat plate x, the momentum thickness  $\theta$  and  $D_e$  of the channel. Reynolds number defined by  $h, x, \theta$  and and  $D_e$  is expressed as:

$$Re_h = \frac{\rho U_\infty h}{\mu} \tag{2.2}$$

$$Re_x = \frac{\rho U_\infty x}{\mu} \tag{2.3}$$

$$Re_{\theta} = \frac{\rho U_{\infty} \theta}{\mu} \tag{2.4}$$

$$Re_{D_e} = \frac{\rho U_\infty D_e}{\mu} \tag{2.5}$$

In the present study, Reynolds number with L = h is used and all numerical simulations are performed at  $3.41 \times 10^4 < Re_h < 1.19 \times 10^5$ , which is defined as high Reynolds number flow. The same Reynolds number for the case with circular cylinder in uniform flow is within the sub-critical and critical flow regime where the wake is completely turbulent (Sumer & Fredsoe, 2006). The flow is already turbulent when reaching the structure due to the fully developed turbulent boundary layer, hence the present study is a fully turbulent problem.

The flow is classified as incompressible, where the density of the fluid  $\rho = constant$  due to negligible compressibility effects. This is valid for Mach number  $Ma \leq 0.3$  where  $Ma = \frac{U}{c}$ and c is the speed of sound in the specified medium. Compressible flow effects are relevant for high-speed aircraft, rockets and missiles while not significant for flow over subsea and offshore structures. Temperature fluctuations in the flow domain is small in the present study and can be ignored, hence isothermal conditions is valid. As a result, it is not necessary to solve the energy equation, saving computational cost. A consequence of the assumptions of incompressible and isothermal flow is that  $\mu = constant$  and kinematic viscosity  $\nu = constant$  in the fluid domain. The turbulence makes the fluid appear more viscous and this is described with the turbulence viscosity  $\mu_t$ . Gravitational force does not influence the fluid flow in the present study.

### 2.2 Turbulence

There are two flow conditions called laminar flow and turbulent flow. Low values of Re are associated with laminar flow, where the flow is smooth and fluid particles follow streamlines. Laminar flow is observed in experiments to be below a critical Reynolds number  $Re_{crit}$ . When  $Re > Re_{crit}$  the flow character change radically and become random and chaotic with increasing value of Re. This is characterized as turbulent flow. Uniform flow over a circular cylinder with a smooth wall has  $Re_{crit} \approx 200$ . Flow in a circular pipe has  $Re_{crit} \approx 2300$ . The value of  $Re_{crit} \approx 10^5$  when uniform flow hit a flat plate and boundary layer flow starts to develop (Yunus & Cimbala, 2006). Fully developed turbulent boundary layer is used in the present numerical simulations. The small horizontal velocity fluctuations u'(t) around U at a point in a turbulent flow is shown in figure 2.1.





The velocity fluctuations are caused by small eddies and stochastic behavior of turbulent flow. The horizontal velocity u(t) at a point in turbulent fluid can be decomposed and described by the expression

$$u(t) = U + u'(t)$$
 (2.6)

In RANS equations, the turbulent flow is expressed by the mean values of flow properties such as U and pressure p. Turbulent fluctuations always have a three-dimensional spatial character even when the velocity is two-dimensional. This three-dimensional component in two-dimensional flow is small for boundary layer flow over surface mounted long structure with square cross section (Martinuzzi & Tropea, 1993; Ryu et al., 2007; Keshmiri, 2012).

Turbulent flow consist of turbulent eddies with a wide range of length scales. The larger eddies have a characteristic length scale which is in the same order as L. The velocity of the eddies is also in the same order as  $U_{\infty}$ . These eddies are dominated by inertia effects and viscous effects are negligible. The smaller eddies may follow the motion of the larger eddies and the kinetic energy is handed down from large eddies to smaller eddies in what is termed the energy cascade. When the eddies reach a length scale of the order of 0.10 to 0.01 millimeters in typical engineering flows, the viscous effects become important. The eddy motions are dissipated and converted into thermal internal energy (Versteeg & Malalasekera, 2007).

### 2.3 Turbulent Boundary layer

When the Reynolds number is sufficiently large during the development of a boundary layer along a surface, turbulence will arise. Figure 2.2 illustrates uniform fluid flow hitting a horizontal flat plate and how the boundary layer develop along the wall of the plate. It starts being laminar with streamlined velocity components to a turbulent state with

#### CHAPTER 2. THEORY

its characteristic eddies that give rise to fluctuating velocities at a point in the boundary layer.



Figure 2.2: Developing boundary layer on a horizontal plate modified from Frei (2013)

There are several parameters that are relevant towards describing the boundary layer profile and its properties such as  $\delta$ , displacement thickness  $\delta^*$  and  $\theta$ . Expressions of  $\delta^*$  and  $\theta$  are

$$\delta^* = \int_0^\infty \left(1 - \frac{U}{U_\infty}\right) dy \tag{2.7}$$

and

$$\theta = \int_0^\infty \frac{U}{U_\infty} \left( 1 - \frac{U}{U_\infty} \right) dy \tag{2.8}$$

where dy is the differential vertical distance.

The expressions for the velocity profile of a boundary layer in turbulent flow are based on both analysis and measurements (Versteeg & Malalasekera, 2007). Turbulent flow along a smooth wall can be considered to consist of multiple layers with different properties.

The layer properties can be described by the non-dimensional variables for velocity and vertical distance from the wall  $u^+$  and  $y^+$  respectively. They are expressed as

$$u^+ = \frac{U}{u_\tau} \tag{2.9}$$

and

$$y^{+} = \frac{\rho y u_{\tau}}{\mu} \tag{2.10}$$

where y is the distance from the wall,  $u_{\tau} = \sqrt{\frac{\tau_w}{\rho}}$  is the shear velocity and  $\tau_w$  is the surface shear stress.

These parameters are used to describe the boundary layer and the inner region can be expressed as

#### CHAPTER 2. THEORY

$$u^{+} = f(\frac{\rho u_{\tau} y}{\mu}) = f(y^{+})$$
(2.11)

where  $f(\frac{\rho u_{\tau} y}{\mu})$  and  $f(y^+)$  are functions of given variables. Equation 2.11 is called the law of the wall.

Far away from the wall, in the outer region of the boundary layer, the velocity at a point can be described by the following expression

$$\frac{U_{\infty} - U}{u_{\tau}} = g(\frac{y}{\delta}) \tag{2.12}$$

where  $g(\frac{y}{\delta})$  is a function of given variables. Equation 2.12 is referred to as velocity-defect law.

The first layer closest to the wall is the viscous sublayer, where viscous stresses dominates. It is also referred to as the laminar sublayer. The velocity profile in this layer is nearly linear. The thickness of the viscous sublayer is small, typically much less than 1.0% of h or  $y^+ < 5$ . The velocity gradient in the viscous sublayer remains nearly constant and the flow velocity only depends on  $y^+$  within the layer. It can be shown that the linear relationship for dimensionless velocity within the viscous sublayer on a smooth wall can be expressed as

$$u^{+} = y^{+} \tag{2.13}$$

After the viscous sublayer comes the buffer layer, where viscous and turbulent stresses are important. Subsequently is the log-law layer where the turbulent stresses dominate. The log-law layer is usually within the region of  $30 < y^+ < 500$  or  $0.02 < y/\delta < 0.20$  where the the shear stress varies slowly with distance from the wall (Versteeg & Malalasekera, 2007). The upper limit of the log layer where the log-law is valid varies and in some cases the limit can be as low as  $y^+ < 100$ . The upper limit of the log-law region may also be  $y^+ > 1000$  for very high Reynold number flow, which is the case for ships and aircrafts (LEAP, 2012). An expression for the non-dimensional velocity for smooth walls within this layer is:

$$u^+ = \frac{1}{\kappa} ln(Ey^+) \tag{2.14}$$

where the von Kármán constant  $\kappa = 0.41$  (Ferziger & Perić, 2002; ANSYS, 2013) and the log-law constant is E = 9.793 in ANSYS CFX (ANSYS, 2013) for smooth walls. The equation is referred to as the log-law. Wall roughness cause a decrease in the value of E (Versteeg & Malalasekera, 2007) and hence lessened velocity increase with higher value of  $y^+$  compared to smooth wall. This result in thicker boundary layer. There is close agreement between experimental data for boundary layer over a smooth wall, and equation 2.13 and 2.14 as seen in figure 2.3.



Figure 2.3: Non-dimensional velocity distribution near a smooth wall adapted from Versteeg and Malalasekera (2007)

The log-law function is used in Computational Fluid Dynamics software such as ANSYS CFX when wall function is selected. The wall function is used to save computational resources by acting as a model for the inner boundary layer flow. This requires a courser mesh compared to if a wall function is not used, hence lower computational costs.

After the log-law layer is the outer layer where inertial effects dominate and viscous effects are negligible. An expression for the non-dimensional velocity profile is called law of the wake and can be expressed as

$$\frac{U_{\infty} - U}{u_{\tau}} = \frac{1}{\kappa} ln(\frac{y}{\delta}) + A^*$$
(2.15)

where  $A^*$  is a constant.

Log-law and law of the wake have the same value at the point of transition between them.

### 2.4 Conservation of mass and momentum

A fluid particle will respond to a force in a similar fashion of a solid particle. If a force is applied to a particle, acceleration will be the result as governed by Newton's second law of motion. The governing equations for incompressible and isothermal fluid flow are the continuity equation and momentum equations also known as Navier-Stokes equations. The continuity equation forces the mass for a closed system to remain constant over time. No mass are created or disappear within this closed system. Navier-Stokes equations states that the inertial forces acting on a fluid element are balanced by the surface and body forces (Yunus & Cimbala, 2006). The equations for conservation of mass and momentum can be expressed as tensors:

#### CHAPTER 2. THEORY

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2.16}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\nu}{\rho} \frac{\partial^2 u_i}{\partial x_i^2}$$
(2.17)

where  $i, j = 1, 2, u_i$  and  $u_j$  are the Cartesian velocity components, t is the time, and  $x_i$  and  $x_j$  are Cartesian direction components.

The influence of gravitational forces on the present fluid flow problem is negligible, hence not included in the equations.

### 2.5 Forces acting on the body

The problem consists of a cross-section of a two-dimensional square body immersed in a turbulent boundary layer. As the fluid moves over the solid body, it exerts pressure forces normal to the surface and shear forces parallel to the surface of the body. Twodimensional flow may be valid if the body is sufficiently long and the flow is normal to the body. End effects are exceedingly relevant as the length compared to the height and width of the structure gets lower.

The drag force  $F_D$  consists of the resulting pressure and shear force in the flow direction while the lift force  $F_L$  is the resulting force in the normal direction of the flow. The drag and lift force can be expressed by the following integral form:

$$F_D = \int_A (-p \cos(\theta_n) + \tau_w \sin(\theta_n))$$
(2.18)

$$F_L = \int_A dF_D = -\int_A (p \, \sin(\theta_n) - \tau_w \sin(\theta_n)) \tag{2.19}$$

where  $dF_D$  is the differential drag force,  $dF_L$  is the differential lift force, dA is the differential area and  $\theta_n$  is the angle that the outward normal of dA makes with the positive flow direction.

The velocity field and the force on the body are influenced by the shape of the body. Bodies may be classified as being streamlined or bluff. A streamlined body is made to align its body with the streamlines in the flow. Examples of these bodies are airplanes and submarines. A bluff or blunt body is characterized by blocking the fluid and the flow tend to separate from the body rather than to follow its shape. Examples of blunt bodies are buildings and buses. Streamlined bodies tend to have less drag force in comparison to blunt bodies when subjected to fluid flow. The shear force component is small compared to the pressure component for the total drag of a blunt body while more important for the total drag of a streamlined body. All structures or bodies used in the present numerical simulations are bluff, hence the pressure component is dominant. The viscous component

#### CHAPTER 2. THEORY

can be neglected for flat bottom surface mounted rectangular structures with b/h < 6 subjected to boundary layer flow (Fujimoto et al., 1975).

One way of expressing the total drag and lift is by force coefficients. This is convenient since the coefficients can be used to find the force for various dimensions of a structure with the same geometry subjected to similar flow conditions. The drag and lift coefficients are  $C_D$  and  $C_L$  respectively.

The expressions for  $C_D$  and lift coefficient  $C_L$  are:

$$C_D = \frac{F_D}{\frac{1}{2}\rho U_\infty^2 A} \tag{2.20}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho U_\infty^2 A} \tag{2.21}$$

where A is the projected front area of the body (Yunus & Cimbala, 2006).

The non-dimensional pressure along the body  $C_p$  is calculated along the path S. The fluid pressure can be expressed with a pressure coefficient as follows:

$$C_p = \frac{p - p_0}{\frac{1}{2}\rho U_{\infty}^2}$$
(2.22)

where  $p_0$  is defined as the inlet pressure  $\delta$  from the wall.
# Chapter 3

# **Computational Fluid Dynamics**

The purpose of this chapter is to give an understanding of CFD, meshing basics and turbulence modeling. The  $k - \epsilon$  turbulence model is described and related equations illustrated.

# 3.1 Introduction to CFD

Computational Fluid Dynamics are useful in a wide variety of applications. CFD can be used to simulate the flow over vehicles, structures submerged in water and inside pipes. It can also be used to simulate temperature distributions in engines or mixing manifolds and numerous other flow problems.

Numerical algorithms are used to tackle the fluid flow problems. Most commercial CFD packages includes a user interface where problem parameters can be set and results can be shown. ANSYS CFX is an example of such a commercial CFD code.

Working with CFD involve three main elements which are pre-processing, solving and post-processing. Pre-processing is where the geometry is defined. Computational domain and mesh generation is also part of this process. A mesh, or a grid, is the cells which the domain is divided into where each cell is part of the discretization of the domain. A cell is also referred to as an element. More accurate numerical results are obtained by increasing the amount of elements in general. Finer mesh is required in areas where larger velocity and pressure gradients occur. Higher computational cost is associated with finer mesh, hence the resolution of mesh is governed by the required numerical accuracy and available computational cost. Over 50% of the time spent on solving CFD problems in the industry is spent making the geometry and mesh (Versteeg & Malalasekera, 2007).

Fluid properties and boundary conditions are also specified in the pre-processing part. Many commercial CFD software packages use the Finite Volume Method (FVM). ANSYS CFX use the a vertex-based FVM approach for discretization. Control volumes are constructed around each cell node, or corner, where the fluid variables are stored. RANS equations are integrated over each control volume and discretized equations are solved (Stenmark, 2013). This is the solving part of a CFD problem. The Post-processing involves extraction and visualization of results.

CFD is attractive to the industry since it is in many cases more cost-effective than physical testing (Sayma, 2009). However, one must note that complex flow simulations are errorprone and it takes correct input of mesh, domain, geometry, boundary conditions and good sense of physical understanding and preferably reference experiments to obtain reasonable results.

## 3.2 ANSYS CFX

ANSYS 15 with ANSYS CFX is applied as the solver of the fluids governing equations. ANSYS CFX is a commercial software and is a general purpose fluid dynamics program that has been used for over 20 years. It has a user friendly GUI and the ANSYS Workbench platform connects the different software packages for making geometry, mesh, solve and post-process results by drag and drop on the screen. It contains numerous solvers and utilities covering a wide range of fluid flow problems. Figure 3.1 illustrates the structure of Workbench used in ANSYS 15:



Figure 3.1: Setup in ANSYS Workbench and the connection between the software packages

As seen in figure 3.1, the whole fluid problem is divided in sub-programs. ANSYS ICEM CFD is the software used for making the geometry and mesh. Information from ANSYS ICEM CFD is further used in the setup of the fluid problem in ANSYS CFX-pre. Fluid properties, flow conditions, boundary conditions, turbulence model, solver and output control are defined in ANSYS CFX-pre. The solver of the governing equations are performed in ANSYS CFX-Solver Manager. The amount of processor cores used for solving and initial values are specified in ANSYS CFX-Solver Manager. Momentum and mass residuals as well as drag and lift are shown during the numerical simulation. When residual target is reached, and drag and lift coefficient have become stable, ANSYS CFD-Post is used for post processing and visualization of results.

### CHAPTER 3. COMPUTATIONAL FLUID DYNAMICS

# 3.3 Mesh

The flow is solved and values of velocity and pressure is calculated for each element in the mesh. For three-dimensional mesh, the cells are enclosed volumes. Information from cells are used in the neighboring elements. The grid has a significant impact on rate of convergence, solution accuracy and CPU time required. Relevant factors which influence this are grid density, skewness, non-orthogonality, cell growth rate and aspect ratio (Bakker, 2006).

Typical cell shapes for two-dimensional mesh used in research and industry are triangular elements and quadrilateral elements. The latter give more accurate results with less elements compared to triangular elements and is almost always used close to a wall where boundary layer need to be resolved, or modeled by a turbulence model. These elements are applied for the present analysis.

There are two types of grids, structured and unstructured grid. A structured grid can be numbered according to indices  $i^*$  and  $j^*$  that do not necessarily correspond to coordinates  $x_1$  and  $x_2$  (Yunus & Cimbala, 2006). Structured grids can be in a single block or a multiblock arrangement where each block contains a structured mesh and are connected to other blocks. It is also possible to divide blocks into structured and unstructured meshes. An unstructured grid consists of cells of various shapes which are arranged in an arbitrary fashion. They are usually made by specifying areas where a minimum and maximum cell size is going to be constructed, then the computer auto-generates the mesh. Unlike the structured grid, one cannot uniquely identify cells in an unstructured grid by the indices  $i^*$  and  $j^*$ . A structured grid can in some cases be constructed in such way that less cells are needed compared to unstructured grid which results in less computational cost. A structured mesh of excellent quality is also associated with more reliable and consistent results. Figure 3.2 illustrates a structured and unstructured mesh for quadrilateral cells.



Figure 3.2: Structured and unstructured quadrilateral grid modified from Yunus and Cimbala (2006)

## 3.4 Turbulence model

Turbulence may be defined as nearly random fluctuations in velocity and pressure in both space and time. The behavior of the turbulent regime can be found by experiments or Direct Numerical Simulations (DNS).

DNS use computers to resolve the unsteady Navier-Stokes equations, resolving all flow details such as the fluctuations of velocity and pressure. The grids applied for DNS are sufficiently fine so they can resolve the smallest eddies where energy dissipation takes place. To resolve these fluctuations, t needs to be small enough to resolve the period of the fastest fluctuations. These requirements result in high computational costs, hence the method is not used for industrial flow computations (Versteeg & Malalasekera, 2007).

Large Eddy Simulation computes the larger eddies and model the small eddies of the flow. This is obtained by filtering the unsteady Navier-Stokes equations before the computations start and only compute eddies above curtain size. The method is associated with less computational cost than DNS, although higher compared to the RANS method.

RANS equations are used for CFD codes to reduce the time used to solve the flow problem. RANS equations are a time average of the Navier-Stokes equations which use average values of small fluctuating velocities to simplify the problem. A fluctuating Reynolds stress component  $\overline{u'_i u'_j}$  comes from time averaging the Navier Stokes equations, and this term is modeled by applying a turbulence model.

In the present study the standard  $k - \epsilon$  turbulence model (Launder & Spalding, 1974)

#### CHAPTER 3. COMPUTATIONAL FLUID DYNAMICS

is applied to solve the RANS equations. The Reynolds-Averaged Navier-Stokes equations which need to be solved based on conservation of mass and conservation of fluid momentum is given by:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{3.1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\mu}{\rho} \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\delta u'_i u'_j}{\delta x_j}$$
(3.2)

An expression of  $\overline{u'_i u'_j}$  is:

$$-\overline{u_i'u_j'} = \frac{\mu_t}{\rho} \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3\rho} k \delta_{ij}$$
(3.3)

where  $\delta_{ij}$  is the Kronecker delta.

The  $k - \epsilon$  turbulence model assumes that  $\mu_t$  is linked to the turbulent kinetic energy k and the rate of dissipation of turbulent kinetic energy  $\epsilon$  as seen in the following equation

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon} \tag{3.4}$$

The  $k - \epsilon$  turbulence model consists of two equations, one for  $\epsilon$  one for k. The differential operator of  $\epsilon$  and k with respect to t may be expressed as tensors in the following way:

$$\frac{D\epsilon}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{C_1 \mu_t}{\rho} \frac{\epsilon}{k} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - C_2 \frac{\epsilon}{k}$$
(3.5)

$$\frac{Dk}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + \frac{\mu_t}{\rho} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \epsilon$$
(3.6)

where the value of the coefficients are:  $C_1 = 1.44$ ,  $C_2 = 1.92$ ,  $C_{\mu} = 0.009$ ,  $\sigma_k = 1.0$  and  $\sigma_{\epsilon} = 1.3$  (Launder & Spalding, 1974).

## 3.5 Courant-Friedrichs-Lewy number

It is not possible to carry out an exact stability analysis for the Euler or Navier-Stokes equations, hence a stability analysis of simpler model equations is used to approximate stability. This is a numerical scheme. There is usually a maximum allowable time-step which makes the numerical scheme unstable and the numerical error would grow exponentially. There are two types of numerical schemes, explicit and implicit. Courant-Friedrichs-Lewy (CFL) number, also referred to as the Courant number, usually needs to be lower than or equal to 1 in the whole domain for an explicit scheme (Ferziger & Perić,

#### CHAPTER 3. COMPUTATIONAL FLUID DYNAMICS

2002). Courant number is defined as the number of elements a fluid particle travels in one time-step and can be expressed as:

$$C = U \frac{\Delta t}{\Delta x} \ge C_{max} \tag{3.7}$$

where  $\Delta t$  is the time step,  $\Delta x$  is the stream-wise length of the element and  $C_{max}$  is the maximum value before the solution diverge. This requirement yields for the whole domain.

Implicit schemes applied to Euler or Navier-Stokes equations allows larger Courant number than explicit schemes. It is often advantageous to set the Courant number as large as possible within the limits of stability to obtain the fastest convergence. Changing  $\Delta t$ may alter the results to some extent, hence a time step convergence study is helpful for studying this effect.

The Second Order Backward Euler scheme is applied for transient simulations due to its robustness and second order accuracy in time (ANSYS, 2013). The numerical scheme is implicit and does not have a time step size limitation.

## **3.6** Post-processing

The solution is converged when the hydrodynamic quantities  $C_D$  and  $C_L$  have become stable at a close to constant value.

The hydrodynamic quantities  $C_D$  and  $C_L$  are accessed from CFX-Solver Manager. The values of  $x_R$ , p and  $u_1$  are obtained from ANSYS CFD-Post.

Tecplot360 is used for visualization of the results, such as illustrating velocity field, pressure field and streamlines. All graphs are created with SigmaPlot and drawings are created in Visio.

# Chapter 4

# Numerical set-up

This chapter explains the domain, mesh, boundary conditions and numerical boundary layer used. The same domain and boundary conditions are applied for all simulations while the geometry is changed to obtain the desired values of  $\delta/h$  and b/h. This is also the case for the simplified geometries of GRP covers.

## 4.1 Domain

A computational domain is defined for every CFD problem and the size of the domain is of great importance when it comes to accuracy of the solution and computational costs for external flows. If the computational domain is made too small, the surfaces of the domain may influence the results such as the hydrodynamic quantities  $C_D$  and  $C_L$  on a structure. The outcome of an exceedingly large domain are additional cells and unnecessary additional computational costs. Due to the optimal choice of a computational domain, it is important to use a domain size which is tested on similar flow problems or perform a domain size convergence test. A domain size convergence test is where various domain size parameters are changed to see how much it affects the results of the problem. The parameters can be the length from inlet to obstacle of interest, height of domain and length from obstacle of interest to the outlet.

The computational domain used for the square and rectangular cross sections of a long structure is shown in figure 4.1. The unit of length shown for the domain size is  $\delta$  due to the decision of using a constant value of boundary layer thickness and change b and h to obtain right  $\delta/h$  and b/h ratio. The domain size is selected based on the size of experimental setup that results are compared with, previous numerical experiments and a domain convergence study on the height of the domain.

The results from Ong et al. (2010) indicates that numerical simulation of flow around a circular cylinder close to a flat seabed at high Reynolds numbers using a  $k - \epsilon$  model can be physically sound. The domain used by Ong et al. (2010) had a distance from the inlet to the center of the cylinder of 10 cylinder diameters. This is sufficient length to avoid far field effects on the flow upstream of the cylinder, hence it is assumed that the inlet

needs to be equal or longer than this value to neglect far field effects in the present study. The distance is set to be  $18\delta$  or 13.1h of the largest structure in the present simulations to make sure no far field effects are present.

Wind tunnel experimental data (Fujimoto et al., 1975) is used for comparison of the numerical results. The wind tunnel used in these experiments have a height of  $23.5\delta$ . The roof of the wind tunnel is made of flexible ceiling which may remove some of the longitudinal pressure gradient. The experimental set-up is adjusted so that the blockage effect due to the presence of the structure or obstacle is small.

Previous numerical simulation (Ong et al., 2010) comments that the distance from the center of the cylinder and out to the top boundary may vary from 8.5 to 9.4 cylinder diameters without having an effect on the flow around the cylinder and the flat seabed.

To ensure that the height of the domain did not effect the solution, a domain size convergence test was conducted. Domain sizes of  $30\delta$ ,  $40\delta$  and  $50\delta$  were used in the numerical simulations. The relative change of drag coefficient is less than 1.0% when increasing the height of the domain from  $30\delta$  to  $40\delta$ . The change of  $C_L$  is 1.3%. If the height of the domain is increased further from  $40\delta$  to  $50\delta$ , the change in drag and lift is negligible. A domain size of  $40\delta$  is chosen due to no significant change in drag by increasing the height of the domain.



Figure 4.1: Computational domain used for square and rectangular structures as well as cross section of simplified GRP covers

The length of the computational domain from the front of the structure and the outlet is set to be  $55\delta$  or 40.1h where h is the height of the largest structure used. This is more than sufficient according to previous numerical simulation (Ong et al., 2010) where the distance from the center of the cylinder to the outlet is 20 diameters to eliminate the far field effects on the structure.

The aforementioned domain is used for mesh convergence study and further analysis for

a square and rectangular cross section as well as the shape of a GRP Cover.

## 4.2 Mesh set-up

The mesh set-up in the present study uses a structured mesh in multiple blocks. This makes it simple for the user to define mesh properties in each block. Making a good quality mesh is essential for consistent and reliable results.

Four measures of mesh quality of each cell is skewness, orthogonality, change in size between neighboring cells and aspect ratio. Skewness for a quadrilateral element is based on the deviation of the inner angle of the elements from being  $90^{\circ}$ . Orthogonality is based on how close the angles between adjacent element edges, or adjacent element faces, are to some optimal angle. This optimal angle is 90° for quadrilateral elements. Change in element size between neighboring size should be smooth and typically less than a ratio of 1.2. Aspect ratio is the length of the elements longest edge divided by the shortest edge. The ideal ratio should be equal to one for a quadrilateral element, but modern CFD codes, such as ANSYS CFX, can handle much larger aspect ratios. The largest aspect ratio for any cell in the present study is kept lower than 100 for all the final mesh configurations used. ANSYS CFX allow aspect ratios up to 1000 (ANSYS, 2013). It is beneficial to have the cell aspect ratio close to one where flow is multi-dimensional, but can be stretched in the same direction of the flow if it is one-dimensional which is the case close to a wall. This is practical when making mesh in boundary layer region along the sea bed or bottom of the domain. Table 4.1 illustrates mesh quality in terms of skewness. Skewness is defined in a spectrum ranging from 0 to 1, where 0 is zero skew and a spectrum value close to 1 is badly skewed cells. Usually there is a connection between the quality of elements in terms of skewness and orthogonal quality.

Table 4.1: Skewness mesh metrics spectrum adapted from Bakker (2006)

Excellent	Good	Acceptable	Poor	Sliver	Degenerate
0 - 0.25	0.25 - 0.50	0.50 - 0.80	0.80 - 0.95	0.95 - 0.99	0.99 - 1.00

Skewness should not exceed 0.85 for quadrilateral and hexahedral elements. To make sure that the mesh in the present study is of excellent quality, skewness is kept below 0.25 for all numerical simulations. This is defined as excellent quality in terms of skewness from table 4.1.

As already mentioned, ANSYS ICEM CFD is the software used for making the twodimensional mesh for the present study. A bottom-up mesh is applied where vertices, or points, are first created and then connected with edges, also referred to as lines, to create the geometry and the domain. ANSYS CFX can only implement a three-dimensional mesh, hence a three-dimensional input needs to be created from the two-dimensional problem. This is solved by adding a thickness of the domain and use only one element in this direction.

A multi-block structured mesh is used in the present study. A block-topology used for square is illustrated in figure 4.2 and 4.3. The square cross section is colored grey and each enclosed area with black lines is a block where the mesh properties is controlled such as number of elements, spacing and growth rate. Similar block topology is applied for rectangular cross section. Compatibility is assured where the blocks meet.

To keep cell count down, and maintain the same amount of accuracy, a fine mesh of good quality is applied at areas of interest such as around the square cross section of the structure. The enclosed areas close to the structure is 0.2h out from the structure. They ensure good control over the cell height of the first cell close to the wall so that the value of  $y^+$  is kept between 30 and 32. Zero growth rate of the cells within 0.2h of the wall is applied. This is to maintain the level of refinement necessary for accurate results. All cells in the mesh for the square and rectangular structures have zero skew and perfect orthogonal quality.



Figure 4.2: The block-topology used for the whole domain of square cross section



Figure 4.3: The block-topology used close to the square cross section

An illustration of the block topology used for the GRP cover referred to as GRP 1 is seen in figure 4.4. The blocks close to the wall upstream of the structure and the wall in the wake of the structure are larger compared to the same blocks for square and rectangular cross sections. These are the only blocks that have elements which are slightly skewed.

The skewed elements is due to the shape of the structure. A similar block topology is applied for the GRP cover referred to as GRP 2.



Figure 4.4: The block-topology used close to GRP 1

Two additional blocks were necessary when including the mudmat in the geometry of GRP 1 as seen along the bottom surface in figure 4.5. They are made to obtain control over the first cell height close too the flat seabed and the cell count along the height of the mudmat.



Figure 4.5: The block-topology used close to GRP 1 with mudmat

When using  $k - \epsilon$  turbulence model, a wall function is applied close to the walls. The boundary layer is not resolved, but modeled by using the log-law equation. This is an attractive method as less computational cost is achieved due to fewer cells necessary compared to resolving the boundary layer. The height  $h_p$  of the first cell from the wall is relevant towards the type of turbulence model used, and if wall function or full boundary layer resolution is going to be practiced. The definition of  $h_p$  is illustrated in figure 4.6. The figure show a gradual growth rate after the first cell.



Figure 4.6: Mesh close to a wall with illustration of  $h_p$ 

High Reynolds RANS models produce incorrect results in the viscous sublayer at  $y^+ < 10$  or 20 (Code-Saturne, 2014). Scalable wall functions is used in ANSYS CFX to reduce this problem. The scalable wall function ensures that all mesh points are outside the viscous sublayer for  $k - \epsilon$  turbulence model (ANSYS, 2013).

The first node in the present numerical simulation is set to be close to the recommended value of  $y^+$  for RANS code with  $k - \epsilon$  model, which is close to 30 (Code-Saturne, 2014). The mean value  $\overline{y}^+$  of dimensionless height of the first cell closest to the wall are set to be between  $30 < \overline{y}^+ < 32$  for the flat bottom surface and the wall of the structure. This strict value of  $\overline{y}^+$  is applied for all present numerical simulations. The value of  $\overline{y}^+$  is found from the converged solution when  $C_D$  and  $C_L$  have become stable. The growth rate of the following cells need to be smooth to obtain stable and reliable results.

A mesh is created with the focus of making a fine and good quality grid towards the structure, and gradually let the mesh get courser further away from the body. Layers of rectangular elements parallel to the flow direction are used to resolve the boundary layer along the flat bottom surface. This ensure accurate and reliable results with a minimum of computational costs. The global mesh of the square structure at b/h = 1 with  $\delta/h = 1.70$  is illustrated in figure 4.7. The local mesh of the same case is shown in figure 4.8. Figure 4.9 demonstrates the local mesh for rectangular cross section when b/h = 5 with the same boundary layer height to the height of the structure as for square cross section.



Figure 4.7: Global mesh for square cross section at  $\delta/h = 1.70$ 



Figure 4.8: Local mesh for square cross section at  $\delta/h = 1.70$ 



Figure 4.9: Local mesh for rectangular cross section at b/h = 5 and  $\delta/h = 1.70$ 

Perfect square shaped cells are chosen at the corners where the largest pressure and velocity gradients occur. There is no growth rate for the cells within 0.2h of the wall and a slight increase of cell size after that. This is important when dealing with high Reynolds number flow in terms of getting stable results.

A mesh structure similar to the one used for square and rectangular cross sections is applied for the GRP covers. The grid close to the structure is illustrated in figure 4.10, 4.11 and 4.12 for GRP 1, GRP 1 with mud mat and GRP 2 respectively.



Figure 4.10: Local mesh for GRP 1



Figure 4.11: Local mesh for GRP 1 with mudmat



Figure 4.12: Local mesh for GRP 2

One of the differences between the mesh used for GRP covers and the square structures, are the presence of skewed and non-orthogonal cells close to the sides of the covers. This is not regarded as a problem due the value of skew and non-orthogonality are within recommended limits with a good margin.

## 4.3 Boundary Conditions

Initial and boundary conditions are specified in CFD problems and they are an important part of modeling the physics correctly. Inlet, outlet, wall and symmetry boundary conditions are specified in ANSYS CFX-pre for the present study, and an overview over the boundary conditions are illustrated in figure 4.13.



Figure 4.13: Boundary conditions applied

A log profile for the boundary layer flow is specified in the inlet for the value of  $u_1(y)$ . The vertical velocity has a value of  $u_2 = 0$  at this location. The log profile is taken from curve fitting of an experimental boundary layer profile (Fujimoto et al., 1975). The value of k and  $\epsilon$  can be expressed as functions of y at the inlet (Ong et al., 2010). The expressions are

$$k(y) = max \left\{ C_{\mu}^{-\frac{1}{2}} \left( 1 - \frac{y}{\delta} \right) \left| 1 - \frac{y}{\delta} \right| u_{\tau}^{2}, 0.00001 U_{\infty}^{2} \right\}$$
(4.1)

$$\epsilon = \frac{C_{\mu}^{\frac{3}{4}}k(y)^{3/2}}{\ell}$$
(4.2)

where  $C_{\mu}$  is a turbulent-viscosity constant in the  $k - \epsilon$  turbulence model. The term  $0.00001U_{\infty}$  is specified to ensure that k(y) has some finite small value as y approach  $\delta$  and beyond. The estimate of the turbulent length scale  $\ell$  is expressed by the following equation

$$\ell = \min\left\{\kappa y \left(1 + 3.5 \frac{y}{\delta}\right)^{-1}, C_{\mu} \delta\right\}$$
(4.3)

A wall function is adapted for all numerical simulations along the flat bottom surface and the structure. Equation 2.14 is used as the wall function in ANSYS CFX.

In the outlet boundary the static pressure  $p_{stat} = 0$ .

The top boundary has a free slip boundary condition where  $u_1 = U_{\infty}$  and  $u_2 = 0$  since the top wall is sufficiently far away from the bottom and the structure. The value of  $\tau_w = 0$  at this boundary.

A no slip boundary condition is specified along the flat bottom surface and the wall of the structure. This implies that the velocity  $u_1=u_2=0$ .

The front and back planes have the symmetry boundary condition which suggest the velocity normal to the plane  $u_n = 0$  (ANSYS, 2013). This implies that the fluid does not flow through this boundary.

# 4.4 Numerical Flow Characteristics

To make sure that the results from the numerical simulation could be compared to physical experiments, the flow characteristics have to be similar in both cases. This includes similarity in boundary layer profile,  $\delta/h$  and shape factor of boundary layer  $H = \delta^*/\theta$ . The value of H = 1.36 in the present study is the same as in the experiments performed by Fujimoto et al. (1975). Comparison of boundary layers from numerical simulations and the experiments are illustrated in figure 4.14. The location of the boundary layer is at the upstream face of the structure, with the structure not being present, as performed by Fujimoto et al. (1975).



Figure 4.14: Comparison of numerical boundary layer in present study and experimental boundar layer from Fujimoto et al. (1975) at the location of the upstream facing side of the structure, without the structure itself being present

The numerical boundary layer profile is obtained by curve fitting of the boundary layer

used in the experiments by Fujimoto et al. (1975). The value of  $u_1$  along the vertical distance of the bottom wall to the location where  $u_1 = U_{\infty}$  is taken from the experiments. This boundary layer profile is obtained at the location of the upstream facing side of the structure, without the structure itself being present. The boundary layer profile is implemented in MATLAB curve Fitting Tool, where a logarithmic curve fitting function is used. The log-profile from MATLAB is included in the inlet boundary condition. The resulting boundary layer in the present numerical simulation is in good agreement with the experimental boundary layer, as seen in figure 4.14.

Other factors that need to be considered in the numerical and the experimental set-up are the similarities in  $Re_h$ ,  $Re_{\theta^*}$  and  $u_{\tau}/U_{\infty}$ . These values are set to be the same in the present study as in the experiments where  $Re_h = 1.19 \times 10^5$ ,  $5.12 \times 10^4$  and  $3.41 \times 10^4$  for  $\delta/h = 0.73$ , 1.7 and 2.55 respectively. The values of  $Re_{\theta} = 9380$  and  $u_{\tau}/U_{\infty} = 0.0366$ . A surface roughness to boundary thickness ratio of  $z_w/\delta = 2.43 \times 10^{-4}$  is applied at the flat bottom surface to obtain the desired value of  $u_{\tau}/U_{\infty}$ . The wall of the structure is set to be smooth with no surface roughness.

# Chapter 5

# Square and rectangular structures

This chapter provides further explanation of the numerical set-up for simulation of boundary layer flow over surface mounted square and rectangular structures. Results are illustrated and discussed for  $\delta/h = 0.73$ , 1.70 and 2.55. Numerical simulation where b/h = 1, 3 and 5 at  $\delta/h = 1.70$  are also performed and presented.

## 5.1 Mesh Convergence

To make sure that the solution is sufficiently mesh independent, a mesh convergence study must be performed. This is done by increasing the total amount of elements in the whole domain and observe how the solution change. A mesh convergence study will also make it possible to decide how many cells that are necessary to get the desired numerical accuracy of the results. Few cells can result in large numerical error and too many cells may result in large computational costs. Usually a mesh convergence test involves illustrating a course, medium and fine mesh, where the medium mesh should have shown sufficient converged solution with reasonable computational cost. The CFD analyst will in most cases proceed with the results from the medium mesh due to the optimal balance of numerical accuracy and computational cost.

The mesh convergence study for the square and the rectangular structures with various values of  $\delta/h$  and b/h is illustrated in table 5.1, where  $C_D$ ,  $C_L$  and  $x_R/h$  is shown for each mesh.

Table 5.1: Hydrodynamic quantities from grid convergence study for square and rectangular cross section at given values of  $\delta/h$  and b/h

b/h	$\delta/h$	Elements	$C_D$	$C_L$	$x_R/h$
1	0.73	28571	1.081	0.660	13.68
1	0.73	42767	1.083	0.660	13.64
$1^{*}$	0.73	65136	1.085	0.662	13.58
1	0.73	99515	1.086	0.662	13.56
1	1.70	21751	0.834	0.572	12.64
1	1.70	33618	0.835	0.570	12.64
$1^{*}$	1.70	51989	0.837	0.568	12.63
1	1.70	77039	0.837	0.568	12.61
1	2.55	19591	0.748	0.536	12.19
1	2.55	30575	0.749	0.528	12.19
$1^{*}$	2.55	47315	0.750	0.525	12.20
1	2.55	71243	0.750	0.525	12.21
3	1.70	25003	0.698	1.782	10.37
3	1.70	38302	0.698	1.789	10.33
$3^{*}$	1.70	58869	0.698	1.789	10.30
3	1.70	86135	0.698	1.789	10.27
5	1.70	26283	0.599	2.544	9.67
5	1.70	40262	0.599	2.544	9.63
$5^{*}$	1.70	61894	0.599	2.546	9.60
5	1.70	92885	0.599	2.548	9.60

The present mesh convergence study shows a good mesh independence of the grid since there are little change in global hydrodynamic quantities as the amount of elements are increased. The largest difference of any hydrodynamic quantity observed between mesh refinements is for square cross section when  $\delta/h = 2.55$ . The value of  $C_L$  differ with a relative difference of 2.1% between the courses and finest mesh for this case. This indicates that the grid is of good quality. The scalable wall functions applied in ANSYS CFX may be a contributing factor of the low difference in hydrodynamic quantities between mesh refinements. The reason for using this fine mesh is to ensure that there are sufficient amount of elements close to the structure and the flat bottom surface to capture accurate local flow details. It also ensures that the elements far from the structures avoid a aspect ratio which is greater than 1000. Aspect ratio of more than 1000 is not recommended when using ANSYS CFX (ANSYS, 2013). Further analysis of results from set-up marked with a symbol (\*) are going to be described.

A domain size convergence study is performed for different domain heights to ensure that the hight of the domain does not influence the solution. The results are shown in table 5.2.

Table 5.2: Hydrodynamic quantities from given values of height between the flat bottom surface and the upper boundary when  $\delta/h = 0.73$  and b/h = 1

Height of domain / $\delta$	Elements	$C_D$	$C_L$	$x_R/h$
50	75975	1.085	0.659	13.53
$40^{*}$	65136	1.085	0.662	13.58
30	53355	1.089	0.671	13.60

There is a relative difference of 1.3% for  $C_L$  when increasing the domain height from 30 $\delta$  to 40 $\delta$ . Less than 1.0% relative change in any hydrodynamic quantity is present when increasing the domain height from 40 $\delta$  to 50 $\delta$ . A domain height of 40 $\delta$  is chosen to be able to neglect any influence of the upper boundary layer on the hydrodynamic quantities. The domain height used for further analysis is marked with a symbol (\*).

# 5.2 Results and discussion

## 5.2.1 Force coefficients

Experiments show velocity and pressure fluctuations present for boundary layer flow over two-dimensional long square cross section Liu et al. (2008). They are small compared to the mean velocities and pressures, hence small fluctuations in  $C_D$  and  $C_L$  may be assumed.

A transient simulation of square geometry at  $\delta/h = 1.70$  is performed to see if fluctuations of  $C_D$  and  $C_L$  is present. The time step is set to be  $\frac{\Delta t U_{\infty}}{h} = 0.0127$  which resulted in a stable maximum value of C = 0.46. Fluctuations of hydrodynamic quantities is not found. The present transient and steady state numerical simulation seem to give the same results of hydrodynamic quantities.

The values from the present numerical study of  $C_D$  are compared to published experimental data from Fujimoto et al. (1975). Table 5.3 illustrates  $C_D$  for a square cross section where b/h = 1 for various values of  $\delta/h$ . Table 5.4 shows  $C_D$  for various values of b/h at a constant value of  $\delta/h = 1.70$ . Quantities that characterize the flow such as  $Re_h$ ,  $Re_{\theta}$ ,  $U_{\infty}$ , H,  $\theta$  and boundary layer profile at the location of the structure are the same in the numeric set-up as in the experiments.

Table 5.3: Hydrodynamic quantities for square cross section compared to equivalent values from experiments (Fujimoto et al., 1975)

b/h	$\delta/h$	$C_D$ , present study	$C_D$ , experiment
1	0.73	1.085	0.956
1	1.70	0.837	0.824
1	2.55	0.750	0.754

Table 5.3 shows good agreement between numerical results and experimental data for  $\delta/h \leq 1.70$ . The maximum relative change of  $C_D$  is found to be less than 1.6% for these boundary layer heights. A slight over-prediction seems to be present for  $\delta/h \geq 0.73$  where the relative change is 11.9%. The drag coefficient can be described as a linear function of  $log_{10}(\delta/h)$  up to  $\delta/h \geq 0.73$  according to experiments (Fujimoto et al., 1975), for a square cross section of a long structure. This is only true when  $\delta/h \geq 1.20$  for two-dimensional bluff-plates immersed in turbulent boundary layers (Good & Joubert, 1968). The drag coefficient increases even more rapidly with  $\delta/h$  when  $\delta/h \leq 1.20$  for the bluff-plates. This indicates that the same phenomenon may be present for a two-dimensional square structure as well. The drag coefficient on a two-dimensional square structure can be expected to be 1.2 when  $\delta << h$  with an error in  $C_D$  of  $\pm 20.0\%$  (Blevins, 2003). The value of drag coefficient is below 1.2 from the numerical simulations, which is physically sound since the boundary layer thickness is not small compared to the height of the structures.

Further study of the effect on the hydrodynamic coefficients for various values of b/h when  $\delta/h = 1.70$  is performed. This is due to the accurate results obtained for this boundary layer height compared to experiments. The results are presented in table 5.4.

Table 5.4: Hydrodynamic quantities for square and rectangular cross section with given value of b/h compared to equivalent values from experiments (Fujimoto et al., 1975)

b/h	$\delta/h$	$C_D$ , present study	$C_D$ , experiments
1	1.70	0.837	0.822
3	1.70	0.698	0.627
5	1.70	0.599	0.536

The relative change of the numerical simulations when b/h > 1 seem to increase. There is a maximum relative change of 9.9% for any value of b/h when compared to experimental values.

Figure 5.1 represent  $C_D$  from the present study and experiments as well as  $C_L$  for square cross section with  $\delta/h = 0.73$ , 1.70 and 2.55. Figure 5.2 illustrates the same hydrodynamic quantities for b/h = 1, 3 and 5 when  $\delta/h = 1.7$ .



Figure 5.1: Drag coefficient for square cross section at given  $\delta/h$  from present study and experiments (Fujimoto et al., 1975)



Figure 5.2: Drag coefficient for square and rectangular cross section at  $\delta/h = 1.70$  and given value of h/h from present study and experiments (Fujimoto et al., 1975)

Results from the numerical study seem to be in good agreement with the published experimental data in general. Hence the method is believed to give accurate results for analyzing boundary layer flow over simplified subsea structures such as simplified GRP covers.

## 5.2.2 Pressure

The pressure coefficient is useful for illustrating a non-dimensional representation of pressure along the wall of the structures. This may give a better physical understanding of why different values of  $\delta/h$  result in change of  $C_D$  and  $C_L$ .

A plot of  $C_p$  along the path S for  $\delta/h = 0.73, 1.70$  and 2.55 is seen in figure 5.3. The graph reveals a positive pressure in front of the structure and a negative pressure on the top and at the back. Lower value of  $C_p$  with increasing value of  $\delta/h$  at the front of the structure is observed at the front facing side. An increase in  $C_p$  is present at the downstream side of the structure for increasing value of  $\delta/h$ . These observations indicate that a lower value of  $C_D$  is expected as  $\delta/h$  increases, which is found to be correct from the hydrodynamic quantities obtained. The value of  $C_p$  increases on top of the structure, hence lower value of  $C_L$  occurs with increasing  $\delta/h$ .



Figure 5.3: Dressure coefficient for square cross section at given value of  $\delta/h$ 

Local minimum pressure peaks are observed just behind the upstream corner where S/h = 1 and in front of the downstream corner at S/h = 2, which is physically sound.

In addition to illustrating the pressure coefficient along the wall of the structure, contour plots of non-dimensional pressure close to the structure is seen in figure 5.4.



Figure 5.4: Pressure contour plot of square and rectangular structure when  $\delta/h = 1.7$  for given values of b/h

From figure 5.4 it is observed that there is a sizable area downstream of the cylinder with low pressure which is associated with the largest vortex in the wake of the structure. Figure 5.4 (a) agrees with the plot of  $C_p$  for  $\delta/h = 1.70$  in figure 5.3.

## 5.2.3 Velocity

Local quantities such as velocity along vertical sections at various location close to the structure is found and compared to the equivalent values from experiments. This is relevant towards verifying that the numerical simulations are able to predict the physics of the problem. The horizontal velocity profile along a horizontal line at six locations is illustrated in figure 5.5. The profile from the present study for square where  $\delta/h = 0.73$  is compared to published experimental results from Liu et al. (2008).



Figure 5.5: Time average of horizontal velocity along vertical lines at given location along the horizontal axis from present numerical simulations of square cross section compared to experiments (Liu et al., 2008)

Liu et al. (2008) conducted experiments with square cross section of a long structure with two-dimensional properties at the center of the length in a wind tunnel. It appears as if the numerical simulations are able to predict the physics when looking at figure 5.5. The velocity profile of the present study follows the pattern of experimental profile. The present study simulates flow where  $Re_h = 1.19 \times 10^5$  while the experiments had  $Re_h = 1.32 \times 10^4$ . The value of  $\delta/h = 0.73$  and H = 1.36 in the present study, where Liu et al. (2008) used  $\delta/h = 0.75$  and H = 1.30.

Figure 5.6 show the time average of velocity contours in the stream-wise direction for the same case as in figure 5.5.



Figure 5.6: Time average of horizontal velocity contour plot of square cross section at  $\delta/h = 0.73$ 

A retardation of the flow in front of the structure is clearly visible in figure 5.6 as well as negative flow velocity in the wake of the structure due to the large vortex in this area. The large vortex in the wake of the cylinder force fluid particles to travel over this recirculating flow, resulting in larger fluid velocity as seen in the area where  $U_{1,avg}/U_{\infty} \geq 1.1$ .

### 5.2.4 Streamlines

There are a total of four vortices for all geometries simulated as seen in figure 5.7 and 5.8. Three vortices can be identified from the separation point at the upstream corner of the structure to the flat bottom surface downstream. The first vortex is formed upstream of the structure. A second vortex can easily be seen over the structure for b/h = 3 and 5. This vortex is barely present for b/h = 1 at any value of  $\delta/h$ . The third vortex is the largest vortex in the wake of the structure. The fourth vortex is formed close to the bottom surface and the wall in the wake of the structure. This vortex is barely visible in figure 5.7 and 5.8.

The flow characteristics around square and rectangular cross section of long structures change as  $\delta/h$  and b/h are varied. The size of the recirculating zone from the upstream corner of the structure to the reattachment point downstream on the flat bottom surface

decrease as  $\delta/h$  and b/h increase. This indicates that the flow particles that travel over the recirculating flow have to pass a shorter distance. A larger vortex system can be correlated with more drag force on the structure.

The reattachment point downstream of the structure moves closer to the structure when  $\delta/h$  increase as seen in figure 5.7. The reattachment point is marked with the a streak and a red circle. The length  $x_R/h = 13.58$ , 12.63 and 12.20 for  $\delta/h = 0.73$ ,  $\delta/h = 1.70$  and  $\delta/h = 2.55$  respectively.



Figure 5.7: Streamlines contours over square cross sections for given value of  $\delta/h$ 

The length of the vortex upstream of the structure is reduced with increasing value of boundary layer thickness. When  $\delta/h = 0.73$ , 1.70 and 2.55 the vortex lengths are 0.67*h*, 0.49*h* and 0.46*h* respectively. Experiments were  $Re_h = 1.32 \times 10^4$ ,  $\delta/h = 0.75$  and H = 1.30(Liu et al., 2008) show that the equivalent value is 1.0*h*, which is higher than the present numerical study of 0.67*h* when  $Re_h = 1.19 \times 10^5$ ,  $\delta/h = 0.73$  and H = 1.36. The same experiments found  $x_R/h = 10.75$  which is lower than the present study of  $x_R/h = 13.58$ . The discrepancy may be explained by difference in  $Re_h$ ,  $\delta/h$  and H.



Figure 5.8: Streamline contours over square and rectangular cross section when  $\delta/h = 1.70$  for given value of b/h

There is no reattaching flow on the structure itself for when b/h = 1 and 3, although reattachment do occur for b/h = 5. This is also observed to be true for experiments (Fujimoto et al., 1975). The reattachment point is found to be at  $x_1/h = 2.88$  in present numerical simulations when b/h = 5. The separation point for the recirculation upstream is 0.49*h* from the structure for any value of b/h when  $\delta/h = 1.70$ .

# Chapter 6

# Simplified GRP Cover

This chapter gives a presentation of the results obtained from numerical simulations of boundary layer flow over different geometries of GRP cover. The results from two simplified geometries of, GRP 1 and GRP 2, are discussed. Mudmats are included for GRP 1 to study the effect of its presence on hydrodynamic quantities.

# 6.1 Simplified Subsea Structures

Two different GRP covers are analyzed with geometrical simplifications. A two-dimensional cross section is used in the numerical simulations to save computational costs. The two-dimensional cross sections is also used to be able to compare results to previous quantities obtained from square and rectangular cross sections.

GRP covers have many details such as weights, flanges and other parts that stick out from the trapezoid cross section. Modeling these parts require detailed design and fine mesh at the location of details to get reliable results. The outcome is higher computational cost and the hydrodynamic quantities obtained may not be much different than for an analysis of a geometry with the same global shape properties.

Two-dimensional flow properties are present at the middle cross section when l/h > 6 (Martinuzzi & Tropea, 1993). As the aspect ratio l/h decrease, more fluid is channeled along the sides of the structure and correspondingly less must flow over the top. The results are a lower value of  $x_R/h$  and a shorter reattachment length of the fluid on the structure itself. Reattachment on the structure may occur at b/h < 4 when the structure is three-dimensional. As found from the present study of boundary layer flow over square cross sections, larger recirculation zone in the wake of the cylinder is associated with higher values of  $C_D$  and  $C_L$ . This indicate that two-dimensional simulation of GRP covers give conservative results in terms of engineering design and stability analysis.

The geometry of two GRP covers are provided by Subsea 7 and they are illustrated in in figure 6.1. These covers are used as baseline for the simplified version implemented in the numerical simulation. Figure 6.2 and 6.3 show the cross section and dimensions in millimeters of GRP 1 and 2 respectively.



Figure 6.1: Detailed three-dimensional drawing of GRP covers used in present study



Figure 6.2: Detailed cross section including dimensions of GRP 1



Figure 6.3: Detailed cross section including dimensions of GRP 2

GRP 1 and 2 have the aspect ratio l/h = 5.36 and l/h = 7.53. This is somewhat short for the assumption of two-dimensional flow which may result in conservative values of hydrodynamic quantities. In some cases the GRP covers are connected in a series of covers, making the value of l/h > 10, and hence the flow at the middle cross section can be assumed fully two-dimensional.

The simplified geometry of GRP 1 and 2 can be seen in figure 6.4 and 6.6 respectively. A numerical simulation of GRP 1 including mudmats is performed to investigate the effect of the mudmat on the hydrodynamic quantities. Its geometry is illustrated in figure 6.5. Mudmats are used to provide additional support for equipment on the sea floor when the seabed is too soft to support the equipment (Tech-Fab, 2015). The present mudmat is an integrated part of the GRP cover. All dimensions are based on the geometry of the GRP covers provided by Subsea 7.



Figure 6.4: Geometry and dimensions for simplified version of GRP 1



Figure 6.5: Geometry and dimensions for simplified version of GRP 1 with mudmats



Figure 6.6: Geometry and dimensions for simplified version of GRP 2

Sharp edges are modeled on the GRP covers instead of rounded corners so that the separation point are fixed at these locations. This makes it easier to evaluate the hydrodynamic quantities against the results for square and rectangular cross sections. This is due to the geometric similarities. Similar numerical set-up as for the square and rectangular cross section is applied for the simulation of GRP covers since this method was proven to give reliable result when compared to the experiments. The same boundary layer profile is applied with  $\delta/h = 1.70$  since simulations where  $\delta/h \ge 1.70$  have proven  $C_D$  to be close to experimental values. This is an indication that the method is well suited for simulations of GRP cover.

## 6.2 Mesh Convergence

A mesh convergence study was performed for the numerical simulations of the GRP covers. Hydrodynamic quantities for the GRP 1 and 2 are seen in table 6.1. The mesh convergence study for GRP 1 including mudmat is illustrated in table 6.2.

Table 6.1: Hydrodynamical quantities from grid convergence study for cross sections of simplified GRP covers

b/h	$\delta/h$	Elements	$C_D$	$C_L$	$x_R/h$
1.60	1.70	25879	0.797	0.759	12.37
1.60	1.70	39878	0.797	0.762	12.30
$1.60^{*}$	1.70	62327	0.799	0.763	12.23
1.60	1.70	91311	0.799	0.763	12.23
3.87	1.70	28439	0.614	1.975	9.80
3.87	1.70	43798	0.613	1.979	9.73
$3.87^{*}$	1.70	68377	0.612	1.982	9.70
3.87	1.70	100677	0.612	1.983	9.70

Table 6.2: Hydrodynamical quantities from grid convergence study for cross sections of GRP 1 with mudmat

b/h	$\delta/h$	Elements	$C_D$	$C_L$	$x_R/h$
1.60	1.70	26494	0.796	0.762	12.30
1.60	1.70	40740	0.797	0.763	12.30
$1.60^{*}$	1.70	61774	0.797	0.764	12.23
1.60	1.70	91396	0.799	0.764	12.23

The mesh convergence study clearly proves mesh independence on global hydrodynamic quantities. The mesh refinements result in a maximum relative difference less than 1.0% for  $C_D$  and  $C_L$  between the coarsest and finest mesh for all simulations of GRP covers. The maximum relative difference between coarsest and finest mesh of  $x_R/h$  is for GRP 1 with 1.1%. A symbol signals the set-up used for further analysis (\*).

# 6.3 Results and discussion

## 6.3.1 Force Coefficients

The hydrodynamic coefficients obtained from simulation of boundary layer flow over the GRP covers is seen in table 6.3. The hydrodynamic quantities of GRP 1 with and without a mudmat is also seen in the table.

Geometry	b/h	$C_D$	$C_L$	$x_R/h$
GRP 1	1.60	0.799	0.763	12.23
GRP 1 with mudmat	1.60	0.797	0.764	12.23
GRP 2	3.87	0.612	1.982	9.70

Table 6.3: Hydrodynamic quantities for GRP covers

The value of  $x_R/h$  is the same for GRP 1 with and without mudmat although  $C_D$  and  $C_L$  have a relative change of less than 1.0%. This suggests that the effect of including a mudmat can be neglected. The resulting hydrodynamic quantities  $C_D$  and  $C_L$  for GRP 1 and 2 is compared to equivalent values obtained from square and rectangular structures. The comparison is presented in figure 6.7 where the square and rectangular geometry is defined as simple geometries.



Figure 6.7: Drag and lift coefficient for square, rectangle and GRP covers at given value of b/h from present study and experiments (Fujimoto et al., 1975)

Resulting  $C_D$  and  $C_L$  from the simulation of boundary layer flow over simplified geometry of GRP cover seem to be correct when compared to hydrodynamic quantities obtained for square and rectangular cross section. GRP 1 has  $C_D = 0.797$  with b/h = 1.60, which are between the equivalent values for square structure where  $C_D = 0.837$  and the result for rectangular structure  $C_D = 0.696$  with b/h = 3. The lift force on GRP 1 also seem to be between equivalent value as for square and rectangular structures at b/h = 1 and b/h = 3. Similar behavior of  $C_D$  and  $C_L$  for GRP 2 is also found.

## 6.3.2 Streamlines

Streamlines are plotted for GRP 1 and 2. Streamlines are are also plotted for GRP 1 including mudmat. The plots are illustrated in figure 6.8. The structure of the streamlines is clearly similar between figure 6.8 a) and b). This is expected for GRP 1 both with and without mudmats due to similarity in  $C_D$  and  $C_L$ .
#### CHAPTER 6. SIMPLIFIED GRP COVER



Figure 6.8: Streamlines contours over GRP covers

There is no reattachment of the flow on GRP 1, while reattachment occur for GRP 2 at  $x_1/h = 2.96h$ . This is the equivalent of 2.75*h* after the separation on the upstream corner, and 4.5% shorter length compared to the reattachment length on a rectangular structure with b/h = 5. This may be due to the angle on which the horizontal velocity component has with the upstream wall of the geometry of GRP cover. According to experiments (Liu et al., 2008), reattaching flow on the structure occur for b/h = 4 and does not happen for b/h = 3. The width of GRP 2 is between these values, hence the reattaching flow on the structure seem to be physically sound.

## Chapter 7

### Conclusion

Numerical simulations of boundary layer flow over simplified subsea structures on a flat seabed have been performed. A two-dimensional RANS method with a standard high Reynolds number  $k - \epsilon$  turbulence model have been applied.

Resulting values of  $C_D$  are in good agreement with published physical experiments for  $\delta/h = 0.73$ ,  $\delta/h = 1.70$  and  $\delta/h = 2.55$ . Different values of width to height ratio for  $\delta/h = 1.70$  also prove similar results between numerical simulation and experiments. Reattachment is present when b/h = 5 while not present for b/h = 3, as observed in published experiments. Local velocity properties for square cross section with  $\delta/h = 0.73$  from present study is also validated against published experiment. The resulting trend from the present study seem to agree with the experiment.

Results from numerical simulations show excellent agreement of  $C_D$  when compared to experiments for  $\delta/h \geq 1.70$ , hence further study of various width to height ratio b/h when  $\delta/h = 1.70$  is performed. The resulting value of  $C_D$  from the numerical simulations indicated a slight over prediction when  $\delta/h \leq 0.73$ .

Transient (Unsteady RANS) and steady-state (RANS) simulations are carried out. The steady-state runs did not have fluctuating values of  $C_D$  and  $C_L$  in any simulation, indicating that two-dimensional boundary layer flow over simple structures with sharp corners may be treated as a stationary problem.

Resulting hydrodynamic quantities for simplified geometries of GRP covers are found to be similar to equivalent values obtained from square and rectangular cross sections. It is also found that mudmat can be neglected in further analysis of GRP 1.

The numerical method used is capable of producing physically sound hydrodynamic quantities for simplified subsea structures on the seabed. This is relevant for GRP covers which may move due to hydrodynamic forces from extreme currents and the added weight needed to avoid displacement.

# Chapter 8

## **Recommendations for Further Work**

Boundary layer flow around subsea structures on a flat seabed is a relevant topic since the amount of subsea eqipment used is escalating world wide. Using GRP covers for protection of this equipment have also increased over the last decade. The stability of these light covers is an issue the industry addresses, and there is uncertainty of how much weight these covers need in order to avoid being moved by current and wave induced velocities.

Various boundary layer thickness is occurring along the seabed. The present study indicates that smaller boundary layer thickness compared to the height of the structure result in larger drag and lift forces. Hence it would be interesting to analyze smaller values of  $\delta/h$  to gain more knowledge about the possible maximum forces on the covers.

The largest uncertainty is associated with the lift force from extreme current or waves. Gratings, or holes, are often used on the top of the GRP covers and this may reduce the lift force. It would be an interesting subject to see how much this would effect the lift force on the covers compared to not including the gratings.

A fully developed boundary layer flow from a current with a constant free stream velocity is adapted in the present study. This is of interest at large water depths where extreme waves do not influence the seabed which is the case at the Aasta Hansteen field. It would be interesting to investigate the effect of extreme wave induced velocities on the covers in shallow waters. This is relevant at the Gullfaks field. The forces from the wave interaction with the covers are of interest.

Results from the numerical simulations may be closer to the real physical forces on the GRP covers on the seabed if full three dimensional simulations were conducted. This is due to the three-dimensional effects on the covers. It would be of interest to study the hydrodynamic quantities of a full three-dimensional analysis and compare them with results from two-dimensional simulations. By using LES instead of RANS approach, more detailed flow properties can be obtained. Three-dimensional simulation and LES would require transient simulations, and the computational cost would be drastically increased compared to the present numerical simulations.

## References

- Adams, E. W., & Johnston, J. P. (1988). Effects of separating shear laeyer on the reattachment flow structure Part 2: Reattachment lenght and wall shaer stress. *Experiments in Fluids*, 6, 493–499.
- ANSYS. (2013). Ansys cfx-solver theory guide. Retrieved 2015-05-21, from http:// 148.204.81.206/Ansys/150/ANSYS%20CFX-Solver%20Theory%20Guide.pdf
- Bakker, A. (2006). Lecture 7 Meshing, lecture notes distributed in Computational Fluid Dynamics (ENGS 150) at Darthmouth Collage. Retrieved 2015-05-21, from http://www.bakker.org/dartmouth06/engs150/07-mesh.pdf
- Blevins, R. D. (2003). Applied Fluid Dynamics Handbook (Third ed.). Krieger Publishing Company.
- Castro, I. P. (1984). The Flow Past a Surface-Mounted Obstacle. *Journal of Fluids* Engineering, 105(4), 461–463.
- Code-Saturne. (2014). Cell size at the wall. Retrieved 2015-03-11, from http://code-saturne.org/cms/sites/default/files/fileattach/BPG/ BPG-3-Cell-Size-At-The-Wall.pdf
- Ferziger, J. H., & Perić, M. (2002). Computational Methods for Fluid Dynamics (Third ed.). Springer.
- Frei, W. (2013). Which turbulence model should i choose for my cfd application? Retrieved 2015-02-19, from http://www.comsol.com/blogs/which-turbulence -model-should-choose-cfd-application/
- Fujimoto, H., Isomura, M., & Ajisaka, K. (1975). Flow over Rectangular Cylinders Immersed in a Turbulent Boundary Layer. Bulletin of the JSME.
- Goldstein, S. (1969). The Mechanics in the First Half of This Century. Annual Review of Fluid Mechanics, 1–28.
- Good, M. C., & Joubert, P. N. (1968). The form drag of two-dimensional bluff-plates immersed in turbulent boundary layers. *Journal of Fluid Mechanics*, 31, 547–582.
- Keshmiri, A. (2012). Numerical sensitivity analysis of 3- and 2-dimensional rib-roughened channels. *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, 48, 1257–1271.
- Launder, B., & Spalding, D. (1974). The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3(2), 269–289.
- LEAP. (2012). Tips and tricks: Turbulence part 2 wall functions and y+ requirements. Retrieved 2015-03-10, from http://www.computationalfluiddynamics.com.au/ tips-tricks-turbulence-wall-functions-and-y-requirements/
- Liu, Y. Z., Ke, F., & Sung, H. J. (2008). Unsteady separated and reattaching turbulent flow over a two-dimensional square rib. *Journal of Fluids and Structures*, 24, 366– 381.

#### References

- Martinuzzi, R., & Tropea, C. (1993). The Flow Around Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow. Journal of Fluids Engineering, 115, 85.
- Ong, M. C., Utnes, T., Holmedal, L. E., Myrhaug, D., & 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. *Coastal Engineering*, 57(10), 931–947.
- Reinertsen. (2015). Aasgard subsea compression. Retrieved 2015-05-25, from http://www.reinertsen.com/subsea/project?pid=73
- Ryu, D. N., Choi, D. H., & Patel, V. C. (2007). Analysis of turbulent flow in channels roughened by two-dimensional ribs and three-dimensional blocks. Part II: Heat transfer. *International Journal of Heat and Fluid Flow*, 28, 1112–1124.
- Sayma, A. (2009). *Computational Fluid Dynamics*. Abdulnaser Sayma & Ventus Publishing ApS.
- Stenmark, E. (2013). On Multiphase Flow Models in ANSYS CFD Software ANSYS CFD Software (Unpublished doctoral dissertation).
- Sumer, B. M., & Fredsoe, J. (2006). Hydrodynamics Around Cylindricalstructures. World Scientific Pub Co Inc.
- Tech-Fab. (2015). Subsea Mudmat. Retrieved 2015-05-25, from http://www.tech-fab..com/subsea-mudmat.htm
- Versteeg, H., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: the finite volume method. Pearson Education Limited.
- Yunus, A. C., & Cimbala, J. M. (2006). Fluid mechanics: fundamentals and applications. International Edition, McGraw Hill Publication, 185–201.