A first principle approach to scour modeling around offshore structures

Pang, Andrew Li Jian

2016

Pang, A. L. J. (2016). A first principle approach to scour modeling around offshore structures. Doctoral thesis, Nanyang Technological University, Singapore.

<http://hdl.handle.net/10356/69118>

<https://doi.org/10.32657/10356/69118>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**A FIRST PRINCIPLE APPROACH TO SCOUR
MODELING AROUND OFFSHORE
STRUCTURES**

PANG LI JIAN ANDREW

**INTERDISCIPLINARY GRADUATE SCHOOL
ENERGY RESEARCH INSTITUTE @ NTU (ERI@N)**

2016

A FIRST PRINCIPLE APPROACH TO SCOUR MODELING AROUND OFFSHORE STRUCTURES

PANG LI JIAN ANDREW

Interdisciplinary Graduate School
Energy Research Institute @ NTU (ERI@N)

A thesis submitted to the Nanyang Technological University
in fulfilment of the requirement for the degree of
Doctor of Philosophy

2016

Acknowledgements

The author will like to thank his academic supervisors, Asst Prof Martin Skote and Assoc Prof Lim Siow Yong, as well as his industrial supervisors, Dr Zhang Yali and Dr Gullman-Strand Johan, for the support and guidance they have provided over the course of this work. Their encouragement and care they have shown goes beyond the call of duty and is deeply appreciated.

The author will also like to show gratitude to Assoc Prof Ng Yin Kwee for his mentorship during the undertaking of this work. The wisdom imparted will be invaluable to the author all through his life. Likewise, the author will like to thank Assoc Prof David R. Fuhrman for hosting and guiding the author during a 6-month attachment to the Technical University of Denmark which his truly grateful for.

In addition, the author will like to acknowledge the support of Lloyd's Register (Global Technology Center), Energy Research Institute @ NTU (ERI@N) and the Interdisciplinary Graduate School (IGS) for their support in undertaking this work. In particular, the author will like to highlight the support provided by ERI@N Joint Industry Programme (JIP) on Offshore Renewables in providing much opportunities and resources needed during the course of this project. Without the support of these organizations, the success of this work will not have been possible.

The author is very appreciative of the friends, colleagues and lab mates that went on this journey together with him. Working, meeting and interacting with peers this fun loving and caring has made these years doing the Ph.D. a memorable one and will be treasured for life.

Most importantly, the author will like to thank his parents (Pang Hong Yiaaw and Molly Chan Geok Cheng) and his family members (Joy Pang Yan Ling and Chen Kok Hao) for their support all these years. Family is a gift that lasts forever, and I'm truly appreciative of them.

-Thank You-

Contents

Acknowledgements	i
List of Tables	vii
List of Figures	xi
Abstract	xvii
1 Introduction	1
1.1 Background and Motivation	1
1.2 Objective and Scope	4
1.3 Thesis Outline	4
2 Literature Review	7
2.1 Background of Scour Assessments	8
2.1.1 Local Scouring Mechanism and Influencing Factors	8
2.1.2 Empirical Scour Models	9
2.2 Numerical Scour Models	12
2.2.1 Scour Models for Piles	12
2.2.2 Scour Models for Complex Structures	15
2.2.3 Hybrid Scour Models	16
2.2.4 Alternative Scour Models	17
2.3 Specific Topics	19
2.3.1 Flow Around a Cylinder	19
2.3.2 Lagrangian-Based Sediment Models	21
2.3.3 Needs of the Industry	23
2.4 Knowledge Gaps	25

3	Turbulence Modeling Around Cylindrical Bluff Bodies	27
3.1	Introduction	27
3.2	Theory	29
3.2.1	Mechanism of Flow Around a Cylinder	29
3.2.2	RANS turbulence modeling	31
3.2.2.1	Two-Equation Turbulence Models	32
3.2.2.2	One-Equation Turbulence Models	37
3.3	Model Setup	40
3.4	Results and Discussion	50
3.4.1	Two-equation Turbulence Models	50
3.4.2	One-equation Turbulence models	59
3.4.3	Flow Within a Scour Hole	67
3.5	Conclusion	72
4	Development of a Full Numerical Scour Model	75
4.1	Introduction	75
4.2	Theory	78
4.2.1	Scour Model Overview	78
4.2.2	Turbulence Model	81
4.2.3	Bed Parameters Module	82
4.2.4	Sediment Transport Module	83
4.2.5	Bed Morphological Module	84
4.2.6	Sand-Sliding Module	85
4.2.7	Code Algorithm	87
4.3	Model Setup	88
4.4	Results and Discussion	91
4.4.1	Scour Model with $k - \omega$ (SST) Turbulence Model	92
4.4.1.1	Scour Depth-Time Development($k - \omega$ (SST) Model)	92
4.4.1.2	Scour Initiation Mechanism($k - \omega$ (SST) Model)	95
4.4.1.3	Scour Equilibrium Depth Projection($k - \omega$ (SST) Model)	99
4.4.2	Scour model with Spalart-Allmaras Turbulence Model	101
4.4.2.1	Scour Depth-Time Development(Spalart-Allmaras Model)	101
4.4.2.2	Scour Initiation Mechanism(Spalart-Allmaras Model)	104
4.4.2.3	Scour Properties Benchmarking	108

4.4.3	Live-Bed Scour Model	111
4.5	Conclusion	115
5	RANS-DEM Sediment Transport Model	117
5.1	Introduction	117
5.2	Theory	120
5.2.1	Turbulence Model	120
5.2.2	Sediment Transport Model	121
5.2.3	Morphological Model	125
5.2.4	Code Algorithm	127
5.3	Model Setup	130
5.3.1	Bed load and Morphological Model	130
5.3.2	Suspended load Model	133
5.4	Results and Discussion	135
5.4.1	Sediment Settling Velocity	135
5.4.2	Bed load Model	137
5.4.3	Bed Morphological Model	139
5.4.4	Suspended load Model	144
5.5	Conclusion	152
6	Scour Model for Industrial Application	155
6.1	Introduction	155
6.2	Theory	156
6.2.1	Turbulence Modeling	158
6.2.2	Bed Parameter Calculations	158
6.2.3	Objective Function and Optimizer	160
6.3	Model Setup	162
6.4	Results	166
6.4.1	Flow Amplification Validation	166
6.4.2	Scour Hole Evolution	169
6.4.3	Scour Depth Optimization	173
6.5	Industrial Applications	178
6.6	Conclusion	181

7	Conclusions	183
7.1	Thesis Summary	183
7.2	Significant Contributions	188
7.3	Future Works	191
7.4	Final Comments	194

List of Tables

3.1	Constants of the Spalart-Allmaras turbulence model proposed by Spalart and Allmaras (1994)	39
3.2	Boundary conditions and input variables for the simulations performed in this work	45
3.3	Results of the mesh sensitivity study for the $k - \omega$ (SST) model . . .	48
3.4	Results of the mesh sensitivity study for the $k - \omega$ (SST) model with a roughness height of 4.24×10^{-2} m.	49
3.5	The coefficient of pressure-base, angle of separation, coefficient of drag, Strouhal number and coefficient of lift-rms of the two-equation turbulence models simulation results and experimental observations. .	51
3.6	The coefficient of pressure-base, angle of separation, coefficient of drag, Strouhal number and coefficient of lift-rms of the $k - \omega$ (SST) simulations results with and without a curvature correction and experimental observations.	54
3.7	The roughness height, K_s/D ratio and roughness constant used in the simulations around a rough cylinder using the $k - \omega$ (SST) model.	56
3.8	The coefficient of pressure-base, coefficient of drag, coefficient of lift-rms, angle of separation and Strouhal number for flows around a cylinder with varying surface roughness when utilizing the $k - \omega$ (SST) model	57
3.9	The coefficient of drag and coefficient of lift (rms) from experiments, $k - \omega$ (SST) model and the Spalart-Allmaras model with $C_{prod} = 1, 0$ & 2 respectively.	63
3.10	The coefficient of drag and coefficient of lift (rms) from experimental data and simulation results for Reynolds numbers ranging from 3.5×10^6 to 8.5×10^6	64

4.1	Simulation model pile diameter and boundary conditions utilized. . .	89
4.2	Input variables utilized in the simulation models similar to that of experimental setup by Melville and Chiew (1999)	89
4.3	Scour depth at $t = 200$ sec predicted by the simulation model and empirical equation proposed by Sumer et al. (1992a).	95
4.4	Average scour rate at $t = 200$ sec predicted by the simulation model and empirical equation.	95
4.5	The average variance between the simulated results and empirical equation from $t = 0 - 200$ sec.	100
4.6	Scour depth at $t = 200$ sec predicted by the simulation utilizing a Spalart-Allmaras model, $k - \omega$ (SST) model and empirical equation (Sumer et al., 1992a).	104
4.7	Simulation setup of the live-bed condition.	111
4.8	Boundary conditions utilized in the live-bed scour simulation. . . .	112
5.1	The model conditions utilized in the bed load and morphological sim- ulations.	132
5.2	The mean flow velocities and sediment particle sizes utilized for each bed load and morphological model validation test.	133
5.3	The sediment size and Rouse number utilized to validate the sus- pended load model.	134
5.4	The simulation result, theoretical calculations and experimental ob- servations for the settling velocity of the sediment particle.	136
5.5	The simulated and theoretical bed load velocity of the bed load sediment	138
6.1	Model setup for the flow validation simulation around the structure and within the scour hole	164
6.2	Model setup for the bed shear stress evolution and mesh optimization simulations.	164
6.3	Mesh sensitivity study for a flat bed and equilibrium scour hole con- figuration.	165
6.4	Equilibrium scour hole obtained by Melville and Chiew (1999) for run N1, N7 and N9.	170

6.5	Simulation results of the predicted scour depth utilizing a Torczon optimizer (Torczon, 1991).	173
-----	---	-----

List of Figures

2.1	Overview of the literature review presented in the thesis.	7
2.2	The formation of the horseshoe and wake vortex as the fluid flows around the monopile (Jensen et al., 2006).	9
2.3	DNS-DEM simulation model by Kidanemariam and Uhlmann (2014). Large number of particles inject into simulation model to reproduce bed morphology.	22
3.1	The variation of the coefficient of drag, coefficient of lift and Strouhal number during the drag crisis (Sumer and Fredsøe, 1997).	30
3.2	The geometrical dimension, meshing and boundary conditions of the computational domain for the x-y plane used in the simulations. . . .	42
3.3	The geometrical dimension, meshing and boundary conditions of the computational domain in the z-direction used in the simulations. . . .	42
3.4	The scour hole mesh and dimensions used in the simulations. The depth of the intermediate and equilibrium scour hole was identical to that of Melville and Raudkivi (1977) experimental setup.	44
3.5	The boundary layer velocity plots presented by Richardson and Pan- chang (1998) against that obtain in the current simulation using sim- ilar conditions.	46
3.6	The coefficient of pressure profile when a curvature correction modifi- cation and varying surface roughness are incorporated into the $k - \omega$ (SST) model.	53
3.7	Mean velocity profile for (a) Spalart-Allmaras model with no rota- tion correction, (b,c) Spalart-Allmaras model with $C_{prod}=2.0$ & 1.0 respectively and (d) $k - \omega$ (SST) turbulence model.	60

3.8	Mean turbulence viscosity along the line of symmetry for Spalart-Allmaras model with no rotation correction, Spalart-Allmaras model with $C_{prod}=2.0$ & 1.0 respectively and $k-\omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 5.2×10^6 .	61
3.9	Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k-\omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 5.2×10^6	65
3.10	Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k-\omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 3.5×10^6	66
3.11	Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k-\omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 8.5×10^6	66
3.12	Flow vector for an initially flatbed and an intermediate scour hole using a $k-\omega$ (SST) model and the flow profile observed in experiments by Melville and Raudkivi (1977).	68
3.13	Flow vector for an initially flatbed and an intermediate scour hole using a Spalart-Allmaras model with $C_{prod} = 3.5$ and the flow profile observed in experiments by Melville and Raudkivi (1977).	69
3.14	Flow profile for an intermediate scour hole using a $k-\varepsilon$ (RNG) model obtained by Richardson and Panchang (1998) and the flow profile observed in experiments by Melville and Raudkivi (1977).	70
3.15	Normalized velocity profile for the equilibrium scour hole configuration using the Spalart-Allmaras model with $C_{prod} = 3.5$, the $k-\omega$ (SST) model, the flow profile observed in experiments by Melville and Raudkivi (1977) and the numerical results using the $k-\varepsilon$ (RNG) model by Richardson and Panchang (1998)	70

4.1	Flow diagram of the numerical scour model developed in this work. .	79
4.2	Overview of the interDymScourFoam developed in OpenFOAM. . . .	80
4.3	The process flow of the sand-sliding model implemented in the inter- DymScourFoam solver.	86
4.4	The algorithm of the interDymScourFoam solver developed.	87
4.5	The scour depth-time evolution of simulation N1 from $t = 1 - 200sec$	93
4.6	The scour depth-time evolution of simulation N7 from $t = 1 - 200sec$	94
4.7	The scour depth-time evolution of simulation N9 from $t = 1 - 200sec$	94
4.8	The scour simulation using the $k - \omega$ (SST) turbulence model for case N7 from $t = 0 - 200sec$, whereby the contour plot indicates the scour depth.	96
4.9	The scour initiation mechanism of simulation N7 using the $k - \omega$ (SST) turbulence model.	98
4.10	The projection of the simulation model scour depth as it reaches its equilibrium depth.	100
4.11	The scour depth-time evolution of simulation N1 utilizing the Spalart- Allmaras model from $t = 1 - 200sec$	102
4.12	The scour depth-time evolution of simulation N7 utilizing the Spalart- Allmaras model from $t = 1 - 200sec$	102
4.13	The scour depth-time evolution of simulation N9 utilizing the Spalart- Allmaras model from $t = 1 - 200sec$	103
4.14	The scour simulation using the Spalart-Allmaras turbulence model for case N7 from $t = 0 - 200 sec$, whereby the contour plot indicates the scour depth.	105
4.15	The scour initiation mechanism of simulation N7 using the Spalart- Allmaras turbulence model.	107
4.16	The normalize Shields parameter of the simulation utilizing the $k - \omega$ (SST) and the Spalart-Allmaras turbulence model at $t = 200 sec$. . .	108
4.17	The velocity profile along the upstream slope of the scour hole, 10 mm in front of the pile as pointed out by the orange arrow.	110
4.18	The scour depth-time evolution of a live-bed simulation utilizing the $k - \omega$ (SST) and Spalart-Allmaras turbulence model.	112
4.19	The unstable bed ripples formed at the upstream section of the pile. .	113

5.1	The algorithm of the interDymSediFoam solver developed.	128
5.2	The overview of the interDymSediFoam developed in OpenFOAM. . .	129
5.3	The simulation model setup for the validation of the bed load and morphological model based on an idealize experimental setup. . . .	130
5.4	The mesh and boundary setup utilized in the simulations performed. The model was extruded in the y-direction (seen on the right) for simpler visualization purposes.	131
5.5	The simulation of the settling velocity of the particle.	136
5.6	The friction velocity mapped from the bed boundary into the bed load region internal mesh.	137
5.7	Simulation model of Case 5a.	140
5.8	The bed elevation graph over time for simulation Case 5a.	141
5.9	The erosion and deposition along a longshore caused by a groyne interference (Fredsoe and Deigaard, 1992).	141
5.10	The bed elevation graph over time for simulation Case 5b.	142
5.11	Simulation model of Case 5b.	143
5.12	The sample of the particle position over a time period of 25,000 sec. .	146
5.13	Suspended load distribution for Rouse number of 1.93	147
5.14	Suspended load distribution for Rouse number of 1.20	149
5.15	Suspended load distribution for Rouse number of 0.95	149
5.16	Suspended load distribution for Rouse number of 0.795	150
5.17	Suspended load distribution for Rouse number of 0.181	151
6.1	The process flow of the proposed hybrid numerical scour model. . . .	157
6.2	The visual illustration of the Torczon algorithm (Torczon, 1991) in determining the minimum functional value of the objective function. .	161
6.3	Simulation results and experimental data from Melville and Raudkivi (1977) of the upstream flow field profile (a,b) and the normalized velocity profile around the pile (c,d) for a flat bed.	166
6.4	Simulation results and experimental data from Melville and Raudkivi (1977) of the upstream flow field profile (a,b) and the normalized velocity profile around the pile (c,d) for an intermediate scour hole. .	168
6.5	Amplification factor around a pile for a flat bed configuration similar to that performed experimentally by Melville and Raudkivi (1977). .	169

6.6	Simulation results of the evolution of the maximum normalized Shields parameter with increasing scour depth.	171
6.7	Scour depth evolution of simulation N9 which utilizes a Torczon optimizer (Torczon, 1991).	174
6.8	Objective function evolution of simulation N9 which utilizes a Torczon optimizer (Torczon, 1991).	174
6.9	Scour depth evolution of simulation N1 which utilizes a Torczon optimizer (Torczon, 1991).	176
6.10	Objective function evolution of simulation N1 which utilizes a Torczon optimizer (Torczon, 1991).	176
6.11	The sedimentary mobility flow chart presented by Whitehouse (1998).	180
7.1	The overview of the work done in the current thesis.	184
7.2	The landscape of the thesis contribution and the proposed future works.	193

Summary

Scouring around an offshore structure has the potential to result in the weakening of the foundation and possible pre-mature failure of the structure. To evaluate the risk of scouring, empirical formulations and experimental model testing are typically utilized in scour assessment. While this has worked reasonably well for standard structures, uncertainty arises when applying such models to new foundational design and untested environmental conditions. Hence, there is a need for a more general approach to model scour around an offshore structure, which has provided the motivation for this work.

The aim of this study is to develop a first principle scour model for offshore structures. For the model to be a first principle approach, it is proposed that the scour process be modeled numerically utilizing a multi-scale modeling approach (whereby the flow is modeled utilizing turbulence models, and the sediment is modeled based on a Lagrangian approach). However, achieving such a complex model is not trivial as it comprises of multiple physical concepts (turbulence modeling, numerical methods, Lagrangian modeling) working in unison with each other. Hence, to make such a complex scour model more achievable, the work performed in this thesis was divided into three parts as described below.

Initially, turbulence models used together with the Reynolds Averaged Navier-Stokes (RANS) equations was evaluated for flow around a cylindrical bluff body. This was done to determine the optimal approach to model the turbulent flow around the structure, which is the key driving force for sediment motion. Based on the two-equation turbulence models simulation results benchmarked against experimental data, it was determined that the $k - \omega$ model with the Shear Stress Transport (SST) formulation is superior over other RANS models. In an attempt to reduce the computational time of the turbulence model further, the one-equation Spalart-Allmaras model was modified for bluff body flows, and the simulation results from this model were shown to agree well with experiments.

Based on the results obtained from the turbulence model study, a numerical scour model utilizing a sediment transport equation to model the sediment behaviour was developed in OpenFOAM. In this part of the work, both the $k - \omega$ (SST) and the Spalart-Allmaras model was utilized to model the flow. Based on the scour simulations performed, it was demonstrated that the numerical scour model developed

could predict the scour initiation mechanism well. Likewise, the scour depth-time development was shown to agree well with empirical equations. In addition, utilizing the Spalart-Allmaras turbulence model showed that it had similar performance to that obtained from the more expensive $k-\omega$ (SST) model. Hence, this demonstrates that there is a potential of utilizing such lower-cost one-equation turbulence model for complex scour simulations.

The numerical scour model described earlier utilized an empirical bed load equation to model the sediment behaviour. In an attempt to reduce the model reliance on empirical based formulations, a Lagrangian-based RANS-DEM (Discrete Element Method) sediment transport model was proposed. It was demonstrated that the RANS-DEM sediment transport model was able to reproduce key features of the bed and suspended load behaviour. In addition, a thought experiment was performed in an attempt to translate the sediment behaviour into bed morphological changes. Based on the simulations performed, the mechanism of the scour process was reproduced reasonably well, but there was a discrepancy in the scour depth achieved. Future works are proposed to develop this model further.

The scour and sediment transport models developed in this work acts as the cornerstone for the eventual development of a full first principle scour model in the near future. While such models provide much insight into the scour process, the main drawback is the high computational time requirement to perform each simulation, which makes it difficult to be applied to industrial practice in the short-run. Hence, in this work, a simplified numerical scour model was developed in FLUENT based on an optimization function. It was shown that utilizing this optimization approach, the predicted equilibrium scour depth obtained was in reasonable agreement with experimental data. However, there are limitations to this model based on the sensitivity of the objective function set. Hence, it is proposed that additional optimization algorithm be explored in future works.

In this work, both the foundational work for a full first principle numerical scour model and a simplified numerical scour model are presented. As computational capabilities continue to increase over time, such continual development of numerical scour modeling will progress this field moving into the future.

Chapter 1

Introduction

1.1 Background and Motivation

Local scouring is a process whereby bed materials are removed around a marine structure due to the resultant disturbance of the uniform flow which typically results in higher degree of turbulence and erosion capacity. When it is not managed properly, scouring can result in a significant reduction in the stability of a foundation and a premature failure of a hydraulic structure. Early studies (Lagasse et al., 2007; Wardhana and Hadipriono, 2003) have shown that in the United States, up to 60% of bridge failures are related to scouring. As such, a significant amount of work has been done over the past decades (most commonly pertaining to bridge structures) to understand and predict this phenomenon better. These knowledge and experience working with scour have been consolidated into numerous textbooks. The most popular of these textbooks are *The mechanics of scour in the marine environment* by Sumer and Fredsøe (2002) , *Scour at marine structures: A manual for practical application* by Whitehouse (1998) and *Bridge scour* by Melville and Coleman (2000).

Earlier works on scour has been typically associated with bridge structures and oil and gas offshore platforms. A relatively new sector which has taken an interest in this issue of scour is the wind energy industry. This is because recently there has been a push by this industry to install offshore bottom fixed wind turbines. These winds turbines are generally installed in shallow water regions($< 30m$) and experience harsh environmental conditions (where stronger prevailing wind are present).

Such a combination of factors typically leads to significant scour around foundations installed at these locations. A study by the United Kingdom (UK) government, Department of Energy and Climate Change (DECC) (Whitehouse et al., 2008), on reviewing the scour around early UK offshore wind farm projects (Barrow, Kentish Flats, Scroby Sands, North Hoyle, Arklow Bank) highlighted scour holes around the wind turbine foundations of up to 6m and numerous scour issues with the inner-array cabling and J-tubing at the base of these foundations. Similarly, the government of the United State of America (USA), the Bureau of Ocean Energy Management, Regulation and Enforcement (BOEMRE), conducted a feasibility study (McNeilan and Smith, 2011) on developing offshore wind farms in the Atlantic Outer Continental Shelf (OCS) and concluded that scouring is a potential hazard due to the dynamics ocean conditions as well as the complexity of the seabed in the Atlantic OCS. These pre-project studies and past experiences of scouring around early offshore wind turbine foundations clearly highlight the need to understand the potential scour risk for future offshore wind energy projects.

When dealing with the problem of scouring, matured industries (e.g. bridge building and oil and gas sectors) typically opt to over-design the scour protection around the structure foundations to mitigate risk. Such an approach is commercially feasible for such industries as each project has only a small number of foundation installations and the cost of each foundation typically makes up only a small portion of the total project budget. For an offshore wind farm project however, the foundation cost takes up a significantly higher proportion of the total cost of each project. The National Renewable Energy Laboratory (NREL) based in the USA conducted a cost analysis of an offshore wind turbine (Fingersh et al., 2006) and estimated the cost of the monopile foundation with its scour protection to be approximately 20% of the total project cost. Byrne and Houlsby (2003) has also estimated the foundation cost of wind turbines to be significantly large, with estimates being as high as 35% of the total cost. Hence, with a large number of wind turbine foundations in each wind farm project (e.g. London Array offshore wind farm (London-Array, 2011) which has 175 turbines), this makes such an option of over-designing not financially feasible. With increasing economical pressure to keep the cost of wind power low, optimizing foundation designs and minimizing the need of excessive scour protection could keep offshore wind energy a competitive source of energy.

Currently, evaluating the scour around offshore foundations is primarily performed based on empirical approaches. For an offshore wind turbine monopile foundation in particular, the formula $S/D = 1.3 \pm 0.7$ proposed by Sumer et al. (1992b) is one of the most popular equation used to predict the equilibrium scour depth. This equation has since been adopted by the Det Norske Veritas (DNV) standard, DNV-OS-J101 (DNV, 2010), to evaluate the equilibrium scour depth around a monopile foundation for wind turbines. While such an approach provides good general predictions of scour depths of slender monopiles, the level of uncertainty is unclear for large monopile foundations due to wave diffraction and scaling issues as that discussed by Whitehouse et al. (2008) in the recent DECC study. Furthermore, with most of the scour formulas derived based on laboratory data of cylindrical foundations, uncertainties arises when evaluating non-standard or complex foundations for scouring, such as the jacket foundation design (Rudolph et al., 2009).

Hence, there is a gap between the relative static/empirical approach to derive the formulations for scouring and the dynamic requirements of the industry, whereby foundation designs are constantly evolving and scaling up in size. To bridge such a gap, a scour prediction model based on fundamental physics and scour mechanism should be derived. Such theoretical models typically require a large amount of iterative calculations and correspondingly a significantly high computational requirement (both in terms of computational speed and capacity). Obtaining such computational requirement was non-trivial in the past, which makes such theoretical model development challenging previously. However, with the recent advancement of computational capabilities, which is more readily available, the potential to capitalize on this powerful computational resource has opened up the possibility of developing a first principle (numerical) approach to scouring and will be the focus of this research.

1.2 Objective and Scope

The objective of this project is to develop a first principle based approach to model the local scouring process around a monopile of an offshore structure, in particular that of an offshore wind turbine foundation. Such a first principle model constitutes of physical concepts including computational fluid dynamics, turbulence models and sediment transport models. In addition, to gain insight into the sediment behaviour, a particle-based sediment transport model was developed as a proof-of-concept for future multi-scale numerical scour modeling. The scope of this project was limited to looking at monopile foundations with non-cohesive sediments.

1.3 Thesis Outline

To achieve the objective of this project, the work in this thesis is divided into 4 parts; (1) review and develop turbulence models for bluff body flows, (2) develop an Eulerian-based scour model, (3) develop a Lagrangian-based sediment transport model and finally (4) develop a optimization-based scour model for industrial application within the near future.

Part 1 - Turbulence modeling

In the initial part of the project, the turbulence modeling around a cylinder/monopile was evaluated. This was to identify the optimal turbulence model which most accurately reproduces the key flow parameters that drive the scour process. In this work, the focus was placed on the Reynolds Averaged Navier-Stokes (RANS) turbulence model, specifically that of the one- and two- equation RANS turbulence models. It was identified that the two-equation $k - \omega$ model with the Shear Stress Transport (SST) formulation was superior to the other RANS turbulence model. In addition, when analysing the one-equation Spalart-Allmaras turbulence model for bluff body flows, it was shown that by modifying the coefficients within the Spalart-Allmaras turbulence model, the predicted flow had good agreement with experimental data, highlighting the potential of this model.

Part 2 - Eulerian-based scour modeling

Based on the turbulence model study, an Eulerian-based scour model was developed in OpenFOAM named interDymScourFoam solver. In this solver, the RANS turbulence model was used to model the flows profile and corresponding flow parameters. Based on the flow parameters, the sediment transport along the bed boundary was determined based on sediment transport equations. The net movement of the sediment was subsequently translated into mesh deformation along the bed boundary based on the Exner equation. To account for sand sliding, a sand sliding model was included into the scour model to smoothen out unphysical steep slopes along the bed boundary. The simulation results from this full numerical scour model was shown to agree well with empirical equations.

Part 3 - Lagrangian-based sediment transport modeling

To deepens the first principle nature of the numerical scour model, the work was extended to determine if the sediment transport in the scour model could be modeled using a Lagrangian-based approach. This led to the development of a RANS-DEM sediment transport model and an OpenFOAM solver named interDymSediFoam. For this part of the work, a simpler spillway case study was utilized to validate the Lagrangian-based (RANS-DEM) sediment transport model. It was shown that the RANS-DEM sediment transport model could reproduce both bed load and suspended load behaviour well. Attempts to translate sediment behaviour into bed morphology was likewise performed. The RANS-DEM sediment transport model derived acts as a stepping stone for future Lagrangian-based scour models.

Part 4 - Optimization-based scour modeling

To address the requirements of the industry for a less computationally expensive numerical scour model, a scour model based on mesh optimization techniques was developed. In this proposed model, the RANS turbulence approach was utilized to predict key flow drivers of the scour process (e.g. shear stress, Shields parameters). Based on these parameters, the mesh is deformed in pre-defined configurations and an optimization algorithm guides the simulation to the equilibrium scour depth within a limited number of iteration. This simplification of the numerical scour model makes the computational requirements reasonable for industrial application.

In the last chapter of the thesis, a summary of the work done over the duration of the course and the significant it plays in advancing the knowledge of numerical scour modeling is presented. In addition, future works are proposed on developing the numerical scour model further and knowledge gaps which need to be addressed in future research projects are discussed.

Chapter 2

Literature Review

Over the past century, empirical methods have predominantly been used to gain insight into the scouring process as well as to derive key governing equations of this phenomenon. With this wealth of work done, such an empirical approach is still the most popular method when performing a scour analysis for an offshore structure. However, with the advancement in computational capabilities in the past decade, there have been an increasing number of attempts to numerically model the scour process with varying levels of success. In this chapter, a literature review is presented which will cover both the background of scour assessment used in current engineering practice and recent works on numerical scour modeling.

Figure 2.1: Overview of the literature review presented in the thesis.

Numerical scour modeling is a multi-disciplinary problem which requires an understanding in multiple fields/areas such as flow field modeling and sediment transportation modeling. In this chapter, the literature review will be structured in 3 sections as seen in Figure 2.1. In the first section (Background of scour assessments),

an overview of the current understanding of the scour process (scour mechanism and influencing factors) and empirical equations on assessing scour around a pile structure are presented. In the next section (Numerical scour models), recent works on numerical scour models are presented. Initially, numerical scour models applied to a pile structure are discussed, including specific issues pertaining to such a numerical model (e.g. effects on turbulence on sediment transport). Subsequently, scour modeling on more complex structures, such as a tripod foundation, are presented. While this research focuses on pile structures, there is significant value in presenting scour models applied to other structural configurations due to the similarities in the underlining physical principles of such models. Lastly, hybrid and alternative scour models (e.g. lattice Boltzmann approaches) are presented and discussed in this section. In the final section (Specific topics), a focused review is presented on specific areas of the numerical scour model where gaps exist which will be addressed in the work done for this thesis (e.g. flow around cylinders, Lagrangian modeling). In addition, the landscape of scour assessment approaches and needs within the industry are discussed which provides relevant justification for the development of a hybrid scour model presented in Chapter 6.

2.1 Background of Scour Assessments

2.1.1 Local Scouring Mechanism and Influencing Factors

When fluid flows pass a monopile, the disturbance due to the monopile will result in the acceleration of the flow and the formation of horseshoe vortex and wake vortex at the base and downstream of the structure respectively as seen in Figure 2.2. The acceleration of the flow and vortex formed transports sediment away from the foundation resulting in a scour pit being formed. The sediment transported away from the monopile moves downstream where the wake vortex diminishes quickly and this results in a high deposition rate of sediment close behind the monopile. Over time as the scour pit depth increases, the strength of the horseshoe vortex decreases up to a point where the equilibrium scour pit is reached and the scour process stops.

Figure 2.2: The formation of the horseshoe and wake vortex as the fluid flows around the monopile (Jensen et al., 2006).

Numerous factors influence the scour process (both in terms of equilibrium depth and scour rate). These factors (Jensen et al., 2006) are the flow rate, water depth, waves characteristic (including breaking waves), sedimentation type, sediment size and size distribution, monopile shape and diameter, and monopile alignment. In this work, the focus is placed on cylindrical monopile foundations in shallow water condition with no wave present. Non-cohesive sediment, such as that of natural sand, will be likewise used in this study.

2.1.2 Empirical Scour Models

There are currently numerous empirical scour models available, each derived from a specific set of data and taking into accounting different influencing factors. Reviews of these equations and models are consolidated in various review papers (Sumer et al., 2001) and scour related textbooks (Sumer and Fredsøe, 2002; Whitehouse, 1998). For an offshore wind turbine monopile foundation, the formula proposed by Sumer et al. (1992b), as shown in Equation 2.1, is the current approach suggested by Det Norske Veritas (DNV), DNV-OS-J101 (DNV, 2010), to determine the scour depth around cylindrical monopiles.

$$\frac{S}{D} = 1.3\{1 - \exp[-0.03(KC - 6)]\} \quad (2.1)$$

where S is the scour depth, D is the pile diameter and KC is the Keulegan-Carpenter number. This formula is a very popular scour equation due to its simplicity and its

ability of account for wave action. Derived from a data set with a standard deviation of 0.7, it is suggested that the maximum scour depth of $S/D = 2.0$ be used during the design phase (Whitehouse et al., 2008). While such an approach provides good general predictions of scour depths of slender monopile foundations, the level of uncertainty is unclear for large monopiles due to wave diffraction and scaling issues as that discussed by Whitehouse et al. (2008). With wind turbine piles size increasing rapidly (up to a diameter of 7.5m (Dong-Energy, 2015)), this uncertainty can be a significant issue for wind farm projects in the near future. Furthermore, given that Sumer et al. (1992b) equation was empirically derived utilizing cylindrical structures, uncertainties arise when utilizing this approach to predict the scour depths around non-standard foundations. Currently, there are no industrial guidelines available for non-standard foundations for offshore wind turbines.

Given the wealth of experience gained from scouring around bridge structures, the offshore wind industry could adopt various models from this field and apply it to wind turbine foundations. One such model is that of the CSU/HEC 18 formula (Richardson and Davis, 2001). This model is used by the US Department of Transportation as a guideline for evaluating scour at bridges for both clear-water and live-bed scouring. The CSU/HEC 18 formula is presented in Equation 2.2.

$$\frac{S}{h} = 2.0K_1K_2K_3K_4 \left(\frac{h}{D} \right)^{0.65} Fr^{0.43} \quad (2.2)$$

where h is the water depth, K_1 is the pile shape correction factor, K_2 is the angle of flow attack correction factor, K_3 is the bed condition correction factor, K_4 is the bed sediment size correction factor and Fr is the Froude Number. The approach of using correction factors in the CSU/HEC 18 formula provides greater flexibility in applying such an equation in different conditions. Although originally being developed for river flows, the CSU/HEC 18 formula was shown to correlate well with field data obtained from Scroby Sand wind farm (Jensen et al., 2006). It was noted by Jones and Sheppard (2000) that the data used to derive CSU/HEC 18 formula included cylindrical piles with h/D less than 1.0, making such an approach applicable to shallow water depths. However, care must be taken when applying this equation to scour around a pile in deeper water.

For deep water conditions, utilizing an equation derived by Breusers et al. (1977) reduces the uncertainty pertaining to water depth limitations. The general form of Breusers et al. (1977) formula is shown in Equation 2.3.

$$\frac{S}{D} = f_1 \left[1.5 \tanh \left(\frac{h}{D} \right) \right] f_2 f_3 \quad (2.3)$$

where f_1 is determined based on the ratio of average current velocity to the critical velocity of the flow, f_2 is the shape factor and f_3 is the angle of attack factor. For a cylindrical pile, Breusers et al. (1977) formula simplifies to Equation 2.4 as shown below.

$$\frac{S}{D} = 1.5 \tanh \left(\frac{h}{D} \right) \quad (2.4)$$

Using this simplified equation, the scour depth predicted at the Scroby Sand wind farm (Jensen et al., 2006) was shown to correlate well with scour depths measured in the field.

It must be noted that for most empirical work done on scouring, including those discussed in this section, the formulations were typically derived based on experimental data which utilized a unidirectional uniform flow. However, for most offshore wind farms, the monopile foundations are generally exposed to tidal conditions. Such tidal environments make the prediction of scouring very complex because of the continuous variability of influencing factors such as flow direction, flow velocity, water depth and tidal durations. These factors coupled with sporadic events such as storm surges and astronomical tidal cycles make scour predictions in this environment very challenging. Currently, there is very minimal work done on scouring in a tidal environment. Escarameia (1998) is one of the few works available which studied scouring in a tidal flow where he looked at the effects of flow reversal, tidal cycle duration, and water depth. Based on his study, a tidal/bi-directional flow gave a shallower equilibrium scour depth than a unidirectional flow. In other works, Noormets et al. (2003) used data obtained from Otzumer Balje tidal inlet and benchmarked it against calculations made using the CSU/HEC 18 formula. It was found that the CSU/HEC 18 formula was 28% over-conservative. Hence, when using empirically derived scour formulas, which are typically derived from uniform flow experiments, care must be taken when applying it to a tidal environment.

2.2 Numerical Scour Models

With the advancement of computational capabilities, numerical scour modeling is increasingly coming achievable. Such numerical models hold the possibilities of providing greater insight into the scour processes/mechanisms as well as overcome scaling limitations of physical modeling. Although these numerical models show strong potential, developing such scour models are non-trivial even at the current level of computational capabilities. Furthermore, the computational resources required, especially for three-dimensional scour simulations, are significant and less readily available to researchers working on scour modeling (Zhao et al., 2010). As such, numerically modeling the scour process is still a developing area of research.

There are a couple of approaches which can be taken in developing a numerical scour model. These approaches are (1) a Finite Volume Method (FVM) (Roulund et al., 2005) and (2) a Lattice-Boltzmann (LB) approach (Alam and Cheng, 2010). Of these two approaches, the FVM is the most popular approach, whereby a hydrodynamic model, sediment transport model and morphological model are coupled together to determine the scour around a structure (Sumer, 2014a). Such a FVM approach will be adopted in this work in developing a full numerical scour model. Reviews of the state-of-the-art numerical scour models have been compiled by Sumer (2007, 2014a,b), which provides a comprehensive review of the current advancements in this field.

2.2.1 Scour Models for Piles

For the FVM approach using a hydrodynamic and morphological model, one of the first attempts to simulate a three-dimensional scour process around a pile was done by Olsen and Melaaen (1993) using a steady-state Navier-Stoke solver and a $k - \varepsilon$ turbulence model. Olsen and Kjellesvig (1998) subsequently extended this work further to include in the time-dependent transient component into the earlier work done. The maximum scour depth obtained by Olsen and Kjellesvig (1998) agreed well with that calculated using empirical formulas and experimental observations. Olsen and Melaaen (1993) and Olsen and Kjellesvig (1998) works assumed clear-water scouring only. Inspired by Olsen's work and to gain better insight into the

fluid flow, Richardson and Panchang (1998) modeled the flow within a scour hole utilizing a frozen bed. Using the solver FLOW 3D and the $k - \varepsilon$ (RNG) turbulence model, his simulation results correlated well with that obtained experimentally by Melville and Raudkivi (1977).

Turbulence modeling for numerical scour models

While the simulation done by Olsen and Melaaen (1993) correlated well with experiments, there was some criticism (Tseng et al., 2000) that not much emphasis was placed on the flow field (i.e. modeling the wake and horseshoe vortex using first-order schemes) and the resulting scour mechanism. Hence, Tseng et al. (2000) utilized a Large Eddy Simulation (LES) to model the flow around a square and cylindrical pile. The simulation results were shown to correlate well with that observed experimentally by Dargahi (1989). It must be noted that while no actual scour simulations were performed in this work, Tseng et al. (2000) inferred the observations he obtained of the flow mechanism with that to the scour process. While Tseng et al. (2000) showed the capabilities of the LES model in reproducing key flow drivers (horseshoe and wake vortex) in the scour mechanism, the computational cost of performing a LES simulation was high which makes this turbulence model impractical for scour modeling. A reasonable turbulence model for scour will be that of a RANS approach which has been used in most scour models since (Roulund et al., 2005; Fuhrman et al., 2014).

Development of full numerical scour models

The numerical scour models described earlier were for clear-water flows and lacked various physical considerations such as accounting for the angle of repose. For the live-bed regime, Roulund et al. (2005) developed a numerical scour model for such a flow case. In his model, he utilized a $k - \omega$ (SST) turbulence model to obtain the flow profile, a bed load equation proposed by Engelund and Fredsøe (1976) to model the sediment transport behaviour and an Exner equation to determine the bed morphology. In addition, a bed correction factor and a sand sliding model were used to account for bed slopes angle variability and the angle of repose. To the author's knowledge, the work by Roulund et al. (2005) provided the first full numerical scour model. Based on the simulation results, it was found that the equilibrium scour

depth was about 15% - 30% smaller than that observed in experiments. However, being a steady-state simulation, Roulund et al. (2005) attributed this difference between the simulation and experimental results to the fact that the unsteady effects of the flow were not accounted for in the simulation performed. In addition, turbulence on the sediment transport, which influences the scour process as discussed in later works (Dixen et al., 2013), was not accounted for in Roulund et al. (2005) model.

Following the work by Roulund et al. (2005), there have been similar works done by various researchers (Baykal et al., 2015; Stahlmann, 2013; Zhao et al., 2010). For Zhao et al. (2010), he used a $k - \omega$ turbulence model to simulate the scour around a submerged cylinder. Both bed load and suspended load were accounted for in his sediment transport models, where the sediment concentration of the suspended load was determined using a convection-diffusion equation. The predicted scour depth using Zhao et al. (2010) numerical model was about 10% - 20% smaller than that observed in experiments. Work by Stahlmann (2013) likewise incorporated an additional suspended load transport equation into the numerical scour model. In Stahlmann (2013) work, a monopile configuration identical to Roulund et al. (2005) of $D = 0.1m$ was utilized. Stahlmann (2013) showed that by including the suspended load, the scour depth obtained correlated well with experimental data, better as compared to that obtained by Roulund et al. (2005).

The most recent work on numerical scour modeling is that by Baykal et al. (2015). In this work, Baykal et al. (2015) extended the numerical scour model to account for both bed load and suspended load sediment transport and unsteady flow features. A significant focus of this work was on understanding the vortex behavior during the scour process and the visualization of the counter-rotating vortex downstream of the pile. In addition, by turning ‘off’ and ‘on’ the suspended load model and unsteady flow term, Baykal et al. (2015) found that without the suspended load, the upstream scour depth decreases by 50% while turning ‘on’ the unsteady flow vortex shedding had minimal effect on the simulation.

For all the works discussed, one major drawback about numerically modeling the scour process is its very high computational time. For Zhao et al. (2010), a 3 hours scour model required a computational time of 2 weeks using 64 grids nodes. As for Baykal et al. (2015), 1 minute of physical time requires approximately 10 days on

8 processes. Such high computational time limits the application of such models to academic research. Currently, no scour codes are available in the public domain.

Influence of turbulence on sediment transport

For the earlier numerical scour models (Roulund et al., 2005), it was noted that the predicted scour depth were typically underestimated. It was proposed by Diken et al. (2013) that such an underestimation can be attributed to the effects of external turbulence on the sediment transport properties. To account for the external turbulence, Diken et al. (2013) characterize the sediment transport function of the flow using $\frac{\sqrt{U'^2}}{U_f}$, whereby U' is the fluctuating velocity and U_f is the friction velocity. Based on the characterization function, Diken et al. (2013) proposed varying the non-dimensional sediment transport variable to decrease the time-scale of the scour process and increase the scour depth as will be that expected with increased turbulence.

2.2.2 Scour Models for Complex Structures

Much of the work done on numerical scour modeling has been limited to relatively simple structural configurations (cylindrical monopile). In many practical applications however, the structural configurations of the foundations can be very complex (e.g. Jacket foundation, piggy-bag pipeline). For complex structures, Stahlmann (2013) performed scour simulation around a tripod foundation. In his work, the front piles of the foundation had good agreement with experimental data. However, the scour depth under the main column was 40% higher than that measured in experiments while the scour depth at the rear piles was 20% lower as compared to experimental observation. While there were some discrepancy of the scour depth within the model, in general, Stahlmann (2013) concluded that the model could be applied for local scour investigations.

In addition to foundational structures, various work has been done to analyse scour around various submerged structures. Diken et al. (2013) performed scour analysis around a half-buried sphere and his simulation results showed good correlation with experimental data when the effect of turbulence on the sediment trans-

port was accounted for. Likewise, attempts to develop a generalized scour model for shipwrecks have been proposed by various researchers (Melling et al., 2011; De-Hauteclocque et al., 2007). However, the outcome of this work for such a generalized submerged model is yet to be known.

For pipelines, numerical works (Brørs, 1999; Fuhrman et al., 2014) typically focus on scouring around single cylindrical pipes. For complex pipeline structures, Zhao and Cheng (2008) simulated the scour depth of a piggyback pipeline with different critical gap ratio. The data obtained by Zhao and Cheng (2008) highlighted the potential of utilizing a validated numerical scour model to obtain scour analysis on new configurations.

Modeling the backfilling process

An area which has gained much interest in recent times is that of the backfilling process which has been performed for pipelines. A backfilling simulation of pipelines was initially performed by Liang et al. (2005) as part of a self-burial pipeline study. More recently, Fuhrman et al. (2014) performed wave-induced scour simulations for a pipeline and the corresponding backfilling simulation due to the variation of the Keulegan-Carpenter (KC) number. Similar to that noted in experimental observation, the simulations performed by Fuhrman et al. (2014) backfilled towards the equilibrium scour depth of the new KC number. This evolution was characterized by an initial quick redistribution of sediment around the pipeline and subsequently slow backfills towards the equilibrium scour depth. This 2 stage backfilling process was likewise observed in experimental and simulations studies for a monopile done by Sumer et al. (2012) and Baykal et al. (2015) respectively.

2.2.3 Hybrid Scour Models

While the numerical scour models discussed in the earlier sections have been shown to predict the scour process with reasonable accuracy, one major drawback is the high simulation time required for each model. This makes such scour models impractical for industrial applications, which generally have a time constraint on each analysis. Hence to address this issue, commercial numerical scour codes utilizing a hybrid scour model have been proposed. Such models mix empirical equations

and scour data together with numerical simulations of the flow field to obtain the equilibrium scour depth around a structure. The most recent hybrid scour model which is commercially available is that developed by the Danish Hydraulic Institute (DHI) called WiTus (DHI, 2012). In this hybrid scour model (Dixen et al., 2012), the flow around a monopile is calculated using a $k - \omega$ (SST) turbulence model. Based on the flow field, the scour rate is read off a table to determine the scour development around the pile. This scour-rate table was derived based on three-dimensional scour simulations with both flow and sediment transport being model for different scour configurations. The results of the simulations were shown to be promising. WiTus has yet to be incorporated into DHI suite of software called MIKE 21 (Warren and Bach, 1992). Currently, the scour model within the MIKE 21 tool box utilizes a time-depend empirical approach to estimate scour depth around a pile.

2.2.4 Alternative Scour Models

It is worth mentioning that in addition to the numerical scour models discussed earlier utilizing the FVM approach, there have been other attempts to model the scour process numerically using other approaches. These include using Euler-Euler methods, lattice Boltzmann approaches and neural networks theories.

For the Euler-Euler method (Zhao and Fernando, 2007; Yeganeh-Bakhtiary et al., 2011), the sediment in the simulation is modeled as one phase with its own set of continuity and momentum equation. The sediment and fluid phase are subsequently coupled with each other via a transfer of momentum between the 2 phases. One of the most recent attempts to use this Euler-Euler method is that done by Yeganeh-Bakhtiary et al. (2011) to model the scouring around a pipeline. In this work, they utilized a modified $k - \varepsilon$ turbulence closure model (Elghobashi and Abou-Arab, 1983) to account for the effects the sediment has on the turbulence production and dissipation in the fluid. Benchmarking their results with observations made by Mao (1987), it was shown that the simulations agreed well with that of experimental observations.

Moving towards a Lagrangian approach to model the scouring process, there has been earlier works (Chopard et al., 2000; Dupuis and Chopard, 2002; Alam and Cheng, 2010) on using a lattice Boltzmann method to model the scour process. In the most recent work, Alam and Cheng (2010) performed a scour analysis around a pipeline using a lattice Boltzmann method and Smagorinsky sub-grid turbulence model. The sediment behaviour/transport in the simulation were modeled based on transportation, deposition, toppling and scour rules defined within the model. The simulations showed fair agreement with experimental observations. In addition, the key flow features and properties, such as the spanwise and spiral (in the proximity of the span shoulder) vortex, were reproduced well.

With the difficulty in finding a physical relationship between the different influencing factors of scouring, it was proposed that a neural network method could find correlations between these factors using various experimental data sets and a learning function. Recent works on using neural networks to predict the equilibrium scour depth around a monopile includes Bateni et al. (2007); Zounemat-Kermani et al. (2009). Based on the neural network simulation, Zounemat-Kermani et al. (2009) concluded that the results were ‘more satisfactory’ than empirical methods.

For this project, the focus will be on developing a numerical scour model based on a FVM approach and a sediment transport equation as discussed in the earlier section. However, having an overview of the different numerical approaches taken to predict the scour process provides completeness to understand the landscape of the works done on numerical scour modeling.

2.3 Specific Topics

2.3.1 Flow Around a Cylinder

For numerical scour modeling, obtaining an accurate flow profile is critical. Hence, identifying the most appropriate turbulence models to adopted in the scour model is important and will be addressed in this thesis. The literature review for such flows around a cylinder is given in this section.

For the flow around a cylinder, earlier works have relied significantly on an empirical approach to understand such flows behavior. While empirical studies have provided much insights, experimental limitation (eg. physical size of experiments) have typically confined most of these earlier works to lower Reynolds number flow regimes. For high Reynolds number flows, within the trans-critical regime, various researchers (Schmidt, 1966; Achenbach, 1968; Jones et al., 1969; James et al., 1980; Schewe, 1983) have performed experimental work on the flow around a cylinder for Reynolds number up to $Re = 10^6$. However, due to the significant experimental challenges involved, minimal work has been done within the $Re = 10^7$ domain, with one of the highest Reynolds number of $Re = 1.87 \times 10^7$ being obtained by Jones et al. (1969) using Langley’s transonic dynamics tunnel.

For numerical modeling of such flows, advancements in computational capabilities have allowed computationally expensive high Reynolds number flow simulations to become increasingly accessible. Generally, there are three approaches to model turbulence flows, which are the Reynolds-Averaged Navier-Stroke (RANS) approach, Large Eddy Simulation (LES) model and Direct Numerical Simulation (DNS) model. Ideally, a DNS simulation should give the most accurate results whereby the full Navier-Stokes equation is solved up to the Kolmogorov length scales. However, due to the extremely fine grid size needed to resolve the Kolmogorov length scales, the computational cost requirement results in a DNS simulation being not feasible, especially for high Reynolds number flow. The next closest alternative will be to perform a Large Eddy Simulation (LES) or a Detached-Eddy Simulation (DES). Squires et al. (2008) performed simulations using a DES and DDES (Delayed Detached-Eddy Simulation) turbulence model for the flow around a cylindrical bluff body. The study showed that both the DES and DDES models agreed reasonably well with experiments and gave approximately identical results. A similar

study was done by Catalano et al. (2003) using a LES approach and it was shown that the results from the LES simulation correlated well with experiment data for $Re = 1 \times 10^6$. However, extending the Reynolds number to $Re = 2 \times 10^6$, Catalano et al. (2003) noted that the discrepancy between the simulation results and experimental data increased due to the lack of grid resolution at the wall surface. Hence, even with current computational capabilities, the grid requirement when modeling a high Reynolds number flow around a cylinder is still a significant challenge.

A less computationally expensive approach to model the turbulent flow around a cylinder is that of a Reynolds-Averaged Navier-Stokes (RANS) approach. Such an approach uses the time-averaged properties of the flow when solving the Navier-Stokes equation, making such a model less computationally intensive. Two recent investigations by Catalano et al. (2003) and Ong et al. (2009) evaluated the ability of the standard $k - \varepsilon$ model to predict the flow around a cylinder. Ong et al. (2009) concluded that the predicted flow using the standard $k - \varepsilon$ model agrees reasonably well with that of experimental data up to a Reynolds number of $Re = 3.6 \times 10^6$ and is a reliable turbulence model for engineering design purposes. However, it was also highlighted by Ong et al. (2009) that there were some inaccuracies in this model due to the strong anisotropy of the turbulence in such a flow around a bluff body. Unal and Goren (2006) did a comparative study of the one- and two- equation RANS turbulence model (Spalart-Allmaras, $k - \varepsilon$ and $k - \omega$ models) applied to the flow around a cylinder in the sub-critical flow range ($Re = 10^4$) and concluded that the $k - \omega$ (SST) model correlates best with experimental observations.

For most earlier works using the RANS turbulence models, more emphasis was placed on the two-equation turbulence models. For the one-equation models, such as that of the Spalart-Allmaras model, less work has been done to explore the potential of using such a low-cost model for flows around cylindrical bluff bodies. While Unal and Goren (2006) did present simulation results using a Spalart-Allmaras model, minimal details on the setup of the Spalart-Allmaras model was given.

Based on these earlier works, there are gaps in our understanding of such bluff body flows in higher Reynolds number flow regimes (within the trans-critical flows range). This flow regime is that typically experienced by most offshore wind turbine foundations.

2.3.2 Lagrangian-Based Sediment Models

In this thesis, the aim is to develop a first principle approach to model scour. Hence, to achieve this, it is proposed by the author that the sediment be modeled utilizing a Lagrangian-based approach. In this section, Lagrangian-base sediment models are reviewed.

In relation to scour, there are no scour models which solely utilizes a Lagrangian-based approach to model the sediment transport process. The closest resemblance of a particle-based scour model was that proposed by Alam and Cheng (2010) whereby a Lattice-Boltzmann (LB) approach was utilized to model the scouring process mentioned earlier. It is important to note that while Alam and Cheng (2010) simulation results had reasonable agreement with experimental data, they did not make any correlation between the Lattice-Boltzmann model and the actual behaviour of the sediment particles.

Although currently there has not been a particle-based scour model, a recent work by Escauriaza and Sotiropoulos (2011) attempted to model the bed load sediment behaviour around a wall-mounted obstacle (cylinder) using a Lagrangian-based modeling approach. They demonstrated that with a simple Lagrangian-based sediment model, the initiation motion of the particles upstream of a cylinder could be replicated well. In their simulations, a Detached Eddy Simulation (DES) and a Discrete Element Method (DEM) was utilized which neglected Basset effects and particle collisions. While the motion of the particles did not influence the morphology of the bed in the model proposed by Escauriaza and Sotiropoulos (2011), their work showed the potential of applying such Lagrangian-based sediment models for scour applications.

When focusing on the sediment transportation only, there has been some earlier works (Schmeeckle and Nelson, 2003; Heald et al., 2004; Kidanemariam and Uhlmann, 2014) on using a Lagrangian-based approach to model the sediment transport process. Most recently, Kidanemariam and Uhlmann (2014) utilized a Direct Numerical Simulation (DNS) coupled with a DEM approach to model pattern formation in sediments. Based on their simulation results, the formation of small dunes for a laminar flow case and vortex dunes for a turbulence flow case was observed. This formation is consistent with that observed experimentally by Ouriemi et al. (2009). While these earlier works on Lagrangian sediment transport models show

promising results, a major drawback is the high computational time needed for each model. In the work by Kidanemariam and Uhlmann (2014), approximately 5,000,000 computational core hours were utilized for the simulations performed. This can be attributed to both the computationally intensive turbulence model used (DNS) for such simulations and the large number of particles injected into each simulation model in order to reproduce bed morphology motion/evolution as shown in Figure 2.3 (note a significant amount of particle do not participate in the bed load transportation at each time-step and are primarily injected into the model to reproduce bed morphology). Hence, with the high computational time needed, applying such Lagrangian-based sediment transport models into more complex scour simulations can make such simulations infeasible even with current advancements in computational capabilities. As such, there is a need to develop a cost-efficient particle-based model to simulation sediment transport behaviours to be applied to scour modeling.

Figure 2.3: DNS-DEM simulation model by Kidanemariam and Uhlmann (2014). Large number of particles inject into simulation model to reproduce bed morphology.

To reduce the computational time for such sediment model simulations, two components of the model has to be addressed in this thesis given as follows:

- Utilization of a cost-efficient turbulence model.
- Reducing the number of particles involved in each simulation model.

2.3.3 Needs of the Industry

In this thesis, one of the aims was to develop a numerical scour model which can address the current gaps in the industry and likewise be adopted by engineers in the short run. Hence, a review of the industry current practices and needs are presented in this section.

Modeling and predicting the scour process in a laboratory environment is already challenging, let alone for complex offshore conditions. This complexity has resulted in coastal engineers being generally conservative when predicting scour of offshore structures. As mentioned earlier, looking at the oil and gas industry, the approach taken to tackle scour is typically to overdesign, have real-time monitoring systems or have a regular inspection program setup to mitigate these problems. While such strategies work reasonably for this sector, less can be said for the offshore wind industry whereby the cost of protecting a large number of foundations of the wind turbines can be significant. This has motivated the industry to search for improved scour prediction methodologies, especially so with the recent increase in the number of offshore wind farms installations.

For offshore wind installations in particular, the industry currently relies on experience and empirically derived equations such as that proposed by Sumer et al. (1992b) and recommended by Det Norske Veritas (DNV), DNV-OS-J101 (DNV, 2010):

$$\frac{S}{D} = 1.3\{1 - \exp[-0.03(KC - 6)]\} \quad (2.5)$$

where S is the scour depth, D is the pile diameter and KC is the Keulegan-Carpenter number. Equation 2.5 has been validated for smaller lab-scale structures by Sumer et al. (1992b). While this equation holds for slender monopile, uncertainty arises when it is applied to large foundations larger than 5 meters (diameter calculated based on the wave periods in the North Sea (Draper, 1991) as describe below) which are the typical pile diameter for new offshore wind turbine foundations. For large monopiles ($D/h = 0.5$, where h is the water depth), wave diffraction and reflection around the structure become important, especially when $D/L > 0.2$ (Rance, 1980), where L is the wavelength. Likewise, physical tests have shown that scour depth around large piles scale differently from that of slender piles (Rance, 1980; Katsui and Toue, 1992). Furthermore, the location of the maximum scour depth moves

from the upstream face of the pile to 45° along the pile (Torsethaugen, 1975) as the pile size increases. Personal communication with various practicing geotechnical engineers reveals a general sentiment that current scour prediction tools are overly conservative for large piles. In addition, the cost implication of providing 1.3D scour allowance for large piles (5m and greater) can be high and this have motivated wind farm owners to place more emphasis on obtaining more detailed analysis of the potential scour around each foundation.

From an industrial perspective, the confidence in using computational modeling is increasing in recent time, and numerical tools are being used more widely as an engineering design tool. Hence, the potential of performing scour analysis numerically could be realized in the near future. For a full numerical scour models, such as that developed in this thesis as well as that from other earlier works (Fuhrman et al., 2014; Stahlmann, 2013) the codes were developed in-house and hence, generally unavailable for public use. While these works on numerically modeling the scour process have shown good correlation with experimental results, it is to the author's opinion that these full numerical scour models are still an academic field and can yet to be applied to industrial application. This is primarily due to the high computational time required for each simulation (about 2-4 week simulation time for a lab scale model).

In practical engineering, a quick scour analysis that is able to take into account the complexity and uncertainty of the scour process is needed before it can be adopted for industrial use. As such, there are recent works on developing hybrid numerical scour models for industrial applications. One of the most significant developments is by Danish Hydraulic Institute (DHI). The DHI approach performs simulations around the structure based on fluid dynamics calculations and predefined scour rates derived from full scour model simulations done earlier (Dixen et al., 2012). This concept has since been developed into a commercial tool called WiTus (DHI, 2012). In this work, the aim is to adopt a similar path to develop a hybrid numerical scour model which is based on a hydrodynamics approach and an optimization function.

2.4 Knowledge Gaps

Based on the literature review, key knowledge gaps are identified and will be addressed in this thesis.

Turbulence model

Most offshore foundations (which are typically bluff body in nature) experiences high Reynolds number flow, within the trans-critical flow regime. While there have been works on turbulence modeling around a bluff-body, much of these earlier works done have been limited to that of lower Reynolds numbers conditions, resulting in knowledge gaps for higher flows regimes. For scour modeling, ensuring that the flow is modeled well is critical as this flow is the key driving force in the scour process. Hence in this work, to obtain greater insight into this gap, the one- and two-equation RANS turbulence models are evaluated for such high Reynolds number flows. Furthermore, as computational time can be expensive for scour models, the development and validation of a one-equation Spalart-Allmaras turbulence model for bluff-body flows, which requires less computational time, will be valuable for future computationally expensive simulations.

Numerical scour model

While there have been various numerical scour models developed by different researchers, the knowledge and experience available of such scour modeling is still very limited. Various question still exists such as,

- Can a numerical scour model capture fine details of the scour initiation mechanism?
- How does a numerical scour model perform in terms of scour depth time development?
- How does the different turbulence model affect a numerical scour model and can a one-equation turbulence model be used for such a complex simulations?
- What are the challenges in developing and utilizing a numerical scour model?

By developing a numerical scour model in OpenFOAM, this thesis aim to address these questions and bring greater insight into modeling the full scour process numerically.

Lagrangian-based sediment transport model

For most current numerical scour models, the sediment transport behaviour is modeled by empirically derived sediment transport equations. It is proposed that by moving towards a Lagrangian-based sediment transport model, this could lead to greater insight into the sediment behaviour during the scour process. While there are Lagrangian-based scour models such as that by Alam and Cheng (2010) using a lattice Boltzmann method, the understanding of the particle behaviour during the scour process from such models is minimal. Furthermore, the computational requirements can be very expensive for these models due to the number of particles needed. Hence, developing a cost-efficient Lagrangian-based sediment transport model could provide a stepping stone to make modeling the scour process using particles more accessible in the future.

Hybrid scour model

On a practical aspect of scour modeling, computational time limitation is a big barrier for a numerical scour model to be adopted by the industry. Currently, the model developed by Dixen et al. (2012) and DHI (2012) are the only available hybrid scour models. The opportunities of developing a simplified numerical scour models can have significant impact in providing a practical scour assessment tools for the industry, which can be adopted rapidly in the near future.

These knowledge gaps have motivated the work done in this project and will be discussed and addressed in greater detail in each subsequent chapters.

Chapter 3

Turbulence Modeling Around Cylindrical Bluff Bodies

In this chapter, the focus was to obtain a good understanding of the flow around a cylindrical bluff body. In particular, efforts were made to determine the most appropriate turbulence model which can best model the complex flow around a cylindrical bluff body while maintaining a reasonable computational time.

3.1 Introduction

The flow around a cylindrical bluff body is a classical, yet still currently challenging flow problem. Although a significant amount of work has been done in this area, there are still numerous gaps in the current understanding of this phenomenon as discussed about in chapter 2 earlier. This is especially so for high Reynolds number flow around a cylinder where limitations in the scaling of an experimental setup and the compressibility effects of the fluid present significant challenges, and this has limited the understanding of this flow regime. With increasing computational capabilities, numerically modeling high Reynolds number flows have become increasingly feasible which can provide greater insight into such flows.

Hence, in this chapter, the work done will be focused on providing greater insight into modeling a high Reynolds number flow around a cylindrical bluff body using the one- and two- equation RANS turbulence model. In particular, the objective of this chapter is to conduct a comparative study of the one- and two-equation RANS turbulence models on its ability to simulate a high Reynolds number ($Re = 10^6$) flow around a cylindrical bluff body. The list of the RANS turbulence models used in this study are as follow:

- Spalart-Allmaras
- $k - \varepsilon$ (Standard)
- $k - \varepsilon$ (RNG)
- $k - \varepsilon$ (Realizable)
- $k - \omega$ (Standard)
- $k - \omega$ (SST)

Particular attention will be paid to the $k - \omega$ (SST) model as earlier works have indicated its superior performance for lower Reynolds number flow. In addition, the ability of improving the $k - \omega$ (SST) model using a curvature correction and in capturing the effect of surface roughness was evaluated.

As it was anticipated that the computational requirements for the numerical scour model to be high, an evaluation of the one-equation Spalart-Allmaras model for bluff-body flows and flows within a scour hole was performed. A rotation correction coefficient for the Spalart-Allmaras model was tuned for such bluff body flow applications.

3.2 Theory

3.2.1 Mechanism of Flow Around a Cylinder

Earlier works on the flow around a cylindrical bluff body have provided a good understanding of this flow mechanism up to a Reynolds number of about 10^6 . A detailed description of such a flow mechanism is presented by Sumer and Fredsøe (1997). Starting from a laminar creep flow at $Re < 5$, laminar vortex street will begin to form at the leeward side of the cylinder. This flow pattern becomes increasingly turbulent with higher Reynolds number and will finally be fully turbulent at approximately $Re > 4 \times 10^6$. This evolving flow pattern with increasing Reynolds number results in the parameters of the flow (coefficient of drag, coefficient of lift, coefficient of pressure, Strouhal number, angle of separation) to similarly vary with Reynolds number.

One unique phenomenon of such flows around a cylindrical bluff body is the drag crisis. The pressure distribution around a cylinder for the subcritical region/pre-drag crisis ($Re = 1 \times 10^5$) and supercritical region/post-drag crisis ($Re = 8.5 \times 10^5$) is shown in Figure 3.1, where S is the point of separation. During the drag crisis, the angle of separation will shift backward from 78 degrees ($Re = 1 \times 10^5$) to 140 degrees ($Re = 8.5 \times 10^5$). This backward shift of the angle of separation will result in a narrower wake region, resulting in a pressure distribution around the cylinder to become increasingly asymmetrical in shape. The weaker negative pressure at the wake region results in the drop of the coefficient of drag (as seen during the drag crisis Figure 3.1(a)). In addition, due to the narrower wake region, the vortex shedding frequency will increase, resulting in a higher Strouhal number as seen in Figure 3.1(c). Hence, when evaluating such flows around cylindrical bluff bodies, it is critical to identify which regime the flow is under.

Figure 3.1: The variation of the coefficient of drag, coefficient of lift and Strouhal number during the drag crisis (Sumer and Fredsøe, 1997).

3.2.2 RANS turbulence modeling

For the RANS turbulence model, it utilizes a Reynolds decomposition which defines the instantaneous flow property ($\tilde{\phi}$) to be the superposition of its mean component (ϕ) and its fluctuating component (ϕ') as shown in Equation 3.1.

$$\tilde{\phi} = \phi + \phi' \quad (3.1)$$

Substituting this Reynolds decomposition equation into the Navier-Stokes equation and taking the time average, the resultant continuity and momentum equation are given as:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3.2)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \left(\frac{\partial p}{\partial x_i} \right) + \nu \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial \overline{u'_i u'_j}}{\partial x_j} \quad (3.3)$$

where p is the dynamic pressure, u_i and u_j are the mean flow velocities, ν is kinematic viscosity and ρ is the density. These equations have a similar form to the original Navier-Stokes equation. However, an additional unknown stress term, called the Reynolds Stresses ($-\rho \overline{u'_i u'_j}$), appears in the momentum equation which results in this set of equation being open.

There are two main approaches to model this Reynolds stresses, which are the Boussinesq approach and the Reynolds Stress Transport Model (RSM). In this work, the Boussinesq approach will be used to model the Reynolds stresses given as:

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\rho k) \delta_{ij} \quad (3.4)$$

where μ_t is the turbulent viscosity, k is the turbulent kinetic energy and the δ_{ij} is the Kronecker delta. Using this Boussinesq approach, additional transport equations must be solved to obtain the turbulent viscosity (μ_t) in Equation 3.4. In this work, focus will be placed on the one- and two- equation RANS turbulence models. Initially, the more popular two-equation turbulence models will be discussed and analysed. Subsequently, the one-equation model, which was originally developed for aerodynamic flows, will be reviewed and developed for cylindrical bluff body flows.

3.2.2.1 Two-Equation Turbulence Models

The $k - \varepsilon$ and $k - \omega$ turbulence models are popular two-equation closure models which solve the turbulent viscosity (μ_t) by modeling the transport equations of either the turbulence kinetic energy (k) and turbulence dissipation rate (ε) or the turbulence kinetic energy (k) and specific dissipation rate (ω). For each turbulence model, modifications have been proposed by various researchers resulting in various versions of the $k - \varepsilon$ model (standard, RNG and realizable) and $k - \omega$ model (standard and SST).

$k - \varepsilon$ model

For the standard $k - \varepsilon$ model (Launder and Spalding, 1972), it is a semi-empirical formula whereby the turbulence kinetic energy (k) was derived from an exact equation and the turbulence dissipation rate (ε) was determined using physical reasoning. The k and ε transport equations are given as:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (3.5)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (3.6)$$

where G_k and G_b represent the generation of turbulence kinetic energy, Y_M represents the contribution of the fluctuating dissipation in compressible turbulence to the overall dissipation rate, σ_k and σ_ε are the turbulence Prandtl numbers, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants and S_k and S_ε are user-defined source terms. Using k and ε , the turbulent viscosity (μ_t) is subsequently solved using Equation 3.7.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3.7)$$

where C_μ is a constant.

Based on the standard $k - \varepsilon$ model, numerous improvements to the model have been proposed. Two of such models are the $k - \varepsilon$ (RNG) model and the $k - \varepsilon$ (realizable) model. The $k - \varepsilon$ (RNG) model (Yakhot and Orszag, 1986) was derived using the statistical renormalization group theory. The resultant model improved the ε transport equation for rapidly strain flow and is also able to account for the effects of swirling flow. In addition, an analytical formula was obtained for the turbulent Prandtl number as compared to a user-defined constant used in the standard $k - \varepsilon$ model.

The $k - \varepsilon$ (realizable) model (Shih et al., 1994) is another of such modified models to the original standard $k - \varepsilon$ model. This model modified the turbulent viscosity formulation and also derived a new ε transport equation from an exact equation. The resultant model satisfies mathematical constraints on the Reynolds stresses which are consistent with the physics of the turbulent flow, hence making the model ‘realizable’. The $k - \varepsilon$ (realizable) model improved the accuracy of predicting the spreading rate of planar and round jets as well as flows with adverse pressure gradients and flow separation.

As the $k - \varepsilon$ model was developed for turbulent core flows, a wall function is needed to model the near wall regions. In this work, a standard wall function (Launder and Spalding, 1974), which is valid for fully turbulent flows, was used for all $k - \varepsilon$ model simulations performed.

$k - \omega$ model

Similarly for the $k - \omega$ model, the transport equation of the turbulence kinetic energy (k) and specific dissipation rate (ω) are used to close the RANS equations via the Boussinesq hypothesis. For the standard or Wilcox $k - \omega$ model (Wilcox, 1988), it is an empirical based model with the k and ω transport equations given as:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (3.8)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \quad (3.9)$$

where G_k and G_ω represent the production terms of k and ω , Y_k and Y_ω represent the dissipation terms of k and ω , Γ_k and Γ_ω represents the effective diffusivity of k and ω , and finally S_k and S_ω are the user-defined source terms.

The turbulent viscosity (μ_t) used in the Boussinesq hypothesis is subsequently determined using Equation 3.10.

$$\mu_t = \alpha^* \frac{k^2}{\omega} \quad (3.10)$$

where α^* is the damping coefficient of the turbulent viscosity for low Reynolds flows.

A variation of the $k - \omega$ model is the $k - \omega$ (SST) model (Menter, 1994) which capitalizes on the accuracy of the $k - \omega$ model within the near-wall region and the $k - \varepsilon$ model in the far-field region. Such an approach is done by transforming the $k - \varepsilon$ model into a $k - \omega$ formulation and incorporating a blending function between the two regions. As such, the $k - \omega$ (SST) model is able to model a wide range of flow profiles with increased accuracy. The blending function between the near-wall and far-field region is embedded within the derivation of the production, dissipation, diffusivity and cross-diffusion terms of the transport equation given as follows:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \quad (3.11)$$

where ϕ_1 represents any constants in the original $k - \omega$ model, ϕ_2 represents any constants in the transformed $k - \varepsilon$ model, ϕ is the resultant constant of the model and F_1 is the blending function, which is 1 in the near wall region and 0 far away from the surface. Details of the $k - \omega$ (SST) model and its blending function are described fully by Menter (1994).

For the $k - \omega$ model, an enhanced wall function (Kader, 1981) is adopted to model the near wall regions. As such, the mesh along the wall boundary need not be significantly fine ($y^+ \simeq 1$) and a coarse mesh ($y^+ \geq 30$) within the fully turbulent region can be utilized.

Curvature correction

A limitation of an eddy-viscosity based model is that these models are less sensitive to system rotation and streamline curvature. Hence, in order to correct for this limitation, a curvature correction was proposed by Smirnov and Menter (2009), which was based on a rotation-curvature correction suggested by Spalart and Shur (1997) earlier. In this curvature correction modification, a multiplier (f_{r1}) given by Equation 3.12 was incorporated into the production term of the $k - \omega$ (SST) model.

$$f_{r1} = \max\{\min(f_{rotation}, 1.25), 0.0\} \quad (3.12)$$

with

$$f_{rotation} = (1 + c_{r1}) \frac{2r^*}{1 + r^*} [1 - c_{r3} \tan^{-1}(c_{r2} \tilde{r})] - c_{r1} \quad (3.13)$$

where c_{r1} , c_{r2} and c_{r3} are empirical constant and r^* and $\tilde{r}$ are functions of vorticity and strain rate tensors. As such, this result in the k and ω transport equations to be given as:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k f_{r1} - Y_k + S_k \quad (3.14)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + \alpha_\phi \frac{\rho G_k}{\mu_t} f_{r1} - Y_\omega + S_\omega \quad (3.15)$$

where α_ϕ is a constant computed by the blending function from the corresponding constants of the $k - \omega$ and $k - \varepsilon$ models. Hence, with this f_{r1} multiplier influencing the turbulence production term, turbulence production will be suppressed or enhanced accordingly.

For the multiplier, a limit was imposed, $0 \leq f_{r1} \leq 1.25$, to ensure numerical stability (for strong convex curvature, $f_{r1} = 0$) and to prevent over-generation of eddy viscosity in the flow with destabilizing curvature (for strong concave curvature, $f_{r1} = 1.25$).

Surface roughness

To account for surface roughness, the law of the wall can be modified by incorporating an additive constant, ΔB , into the log-law given as follows;

$$\frac{u_{wall}u^*}{\tau_w/\rho} = \frac{1}{\kappa} \ln \left(E \frac{\rho u_\tau d_{wall}}{\mu} \right) + \Delta B \quad (3.16)$$

where d_{wall} and u_{wall} is the wall distance and velocity at the center point of the wall adjacent cell and $u^* = C_\mu^{1/4} k^{1/2}$.

The derivation of ΔB can be determined based on the non-dimensional surface roughness height, K_s^+ , define as:

$$K_s^+ = \rho K_s u^* / \mu \quad (3.17)$$

where K_s is the physical roughness height. Based on the non-dimensional surface roughness height, K_s^+ , Cebeci and Bradshaw (1977) proposed formulas for ΔB based on the surface roughness regime it is in:

Smooth Regime ($K_s^+ \leq 2.25$)

$$\Delta B = 0 \quad (3.18)$$

Transitional Regime ($2.25 < K_s^+ \leq 90$)

$$\Delta B = \frac{1}{\kappa} \ln \left[\frac{K_s^+ - 2.25}{87.75} + C_s K_s^+ \right] \times \sin\{0.4258(\ln K_s^+ - 0.811)\} \quad (3.19)$$

Fully rough Regime ($K_s^+ > 90$)

$$\Delta B = \frac{1}{\kappa} \ln(1 + C_s K_s^+) \quad (3.20)$$

where κ is the von Kármán constant and C_s is the roughness constant. Hence, based on the non-dimensional surface roughness height, K_s^+ , the ΔB proposed by Cebeci and Bradshaw (1977) is added to the log-law and subsequently the shear stresses and turbulent quantities at the wall are determined accordingly.

3.2.2.2 One-Equation Turbulence Models

In addition to the two-equation models, the one-equation turbulence models, such as the Spalart-Allmaras model (Spalart and Allmaras, 1994), was likewise evaluated for bluff body flows. These one-equation turbulence models are generally computationally less intensive turbulence model due to the reduced number of equations to be solved. Though initially developed for aerodynamic flows, it is proposed in this work that the Spalart-Allmaras model can be tuned for bluff body flows, in particular that of a cylindrical configuration, which is the primary focus in this work.

Spalart-Allmaras model

For the Spalart-Allmaras model, a kinematic eddy viscosity ($\tilde{\nu}$) transport equation is utilized given as:

$$\frac{\partial}{\partial t}(\rho\tilde{\nu}) + \frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i) = \frac{1}{\sigma_{\tilde{\nu}}} \left[\frac{\partial}{\partial x_j} \left\{ (\mu + \rho\tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right\} + C_{b2}\rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] + G_{\tilde{\nu}} - Y_{\tilde{\nu}} + S_{\tilde{\nu}} \quad (3.21)$$

where $G_{\tilde{\nu}}$ and $Y_{\tilde{\nu}}$ represent the production and destruction of the eddy viscosity, $\sigma_{\tilde{\nu}}$ & C_{b2} are constants and $S_{\tilde{\nu}}$ is a user-defined source term. The turbulent viscosity needed by the Boussinesq assumption is subsequently obtained by,

$$\mu_t = \rho\tilde{\nu}f_{v1} \quad (3.22)$$

where f_{v1} from Equation 3.22 is obtained from,

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3} \quad (3.23)$$

$$\chi = \frac{\tilde{\nu}}{\nu} \quad (3.24)$$

where C_{v1} is a constant.

The production term, $G_{\tilde{\nu}}$, in the kinematic eddy viscosity transport equation is given as follow,

$$G_{\tilde{\nu}} = C_{b1}\rho\tilde{S}\tilde{\nu} \quad (3.25)$$

where C_{b1} is a constant. The $\tilde{S}$ term in the turbulence production component is given by,

$$\tilde{S} \equiv |\Omega_{ij}| + \frac{\tilde{v}}{\kappa^2 d_{wall}^2} f_{v2} \quad (3.26)$$

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (3.27)$$

where κ is the von Kármán and d_{wall} is the distance of the point to the nearest wall. $|\Omega_{ij}|$ is the magnitude of the vorticity given as,

$$|\Omega_{ij}| \equiv \sqrt{2\Omega_{ij}\Omega_{ij}} \quad (3.28)$$

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (3.29)$$

For the destruction term, $Y_{\tilde{v}}$, in the kinematic eddy viscosity transport equation, it is given as,

$$Y_{\tilde{v}} = C_{w1} \rho f_w \left(\frac{\tilde{v}}{d_{wall}} \right)^2 \quad (3.30)$$

where C_{w1} is a constant. f_w is obtained from the following three equation:

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{1/6} \quad (3.31)$$

$$g = r + C_{w2}(r^6 - r) \quad (3.32)$$

$$r \equiv \frac{\tilde{v}}{\tilde{S} \kappa^2 d_{wall}^2} \quad (3.33)$$

where C_{w2} and C_{w3} are constants.

The values of the constants within the Spalart-Allmaras model used in this work are identical to that originally proposed by Spalart and Allmaras (1994) shown in Table 3.1.

Table 3.1: Constants of the Spalart-Allmaras turbulence model proposed by Spalart and Allmaras (1994)

Spalart-Allmaras Model Constants			
$C_{b1} = 0.1355$	$C_{b2} = 2/3$	$\sigma_{\tilde{v}} = 0.667$	$\kappa = 0.41$
$C_{\omega1} = 3.208$	$C_{\omega2} = 0.3$	$C_{\omega3} = 2$	$C_{v1} = 7.1$

Since the initial proposal of the Spalart-Allmaras model, numerous modifications to this turbulence model have been proposed by various researchers (Rumsey et al., 2010). In the present study, the focus will be on the modification proposed by Dacles-Mariani et al. (1995) whereby a rotation correction is included into the Spalart-Allmaras model. The purpose of the correction is to account for regions where the vorticity exceeds the strain rate and where turbulence is suppressed, such as that in the core of a vortex. This correction is done by reducing the eddy viscosity within such a region by modifying the magnitude of vorticity ($|\Omega_{ij}|$) in the production term as follows.

$$|\Omega_{ij}| + C_{prod} \min(0, |S_{ij}| - |\Omega_{ij}|) \quad (3.34)$$

where

$$|S_{ij}| \equiv \sqrt{2S_{ij}S_{ij}} \quad (3.35)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \quad (3.36)$$

For the constant C_{prod} , it was initially proposed that a value of 2.0 be utilized (Dacles-Mariani et al., 1995). This value was suggested based on an aerodynamic flow case being evaluated by Dacles-Mariani et al. (1995). As such, the necessity to re-calibrated this constant for cylindrical bluff-body flows is likely.

For the wall boundary condition, a wall function which is that of a logarithmic layer was utilized given as,

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho u_\tau d_{wall}}{\mu} \right) \quad (3.37)$$

where u is the velocity parallel to the wall, u_τ is the shear velocity and $E = 9.793$. Such a wall function is valid for fully turbulent flow regions, whereby the $y^+ \geq 30$, and where a coarser mesh can be utilized.

3.3 Model Setup

The simulation model configuration was setup as a cylindrical bluff body within the trans-critical flow regime, similar to that experienced by numerous offshore structures. A diameter of $D = 4.2$ m and an inlet velocity of 1.25 m/s was utilized in the simulations performed. Together with the viscosity of water ($\nu = 1.004 \times 10^{-6}$), the flow parameters results in a Reynolds number of $Re = 5.2 \times 10^6$, which is similar to that experienced by typical offshore wind turbine foundations. The simulations performed in this part of the work were done in FLUENT.

Model geometrical dimension and mesh

In this section, simulations were done for both two-dimensional and three-dimensional models. To evaluate the two-equation turbulence models, a two-dimensional simulation model was utilized. As for the evaluation of the one-equation turbulence model and the flow within a scour hole, both two-dimensional and three-dimensional models were utilized. The process in which the models were created is such that the two-dimensional model was initially built and subsequently, the three-dimensional mesh was obtained by extruding the two-dimensional model in the z-direction. For the simulation within a scour hole, the model was built based on the experimental setup by Melville and Raudkivi (1977), whereby a cylinder diameter of 5.08cm was used. Details of the model will be discussed below.

Two-Dimension model

The geometrical dimension of the computational domain used for all the simulations performed was $25D \times 20D$. From the center of the cylinder, the velocity inlet boundary was $10D$ upstream while the outflow boundary was $15D$ downstream from the center of the cylinder. The upper and lower far-field symmetrical boundaries were $10D$ from the center of the cylinder. This lateral distance chosen for the upper and lower far field is consistent with that proposed by Behr et al. (1995) whereby a distance of $8D$ or greater is recommended for flows around cylindrical bodies.

A hexahedral block mesh was used for all the simulations. The meshing of the model was divided into 2 zones, the inner zone, which is a $4D \times 4D$ block around the cylinder, and the outer zone, which is the zone away from the cylinder. For the inner zone, the grid expansion ratio ranged from 1.01 - 1.05 from the cylinder wall. As for the outer zone, the grid expansion ratio from the inner zone edge was constant at 1.02 downstream and 1.05 for the upper and lower far fields and the upstream direction. Figure 3.2 shows the geometrical dimensions, meshing and boundary conditions of the computational domain for the x-y plane used in the simulations.

Three-Dimension model

As mentioned earlier, the three-dimensional model was obtained by extruding the two-dimensional model in the z-direction by $3D$ as shown in Figure 3.3. The mesh configuration in the x-y plane was identical to that used in the two-dimensional mesh. For the z-direction, the grid expansion ratio of 1.20 from the bottom boundary was utilized.

Figure 3.2: The geometrical dimension, meshing and boundary conditions of the computational domain for the x-y plane used in the simulations.

Figure 3.3: The geometrical dimension, meshing and boundary conditions of the computational domain in the z-direction used in the simulations.

Scour hole model

For the simulations performed for the flow within a scour hole, a monopile diameter of 5.08cm was used, identical to that of Melville and Raudkivi (1977) experimental setup. The size of the simulation domain was identical to that specified in Figure 3.2 and 3.3, whereby $D = 5.08\text{cm}$. The 3 configurations of the scour hole (an initially flatbed, intermediate and equilibrium scour hole) shown in Figure 3.4 are analysed in this work. These 3 configurations are likewise identical to that measured and presented by Melville and Raudkivi (1977). For the initially flatbed configuration, the three-dimensional mesh setup (domain dimensions and grid expansion ratio) was similar to that described above. From the initially flatbed, the intermediate and equilibrium scour hole was obtained by deforming the mesh 4.71 cm and 6.13 cm respectively using a Mesh Morpher module available in FLUENT (ANSYS, 2011). For the Mesh Morpher module, the user defines control points (X_{origin}) along the mesh boundary and specifies the displacement (X_{disp}) needed to obtain the new mesh configuration (X_{new}) given as follows:

$$X_{new} = X_{origin} + X_{disp} \quad (3.38)$$

In the development of the intermediate and equilibrium scour hole configuration, the control points adjacent to the monopile was displaced by 4.71 cm and 6.13 cm respectively. Based on the movement of these control points, the corresponding domain within the control points are deformed based on a free-form deformation method (Sederberg and Parry, 1986) to finally obtain the scour hole configuration shown in Figure 3.4. Full details of the Mesh Morpher module is described by Jemcov and Maruszewski (2009).

Figure 3.4: The scour hole mesh and dimensions used in the simulations. The depth of the intermediate and equilibrium scour hole was identical to that of Melville and Raudkivi (1977) experimental setup.

Boundary conditions and input parameters

The boundary conditions and input parameters of the computational domain used in the simulations are shown in Table 3.2. For the input parameters defined in Table 3.2, , it describes the x-velocity component (U_x), y-velocity component (U_y) and z-velocity components (U_z) respectively.

Table 3.2: Boundary conditions and input variables for the simulations performed in this work

Model Type	Boundary	Boundary Conditions	Input Parameters
All	Upstream	Velocity Inlet	$U_x = 1.25m/s^*$, $U_y = 0m/s$, $U_z = 0m/s$
All	Down-stream	Outflow	$U_x = \text{free}$, $U_y = \text{free}$, $U_z = \text{free}$
All	Upper and Lower Far-Field	Symmetry	$U_x = \text{free}$, $U_y = 0m/s$, $U_z = \text{free}$
All	Cylinder	Wall	$U_x = 0m/s$, $U_y = 0m/s$, $U_z = 0m/s$
3-Dim. /Scour hole	Top	Symmetry	$U_x = \text{free}$, $U_y = \text{free}$, $U_z = 0m/s$
3-Dim. /Scour hole	Bottom	Wall	$U_x = 0m/s$, $U_y = 0m/s$, $U_z = 0m/s$

*For simulations within a scour hole, a velocity inlet of $U_x = 0.25m/s$ was utilized.

For the two-equation turbulence model, the parameters of the inlet free stream flow were obtained from Jones et al. (1969) experiments whereby the turbulence intensity was $I = 0.17\%$. In addition, the turbulence viscosity ratio of $\mu_t/\mu = 1$, which is recommended for external flows (ANSYS, 2011), was used in the simulations performed. The turbulent kinetic energy (k), turbulent dissipation rate (ε) and specific dissipation rate (ω) were subsequently determined as follows:

$$k = \frac{3}{2} (U_m I)^2 \quad (3.39)$$

$$\varepsilon = \rho C_\mu \frac{k^2}{\mu} \left(\frac{\mu_t}{\mu} \right)^{-1} \quad (3.40)$$

$$\omega = \rho \frac{k}{\mu} \left(\frac{\mu_t}{\mu} \right)^{-1} \quad (3.41)$$

where U_m is the mean velocity and $C_\mu = 0.09$ is an empirical constant.

For the Spalart-Allmaras simulation, at the wall boundary, the kinematic eddy viscosity was set as $\tilde{\nu} = 0$. As for the inlet boundary condition, Allmaras and Johnson (2012) suggest values between 3 and 5 for the ratio $\tilde{\nu}/\nu$ for a freestream fully turbulent flow. This results in the kinematic eddy viscosity to be estimated at $\tilde{\nu} = 0.001m^2/s$ for the inlet condition.

In all the simulations performed, a uniform velocity profile was applied at the velocity inlet boundary. For the 3-dimensional simulation models with a wall boundary bottom, the 10D upstream zone of the domain allowed for the development of the boundary layer velocity profile of the flow before reaching the cylinder. When benchmarking the current simulation boundary layer velocity profile at the cylinder against that presented by Richardson and Panchang (1998), whose work will subsequently be utilized to validate the current scour hole simulations, the boundary layer thickness was shown to have good correlation with each other. As shown in Figure 3.5, the boundary thickness obtained by Richardson and Panchang (1998) was approximately $0.013m$, which is in close agreement to the boundary layer thickness of $0.0145m$ obtained in the current simulation of similar conditions.

Figure 3.5: The boundary layer velocity plots presented by Richardson and Panchang (1998) against that obtain in the current simulation using similar conditions.

Numerical scheme

A Finite Volume Method (FVM) approach was used to solve the governing equations in FLUENT. For the velocity-pressure coupling, a SIMPLE algorithm (Patankar and Spalding, 1972) was used while a second order upwind scheme (Barth and Jespersen, 1989), which is stable and has a second-order accuracy, was used for the spatial discretization of the simulations done. While a PISO algorithm (Issa, 1986) is predominately adopted for transient simulations, a SIMPLE scheme was utilized in this work in anticipation for the more complex flow and time-step requirements of the scour model simulations which was performed in the subsequent chapter.

Time-step

In this work, the time-step for the simulations performed was determined based on the Strouhal number observed in experiments. Based on the experimental data, the approximate Strouhal number for a Reynolds number flow of 5×10^6 is $St \cong 0.25$. Hence, the period of oscillations of the flow can be determined based on Equation 3.42 given as,

$$St = \frac{D}{U_m T} \quad (3.42)$$

where T is the period of oscillation of the vortex shedding. Unal and Goren (2006) suggested a time-step period of $(1/200)T$ to capture the rapid changing nature of the flow. In this work, a significantly finer time-step period of $(1/300)T$ was utilized as compared to that suggest by Unal and Goren (2006). This results in a time-step of approximately $0.04sec$ for the simulations performed with a Reynolds number of 5.2×10^6 .

Mesh sensitivity study

A mesh sensitive study was done on each of the turbulence models used in this study to ensure a grid independent solution was obtained for all the simulations. The mesh sensitive study was performed for the 2-dimensional models only as the 3-dimensional models had a mesh configuration in the x-y plane which was similar to the 2-dimensional models. In this sensitivity study, the coefficient of drag (C_D), the coefficient of lift root-mean-square value (C_{Lrms}) and Strouhal number (St) were utilized as the parameters benchmarked for this sensitivity study. The percentage difference presented in Table 3.3 was the difference between the calculated parameter using the current mesh against the calculated parameter for the proceeding mesh (i.e. M1 and M2, M2 and M3). A grid independence result was achieved when the percentage difference was less than 5%.

Table 3.3: Results of the mesh sensitivity study for the $k - \omega$ (SST) model

ID	Mesh Size (Elements)	C_D	Difference (%)	C_{Lrms}	Difference (%)	St	Difference (%)
M1	134900	0.508	10.03%	0.242	23.55%	0.305	5.25%
M2	227232	0.457	0.21%	0.185	2.16%	0.321	0.62%
M3	325244	0.458	-	0.181	-	0.323	-

Table 3.3 shows one of the results of the grid sensitivity study for the $k - \omega$ (SST) model. For M2 (227232 elements), increasing the mesh size by 43.13% to obtain M3 (325244 elements), a 0.21% variation in C_D , 2.16% variation in C_{Lrms} , and 0.62% variation in St was achieved. With this minor variation of C_D , C_{Lrms} , and St values with a significant increase in mesh size, it can be determined that a mesh independence solution was obtained for M2 and hence this mesh was used for the simulations performed.

Table 3.4: Results of the mesh sensitivity study for the $k - \omega$ (SST) model with a roughness height of 4.24×10^{-2} m.

ID	Mesh Size (Elements)	C_D	Difference (%)	C_{Lrms}	Difference (%)	St	Difference (%)
M1	134900	0.661	17.99%	0.290	48.94%	0.221	17.23%
M2	227232	0.806	0.88%	0.568	0.18%	0.267	4.64%
M3	325244	0.799	-	0.569	-	0.280	-

A similar mesh sensitive study was done for the simulation performed with varying surface roughness height. The meshes utilized in this sensitivity study, M1, M2 and M3, was identical to that described earlier. In this mesh sensitivity study, only the model with surface roughness height of 4.24×10^{-2} m was evaluated (which is the largest roughness height utilized in this work). Subsequently, the mesh identified in this sensitivity study was utilized for all the other similar simulations performed in this work, including the models with a smaller surface roughness height. As seen from Table 3.4, mesh independence was achieved for mesh M2 (227232 elements). For M2, when the mesh size is increased by 43.13% (M3), the variation in C_D , C_{Lrms} and St were less than 5%, which is the condition set earlier for grid independence to be achieved. Hence, mesh M2 was utilized for all the simulations performed with varying surface roughness.

3.4 Results and Discussion

3.4.1 Two-equation Turbulence Models

A comparative study was performed between the popular two-equation RANS turbulence models, in particular the $k - \varepsilon$ (Standard) , $k - \varepsilon$ (RNG), $k - \varepsilon$ (Realizable), $k - \omega$ (Standard) and $k - \omega$ (SST) models. The key parameters used to benchmark each turbulence model are the coefficient of pressure-base (at angle 180° , C_{pbase}), coefficient of drag (C_D), coefficient of lift root-mean-square (C_{Lrms}), angle of separation (θ_{sep}) and the Strouhal number (St). These parameters are of significant interest in many practical engineering problems as they provide not only the key elements to predict the loading on a structure due to the fluid flow (C_{pbase}, C_D, C_{Lrms}), but also provide a good quantitative description of the flow patterns (θ_{sep}, St) around the structure. It is noted that a smooth cylinder was used in these initial simulations.

The coefficient of pressure-base (C_{pbase}) obtained from the simulation results are presented in Table 3.5, together with the experimental data. When analyzing the pressure coefficient at the base of the cylinder (C_{pbase}), it was observed that both the $k - \omega$ (SST) model and the $k - \omega$ (Standard) model correlated well with Jones et al. (1969) data of $C_{pbase} = -0.621$. For the $k - \varepsilon$ models, the simulations tend to under-predict the resultant base pressure, approximately $C_{pbase} = -0.37$, as compared to experimental results which typically had a C_{pbase} value of greater than -0.60.

In addition to the coefficient of pressure-base (C_{pbase}), the angle of separation (θ_{sep}), coefficient of drag (C_D), Strouhal number (St) and coefficient of lift-rms (C_{Lrms}) of the simulations are similarly presented in Table 3.5. The coupling between these parameters, as will be explained later, provides a good understanding and justification of the superior nature one turbulence model has over another when modeling a high Reynolds flow around a cylinder.

Table 3.5: The coefficient of pressure-base, angle of separation, coefficient of drag, Strouhal number and coefficient of lift-rms of the two-equation turbulence models simulation results and experimental observations.

Model	Coeff. Of Pressure-base (C_{pbase})	Angle of Sep. (θ_{sep})	Coeff. Of Drag (C_D)	Strouhal number (St)	Coeff. Of Lift rms (C_{Lrms})
$k - \varepsilon$ (Standard)	-0.311	123.47	0.270	0.446	0.0744
$k - \varepsilon$ (RNG)	-0.372	121.83	0.307	0.365	0.1702
$k - \varepsilon$ (Realizable)	-0.385	122.38	0.313	0.426	0.0788
$k - \omega$ (Standard)	-0.594	114.70	0.463	0.320	0.4079
$k - \omega$ (SST)	-0.594	109.76	0.457	0.321	0.1847
Achenbach (1968) $Re = 4.7 \times 10^6$	-	111.94	0.726	-	-
James et al. (1980) $Re = 5.46 \times 10^6$	-	110.94	0.317	0.229	-
Schmidt (1966) $Re = 5.0 \times 10^6$	-0.605	-	0.533	-	0.1379
Roshko (1961) $Re = 5.32 \times 10^6$	-0.853	-	0.697	0.263	-
Schewe (1983) $Re = 5.25 \times 10^6$	-	-	0.532	0.262	0.0467
Jones et al. (1969) $Re = 5.61 \times 10^6$	-0.621	-	0.636	0.253	0.1140

**Wall functions were utilized in the $k - \varepsilon$ and $k - \omega$ models.

For the angle of separation, Table 3.5 shows that the $k - \omega$ (SST) model predicts the separation at $\theta_{sep} = 109.76^\circ$, which correlates well with Achenbach (1968) and James et al. (1980). Similarly, for the $k - \omega$ (Standard) model, a reasonable agreement between the predicted θ_{sep} and experimental data was observed, whereby a reasonable agreement of the simulations is defined as to be within a 25% deviation (generally within engineering tolerance) of that observed in experimental results. However, for all the $k - \varepsilon$ models, the predicted angle of separation was significantly larger than the experimental data. Such an over-prediction of the angle of separation by the $k - \varepsilon$ model was likewise reported by Zhang et al. (2014). For flow around a cylinder, the accurate modeling of the angle of separation is critical. This is because the width of the resultant leeward vortex shedding is highly dependent on the angle at which the flow separates. A smaller angle of separation will have a broader width of the leeward vortex shedding behind the cylinder, and this results in a larger based pressure at the rear of the cylinder and a higher coefficient of drag. The vice-versa is similarly valid for larger angle of separations.

Hence, when analysing the C_D obtained from the simulations, the good agreement using the $k - \omega$ (SST) model is due to the models good prediction of the angle of separation (Table 3.5). For the $k - \varepsilon$ model, the larger angle of separation predicted results in a narrower leeward vortex region, and hence a C_D which slightly underpredicts the smallest C_D obtained by James et al. (1980). For the Strouhal number, both the $k - \omega$ and the $k - \varepsilon$ models over-predicts the St number compared to that observed by experiments. However, the lower St number predicted by the $k - \omega$ models correlate better with experiments compared to the $k - \varepsilon$ models. Again, this trend is attributed to the angle at which the flow separates. For larger angle of separation, the narrower leeward vortex increases the frequency of the vortex oscillations and hence an increase in the Strouhal number and vice-versa for smaller angle of separations.

For the coefficient of lift-rms value, Table 3.5 shows that the $k - \varepsilon$ models have a better correlation with experimental data compared to the $k - \omega$ models, falling within the range of $C_{Lrms} = 0.0467 - 0.1379$ observed by Schewe (1983) and Schmidt (1966) respectively. The $k - \omega$ models tend to over-predict the C_{Lrms} . But the predicted C_{Lrms} value of the $k - \omega$ (SST) model agreed reasonably with the results of Schmidt (1966).

From the results obtained in the simulations, one may conclude that the $k - \omega$ (SST) model agreed best with experimental data for high Reynolds flow (trans-critical flows) around a cylinder. The key parameter predicted well by the $k - \omega$ (SST) model is the angle at which the fluid separates around the cylinder, which resulted in good prediction of the C_D and the St number compared to the other two-equation RANS turbulence models. For the $k - \varepsilon$ models, due to the larger angle of separation predicted, the resultant C_D and the St number did not agree as well with the experimental data as compared to the $k - \omega$ (SST) model.

Curvature correction ($k - \omega$ (SST) model)

A weakness of the eddy-viscosity model is in capturing the effect of the flow curvature and rotation. To sensitize the turbulence model to account for such curvature, a multiplier, as described in the earlier section, can be incorporated into the production term of the turbulence model. To evaluate the ability of improving the $k - \omega$ (SST) model further, additional simulations incorporating the curvature correction were performed. The C_p , C_{pbase} , C_D , C_{Lrms} , θ_{sep} and St of the extended simulations are presented in Figure 3.6 and Table 3.6.

Figure 3.6: The coefficient of pressure profile when a curvature correction modification and varying surface roughness are incorporated into the $k - \omega$ (SST) model.

Table 3.6: The coefficient of pressure-base, angle of separation, coefficient of drag, Strouhal number and coefficient of lift-rms of the $k - \omega$ (SST) simulations results with and without a curvature correction and experimental observations.

		$k - \omega$ (SST) Smooth	$k - \omega$ (SST) with Curvature Correction Smooth
Coefficient of Pressure-Base	Simulations	-0.593	-0.632
	Experiments (Jones et al., 1969)	-0.621	-0.621
Coefficient of Drag	Simulations	0.457	0.520
	Experiments (Jones et al., 1969)	0.636	0.636
Coefficient of Lift (RMS)	Simulations	0.184	0.289
	Experiments (Jones et al., 1969)	0.114	0.114
Angle of Separation	Simulations	109	109
	Experiments (James et al., 1980)	110	110
Strouhal Number	Simulations	0.320	0.291
	Experiments (Jones et al., 1969)	0.253	0.253

Figure 3.6 shows that when the curvature correction is activated, the coefficient of pressure profile is in closer agreement with James et al. (1980) as compared to the original $k - \omega$ (SST) turbulence model. While the curvature correction improves the flow model in the upstream portion of the cylinder, moving towards the rear of the cylinder (at angle 180°), the C_{pbase} of both the $k - \omega$ (SST) model, with and without a curvature correction, approximately converges to similar C_{pbase} values.

Referring to Table 3.6, it can be seen that the $k - \omega$ (SST) models, both with and without a curvature correction, has a coefficient of pressure-base of $C_{pbase} \approx -0.6$. Examining other parameters in addition to the coefficient of pressure-base, the coefficient of drag and coefficient of lift-rms results were likewise evaluated as shown in Table 3.6. As seen in Table 3.6, the curvature correction causes both the coefficient of drag, C_D , and coefficient of lift-rms, C_{Lrms} , to increase. While the resultant C_D is closer to the experimental value observed by Jones et al. (1969) as seen in Table 3.6, an increase in C_{Lrms} caused a larger discrepancy between the simulation and experimental results. For the angle of separation, Table 3.6 shows that the curvature correction had minimal influence. However, the curvature correction caused a decrease in the value of St ($= 0.291$), and hence a better agreement with experimental observations.

It may be concluded that the curvature correction does influence the predicted profiles and characteristics of a high Reynolds number flow around a cylinder. While omitting such a correction factor results in simulations that correlate reasonably well with experiments, it is beneficial to incorporate the curvature correction, which requires minimal additional computational cost, but improves the predicted C_p profile, C_D and St number.

Surface roughness ($k - \omega$ (SST) model)

The simulations performed above were for that of a smooth cylinder. For high Reynolds number flow around a cylinder, earlier works (Roshko, 1961; Jones et al., 1969; James et al., 1980) have suggested that the surface roughness has a significant influence on the resultant flow parameters. Experimental observations by Shih et al. (1993) and Achenbach (1971) on the effects of surface roughness for flows around a cylinder at high Reynolds number noted that an increase in the surface roughness was able to reduce the Reynolds number at which the transition between the flow regimes occurs.

Table 3.7: The roughness height, K_s/D ratio and roughness constant used in the simulations around a rough cylinder using the $k - \omega$ (SST) model.

Model Name	Roughness Height, K_s (m)	K_s/D Ratio	Roughness Constant, C_s
$k - \omega$ (SST) (Rough 1)	1.26×10^{-03}	3.00×10^{-4}	0.50
$k - \omega$ (SST) (Rough 2)	5.04×10^{-03}	1.20×10^{-3}	0.50
$k - \omega$ (SST) (Rough 3)	4.24×10^{-02}	1.01×10^{-2}	0.50

To understand the ability for a $k - \omega$ (SST) model to capture the effect of surface roughness, additional simulations were performed with different surface roughness. The K_s/D ratios used in these simulations were similar to the one used by Shih et al. (1993) as shown in Table 3.7.

Figure 3.6 and Table 3.8 presents the flow parameters obtained from the simulations performed for the flow around a cylinder with varying surface roughness, together with published experimental data. In Figure 3.6, it is shown that when the surface roughness (Rough 1, Rough 2 and Rough 3) is incorporated, the pressure profile around the sides of the cylinder (at angle approx. 60° - 120°) decreases considerably, as expected with an increase in surface roughness. While the general relationship between the pressure profile and K_s/D ratio is not so clear, the trend of the pressure profile variation with surface roughness is consistent with that expected.

Table 3.8: The coefficient of pressure-base, coefficient of drag, coefficient of lift-rms, angle of separation and Strouhal number for flows around a cylinder with varying surface roughness when utilizing the $k - \omega$ (SST) model

		$k - \omega$ (SST) Rough 1	$k - \omega$ (SST) Rough 2	$k - \omega$ (SST) Rough 3
Coefficient of Pressure-Base	Simulations	-0.804	-0.876	-0.940
	Experiments (Shih et al., 1993)	-0.977	-1.049	-1.074
Coefficient of Drag	Simulations	0.705	0.768	0.806
	Experiments (Shih et al., 1993)	0.860	0.970	1.062
Coefficient of Lift (RMS)	Simulations	0.438	0.505	0.568
	Experiments (Shih et al., 1993)	NA	NA	NA
Angle of Separation	Simulations	108	110	109
	Experiments (Shih et al., 1993)	87	86	84
Strouhal Number	Simulations	0.261	0.258	0.267
	Experiments (Shih et al., 1993)	0.242	0.225	0.213

In addition to the pressure profile, the C_{pbase} , C_D , C_{Lrms} , θ_{sep} and St were likewise evaluated and presented in Table 3.8. For the base pressure of the cylinder (at angle 180°), C_{pbase} , prediction by the $k - \omega$ (SST) model shows an inverse relationship with K_s/D , similar to the trend observed experimentally (Table 3.8). While the magnitude of the C_{pbase} tends to be slightly under-predicted, the slight difference can be attributed to uncertainty in the experimental results (Shih et al., 1993). This is especially so as Shih et al. (1993) did highlight difficulties in obtaining the experimental data for $K_s/D = 3.00 \times 10^{-4}$ (Rough 1) and $K_s/D = 4.24 \times 10^{-02}$ (Rough 3) due to gaps in the cylinder/screen surface (Rough 1) and strong vibrations experienced by the cylinder (Rough 3) during the experiments.

For the coefficient of drag, based on Table 3.8, it can be seen that the $k - \omega$ (SST) model predicts an increasing trend of C_D with an increase in surface roughness, similar to that observed in experiments. Although the magnitude of C_D obtained by the model tends to underestimate that observed in experiments of Shih et al. (1993) by approximately 20% – 25%, the increasing C_D trend with higher surface roughness is clearly captured by the $k - \omega$ (SST) model. As for the coefficient of lift-rms, C_{Lrms} , the $k - \omega$ (SST) model predicts that C_{Lrms} increases with K_s/D ratio, as expected due to the added forces on the cylinder as a result of the surface roughness. Experimental results on C_{Lrms} are however not available for comparison with the current simulations.

Data from Shih et al. (1993) shows that the angle of flow separation and Strouhal number is relatively insensitive to the K_s/D ratio, with a slight inverse relationship (Table 3.8). Similarly, the $k - \omega$ (SST) model captures the insensitivity of these parameters, which fluctuates slightly about a very narrow range of $\theta_{sep} = 108^\circ$ to $\theta_{sep} = 110^\circ$ and $St = 0.258$ to $St = 0.267$. Although the $k - \omega$ (SST) model is unable to predict the θ_{sep} and St slight decreasing trend noted by Shih et al. (1993), the relative insensitivity of θ_{sep} and St to the increase in K_s/D ratio is generally captured well by the $k - \omega$ (SST) model. It is noted that there is a constant magnitude difference between the simulation results and Shih et al. (1993) data. Such a difference can be attributed to the challenges and difficulties in conducting a high Reynolds flow experiment. For a smooth cylinder, Shih et al. (1993) obtained a flow separation of $\theta_{sep} = 98^\circ$ which slightly differs from the value obtained in the present simulations, and also from that observed by Achenbach (1968) and James

et al. (1980) of approximately $\theta_{sep} = 111^\circ$. This magnitude difference is consistent between the current simulation results and Shih et al. (1993) data for the flow around the rough cylinders in this evaluation.

Hence, from the simulation results of the $k - \omega$ (SST) model, the evolution of the flow characteristic with increasing K_s/D can be captured reasonably well, for both the minor variation in the angle of separation and Strouhal number, as well as the more substantial variation of the C_p and C_D with increasing surface roughness. While there are slight discrepancies in the magnitude of the parameters due to challenges of performing high Reynolds flow experiments, the ability of the $k - \omega$ (SST) model to capture the changes in flow parameter with K_s/D ratio is demonstrated.

3.4.2 One-equation Turbulence models

As mentioned earlier, the one-equation Spalart-Allmaras model was evaluated to determine if such a cost-efficient turbulence model can model flows around bluff bodies. In particular, focus was on determining if a rotational correction proposed by Dacles-Mariani et al. (1995) can be tuned for such non-aerodynamic flows. For this analysis of the Spalart-Allmaras model, it was performed in two stages. Initially, a two-dimensional simulation around a cylinder was performed to determine the effectiveness of incorporating a rotation correction into the Spalart-Allmaras model for bluff body flows. Subsequently, the simulation was extended to a three-dimensional model to gain future insight into the Spalart-Allmaras model.

Two-Dimensional simulation

The ability of the Spalart-Allmaras model to capture the turbulence around a bluff body was first analyzed on a two-dimensional plane. To provide a good reference point and benchmark for the Spalart-Allmaras model simulations, the $k - \omega$ (SST) turbulence model simulations performed earlier was used. The $k - \omega$ (SST) turbulence model was selected as the benchmark model due to its optimal simulation results obtained in the earlier section. In addition, Unal and Goren (2006) established earlier that the $k - \omega$ (SST) model is the favored RANS turbulence model for such flows in the sub-critical flow regime. Hence, by comparing the results of the Spalart-Allmaras model with the $k - \omega$ (SST) model, the strengths and limitations of

the Spalart-Allmaras model can be determined. In addition, flow parameters from experimental work by various researchers (Schmidt, 1966; Achenbach, 1968; Jones et al., 1969; James et al., 1980; Schewe, 1983) were also used as a benchmark to compare against the Spalart-Allmaras simulation results.

Figure 3.7: Mean velocity profile for (a) Spalart-Allmaras model with no rotation correction, (b,c) Spalart-Allmaras model with $C_{prod}=2.0$ & 1.0 respectively and (d) $k - \omega(SST)$ turbulence model.

Figure 3.8: Mean turbulence viscosity along the line of symmetry for Spalart-Allmaras model with no rotation correction, Spalart-Allmaras model with $C_{prod}=2.0$ & 1.0 respectively and $k-\omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 5.2×10^6 .

Figures 3.7 and 3.8 shows the mean velocity profile around the cylinder and the mean turbulence viscosity profile along the line of symmetry of the cylinder for the Spalart-Allmaras simulations with different rotational correction coefficient ($C_{prod} = 0, 1.0$ & 2.0) together with the $k - \omega$ (SST) simulation. Based on the mean turbulence viscosity profile, without the rotation correction ($C_{prod}=0$), the Spalart-Allmaras model tends to over predict the turbulences production within the wake region resulting in a significantly higher turbulence viscosity downstream of the cylinder. As a result, when looking at the mean velocity profile of this simulation (Figure 3.7(a)), it can be seen that the wake region is elongated as compared to that of the $k - \omega$ (SST) simulation (Figure 3.7(d)). This is as expected as the turbulence is not suppressed within the vortex core which is one of the shortfalls when applying such a standard Spalart-Allmaras model ($C_{prod}=0$) for bluff-body flows.

When the rotation correction is turned on, the turbulence production within the wake region of the flow is suppressed based on Equation 3.34 described earlier. In the original proposal of the rotation correction for the Spalart-Allmaras model (Dacles-Mariani et al., 1995), $C_{prod} = 2.0$ was proposed for aerodynamic flows (which the Spalart-Allmaras model was originally designed for). However, whether this magnitude of the C_{prod} holds for a bluff body flow remains an open question. Hence, utilizing $C_{prod} = 2.0$ initially, it can be seen that with $C_{prod} = 2.0$, the Spalart-Allmaras model over suppresses the turbulences which results in a signif-

icantly shorter wake region (Figure 3.7(b)) and a lower lee-wake turbulence viscosity profile (Figure 3.8) as compared to the $k - \omega$ (SST) simulation. When the rotation correction was originally proposed, no physical reasoning was linked between $C_{prod} = 2.0$ and the aerodynamic flow being modeled and the magnitude of 2 was selected based on the good correlation the simulation results gave with experimental data. Hence, working along similar lines and based on the current simulation results, the necessity to re-tune the C_{prod} term (between $C_{prod} = 0$ to 2) for a bluff-body flow is evident.

Various simulations were performed, with C_{prod} ranging from 0 to 2. Based on the results, it was identified that $C_{prod} = 1.0$ was ideal for cylindrical bluff body flows. For $C_{prod} = 1.0$, the turbulence suppression results in a mean turbulence viscosity profile that correlates well with that of the $k - \omega$ (SST) simulation as seen in Figure 3.8. Likewise, the wake region of the mean velocity profile compares well with the $k - \omega$ (SST) simulation (significantly improved from the simulation utilizing $C_{prod} = 0$ & 2) as seen in Figure 3.7.

Key flow parameters (coefficient of drag, C_D , and coefficient of lift-rms, C_{Lrms}) from the Spalart-Allmaras simulations were benchmarked against experimental data and values predicted by the $k - \omega$ (SST) model as shown in Table 3.9. For both the coefficient of drag and the coefficient of lift (rms), the Spalart-Allmaras simulation results utilizing $C_{prod} = 1.0$ were within range of experimental observations reported by various researchers (Schmidt, 1966; Achenbach, 1968; Jones et al., 1969; James et al., 1980; Schewe, 1983). Likewise, when taking reference with the $k - \omega$ (SST) simulation, the Spalart-Allmaras simulation results utilizing $C_{prod} = 1.0$ had the closest correlation to the $k - \omega$ (SST) results as compared to that utilizing $C_{prod} = 0$ & 2. When $C_{prod} = 2.0$ was utilized, the turbulence gets overly suppressed, and this results in unphysically high coefficient of drag and coefficient of lift (rms) predictions. Alternatively, when the rotation correction was turned off ($C_{prod} = 0$), while the coefficient of drag gave a reasonable correlation with experimental data, the excessive turbulence in the wake of the flow result in no vortex shedding being obtained, hence resulting in zero C_{Lrms} . These observations with the key flow parameters reinforce the suitability of utilizing $C_{prod} = 1.0$ in the rotation correction for cylindrical bluff body flows.

Table 3.9: The coefficient of drag and coefficient of lift (rms) from experiments, $k-\omega$ (SST) model and the Spalart-Allmaras model with $C_{prod} = 1, 0 \text{ \& } 2$ respectively.

	Coefficient of Drag, C_D	Coefficient of Lift (RMS), C_{Lrms}
Experimental Data Range (Schmidt, 1966; Achenbach, 1968; Jones et al., 1969; James et al., 1980; Schewe, 1983)	0.317 – 0.726	0.047 – 0.138
$k - \omega$ Simulation	0.457	0.185
Spalart-Allmaras Simulation ($C_{prod} = 1$)	0.388	0.077
Spalart-Allmaras Simulation ($C_{prod} = 0$)	0.354	0.0
Spalart-Allmaras Simulation ($C_{prod} = 2$)	0.828	0.904

Similar simulations were performed with C_{prod} ranging from 0 - 2 for different flow velocities to determine the Reynolds number independence of the C_{prod} parameter. The Reynolds numbers utilized for this analysis range from 3.5×10^6 to 8.5×10^6 , which is the upper boundary of the upper-transition flow regime to the trans-critical flow regime. The coefficient of drag and coefficient of lift (rms) from the simulation results for Reynolds numbers 3.5×10^6 and 8.5×10^6 are shown in Table 3.10. In addition, experimental data from experimental works with similar Reynolds numbers are likewise presented in Table 3.10. Based on these simulations, it can be seen that utilizing $C_{prod} = 1.0$ results in good agreement between the simulation and experimental data, hence demonstrating the Reynolds number independence of C_{prod} for such bluff body flows within this trans-critical flow regime for cylindrical flow configuration. For lower Reynolds number flows, it is suggested that addition validation studies be performed.

Table 3.10: The coefficient of drag and coefficient of lift (rms) from experimental data and simulation results for Reynolds numbers ranging from 3.5×10^6 to 8.5×10^6 .

	Reynolds Number, Re	Coefficient of Drag, C_D	Coefficient of Lift (RMS), C_{Lrms}
Upper transition flow regime			
Experimental Data (Jones et al., 1969)	3.49×10^6	0.557	0.0839
Spalart-Allmaras Simulation ($C_{prod} = 1$)	3.49×10^6	0.401	0.074
Spalart-Allmaras Simulation ($C_{prod} = 0$)	3.49×10^6	0.367	0.001
Spalart-Allmaras Simulation ($C_{prod} = 2$)	3.49×10^6	0.853	0.931
Higher trans-critical flow regime			
Experimental Data (James et al., 1980)	7.83×10^6	0.363	NA
Spalart-Allmaras Simulation ($C_{prod} = 1$)	8.27×10^6	0.369	0.073
Spalart-Allmaras Simulation ($C_{prod} = 0$)	8.27×10^6	3.39	0.0013
Spalart-Allmaras Simulation ($C_{prod} = 2$)	8.27×10^6	0.823	0.897

Three-Dimensional simulation

Moving forward, a three-dimensional simulation analysis of the Spalart-Allmaras model was performed for a bluff body flow based on the insight gained from the two-dimensional simulations. In particular, a configuration of an open-channel flow (top-Symmetry and bottom-wall boundary as described earlier) around a cylindrical monopile was analyzed. As demonstrated in the two-dimensional simulation, a rotation correction enables the suppression of turbulence production within the vortex core, resulting in the Spalart-Allmaras model predicting bluff body flow well.

Figure 3.9: Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k - \omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 5.2×10^6

The mean turbulence viscosity of the three-dimensional simulation ($Re = 5.2 \times 10^6$) for the Spalart-Allmaras model with varying C_{prod} as well as the $k - \omega$ (SST) simulation are plotted in Figure 3.9. Commencing the analysis with $C_{prod} = 0$ (rotation correction is turned off), similar observations as the two-dimensional simulations were observed, whereby the Spalart-Allmaras model over predicts the mean turbulence viscosity in the wake region of the flow. Subsequently increasing the C_{prod} gradually, the rotation correction provided the suppression required to lower the turbulence viscosity in the lee-wake region. However, it was noted that for the three-dimensional model, when $C_{prod}=1.0$, which is suitable for a two-dimensional bluff body flow, the resultant lee-wake mean turbulence viscosity was still significantly higher than that of the benchmarked $k - \omega$ (SST) simulation. Moving beyond $C_{prod}=1.0$, it was identified that good correlation between the Spalart-Allmaras model and the $k - \omega$ (SST) was achieved when $C_{prod}=3.5$ as shown in Figure 3.9.

It is interesting to note that for a three-dimensional flow, the C_{prod} of the rotation correction was higher as compared to the two-dimensional flow. This discrepancy can be attributed to the higher complexity of a three-dimensional flow which results in higher vorticity within the wake region of the flow. With reference to Equation 3.34, the higher vorticity implies that a larger suppressing term ($C_{prod} \min(0, |S_{ij}| - |\Omega_{ij}|)$) is required, which subsequently translates into a higher C_{prod} value for three-dimensional flows, which was found to be 3.5 in this work.

Figure 3.10: Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k - \omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 3.5×10^6

Figure 3.11: Mean turbulence viscosity for three-dimensional simulations along the line of symmetry for a Spalart-Allmaras model with no rotation correction ($C_{prod} = 0$), Spalart-Allmaras model with $C_{prod}=3.5$ and $k - \omega$ (SST) turbulence model. The Reynolds number for these simulations performed was 8.5×10^6

To further validate the proposed coefficient, $C_{prod}=3.5$, a Reynolds number independence study for the three-dimensional flow was conducted. The range of Reynolds numbers evaluated was similar to that performed for the two-dimensional flow, between Reynolds number 3.5×10^6 and 8.5×10^6 . The mean turbulence viscosity of the three-dimensional simulation for Reynolds number 3.5×10^6 and 8.5×10^6 are presented in Figure 3.10 and 3.11 respectively. In each figure, the Spalart-Allmaras model with $C_{prod} = 0$ and 3.5, as well as the $k - \omega$ (SST) simulation, are plotted.

As can be seen in both Figure 3.10 and 3.11, for a range of Reynolds number, when $C_{prod}=0$, the Spalart-Allmaras model over predicts the mean turbulence viscosity in the wake region of the flow. Hence, tuning the C_{prod} to 3.5 as proposed earlier, the rotation correction lowers the turbulence viscosity in the lee-wake region, resulting in a closer correlation with that obtained from the $k - \omega$ (SST) simulation. Such simulation results with varying Reynolds number provides additional validation of the applicability of $C_{prod} = 3.5$ for three-dimensional flows when adopting a Spalart-Allmaras turbulence model for cylindrical bluff body flows.

3.4.3 Flow Within a Scour Hole

In the earlier sections, the superior performance of the $k - \omega$ (SST) model have been demonstrated and likewise the ability to tune the Spalart-Allmaras model for cylindrical bluff body flows. To gain more insight of the performance of these models with regards to scour modeling, the $k - \omega$ (SST) model and the Spalart-Allmaras model with a $C_{prod}=3.5$ was applied to a more complex three-dimensional flow within a scour hole of a cylindrical monopile foundation. In this part of the work, experimental observations obtained by Melville and Raudkivi (1977) and the corresponding numerical work using a two-equation $k - \varepsilon$ (RNG) model performed by Richardson and Panchang (1998) were used to benchmark the $k - \omega$ (SST) and Spalart-Allmaras simulation results. As described earlier, three configurations of the scour hole (an initially flat bed, an intermediate scour hole and the equilibrium scour hole) were simulated. A cylinder diameter of 5.08 cm and a mean flow velocity of 0.25 m/s were utilized in these simulations as described earlier, similar to that of the experimental and numerical setup by Melville and Raudkivi (1977) and Richardson and Panchang (1998). Likewise, the boundary conditions and simulation domain

size utilized was identical to that in Table 3.2 and Figure 3.2 and 3.3 respectively, except the velocity applied at the inlet boundary was $U_x = 0.25m/s$ which was that used by Melville and Raudkivi (1977).

In Figure 3.12 and 3.13, the flow profile in front of the monopile, along the symmetrical plane of the cylinder, for the $k-\omega$ (SST) model, Spalart-Allmaras model with $C_{prod} = 3.5$ and that observed in experimental work (Melville and Raudkivi, 1977) are shown for an initially flatbed and intermediate scour hole configuration. As mentioned earlier, the simulations conditions were identical to the experimental setup by (Melville and Raudkivi, 1977). Based on the experimental observation by Melville and Raudkivi (1977), one of the key features of the flow within a scour hole is that of the horseshoe vortex at the base of the cylinder/monopile. This horseshoe vortex and the downward flow as the incoming stream reaches the cylinder surface can be clearly seen both in the $k-\omega$ (SST) and Spalart-Allmaras ($C_{prod} = 3.5$) simulations.

Figure 3.12: Flow vector for an initially flatbed and an intermediate scour hole using a $k-\omega$ (SST) model and the flow profile observed in experiments by Melville and Raudkivi (1977).

Flatbed
Intermediate Hole
	Spalart Allmaras ($C_{prod} = 3.5$) Simulation	Experiment (<i>Melville et al.</i>)

Figure 3.13: Flow vector for an initially flatbed and an intermediate scour hole using a Spalart-Allmaras model with $C_{prod} = 3.5$ and the flow profile observed in experiments by Melville and Raudkivi (1977).

While the simulation general flow profiles agree well with experimental plots, the flow in the experimental plot seems to have more significant eddy than the simulated models. This could be attributed to two reasons. Firstly, it should be noted that the vector arrows used by Melville and Raudkivi (1977) were of uniform length, hence resulting in a plot which has a seemingly more pronounced vortex. Secondly, earlier work by Richardson and Panchang (1998) utilizing a $k - \varepsilon$ (RNG) model did observe slight difference between the simulated vortex and experimental data by Melville and Raudkivi (1977) and attributed this to the minor differences in the scour hole geometries. Such minor variation in the scour hole geometries are likewise valid in the current work using the $k - \omega$ (SST) and Spalart-Allmaras model as the very detailed scour hole geometry used in the experimental work by Melville and Raudkivi (1977) is unknown. The vortex within an intermediate scour hole obtained by Richardson and Panchang (1998) is presented in Figure 3.14 which provides a gauge of the expected minor difference between the simulation results and experimental data. However, the overall flow profile of the horseshoe vortex obtained at the base of the cylinder/monopile by the $k - \omega$ (SST) and Spalart-Allmaras ($C_{prod} = 3.5$) simulations had good agreement with experimental data.

Figure 3.14: Flow profile for an intermediate scour hole using a $k - \varepsilon$ (RNG) model obtained by Richardson and Panchang (1998) and the flow profile observed in experiments by Melville and Raudkivi (1977).

Figure 3.15: Normalized velocity profile for the equilibrium scour hole configuration using the Spalart-Allmaras model with $C_{prod} = 3.5$, the $k - \omega$ (SST) model, the flow profile observed in experiments by Melville and Raudkivi (1977) and the numerical results using the $k - \varepsilon$ (RNG) model by Richardson and Panchang (1998)

For the flow velocity profile, the flow within an equilibrium scour hole configuration was analyzed. Figure 3.15 shows the resultant normalized velocity profile of the $k-\omega$ (SST) and Spalart-Allmaras ($C_{prod} = 3.5$) simulation together with the velocity profile obtained experimentally by Melville and Raudkivi (1977) and numerically by Richardson and Panchang (1998) using the $k-\varepsilon$ (RNG) model. The normalization of the velocity was obtained by dividing the velocity magnitude by the free stream velocity, which in this simulation was 0.25m/s.

Based on Figure 3.15, it can be seen that for both the $k-\omega$ (SST) and Spalart-Allmaras simulation, the normalized mean velocity decreases from approximately 0.9 (Figure 3.15(a,e)) in the incoming flow to 0.4 (Figure 3.15(b,f)) near the cylinder wall. Such a profile is similarly observed experimentally by Melville and Raudkivi (1977), as well as numerically by Richardson and Panchang (1998) using the $k-\varepsilon$ (RNG) turbulence models. Moving down into the scour hole, there is a clear velocity gradient from 0.6 (Figure 3.15(c,g)) approximately in the center of the scour hole to 0.1 (Figure 3.15(d,h)) at the downward sloping bed surface in the scour hole. Such a velocity profile gradient is likewise generally observed in experimental observations as well as in earlier numerical works. While it is noted that the detailed velocity profile along the downward sloping bed surface in the scour hole differs for each simulation model and experimental observation, this can be attributed to the high level of turbulence within that region which makes such analysis and measurement very challenging.

Based on the simulations performed, it is concluded that the $k-\omega$ (SST) and the Spalart-Allmaras model with a $C_{prod}=3.5$ is able to predict the flow within a scour hole with reasonable agreement with experimental observations. While the good correlation between the experimental data and that of the $k-\omega$ (SST) model was expected, the good performance of the Spalart-Allmaras model with a $C_{prod}=3.5$ was encouraging. Thus, the results of the Spalart-Allmaras model show promising capabilities of applying such a cost efficient turbulence models for applications in complex bluff body flows, despite being originally developed for aerodynamic flows. As such, moving forward with the development of a numerical scour model, both the $k-\omega$ (SST) and the Spalart-Allmaras model will be utilized. Doing so, it will provide better insight into the relative performs of both the one- and two- equation RANS turbulence models when applied to a scour model.

3.5 Conclusion

In this section, the one- and two- equation RANS turbulence model was evaluated to determine the best approach to capture the high Reynolds number flow around a cylindrical bluff body. This work was done to address knowledge gaps in the understanding of high Reynolds flow around bluff bodies and to determine the ideal turbulence model to adopt moving forward in this work on scour modeling.

For the two-equation models, it was shown that the $k - \omega$ (SST) model had a closer agreement to experimental values as compared to other RANS turbulence models. This good agreement of the $k - \omega$ (SST) model with experimental data is due to the ability of this turbulence model to predict the angle of separation well. Hence, this corresponds to modeling the width of the leeward vortex wake, the Strouhal number and the coefficient of drag similarly well. Advancing the analysis further, the effects of incorporating a curvature correction modification into the $k - \omega$ (SST) model as well as the ability of this turbulence model to capture the surface roughness for a high Reynolds number flow was studied. It was established that the curvature correction improved the coefficient of pressure profile while maintaining or slightly improving the predicted angle of separation, coefficient of drag and Strouhal number. However, the agreement of the coefficient of lift-rms with that of experimental value was worsened when the curvature correction was included. For the surface roughness analysis, the $k - \omega$ (SST) model was observed to be able to reasonably capture the flow characteristic evolution as the magnitude of the K_s/D ratio increases, especially so for the substantial variation of the C_p and C_D with increasing surface roughness. While there were minor discrepancies in the prediction of the angle of separation and Strouhal number in the simulation results, the general trend of these flow parameters to vary minimally as the K_s/D ratio increases agreed well with that observed experimentally by Shih et al. (1993).

For the one-equation model, it is proposed that a Spalart-Allmaras model with a rotation correction is capable of modeling bluff body flows around a cylinder. Starting with two-dimensional simulations around a cylindrical body, it is demonstrated that the rotation correction proposed by Dacles-Mariani et al. (1995) effectively suppresses the turbulence production within the vortex core, where vorticity exceeds the strain rate. However, the $C_{prod} = 2.0$ suggested by Dacles-Mariani et al. (1995) over-suppressed the turbulence and had to be re-tuned for bluff body flows.

It was found that a $C_{prod} = 1.0$ was suitable for cylindrical bluff bodies and this coefficient was Reynolds number independent within the trans-critical regime. For three-dimensional flows, the more complex three-dimensionality of the flow yields higher vorticity, and hence a higher C_{prod} was required to suppress the turbulence within regions where the vorticity exceeds the strain rate. Hence, it is proposed that $C_{prod} = 3.5$ be used for a three-dimensional flow. It should be noted that the proposed rotation correction coefficients have been validated for cylindrical structural configurations, which is the focus of this work. For other structural configuration (e.g. square piles), additional validation studies are suggested to be performed before being applied for engineering applications.

For the flow within a scour hole around a cylindrical monopile, the $k - \omega$ (SST) and the Spalart-Allmaras simulation was benchmarked with experimental observations (Melville and Raudkivi, 1977) and other numerical work (Richardson and Panchang, 1998) using the $k - \varepsilon$ (RNG) model. The key features of the flow, such as the horseshoe vortex, were successfully reproduced in both turbulence models. In addition, the mean velocity profile predicted from the $k - \omega$ (SST) and Spalart-Allmaras model correlated well with experimental observation. It was particularly encouraging to note that the cost-efficient Spalart-Allmaras model showed promising results in modeling such complex bluff-body flows within a scour hole.

For scour modeling, it is critical to obtaining accurate turbulent flow profiles around the structure. This is because these turbulence flows are the key driving force of the sediment transportation and ultimately scour. In this chapter, addressing knowledge gaps in the understanding of such turbulence modeling, as described above, will provide the confidence in applying the appropriate turbulence model in the scour models, which was developed in the subsequent chapter. Hence, moving forward, the $k - \omega$ (SST) will be applied to the numerical scour model in the later works. In addition, parallel simulations utilizing the Spalart-Allmaras model with a $C_{prod} = 3.5$ will be likewise performed to gain better insight into its performance for such complex simulations.

Chapter 4

Development of a Full Numerical Scour Model

4.1 Introduction

In chapter 3 of the dissertation, it was identified that the ideal RANS turbulence model for such a bluff body flow is the $k - \omega$ (SST) turbulence model. In addition, the potential of using the computationally efficient Spalart-Allmaras model for such complex flows was likewise demonstrated. Moving forward from this understanding, in this chapter, the focus was to develop a full numerical scour model for a monopile foundation. Both the $k - \omega$ (SST) and the Spalart-Allmaras turbulence model was utilized in this scour model development. There are a couple of potential computational fluid dynamics codes whereby such developments can be performed in. In this work, three key criteria were used to select which software platform was most ideal for the development of the scour model solver. These criteria are:

- The current capabilities and modulus available in the software which are relevant to scour modeling (e.g. turbulence models, dynamic meshing).
- The ability to couple a user-define solver with the source code of the software.
- The financial cost of the software.

Based on these criteria, in this work, the numerical scour model developed was built in an open source code called OpenFOAM. The OpenFOAM code is an open source C++ library toolbox which utilizes a Finite Volume Method (FVM) approach to solve the flow equations. The C++ library toolbox consists of various features including turbulence model solvers and dynamic meshing capabilities. These features make such a code suitable for the development of a scour model. Furthermore, being an open source code, this allows for greater access to the source files and hence flexibility when developing and coupling the scour solver to other solvers in OpenFOAM. Finally, being an open source code, there is no licensing cost for utilizing this software and this makes such a code more accessible as compared to expensive commercial codes. Details of the scour model and computational code developed will be described in greater detail in the subsequent sections.

As discussed in the literature review presented in chapter 2, there has been some earlier work done in developing a full numerical scour model for offshore structures. In these earlier works, various approaches have been taken to model the full scour process, which includes a Finite Volume Method (FVM) (Roulund et al., 2005) approach, an Eulerian-Eulerian method (Hosseini Kazeminezhad et al., 2011) approach and a Lattice-Boltzmann (LB) approach (Alam and Cheng, 2010). Among these approaches, the FVM approach has been the most popular method adopted and have been applied to model scour around various structural configuration including the monopile foundation (Roulund et al., 2005), pipeline (Fuhrman et al., 2014) and tripod structure (Stahlmann, 2013). In this work, the FVM approach was likewise adopted in the development of the numerical scour model in OpenFOAM.

The earlier works on numerical scour modeling using a FVM approach have shown optimistic simulation results. Both the equilibrium scour depth (Roulund et al., 2005) and backfilling process (Fuhrman et al., 2014) have been shown to agree with experimental observations with minor variation. While these works have provided some insights into numerical scour modeling, the experience available of such modeling is still very limited with many questions unanswered as earlier described in chapter 2. In addition, the development of a numerical scour model isn't trivial and in most of these earlier works, the scour codes were developed in-house and are currently unavailable for public use.

Hence in this chapter, a numerical scour model is developed with the objective to obtain the following:

- Model the details of the scour initiation mechanism.
- Evaluate the scour depth time development of the numerical scour model.
- Evaluate the performance of the numerical scour model utilizing different turbulence model.
- Evaluate the ability of utilizing a one-equation Spalart-Allmaras model for such scour simulations.
- Identify challenges in developing and utilizing a numerical scour model

By achieving these objectives, the advantages and limitations of such a scour model can be identified. This will provide greater confidence and understanding in performing such numerical scour modeling for offshore structures.

4.2 Theory

For a full numerical scour model, various physical concepts (turbulence modeling, bed parameter calculations, sediment transport modeling) work in sync with each other to model the full scouring process. In this section, the theoretical model of each concept is described in detail.

4.2.1 Scour Model Overview

A flow diagram of the approach taken to model the scour process in this work is presented in Figure 4.1. After initializing the simulation model, the flow profile and turbulence are determined using the RANS turbulence model, in particular the $k-\omega$ (SST) and the Spalart-Allmaras model. Based on the derived flow parameters, the bed parameters (Shields parameter, bed shear stress, etc.) are calculated. With the flow and bed parameters identified, the sediment transport equation is solved to determine the net sediment transportation rate at the different regions of the model. In this work, only bed load sediments are considered and the bed load transport equation utilized was that proposed by Engelund and Fredsøe (1976) as would be described later. Based on the net sediment transport within a region and using the Exner equation, the bed morphology is evolved using a dynamic mesh. Once the new bed morphology is obtained, physical constraints (i.e. bed angle) on the model are checked. If the angle of repose has been exceeded, a sand-sliding model will smoothen out the bed morphology up to a point when the bed angle reaches the angle of repose. This process is repeated up to the point where the equilibrium scour depth is reached and the simulation terminates.

Figure 4.1: Flow diagram of the numerical scour model developed in this work.

Based on the flow diagram described above, a solver was developed in OpenFOAM for this work termed as `interDymScourFoam`. The solver was built on top of a pre-existing solver called `interDyMFoam`, which is a multi-phase Volume-Of-Fraction (VOF) solver with dynamic mesh capabilities. The modules that were developed and included into the `interDyMFoam` source code to obtain the final `interDymScourFoam` is as follows:

- Bed Parameter Module
 - Shields parameter calculation
 - Critical shear stress calculation
 - Bed angle calculation
- Sediment Transport Module
 - Solve bed load equation
- Bed Morphological Module
 - Solve Exner equation
- Sand-Sliding Module
 - Mass conservative sand-sliding

The overview of the `interDymScourFoam` solver developed and the interactions between the different modules is presented in Figure 4.2.

Figure 4.2: Overview of the `interDymScourFoam` developed in OpenFOAM.

4.2.2 Turbulence Model

As mentioned earlier, a RANS turbulence modeling approach was adopted in this work to reduce the computational requirements of the model. As demonstrated in chapter 3, the $k - \omega$ (SST) model was shown to be the superior RANS turbulence model for bluff body flows when compared to other one- and two- equation turbulence models. In addition, the Spalart-Allmaras model likewise showed optimistic results for such bluff body flows when a rotational correction with $C_{prod}=3.5$ was utilized. Hence, in this work, both the $k - \omega$ (SST) model and the Spalart-Allmaras model with a $C_{prod}=3.5$ was utilized to model the turbulent flow field.

The $k - \omega$ (SST) model is a two-equation turbulence model which capitalizes on the accuracy of the $k - \omega$ model within the near-wall region and the $k - \varepsilon$ model in the far-field region. This turbulence model utilizes the transport equations of the turbulence kinetic energy (k) and specific dissipation rate (ω) as presented in chapter 3. The detailed description of the $k - \omega$ (SST) model is described fully in chapter 3.

For the Spalart-Allmaras model, this turbulence model is a one-equation RANS turbulence model which utilizes a kinematic eddy viscosity ($\tilde{\nu}$) transport equation as described in chapter 3. The Spalart-Allmaras model constants used in this work were identical to that originally proposed by Spalart and Allmaras (1994). However, for the rotational correction term (C_{prod}), a value of $C_{prod}=3.5$ was utilized for the three-dimensional bluff body flow as proposed earlier. The detailed description of the Spalart-Allmaras model is likewise presented in chapter 3.

4.2.3 Bed Parameters Module

In the numerical scour model, various bed parameters have to be calculated. These parameters include the Shields parameter, critical Shields parameter, the bed angle correction factor and the fluid velocity at the bed. These parameters are determined during the initiation of the simulation and at each time-step of the simulation.

During the initialization of the simulation, the critical Shields parameter is determined based on the equation proposed by Soulsby and Whitehouse (1997),

$$\theta_{cr0} = \frac{0.24}{d_*} + 0.055 [1 - \exp(-0.02d_*)] \quad (4.1)$$

where θ_{cr0} is the critical Shields parameter and d_* is the dimensionless particle diameter given as,

$$d_* = \left(\frac{(s-1)g}{v^2} \right)^{1/3} d \quad (4.2)$$

where v is the kinematic viscosity, d is the sediment size and s is the specific gravity of the sediment. The critical Shields parameter obtained in Equation 4.1 is that for a flatbed condition. To account for the bed angle evolution during the scour process, a bed angle correction factor proposed by Roulund et al. (2005) was utilized given as,

$$\theta_{cr} = \theta_{cr0} \left(\cos\beta \sqrt{1 - \frac{\sin^2\alpha \tan^2\beta}{\mu_s^2}} - \frac{\cos\alpha \sin\beta}{\mu_s} \right) \quad (4.3)$$

where μ_s is the static friction coefficient, α is the angle difference between the flow direction and the steepest bed slope and β is the angle of the steepest bed slope.

Based on the friction velocity (U_f) obtained from the flow simulation, the Shields parameter is obtained as follow,

$$\theta = \frac{U_f^2}{(s-1)gd} \quad (4.4)$$

where θ is the Shields parameter at the location.

Finally, the fluid velocity (U) at the bed is obtained from the friction velocity given as,

$$U = aU_f \quad (4.5)$$

where a is an empirical constant. In this work, $a = 10$ was utilized as proposed by Engelund and Fredsøe (1976), whereby this constant was derived empirically by fitting a bed load particle velocity expression with experimental data.

4.2.4 Sediment Transport Module

To model the sediment transportation, an empirical bed load equation proposed by Engelund and Fredsøe (1976) was utilized given as,

$$q_b = \frac{1}{6}\pi d^3 \frac{PEF}{d^2} U_b \quad (4.6)$$

where q_b is the rate of bed load transport, d is the sediment size, PEF is the probability of sediment motion at the surface of the bed and U_b is the mean transport velocity of the particle.

Engelund and Fredsøe (1976) proposed a semi-empirical equation to determine the PEF given as,

$$PEF = \left[1 + \left(\frac{\frac{1}{6}\pi\mu_d}{\theta - \theta_{cr}} \right)^4 \right]^{-1/4} \quad (4.7)$$

where μ_d is the dynamic friction coefficient.

For the mean transport velocity (U_b), it is determined based on a force balance analysis on the sediment particle. From the force balance analysis, four equations were derived given as,

$$F_D \cos \Psi_1 + W \sin \beta \cos (\alpha - \Psi) - (W \cos \beta) \mu_d = 0 \quad (4.8)$$

$$F_D \sin \Psi_1 - W \sin \beta \sin (\alpha - \Psi) = 0 \quad (4.9)$$

$$U_r \sin \Psi_1 - aU_f \sin \Psi = 0 \quad (4.10)$$

$$U_r \cos \Psi_1 - aU_f \cos \Psi + U_b = 0 \quad (4.11)$$

where F_D is the drag, W is the force due to gravity, U_r is the velocity of the flow relative to the sediment, Ψ is the angle difference between the flow velocity and U_b and the Ψ_1 is the angle difference between the drag force and U_b . The four unknowns in the above four equations (Equation 4.8-4.11) are U_b , U_r , Ψ and Ψ_1 . To solve for the 4 unknowns, a Newton-Raphson method is employed to solve Equation 4.8-4.11 to finally obtain U_b . Hence, by obtaining the PEF using Equation 4.7 and U_b by solving Equation 4.8-4.11, the rate of bed load transportation can be obtained using Equation 4.6 proposed by Engelund and Fredsøe (1976).

4.2.5 Bed Morphological Module

The sediment transport module described earlier is to determine the transport rate of sediments only, without considering the resultant changes in the bed morphology. In the bed morphological module, the transport rate obtained in the earlier module is translated to bed morphological changes. The sediment transport and bed morphological module was separated in this work to allow for more flexible and easier expansion of the solver in future developments.

For the current work, from the bed load transport rate, the bed morphological changes are calculated using a mass conservative Exner equation given as,

$$\frac{\partial h_{bed}}{\partial t} = -\frac{1}{1-n} \nabla \cdot q_s \quad (4.12)$$

where h_{bed} is the bed height, n is the porosity and q_s is the sediment flux, which in this model is limited to the bed load transportation (q_b).

4.2.6 Sand-Sliding Module

When the bed angle exceeds the angle of repose (ϕ_{repose}), sand-sliding occurs. To replicate this physical phenomenon in the scour model, a sand-sliding model was included. The angle of repose for the sediment was calculated using the empirical formula proposed by Lim and Cheng (1998) given as,

$$\tan(\phi_{repose}) = 0.457 \left(\frac{\sqrt{gd_{50}}d_{50}}{v} \right)^{0.05} \left(\frac{\rho_s - \rho}{\rho_s} \right)^{0.106} S_p^{-0.8} \sigma_g^{0.12} \quad (4.13)$$

where d_{50} is the mean particle diameter, ρ_s is the sediment density, S_p is the shape factor and σ_g is the sediment geometric standard deviation. As a uniform sediment size was utilized in this work, $d_{50} = d$.

When the bed angle of the bed boundary cell exceeds the angle of repose, as calculated using Equation 4.13, the sand-sliding model will be activated. The maximum and minimum node height for the cell is identified and the difference between the maximum and minimum node height (Δh_{bed}) is calculated. For the node with the maximum height, it's height is reduced by 25% of Δh_{bed} and correspondingly for the node with the minimum height, it's height is increased by 25% of Δh_{bed} . This simultaneous increase and reduction of node heights make this sand-sliding model mass conservative. The sand-sliding evolution is performed in every cell where the angle of repose condition is not satisfied. Once the calculation is performed for all the cells, the new bed angle is re-calculated to determine if all cells have fulfilled the angle of repose condition. If there are locations along the bed boundary where the bed angle still exceeds the angle of repose, the sand-sliding model calculation is repeated. Figure 4.3 shows the process flow of the sand-sliding model.

Figure 4.3: The process flow of the sand-sliding model implemented in the inter-DymScourFoam solver.

4.2.7 Code Algorithm

The theoretical concepts and process flow of the scour model presented earlier were implemented into an OpenFOAM solver termed the interDymScourFoam. The algorithm of the interDymScourFoam solver is shown in Figure 4.4.

Figure 4.4: The algorithm of the interDymScourFoam solver developed.

4.3 Model Setup

The simulation model was setup similar to that of experimental work done by Melville and Chiew (1999). In particular, run number N1, N7 and N9 from Melville and Chiew (1999) experiments were selected as the sediment size, flow velocity and water depth were identical for these three runs. The key variable in these experimental runs was the pile diameter, which resulted in a range of different Reynolds number.

Mesh geometry

A hexahedral block mesh with the geometrical dimension similar to that in chapter 3 was utilized in this section. In the x-y plane, a geometrical dimension of 25D by 20D was used. Likewise, the location of the cylinder was positioned 10D from the upstream inlet boundary, 15D from the downstream outflow boundary and 10D from the side boundaries. In the z-direction, a height of 200mm was used, which is identical to the experimental setup by Melville and Chiew (1999).

It should be noted that for the current scour simulation, a mesh sensitivity study was not performed and the mesh was set-up based on best practices. This is because in a scour simulation, the continuous deformation of the bed boundary during the scour process results in a constant evolution of the mesh which makes such sensitivity studies non-trivial. Furthermore, the high computational cost of each simulation results in a significantly high amount of resources required for such a sensitivity study. Hence, with these limitations, the mesh was set-up based on best practices. Such an approach is similar to that adopted by earlier scour related works(Roulund et al., 2005). This is an area that could be addressed in future works.

Boundary conditions

The pile diameters and boundary conditions imposed on the simulation model are given in Table 4.1.

Table 4.1: Simulation model pile diameter and boundary conditions utilized.

Simulation Name	Pile Diameter, D (mm)	Boundary Condition			
		Inlet	Outflow	Top & Side	Bed & Cyl.
N1	70	Velocity	Outflow	Sym	Wall
N7	50	Velocity	Outflow	Sym	Wall
N9	38	Velocity	Outflow	Sym	Wall

Model parameters

The flow and sediment parameters utilized in the simulation models for all 3 cases (N1, N7 and N9) are identical to that of the experimental setup by Melville and Chiew (1999). The key difference between the 3 cases is the pile diameter and corresponding Reynolds number. These parameters are shown in Table 4.2 as follows.

Table 4.2: Input variables utilized in the simulation models similar to that of experimental setup by Melville and Chiew (1999)

Flow Velocity	$0.231m/s$
Sediment Size	$0.96mm$
Sediment Density	$2650kg/m^3$
Fluid Density	$1000kg/m^3$
Porosity	0.40
Dynamics friction coefficient	0.51
Static friction coefficient	0.63

Similar to the simulations performed in chapter 3, a uniform velocity profile was applied at the velocity inlet boundary and the upstream portion of the domain was utilized to develop the flow velocity profile. The upstream length of the domain was $10D$, identical to that utilized in chapter 3, which was found to be sufficient for the flow velocity profile to be formed.

For the time-step of the simulation, an adaptive time-step was utilized which was governed based on the maximum courant number of the simulation. For the current simulations, an initial time-step of 0.02 sec was applied and a maximum courant number of 1 was implemented into the simulations. The time-step was subsequently adjusted based on the courant number.

4.4 Results and Discussion

The results of the simulation N1, N7 and N9 are presented in this section. Particular focus in this work was given to the initiation mechanism of the scour process, which is challenging to evaluate in experimental setup due to the short time period it occurs. It is in modeling and understanding such experimentally challenging initiation mechanism where the advantage of numerical scour modeling is fully realized. When evaluating the simulation results, attention was placed on investigating the numerical scour model ability to estimate the scour depth-time development profile and in replicating key scour mechanism processes. In the simulation models, the time development period (t_d) was limited to $t_d = 200\text{sec}$, which was found from the simulation results to be sufficient for scour initiation (the formation of a shallow scour hole) to occur as will be discussed later.

To evaluate the scour mechanism/process, experimental observation described by Breusers and Raudkivi (1991) was utilized. In addition, to analyze the time-dependent scour hole development, an equation proposed by Sumer et al. (1992a) was utilized given as,

$$S_t = S_0 \left(1 - \exp \left(-\frac{t}{T} \right) \right) \quad (4.14)$$

where t is the current time-step, S_0 is the equilibrium scour depth, S_t is the scour depth at the time-step t . T is the time scale of the scour process, whereby the scour depth - time slope at $t = 0$ crosses the equilibrium scour depth. T can be determined from the normalized time scale given as,

$$T = \frac{D^2}{(g(s-1)d_{50}^3)^{1/2}} T^* \quad (4.15)$$

$$T^* = \frac{1}{2000} \frac{\delta}{D} \theta^{-2.2} \quad (4.16)$$

where θ is the Shields parameter and δ/D is the boundary-layer to pile diameter ratio.

The simulation results obtained were evaluated in three parts. Firstly, the scour simulation using the $k - \omega$ (SST) turbulence model was first evaluated. This is to evaluate the scour model developed (interDymScourFoam) in its ability to predict the scour process/mechanism and the scour depth-time behavior of the simulation model. By using a more commonly adopted two-equation turbulence model for bluff body flow, it reduces the uncertainty due to the selection of the turbulence model. Subsequently, a Spalart-Allmaras turbulence model was applied to the interDymScourFoam to evaluate the performance of using such a one-equation turbulence model in simulating the complex scour process. Finally, to evaluate the developed solver further, a live-bed scour simulation was performed utilizing the interDymScourFoam.

4.4.1 Scour Model with $k - \omega$ (SST) Turbulence Model

The numerical scour model utilizing the $k - \omega$ (SST) turbulence model was simulated for 200 sec. Employing a more commonly adopted $k - \omega$ (SST) model initially reduces the uncertainty in the turbulence modeling performance for such bluff body flow as demonstrated in chapter 3. This part of the work provides greater insight into the ability of the scour model developed to predict the scour depth-time evolution and scour initiation mechanism.

4.4.1.1 Scour Depth-Time Development($k - \omega$ (SST) Model)

The scour depth-time development for each simulation run together with that calculated from empirical equation (Sumer et al., 1992a) is plotted in Figure 4.5 - 4.7. The plots show good correlation between the simulated development of the scour hole around a pile and that predicted by empirical equation (Sumer et al., 1992a). It should be noted that the simulated plots are the instantaneous maximum scour depth around the pile and fluctuation about the empirical equation plot/theoretical line is expected. Furthermore, as the current work focuses on the scour initiation process only, the time period of the simulation is relatively short (200sec) compared to the time period to achieve equilibrium scour depth (approximately 1-2 hrs). This short time period results in the fluctuation of the simulated scour depth to be more pronounce than its actually is.

Using simulation N1 as an example, while the simulated scour depth seem to diverge from the theoretical value from $t = 40 \text{ sec} - 100 \text{ sec}$, this is primarily due to portions of the pile with a slight increase in scour rates. As time progresses, instantaneous accumulation of sediment along the pile diameter occurs and gets progressively transported downstream before separating from the pile and getting deposited at the wake region of the pile. Hence during this period, the scour rate decreases slightly, resulting in the scour depth converging back towards the theoretical line (from $t=120\text{sec}$ onwards in Simulation N1). Such a behavior is to be expected until the scour depth reaches it equilibrium depth. With increasing simulation time periods, the fluctuation about the empirical equation plot will be less pronounce as the scour hole reaches it's equilibrium scour depth.

Figure 4.5: The scour depth-time evolution of simulation N1 from $t = 1 - 200\text{sec}$

Figure 4.6: The scour depth-time evolution of simulation N7 from $t = 1 - 200sec$

Figure 4.7: The scour depth-time evolution of simulation N9 from $t = 1 - 200sec$

Evaluating the scour depth at $t=200$ sec, Table 4.3 shows the simulated scour depth and that estimated from empirical equation (Sumer et al., 1992a). The deviation between the simulated and empirically calculated scour depth was no more than 20%, which is within experimental variation.

Likewise, the average scour rate obtained from the simulation results at $t=200$ sec and that estimated from empirical equation are shown in Table 4.4. As seen from Table 4.4, there is a reasonable agreement between the simulated scour rate and that estimated from empirical equation which further validates the physical correlation of the simulation performed.

Table 4.3: Scour depth at $t = 200$ sec predicted by the simulation model and empirical equation proposed by Sumer et al. (1992a).

Simulation Run	Simulated Scour Depth	Empirical Scour Depth	Deviation
N1	0.0202	0.0167	17.3%
N7	0.0134	0.0144	7.5%
N9	0.0128	0.0136	6.3%

Table 4.4: Average scour rate at $t = 200$ sec predicted by the simulation model and empirical equation.

Simulation Run	Simulated Scour Rate	Empirical Scour Rate
N1	6.06 mm/s	5.01 mm/s
N7	4.02 mm/s	4.29 mm/s
N9	3.84 mm/s	4.08 mm/s

4.4.1.2 Scour Initiation Mechanism($k - \omega$ (SST) Model)

The ability of the numerical model to replicate the scour mechanism is likewise critical in evaluating the performance of the scour model. The scour initiation process is an ideal mechanism to evaluate the scour model as this process has various key features (eg. initiation point, scour development direction) which can be identified.

Figure 4.8: The scour simulation using the $k - \omega$ (SST) turbulence model for case N7 from $t = 0 - 200\text{sec}$, whereby the contour plot indicates the scour depth.

The simulation model for case N7 from $t = 0 - 200$ sec is presented in Figure 4.8. From Figure 4.8, starting from a flatbed condition, the horseshoe vortex and flow constriction results in an increased stress amplification around the monopile. This results in scour initiating at the upstream sides of the pile. A trench-like pit surrounding the upstream side of the pile is formed by $t = 75$ sec. The upstream sediment is subsequently transported around the monopile and is deposited downstream where the vortexes start to dissipate. This results in the formation of small sediment pile downstream as seen at $t = 75$ sec. As time progresses, the scour depth upstream of the pile deepens and the trench-like pit elongates downstream along the pile diameter. This trench fully encompasses the diameter of the pile as seen at $t = 200$ sec and a complete shallow scour hole is formed. Downstream of the pile, sediment deposition is accumulated over time (as seen in the increasing high of the pile from $t = 75$ sec to $t = 200$ sec). This scour initiation mechanism described from the simulation model is observed in numerous experimental observations (Breusers and Raudkivi, 1991; Sumer and Fredsøe, 2002; Melville and Coleman, 2000) which validate the ability and performance of the numerical scour model.

An interesting observation of the numerical scour model developed was its ability to replicate fine details of the scour process/mechanism in the simulation. In general, researchers (Sumer and Fredsøe, 2002; Melville and Coleman, 2000) describe the scour initiation as occurring upstream of the pile which subsequently propagates downstream to ultimately form a shallow scour hole which deepens over time. This description however omits an initiation mechanism which occurs rapidly in the first few moments of the scour process. As reported by Breusers and Raudkivi (1991) (Page 64), scour initiates at the sides of the pile where the horseshoe vortex and the flow constriction results in high-stress amplification at the bed. This initial scour hole (at the upstream sides of the pile) move upstream till they meet at the center line of the pile before moving downstream as described in the earlier paragraph. This initial behaviour of the scour mechanism occurs rapidly and is typically omitted.

Figure 4.9: The scour initiation mechanism of simulation N7 using the $k - \omega$ (SST) turbulence model.

From the simulations performed, it was interesting to observe that this initial scour hole development and propagation behavior is captured well as seen in Figure 4.9 for case N7. From Figure 4.9, the scour initiates at approximately 45 degrees to the side of the pile from $t = 0$ to $t = 30$ sec while the toe of the pile remains relatively flat. The 2 small scour pits develop over time and moves upstream to finally meet at the center line of the pile, forming a trench-like scour pit at the upstream portion of the pile at $t = 70$ sec. Once scour initiates upstream of the pile, it propagates downstream to form a shallow scour hole as described in the earlier paragraph.

Hence, the capability of the numerical scour model to reproduce such fine details of the scour initiation mechanism enhances the validation and confidence of the scour model developed. As such, the potential opportunity to shed additional light into the mechanism of the scour process (especially for complex structures) utilizing a numerical scour model is evident.

4.4.1.3 Scour Equilibrium Depth Projection($k - \omega$ (SST) Model)

A challenge for scour modeling in general, and likewise in this work, is that the computational requirement for each simulation is significantly high. While it will be ideal that the equilibrium scour depth be achieved, there were challenges in the parallelization of the interDymScourFoam code development. This made the wall time of each simulation to be a significant limitation. Based on the simulation results presented for the scour initiation process, there is evidence that there is a good correlation between the numerical model prediction and that observed from experiments . Using the current simulation results and its variance with theoretical equation (Sumer et al., 1992a), projections of the scour model equilibrium depths are made. The average variance between the empirical equation and simulated results from $t = 0 - 200$ sec are given in Table 4.5.

Table 4.5: The average variance between the simulated results and empirical equation from t= 0 - 200 sec.

Simulation Run	Average Variance (mm)
N1	4.19
N7	1.13
N9	-0.63

Based on the average variance of the current simulation, the projected simulation results as the model approaches the equilibrium scour depth together with that obtained from the theoretical model (Sumer et al., 1992a) are shown in Figure 4.10 . The current low variance between the simulation model and empirical model results in an optimistic projection of the equilibrium scour depth.

Figure 4.10: The projection of the simulation model scour depth as it reaches its equilibrium depth.

4.4.2 Scour model with Spalart-Allmaras Turbulence Model

In the previous section, a $k-\omega$ (SST) turbulence model was utilized to model the flow field. Such two-equation turbulence models have typically been the favoured turbulence modeling approach adopted by other numerical scour model works (Roulund et al., 2005). Likewise in chapter 3, it was shown that the $k-\omega$ (SST) model was superior over other one- and two- equation RANS turbulence models for bluff body flows. While the $k-\omega$ (SST) model predicts the turbulence well, the computational time of the resultant numerical scour model is still significantly high. Hence, in an effort to reduce the computational requirement further, a one-equation Spalart-Allmaras turbulence model was applied to the interDymScourFoam solver developed. As demonstrated in chapter 3, the Spalart-Allmaras turbulence model with a rotation correction of $C_{prod} = 3.5$ for three-dimensional flows predicts the flow around a bluff body well. Hence, utilizing this modification, the Spalart-Allmaras model was applied to the numerical scour model developed to evaluate the ability of utilizing such a low-cost turbulence model for such complex scour simulations.

4.4.2.1 Scour Depth-Time Development(Spalart-Allmaras Model)

The scour simulation utilizing the Spalart-Allmaras turbulence model was applied to cases N1, N7 and N9. The scour depth - time evolution for each simulation run together with the simulation results from the scour simulation using the $k-\omega$ (SST) turbulence model and that determined from Sumer et al. (1992a) equation is plotted in Figure 4.11 - 4.13 as shown below.

Figure 4.11: The scour depth-time evolution of simulation N1 utilizing the Spalart-Allmaras model from $t = 1 - 200sec$

Figure 4.12: The scour depth-time evolution of simulation N7 utilizing the Spalart-Allmaras model from $t = 1 - 200sec$

Figure 4.13: The scour depth-time evolution of simulation N9 utilizing the Spalart-Allmaras model from $t = 1 - 200sec$

From Figure 4.11 - 4.13, the scour depth-time varying plots of the scour simulation using the Spalart-Allmaras model showed good correlation with that predicted by empirical equation(Sumer et al., 1992a). This prediction utilizing the Spalart-Allmaras model is comparable to that obtained from the $k - \omega$ (SST) turbulence model, showing approximately similar average scour rates between the two simulation models. The oscillatory fluctuation in the Spalart-Allmaras simulation about the empirical model plot/theoretical line is likewise attributed to the instantaneous scour depth behavior as described earlier in the $k - \omega$ (SST) simulation results. While both turbulence models have comparable results, given that the simulation model has not reached its equilibrium scour depth, it is difficult to validate the superior nature of one turbulence model over the other. However, the current simulation results show optimistic prospects of using such a cost efficient one-equation Spalart-Allmaras turbulence model for complex scour simulations.

Table 4.6: Scour depth at $t = 200$ sec predicted by the simulation utilizing a Spalart-Allmaras model, $k - \omega$ (SST) model and empirical equation (Sumer et al., 1992a).

Simulation Run	Simulated Scour Depth, Spalart-Allmaras	Simulated Scour Depth, $k - \omega$ (SST)	Empirical Scour Depth
N1	0.0146	0.0202	0.0167
N7	0.0112	0.0134	0.0144
N9	0.0125	0.0128	0.0136

Based on the simulation results at $t = 200$ sec, the scour depth from the simulation utilizing the Spalart-Allmaras model, $k - \omega$ (SST) model and that calculated from empirical equation (Sumer et al., 1992a) are presented in Table 4.6. The scour depth predicted from the simulation utilizing the Spalart-Allmaras model were within 25% variance from that calculated by empirical equation (Sumer et al., 1992a). This variance is comparable to the $k - \omega$ (SST) simulation performed earlier and further highlights the potential of the Spalart-Allmaras model.

4.4.2.2 Scour Initiation Mechanism (Spalart-Allmaras Model)

Similar to the evaluation of the simulation utilizing the $k - \omega$ (SST) model, Figure 4.14 shows the results of the simulation using the Spalart-Allmaras turbulence model for case N7 from $t = 0 - 200$ sec. It can be seen that the scour mechanism obtained was similar to the $k - \omega$ (SST) simulation results and that observed in experimental observations (Breusers and Raudkivi, 1991; Sumer and Fredsøe, 2002; Melville and Coleman, 2000). Starting from a flat-bed condition, the shear stress amplification around the monopile results in scour initiating at the upstream location. This forms a trench-like pit in front of the pile by $t = 75$ sec. The eroded sediments is subsequently transported around the pile and deposited downstream in small sediment piles. Over time, this scour trench deepens and elongates around the pile to finally form a shallow scour hole is seen at $t = 200$ sec. This correlation of the scour mechanism between the Spalart-Allmaras model results and that observed in experiments further validates the suitability of using such a low-cost turbulence model for numerical scour simulations.

Figure 4.14: The scour simulation using the Spalart-Allmaras turbulence model for case N7 from $t = 0 - 200$ sec, whereby the contour plot indicates the scour depth.

The fine details of scour initiation mechanism modeled by the simulation utilizing the $k-\omega$ (SST) model, whereby the scour initiates at the sides of the pile and moves upstream to form a trench-like scour pit upstream, was likewise captured well in this simulation model utilizing the Spalart-Allmaras turbulence model as seen in Figure 4.15. From Figure 4.15, it can be seen that from a flatbed condition ($t = 0$ sec), small scour pits initiates at the sides of the pile, with the toe of the pile remaining relatively flat ($t = 1 - 30$ sec). This scour pits start moving upstream to ultimately meet at the center line of the pile to form a trench-like pit upstream ($t = 30 - 70$ sec) before finally moving downstream as described in the earlier paragraph. This scour mechanism is identical to that observed in the $k-\omega$ (SST) model simulation and that describe by Breusers and Raudkivi (1991) from experimental observations. Such ability of the Spalart-Allmaras model to replicate the scour initiation mechanism to this level of detail validates the performance and potential of this one-equation turbulence model for complex scour modeling.

Figure 4.15: The scour initiation mechanism of simulation N7 using the Spalart-Allmaras turbulence model.

4.4.2.3 Scour Properties Benchmarking

A key driving force in a scour model is the bed shear stress amplification and corresponding local Shields parameter. The accuracy of the calculated Shields parameter is reliant on the performance and ability of the turbulence model adopted and subsequently in the scour model. Hence, the predicted Shields parameter profile obtained from the simulation utilizing the Spalart-Allmaras turbulence model was benchmarked against that obtained from the $k - \omega$ (SST) model to evaluate the performance of this model.

Figure 4.16: The normalize Shields parameter of the simulation utilizing the $k - \omega$ (SST) and the Spalart-Allmaras turbulence model at $t = 200$ sec.

The normalized Shields parameter of the simulation utilizing the $k - \omega$ (SST) model and the Spalart-Allmaras model at $t = 200$ sec is shown in Figure 4.16. The normalization was obtained by dividing the local Shields parameter at each region by the critical Shields parameter determined from the equation proposed by Soulsby and Whitehouse (1997) described earlier in Equation 4.1. It must be noted that the normalized Shields parameter scale in Figure 4.16 was limited to 5 for visualization purposes. Based on Figure 4.16, the general profile of the normalized Shields parameter obtained from both the $k - \omega$ (SST) model and the Spalart-Allmaras model correlated well with each other. At the upstream toe of the pile (along the scour hole downward slope), the normalized Shields parameter is low (< 1) as would be expected. This is due to the formation of the horseshoe vortex at that location as discussed in chapter 3 earlier. The wake region of the pile is a second area of low normalized Shields parameter and this is likewise expected as this is the region where the lee-wake vortex is formed. Along the pile diameter, higher normalized Shields parameter is noted (especially at the $45^\circ - 90^\circ$ of the pile where the flow constriction is at its maximum) for both the $k - \omega$ (SST) model and the Spalart-Allmaras model. Such high Shields parameter drives the sediment motion around the pile, resulting in a scour hole being formed over time.

It is noted that downstream and away from the pile, the normalized Shields parameters differ between the $k - \omega$ (SST) model and the Spalart-Allmaras model. This differing pattern downstream is due to the difference in the deposition of the sediment downstream. Such difference in deposition pattern is common as it is influenced by the lee-wake vortex, which is significantly irregular in nature.

Figure 4.17: The velocity profile along the upstream slope of the scour hole, 10 mm in front of the pile as pointed out by the orange arrow.

To obtain additional insight into the behaviour of the turbulence models during the scour process, the velocity profile along the upstream slope of the scour hole (10mm in front of the pile, see orange arrow in Figure 4.17) is evaluated. Based on Figure 4.17, both the $k - \omega$ (SST) model and the Spalart-Allmaras model agreed well with each other. While the Spalart-Allmaras model predicts a slightly higher maximum velocity than the $k - \omega$ (SST) model (0.126m/s for the Spalart-Allmaras model against 0.122m/s for the $k - \omega$ (SST) model), it should be noted that the scour hole configuration between the two models are slightly different, despite having similar scour depths. Hence, minor deviation in the both velocity profile is expected.

Based on the benchmarking between the scour simulation utilizing the $k - \omega$ (SST) model and the Spalart-Allmaras model, there is evidence that this one-equation turbulence model could be used for such complex simulations. It is suggested that in future works, more complex geometries could be used to further evaluate the Spalart-Allmaras model and to obtain greater understanding of such a turbulence model when applied to such complex simulations.

4.4.3 Live-Bed Scour Model

In the earlier section, the simulation setup (run N1, N7 and N9) were for that of a clear-water scour scenario. An attempt was made to apply the scour model developed to that of a live-bed condition. The configuration setup of the simulation was similar to that of the experimental and numerical work by Roulund et al. (2005) with the model conditions given in Table 4.7 as shown below.

Table 4.7: Simulation setup of the live-bed condition.

Pile Diameter	0.10m
Flow Velocity	0.460m/s
Sediment Size	0.26mm
Sediment Density	2650kg/m ³
Fluid Density	1000kg/m ³
Porosity	0.40
Dynamics friction coefficient	0.51
Static friction coefficient	0.63

Based on these input conditions, a simulation model was setup using a mesh geometry identical to that described in section 4.3 earlier, where $D = 0.10\text{m}$ in the current simulation model. Similarly, the boundary conditions utilized in this model was identical to that used in the earlier simulations of N1, N7 and N9 given in Table 4.8 below. It should be noted that only the bed load sediment transportation was model in the current simulation without accounting for the suspended load transportation. The exclusion of the suspended load model will be discussed in greater detail when evaluating the live-bed simulation results.

Table 4.8: Boundary conditions utilized in the live-bed scour simulation.

Boundary Condition			
Inlet	Outflow	Top & Side	Bed & Cyl.
Velocity	Outflow	Sym	Wall

Based on the model setup above, the scour depth-time development results for the scour simulation using both the $k - \omega$ (SST) and Spalart-Allmaras turbulence model are shown in Figure 4.18. The theoretical scour depth development calculated from Sumer et al. (1992a) equation is likewise plotted in Figure 4.18. As seen in the scour depth development plots, the results obtained from the simulations were disappointing. While both simulations utilizing the $k - \omega$ (SST) and the Spalart-Allmaras model had approximately similar profiles, the scour depth differed significantly from that predicted by the empirical equation (Sumer et al., 1992a). It is suggested that there are two key reasons for this deviation.

Figure 4.18: The scour depth-time evolution of a live-bed simulation utilizing the $k - \omega$ (SST) and Spalart-Allmaras turbulence model.

Firstly, it is suggested that the primary reason for this poor agreement was due to the mesh instability encountered during the simulation of the live-bed model. During the simulation, bed ripples were noted at the upstream section of the pile (as seen in Figure 4.19). Without intervention, these ripples become unstable which leads to a divergence in the simulation run. The flow regime of the current simulation is within the range where bed formations occur in physical experiments. In the work by Roulund et al. (2005), bed ripples were resolved by applying a perturbation to the bed roughness in the first 30 sec of the simulation. However, in the current simulations done, such a perturbation was not applied. To overcome the bed-ripples instability encountered in the current simulations, a damping function was implemented into the interDymScourFoam solver which limited the deformation of the mesh (dh_{bed}/dt) to a predefined value at each time-step to stabilize the simulation. This function limited the bed-ripples growth rate, especially during the earlier stages of the simulation. Subsequently, once the scour initiation process is completed (whereby a shallow scour hole is formed), a lower value of the limiting function was needed. It is suggested that this damping function resulted in a significantly slower scour hole development as seen in the simulation results (Figure 4.18).

Figure 4.19: The unstable bed ripples formed at the upstream section of the pile.

A second, more trivial reason for the poor correlation of the simulation result is that the current scour model (interDymScourFoam) does not incorporate a suspended load model. For a live-bed condition, the suspended load can account for a significant proportion of the total sediment load transportation. It should be expected that not accounting for the suspended load will result in a slower and shallower scour hole development. While this exclusion of the suspended load model is improbably the main cause of the poor simulation performance in this work, its omission likely contributed to the deviation of the simulation results.

Hence, to account and model live-bed cases in future works, additional considerations and improvements to the interDymScourFoam developed have to be incorporated for it to be applicable to a wider range of cases/conditions. These considerations include (1) incorporating a suspended load model into the interDymScourFoam solver, (2) addressing the issue of bed formations and (3) improving the approach taken to smoothen and stabilize the mesh deformation. These issues should be addressed in future works to finally obtain a comprehensive stable numerical scour model.

4.5 Conclusion

In this chapter, a full numerical scour model for monopile foundations was developed in OpenFOAM termed the interDymScourFoam solver. In this scour model, the flow profile was obtained from a RANS turbulence model. Based on the flow field, the bed parameters are determined and sediment transport is calculated utilizing a bed load equation (Engelund and Fredsøe, 1976). Subsequently, utilizing the Exner equation, the bed morphology is evolved and sand-sliding occurs in regions where the bed angle exceeds the angle of repose. This process is repeated till the equilibrium scour depth is obtained.

Focusing on the scour initiation process, for a clear-water scour, the simulation results using a $k - \omega$ (SST) turbulence model showed very good correlation of its scour depth-time development when benchmarked with the empirical equation proposed by Sumer et al. (1992a). In addition, when evaluating the scour initiation mechanism, the simulation model was able to predict fine details of the mechanism, including the initial movement of the scour hole upstream before elongating downstream to form a shallow scour hole. Such mechanisms are often omitted in experimental studies due to its rapid occurrence but was observed and described by Breusers and Raudkivi (1991). This observation in the simulation results shows the potential of utilizing numerical scour models to identify key scour mechanisms. One of the limitations of the interDymScourFoam solver developed was in performing parallel simulations which resulted in high simulation wall time. As such, the equilibrium scour depth was not obtained. However, the scour depth-time development and scour mechanism obtained from the simulation of the scour initiation process highlights the validity of the developed model.

In addition to the $k - \omega$ (SST) turbulence model, the Spalart-Allmaras model was also applied to the interDymScourFoam solver. The results of the simulation showed that a Spalart-Allmaras model was able to predict the scour process with comparable results to the $k - \omega$ (SST) turbulence model. This simulation results show the possibility of using a lower-cost turbulence models for such complex scour simulations, hence reducing the computational requirements of such models.

For the live-bed scour, the results of the simulation were unfortunately disappointing, with the scour depth prediction deviating significantly from empirical equation. It is suggested that this poor agreement is due to the mesh instability, which resulted in the incorporation of a damping function. In addition, the current scour model lacked a suspended load model which likewise contributed to the poor simulation results. Further developments to the numerical scour model is needed which includes incorporating a suspended load model into the scour model, improving the smoothening approach taken to dampen out the mesh instability and finally in parallelizing the code for shorter simulation times. These future works will increase the applicability of the scour model to more cases and will reduce the computational time requirements to a manageable level.

Chapter 5

RANS-DEM Sediment Transport Model

5.1 Introduction

In the previous chapter, a full numerical scour model was developed in OpenFOAM (interDymScourFoam). It was shown that the scour hole development predicted from the simulation correlated well with empirical formulation. While the simulation has provided much insight into the scour mechanism, especially during the scour initiation process, the detailed behaviour of the sediments during the scour process are to a large extent loss. This is because the scour simulation utilizes a bed load equation to model the sediment transport and corresponding bed morphology. Hence, the actual behaviour of individual sediment particles are not captured in the scour model. The ability to track and observe the behaviour of individual sediment particles during the scour process could provide greater fundamental insights into the sediment behaviour and corresponding scour mechanism (Loth, 2000). Such real-time tracking of sediment particles can be challenging to perform during scour experiments and where numerical models have a significant advantage.

Currently, there are no scour models which utilize a Lagrangian-based sediment transport modeling approach. The closest resemblance of a particle-based scour model was that proposed by Alam and Cheng (2010) whereby a Lattice-Boltzmann (LB) approach was utilized to model the scouring process. While Alam and Cheng (2010) simulation results had reasonable agreement with experimental data, they

did not make any correlation between the Lattice-Boltzmann model and the actual behaviour of the sediment particles.

Although currently there has not been a particle-based scour model, a recent work by Escauriaza and Sotiropoulos (2011) attempted to model the bed load sediment transport around a wall-mounted obstacle (cylinder) using a Lagrangian-based modeling approach. They demonstrated that with a simple Lagrangian-based sediment model, the initiation motion of the particles upstream of a cylinder could be replicated well. While the motion of the particles did not influence the morphology of the bed in the model proposed by Escauriaza and Sotiropoulos (2011), their work showed the potential of applying such Lagrangian-based sediment models for scouring applications.

These earlier works (Alam and Cheng, 2010; Escauriaza and Sotiropoulos, 2011) on modeling scour using particle-based approaches have indicated potential in developing a full particle-based numerical scour model in the future, whereby the sediment transport is modeled using a Lagrangian-based approach. To ultimately achieve such a fundamental but complex scour model, it is proposed that the development be broken down in 3 stages to finally realize such a vision. These proposed stages are given as follow:

- Stage 1: Development and validation of sediment behaviour on flat bed
 - Individual particle behaviour validation (e.g. settling velocity)
 - Bed load and suspended load behaviour validation
 - Translation of sediment behaviour to bed morphology validation
- Stage 2: Application of sediment transport model to simple case studies
 - Bed forms evolution simulations
 - Dredging simulations
- Stage 3: Application of sediment transport model to complex case studies
 - Scour at sluice gate and around abutment simulation.
 - Scour around monopile and jacket foundation simulation.

By adopting this gradual approach in the development of the envisioned Lagrangian-based scour model, the validation of the final particle-based scour model will be more manageable, especially given the complexity of such a scour model.

While the proposed development plan to achieve a full Lagrangian-based scour mode is the long-term vision of the author, in this current work, the focus will be only on stage 1 of the plan. This stage will be to develop and validate a particle-based sediment transport model for a flat bed condition. Such a development will act as the cornerstone for future works to build on, to ultimately achieve a full Lagrangian-based scour mode. With this consideration of the long-term vision, the code development done for the Lagrangian sediment transport model in this chapter was built in the same environment as that of the `interDymScourFoam` solver developed earlier in chapter 4. Doing so, this will enable easier integration between the sediment transport model and the numerical scour model in the future works. Details of the code development will be discussed in-depth in the subsequent sections.

The Lagrangian-based model developed/proposed in this work was that of a RANS-DEM sediment transport model. Through the use of a RANS turbulence model, a coarser mesh can be adopted which lowers the computational resource requirements significantly. In addition, instead of packing the simulation domain with particles to model the bed morphology as was done in earlier works (Kidanemariam and Uhlmann, 2014), it is proposed that a limited number of tracer-particles be used instead. Based on the behaviour of the tracer-particles, the mesh of the bed boundary will be deformed accordingly which results in the corresponding bed morphology evolution. Details of the proposed RANS-DEM sediment model will be discussed in-depth in the subsequent section. Such a Lagrangian-based sediment transport model will act as the cornerstone for future works towards the development of a full Lagrangian-based scour model.

5.2 Theory

In this section, the Lagrangian-based RANS-DEM sediment transport model proposed is described. This model consist of three components which are as follows:

- Turbulence Model
- Sediment Transport Model
- Bed Morphology Model

Starting from the turbulence model, the flow and turbulence profile within the simulation domains are determined using the RANS equations. This flow and turbulence profiles are subsequently utilized to model the bed load and suspended load behaviour based on a force balancing Discrete Element Method (DEM). Based on the predicted sediment behaviour, the bed boundary of the simulation domain is deformed accordingly, hence replicating the formation of the bed morphology and bed-form features. The details of each component are described in-depth in the following paragraphs.

5.2.1 Turbulence Model

The RANS approach was utilized in this work to model the flow and turbulence profile due to its relatively low computational time. The turbulence model utilized to close the RANS equation is that of the two-equation $k - \omega$ (SST) model. The advantage of the $k - \omega$ (SST) model is that it capitalizes on the accuracy of the $k - \omega$ model within the near-wall region and the $k - \varepsilon$ model in the far-field region. In addition, the calculated turbulence kinetic energy (k) and specific rate of dissipation (ω) provides the magnitude and length scale of the turbulent eddies. These turbulence properties will be critical in determining the fluctuating components of the flow at each point within the simulation domain, which is required for the suspended load model, as will be explained in the subsequent section. The details of the $k - \omega$ (SST) turbulence model is fully described in chapter 3.

5.2.2 Sediment Transport Model

The sediment motion and transportation was modeled utilizing a DEM approach, whereby a force balance on each particle determined the trajectory of each particle. The force balance components (drag and gravitational forces) used in this work are given as,

$$m \frac{dv_{particle}}{dt} = \frac{1}{2} \rho C_D U_r^2 A + \left(1 - \frac{\rho}{\rho_s} \right) mg \quad (5.1)$$

where m is the mass of the particle, C_D is the coefficient of the drag of the particle, U_r is the relative velocity, A is the area of projection of the particle, ρ is the density of the fluid and ρ_s is the density of the sediment. The C_D utilized in this work was obtained from an empirical expression proposed by Fredsøe and Deigaard (1992) given as,

$$C_D = 1.4 + \frac{36}{Re} \quad (5.2)$$

where Re is the particle Reynolds number.

While this DEM model utilizes a simplified force balance, which does not account for particle-particle collision and forces attributed to added mass, pressure, and viscous forces, it was found to be sufficient to model the sediment behaviour as will be elaborated on in the later sections.

Sediment regimes

For a sediment particle being transported within a flow, it can be characterized as either being within a bed load regime or a suspended load regime. The regime which the sediment particle is within is determined based on the particle's position/height relative to the bed boundary. In this work, the reference height (b), which separates the bed load and suspended load regimes, is a function of the mean diameter of the sediment given as,

$$b = C_\alpha \times d \quad (5.3)$$

where d is the sediment diameter and C_α is a constant. It is proposed by Fredsøe and Deigaard (1992) that $C_\alpha = 2$.

For each sediment regime, the mechanism by which the flow velocity is applied on each sediment particle is consistent to that observed in experiments. The description of the different sediment regime models (bed load and suspended load models) are described in the following sub-sections.

Bed load model

The bed load is the sediment regime where the particles have a position lower than the reference height, b , and is transported along the bed boundary. Based on experimental observations, sediment particles in the bed load move along the bed by the rolling, sliding and saltating mechanisms. However, in this work, it is assumed that the bed load movement is only driven by the sliding mechanism.

The approach taken in the current work to model the bed load transportation is similar to that proposed by Engelund and Fredsøe (1976) as described earlier in chapter 4 and is likewise presented in the current chapter for easier reference.

$$q_b = \frac{\pi}{6} d^3 \frac{PEF}{d^2} U_b \quad (5.4)$$

where d is the sediment size, PEF is the probability of sediment motion and U_b is the bed load sediment velocity. As seen from the above equation, Engelund and Fredsøe (1976) suggested that the bed load comprises of 3 key components, which are the volumetric properties of the sediment ($\frac{\pi}{6} d^3 \frac{1}{d^2}$), the probability of sediment motion (PEF) and the bed load sediment velocity (U_b). The first two components are directly derived from the sediment properties ($\frac{\pi}{6} d^3 \frac{1}{d^2}$) and empirical equations (PEF). To obtain the bed load sediment velocity however, Engelund and Fredsøe (1976) proposed an analytical equation for U_b based on a balance of agitating and stabilizing force given as,

$$U_b = 10 \left[1 - 0.7 \sqrt{\frac{\theta_{cr}}{\theta}} \right] U'_f \quad (5.5)$$

where θ_{cr} is the critical Shields parameter, θ is the Shields parameter and U'_f is the skin friction velocity.

Adopting the same approach but applied to an Eulerian simulation framework, Roulund et al. (2005) (likewise in the scour model presented in chapter 4) obtained the bed load sediment velocity from a force balance analysis which resulted in 4 unknowns and 4 equations. These equations were solved using a Newton-Raphson technique. Details of the force balance analysis are presented in chapter 4.

In this work, the bed load approach proposed by Engelund and Fredsøe (1976) is applied to a Lagrangian simulation framework. A key benefit of adopting this bed load approach into a Lagrangian-based model is the availability and ease of determining the bed load sediment velocity (U_b) during each simulation time-step. However, as a RANS turbulence model was utilized, obtaining the driving force on the particle directly from the flow velocity cannot be done. To overcome this limitation, instead of utilizing the flow velocity at the particle/sediment location, the friction velocity determined at the bed boundary is utilized as the input into the DEM force balance equation instead. The bed load sediment velocity (U_b) is determined as follows;

1. Determine the friction velocity at bed boundary, where $U_f = \sqrt{\tau_0/\rho}$.
2. Map U_f from bed boundary into the internal mesh of the bed load region.
3. Calculation of fluid velocity on bed load sediment ($U_{b-fluid} = aU_f$) within bed load region. A value of $a = 10$ was proposed by Engelund and Fredsøe (1976).
4. Direction of flow velocity obtained from the unit vector of the cell's flow velocity at the particle/sediment location.
5. Solve force balance on particle based on DEM model to obtain bed load sediment velocity (U_b).

To account for the probability of sediment motion, the percentage of particles in motion in the bed surface layer (PEF) proposed by Engelund and Fredsøe (1976) as presented earlier in chapter 4, was utilized. The calculated PEF is subsequently applied to the bed load sediment velocity (U_b) to obtain the mean bed load sediment velocity ($U_{b(PEF)}$) given as,

$$U_{b(PEF)} = PEF \times U_b \quad (5.6)$$

Suspended load model

The suspended load are the particles within the flow regime that are above the reference height, b , described earlier. In comparison to the bed load particles, the trajectory of the suspended load particles are determined from the flow velocity at its location and the DEM equation (Equation 5.1). While the flow field obtained from the RANS turbulence model is capable of modeling the lateral motions of these suspended particles well, this is not the case for the suspended load distribution along the water column. This is because when applying the RANS turbulence model to predict the flow field, the ‘Reynolds averaging’ involved results in the fluctuating instantaneous flow velocities to be lost. These fluctuating instantaneous velocities are the key components which keep the sediment particles in suspension (Engelund and Fredsøe, 1976). Hence, to model the suspended load well, especially in terms of its load distribution, the need to incorporate/redetermine the fluctuating flow velocities lost by the RANS turbulence model is critical.

In the proposed suspended load model, the fluctuating instantaneous velocity (U') in the z-direction (which keeps the particles in suspension) are redetermined utilizing the turbulence properties (k and ω) together with a mixing length theory. The magnitude of the fluctuating flow velocity in the z-direction is obtained from the turbulence kinetic energy (k) given as,

$$U' = R(n)\sqrt{\frac{4}{9}k} \quad (5.7)$$

where $R(n)$ is a random number with a standard normal distribution. As seen in Equation 5.7, U' accounts for the anisotropy turbulence normal stresses based on $\frac{4}{9}k$ (Fuhrman et al., 2011).

While the turbulence kinetic energy provides the magnitude of the fluctuating flow velocity, the particle time-step likewise has to be determined to account for the size of the turbulent eddy. To obtain the particle time-step, the mixing length (l_m) was first determined as follows,

$$l_m = \frac{\sqrt{k}}{\omega} \quad (5.8)$$

Using the mixing length (l_m), the particle time-step ($t_{particle}$) can subsequently be determined as,

$$t_{particle} = C_{particle} \times \frac{l_m}{U_{rms}} \quad (5.9)$$

where U_{rms} is the root-mean-square value of the particle velocity and $C_{particle}$ is a constant. In this work, it was identified that $C_{particle}$ was a function of the coefficient of drag (C_D) of the sediment particle utilized given as,

$$C_{particle} = C_{sedi} \times C_D \quad (5.10)$$

where $C_{sedi} = 2.1$. The derivation of the $C_{particle}$ and C_{sedi} will be discussed in greater details in the results section.

To validate the suspended load model, the sediment distribution along the water column was benchmarked against the analytical Vanoni-distribution. This will likewise be discussed in greater detail in the results section.

5.2.3 Morphological Model

The sediment transport models described above simulates the bed load and suspended load motion/behaviour only, without accounting for any deformation of the bed. Hence, based on the sediment motion, should the net sediment transportation be non-zero, this will result in morphological changes along the bed boundary. Hence in this work, a thought-experiment was performed in translating the sediment transportation obtained in the earlier models into bed morphology changes (via dynamic meshing) as will be described below. It should be noted that this being a thought-experiment, while attempts to demonstrate the potential capabilities of the morphological model will be made, it is acknowledged that a significant amount of additional work has to be done before this model is fully validated. The purpose of such a thought-experiment is to present the potential capabilities of such a model for future works to expand on.

In the proposed morphological model, instead of utilizing excessive particles to model bed morphology (Kidanemariam and Uhlmann, 2014), a technique of utilizing tracer-particles and deforming the mesh according to these tracer-particles movement is implemented. Such an approach significantly reduces the quantity of particles required and its corresponding computational time.

The morphological model consists of 4 parts, which are the (1) Mesh zoning, (2) Particle injection and transportation, (3) Bed boundary deformation and (4) Particle re-calibration. The sequence of this model is as follow,

1. **Mesh zoning:** The simulation domain is initially sub-divided into zonal sections along the stream-wise direction.
2. **Particle injection and transportation:** For each zone, tracer-particles are injected at the inlet of each zone and allowed to interact with the flow for Δt_{morp} . It should be noted that altering the injection location will require re-calibration of the model, which will not be focused on in this work. The transportation of the particles are governed by the DEM equation (bed load and suspended load model) described earlier.
3. **Bed boundary deformation:** At time $= t + \Delta t_{morp}$, the number of particles in each zone is determined, and the net sediment transport within each zone is calculated. Based on the net sediment transport, the bed boundary within the zone is deformed accordingly using the dynamic meshing in OpenFOAM.
4. **Particle re-calibration:** Upon completion of the mesh deformation, all the particles are removed from the zone and new particles are re-injected into the inlet of each zone (step 2) and this process is repeated.

Hence based on the above morphological model, Equation 5.11 is proposed to couple the interaction between the Lagrangian framework of the sediment transport model and the Eulerian framework of the flow domain and bed boundary given as,

$$\frac{\partial h_{bed}}{\partial t} = \frac{4}{6} \pi d^3 A_{zone} (N_t - N_{t+\Delta t_{morp}}) \quad (5.11)$$

where d is the sediment diameter, A_{zone} is the area of the zone, N_t is the number of particles in a zone at time t and $N_{t+\Delta t_{morp}}$ is the number of particles in a zone at time $t + \Delta t_{morp}$, where Δt_{morp} is the bed morphology time-step.

As mentioned earlier, a key consideration of the morphological model is the location of the particle injection within each zone. Based on the assumption adopted that the particle has to transverse the entire zone to be representative of the net sediment transport of the zone, the particles are injected upstream of the zone. This injection point was shown to reproduce bed morphology well as will be discussed in the results section.

For the bed deformation, a dynamic mesh was utilized to deform the bed boundary of the simulation domain based on Equation 5.11. For the mesh motion in the dynamic mesh, a `dynamicMotionSolverFvMesh` was utilized, which solves a Laplace's equation for the motion displacement to finally obtain the new position of the mesh points.

5.2.4 Code Algorithm

For the RANS-DEM sediment transport model described above, the code development and validation was performed in both OpenFOAM (bed load and morphological model) and Matlab (Suspended load). The suspended load model was evaluated in Matlab rather than in OpenFOAM due to the large number of time-step required to test and benchmark the suspended load distribution with that of the Vanoni distribution.

For the OpenFOAM development, a solver termed the `interDymSediFoam` was developed. Similar to the `interDymScourFoam` described in the earlier chapter, the `interDymSediFoam` was built on top of the pre-existing solver called `interDymFoam`, which is a multi-phase Volume-Of-Fraction (VOF) solver with dynamic mesh capabilities. The `solidParticleFoam` was coupled with the `interDymFoam` to perform the DEM calculations on the particles. Using the coupled solver, the bed load model, suspended load model and morphological model described earlier were incorporated into the `interDymSediFoam`. It must be noted however that the suspended load model was validated in Matlab and incorporating it into the `interDymSediFoam` solver was to provide completeness to this work. The algorithm of the `interDymSediFoam` solver is given in Figure 5.1. Likewise, the overview of the components of the `interDymSediFoam` is shown in Figure 5.2.

Figure 5.1: The algorithm of the interDymSediFoam solver developed.

Figure 5.2: The overview of the interDymSediFoam developed in OpenFOAM.

To validate the bed load model and morphological model, the interDymSediFoam solver was utilized to model the flow and bed evolution over a flat erodible bed. The predicted bed particle velocity and morphology were benchmarked against theoretical and experimental observations.

To validate the suspended load model, a Matlab script was written. By fixing the profile of the flow velocity, turbulence kinetic energy and specific rate of dissipation, a suspended load particle was introduced into the domain. The particle motion and position determined from the suspended load model described earlier are recorded over time. Subsequently, the profile of the particle elevation distribution was plotted and compared with the Vanoni diagram to benchmark the simulation with the theoretical equation. Details of the validation study will be discussed in the subsequent section.

5.3 Model Setup

Simulation models were setup to validate the bed load, suspended load and morphological models proposed earlier. The suspended load validation was performed in Matlab while the bed load and morphological model validation were performed using the interDymSediFoam solver developed in OpenFOAM. To validate the proposed bed load model and morphological model, a numerical setup of a flat erodible bed was utilized. For the suspended load, the sediment distribution along the water depth for a range of sediment sizes was utilized. The validation models are described in the subsequent paragraphs.

5.3.1 Bed load and Morphological Model

The bed load and morphological model was validated in OpenFOAM based on an idealized experimental setup shown in Figure 5.3. As discussed in the earlier section, a simplified computational layout of an erodible flat bed was utilized in this work to provide fundamental validation of the bed load behaviour on simple cases before progressing to more complex domains in future works as described earlier in section 5.1. These complex domains are out of the scope in the current work.

Figure 5.3: The simulation model setup for the validation of the bed load and morphological model based on an idealize experimental setup.

Mesh geometry

A hexahedral block mesh with a geometrical dimension of $1 \text{ m} \times 1 \text{ m}$ in the x- and z-direction was used to perform all the bed load and morphological simulations. While the model was extruded in the y-direction as seen in Figure 5.4, this was done for visualization purposes. In the simulation, the y-axis component was turned off and the particles were constrained within the x- and z- directional plane. The mesh and boundary setup of the model is shown in Figure 5.4.

Figure 5.4: The mesh and boundary setup utilized in the simulations performed. The model was extruded in the y-direction (seen on the right) for simpler visualization purposes.

Boundary conditions

The boundary conditions utilized in the bed load and morphological simulation is given in Figure 5.4 above. For the upstream inlet boundary, an inflow profile for the velocity, turbulence kinetic energy and specific rate of dissipation is utilized given as,

$$u(z) = \frac{U_f}{\kappa} \ln \left(\frac{z}{z_o} \right) \quad (5.12)$$

$$k(z) = C_\mu^{-1/2} \left(1 - \frac{z}{\delta} \right)^2 U_f^2 \quad (5.13)$$

$$\omega(z) = \frac{U_f}{C_\mu^{-1/2} \kappa z} \quad (5.14)$$

where U_f is the friction velocity, κ is the von Kármán constant, z is the water depth variable, δ is the boundary layer height and C_μ is the closure coefficient. For z_o , it is the bed roughness given as,

$$z_o = \frac{K_s}{30} \quad (5.15)$$

where K_s is the height of roughness element, which is given as $K_s = 2.5d$.

Model conditions

The model conditions utilized in the bed load and morphological simulations are given in Table 5.1, whereby the Zone Length is defined as the length of each zone in the x-direction. A visualization of the zones is likewise shown in Figure 5.4.

Table 5.1: The model conditions utilized in the bed load and morphological simulations.

Sediment Density	$2655kg/m^3$
Porosity	0.4
Zone Length	$0.05m$

The flow velocity and sediment size for each validation simulation varied based on the availability of empirical data and the sediment behaviour being modeled in each case. The mean flow velocities and sediment particle sizes utilized in this work are summarized in Table 5.2. Details of the selection of the flow velocities and sediment sizes are justified in the subsequent section.

Table 5.2: The mean flow velocities and sediment particle sizes utilized for each bed load and morphological model validation test.

	Case Name	Mean flow velocity	Sediment Size
Settling velocity validation	Case 1	0.00m/s	2.000mm
Bed load validation	Case 2	0.30m/s	0.125mm
	Case 3	0.50m/s	0.125mm
	Case 4	0.75m/s	0.125mm
Morphological validation	Case 5a/b	0.75m/s	0.125mm

5.3.2 Suspended load Model

The validation of the suspended load model was performed in Matlab. The key feature which had to be validated is the Vanoni distribution of the suspended particles. As described above, the fluctuating velocities, which keeps the particles in suspension, was redetermined based on the turbulence kinetic energy and specific rate of dissipation.

Model geometry

As the validation of the suspended load model was primarily to reproduce the suspended load distribution, a one-dimensional geometrical model was setup in the z-direction (water depth). The sediment's elevation evolved along this z-direction during the simulation based on the suspended load model described earlier. The z-direction limit or water depth used in all simulations was 0.2 m.

Boundary conditions

In the Matlab model, the turbulence kinetic energy and specific rate of dissipation profile was kept constant based on Equations 5.13 and 5.14 described earlier. As the validation of the suspended load model was primarily to reproduce the suspended load distribution, no inflow velocity was utilized and the sediment was kept in suspension based on the turbulence only. At the upper and lower limits of the model, a reflective boundary condition (which reflects the particle back into the domain) was implemented into the simulation model.

Model conditions

A key variable which influences the suspended load distribution is the sediment size and corresponding Rouse number of each simulation case. In this work, 5 simulation cases of decreasing Rouse number was utilized to determine the validity of the proposed suspended load model. The case number, sediment size, and corresponding Rouse number are presented in Table 5.3.

Table 5.3: The sediment size and Rouse number utilized to validate the suspended load model.

Case Name	Sediment Size	Rouse Number
Case 6	1.250mm	1.93
Case 7	0.600mm	1.20
Case 8	0.420mm	0.95
Case 9	0.350mm	0.79
Case 10	0.125mm	0.18

5.4 Results and Discussion

The results of the validation simulations of the RANS-DEM model are presented in 4 parts in this section. In the first part, the force balance in the RANS-DEM model on each particle was validated based on the settling velocity simulation. Subsequently, the bed load model was simulated to determine the ability of the model to predict the mean bed load velocity of the sediment particle. Pushing the analysis further based on a thought-experiment, the validated bed load model was applied to the morphological model to evaluate the translation of the bed load behaviour to morphology evolution. These simulations were performed utilizing the interDymSediFoam solver developed in OpenFOAM. Finally, to validate the suspended load model, a Matlab script was utilized. In the suspended load model, the sediment distribution along the water depth was modeled and benchmarked against the Vanoni distribution. The details of validation studies are described in the following sub-sections.

5.4.1 Sediment Settling Velocity

A simulation of a single free-falling particle, as shown in Figure 5.5, was performed to determine if the model is capable of predicting the settling velocity of the sediment. Such a simulation was done to validate that the numerical setup and DEM force balance model written in the interDymSediFoam solver operated in accordance to that described earlier.

The settling velocity simulation is setup based on the model conditions described in the earlier section, whereby a sediment size of 2mm was utilized. The simulation results obtained was compared with that calculated theoretically from the settling velocity equation proposed by Fredsøe and Deigaard (1992) as follows,

$$w_s = \sqrt{\frac{4(s-1)gd}{3C_D}} \quad (5.16)$$

where s is the relative density of the sediment, g is the acceleration due to gravity and d is the sediment size. The C_D used in this work was estimated based on Equation 5.2, whereby $\frac{36}{Re} \approx 0$ due to the large size of the sediment (2mm) which results in $C_D = 1.4$.

Additional experimentally measured settling velocities obtained by Cheng (1997) were likewise used to benchmark the current simulation results. The simulation result, theoretical calculations and experimental observations are shown in Table 5.4.

Figure 5.5: The simulation of the settling velocity of the particle.

Table 5.4: The simulation result, theoretical calculations and experimental observations for the settling velocity of the sediment particle.

	Settling Velocity
Simulation	0.169 m/s
Empirical Equation (Fredsøe and Deigaard, 1992)	0.176 m/s
Experiment (Cheng, 1997)	0.193 m/s

Based on Table 5.4, the simulation results correlated reasonably well with the empirical equation (Fredsøe and Deigaard, 1992) and experimental observation by Cheng (1997). It should be noted that in this work, the C_D value utilized (Equation 5.2) was in the upper range of the spectrum of proposed C_D values currently

available in the literature. Hence, the simulated results were slightly smaller as compared to experimental observations by Cheng (1997). In Cheng (1997) work, he utilized several empirical equations to determine the settling velocity of a 2mm sediment. The range of settling velocities estimated by Cheng (1997) from the empirical equations ranged from 0.178 m/s - 0.205 m/s. Each empirical equation adopted a slightly different C_D , which highlights the sensitivity of this variable. As the simulation results had good correlation with empirical calculations (Fredsoe and Deigaard, 1992) and reasonable agreement (variance < 15%) with experimental observations, the simulation validates the force balancing DEM setup in the interDymSediFoam solver developed in this work.

5.4.2 Bed load Model

As described earlier, the bed load sediment transport was modeled in a similar approach to that proposed by Engelund and Fredsoe (1976), but on a Lagrangian framework. Based on the flow field obtained from the turbulence model, the friction velocity is calculated at the bed boundary. The friction velocity from the bed boundary is subsequently mapped into the internal mesh of the bed load region as shown in Figure 5.6. Utilizing the friction velocity, the fluid velocity acting on the bed load is calculated to determine the bed load sediment velocity (U_b).

Figure 5.6: The friction velocity mapped from the bed boundary into the bed load region internal mesh.

In general, bed load motion can be described for 3 different flow regimes. These are (1) no bed load transportation ($\theta < \theta_{cr}$), (2) low bed load transportation ($\theta \geq \theta_{cr}$) and high bed load transportation ($\theta \gg \theta_{cr}$). In this work, three simulation test cases (Case 2, 3 and 4) as described earlier was utilized to validate the bed load model. The simulated mean bed load velocity was benchmarked against theoretically calculated mean bed load velocity ($U_{b(PEF)}$) based on the equation proposed by Engelund and Fredsøe (1976);

$$U_{b(PEF)} = PEF \times \left(10 \left[1 - 0.7 \sqrt{\frac{\theta_{cr}}{\theta}} \right] U'_f \right) \quad (5.17)$$

where U'_f is the skin friction of the flow.

Table 5.5: The simulated and theoretical bed load velocity of the bed load sediment

Case Name	Shields Criterion	Flow Velocity	Simulated Results	Theoretical Equation
Case 2	$\theta < \theta_{cr}$	0.30m/s	0.0m/s	0.0m/s
Case 3	$\theta \geq \theta_{cr}$	0.50m/s	0.032m/s	0.026m/s
Case 4	$\theta \gg \theta_{cr}$	0.75m/s	0.170m/s	0.153m/s

The simulation results and theoretical calculations for the bed load models are presented in Table 5.5. For low flow velocity (Case 2) where the Shields parameter is lower than that of the critical value, no bed load transportation occurs as expected. This validates the interDymSediFoam solver threshold function for sediment motion initiation. As the flow velocity increases to slightly above the critical velocity (Case 3), sediment transportation begins at a very slow rate. The simulated mean bed load velocity at this critical flow value agreed well with $U_{b(PEF)}$ obtained from theoretical calculation as shown in Table 5.5. It is noted that the simulation models the average particle velocity ($U_b \times PEF$) rather than the instantaneous velocity of the particles in motion (bed load particle at low Shields flow velocity moves in random intervals (low PEF) but at higher velocity (higher U_b)). While it is possible to model instantaneous velocity of the bed load particles using a random function and an if-else statement, the time-average motion of the bed load particles between the two approaches will be similar.

As the velocity of the flow is increased further beyond the critical value(Case 4), the simulated mean bed load velocity and theoretical calculation likewise agreed well for such high velocity flows as shown in Table 5.5. Hence, with this good agreement between the simulated mean bed load velocity and theoretical equation for different flow profiles/regimes, these results highlight the robustness of this bed load model to predict bed load behaviour/motion for a spectrum of flow cases. Similarly, this has likewise demonstrated that the model proposed by Engelund and Fredsøe (1976) can be applied to a bed load transportation model in a Lagrangian-based framework.

5.4.3 Bed Morphological Model

The bed load model describes the motion and behaviour of the sediment particle along the bed boundary. Based on this sediment motion, a thought-experiment was performed utilizing a proposed bed morphological model to translate the sediment transportation behaviour to bed deformation, which ultimately leads to the formation of key bed features over time. Hence, utilizing the proposed bed morphology model, a live-bed simulation (Case 5) was performed. Based on the model setup as shown earlier in Figure 5.3, two separate simulations were performed. In the initial simulation (Case 5a), a hypothetical case, whereby the total mass of the sediment within the system is conserved (particles are not allowed to leave the outflow boundary), was performed. In this simulation, the morphological model's abilities to account for both erosion and deposition of the bed boundary was evaluated. For the second simulation (Case 5b), the total mass of the sediment within the system is not conserved and particles are able to leave the simulation domain from the outflow boundary. Such a simulation setup is that similar to a case of scour below a spillway and sluice gate, which is a more realistic scour case. For both cases, the simulation was run till minimal bed morphology changes were observable.

The simulation model for the mass conservative simulation (Case 5a) and the bed level elevation results are shown in Figure 5.7 and Figure 5.8 respectively. With reference to Figure 5.8, starting from an initially flat bed condition, the scour initiates upstream of the simulation domain, where there isn't a continuous supply of sediment to replace the sediment getting transported downstream. Over time, the scour trench formed upstream deepens ($t=1000\text{sec} - 1500\text{sec}$) and equilibrates at a depth of approximately 0.015m. Subsequently, the scour trench grows and prop-

agates downstream ($t=2000\text{sec} - 4000\text{sec}$) with minimal variation to the upstream equilibrium scour depth, similar to that observed by Breusers (1966, 1967). Downstream of the simulation domain, deposition of particles occurs as the particles are prevented from leaving the system in Case 5a.

Figure 5.7: Simulation model of Case 5a.

Figure 5.8: The bed elevation graph over time for simulation Case 5a.

In Case 5a, the final bed morphology (at $t=4000\text{sec}$) was that with the maximum scour depth occurring upstream of the domain which gradually increases in height till it reaches the deposition peak at the outflow boundary of the domain. While such a test case (whereby particles are not allowed to exit the domain) is primarily theoretical and difficult to replicate in experimental work, the purpose of the simulation was to evaluate the ability of the bed morphological model to hand both erosion and deposition in a single simulation domain. What was interesting was the resemblance the simulated bed morphology had with the erosion and deposition patterns along a longshore caused by groyne fields interferences as seen in Figure 5.9 (Fredse and Deigaard, 1992). For erosion and deposition within a groyne field, the sediment motion is similar to that of simulation Case 5a, except rather than the bed morphology evolving in the z -direction, the coast morphology is evolving in the y -direction. This similarity between the simulation results of Case 5a and that presented by Fredse and Deigaard (1992) for groyne fields provides optimism of the validity of the morphological model in handling both erosion and deposition.

Figure 5.9: The erosion and deposition along a longshore caused by a groyne interference (Fredse and Deigaard, 1992).

To simulate a more realistic case, a simulation setup (Case 5b), whereby the particle constraint at the outflow boundary is removed, and particles are allowed to exit the domain, was performed. Such a simulation test case is similar to scour at a spillway performed by Breusers (1966, 1967). The bed level elevation results and simulation model for Case 5b are shown in Figure 5.10 and Figure 5.11 respectively. As seen in Figure 5.10, the scour initiates upstream of the domain to form an initial trench ($t = 500\text{sec}$). As time progresses, the scour trench deepens ($t=1000\text{sec}$ - 2000sec) to equilibrate at a depth of approximately 0.015m . Subsequently, the scour trench grows and propagates downstream ($t=3000\text{sec}$ - 7000sec), maintaining the approximate equilibrated scour depth of 0.015m . As particles were allowed to exit the system, no deposition downstream of the domain was obtained. This scour mechanism obtained from the simulation correlates with that reported by Breusers (1966, 1967). However, in the simulation model, an initial deposition of sediment slightly downstream of the initial scour trench was not reproduced. Such initial sediment deposition is likewise observed for scour at sluice gates as described by Melville and Lim (2013). It is suggested that the model's inability to reproduce this fine detail of the initial deposition downstream of the scour trench is due to sediment particles only providing the average behaviour of the sediment particles within the zone it is in. In future works, the sensitivity of the zone size should be evaluated to determine if reducing the coarseness of the zone size improves the level of morphological details of the simulation model.

Figure 5.10: The bed elevation graph over time for simulation Case 5b.

Figure 5.11: Simulation model of Case 5b.

While the mechanism of the scour process between the simulation and that described by Breusers (1966, 1967) had reasonable correlation, the equilibrium scour depth obtained in the simulation was significantly shallower as compared to experimental studies. Utilizing the experimental data obtained by Breusers and Raudkivi (1991), the approximate experimental scour depth for the closest experimental setup (water depth = 1.5m, flow velocity = 0.83m/s, $d = 0.22mm$) to the simulation was 0.04m (Breusers and Raudkivi, 1991). This is deeper than the 0.015m depth obtained in the current simulation. It is suggested that this difference is due to the sensitivity of the zone size and morphological time-step. Such sensitivity of these parameters has to be investigated in greater detail in future works. However, this being a thought-experiment, the reasonable agreement of the scour mechanism between the simulation results and that described by Breusers (1966, 1967) has highlighted the potential of such a morphological model.

5.4.4 Suspended load Model

In the earlier section, the validation of the bed load model and morphological model was performed in OpenFOAM utilizing the interDymSediFoam solver developed. The suspended load model was likewise incorporated into the interDymSediFoam as described in section 5.2.2. For the suspended load, when implementing the model into OpenFOAM, some of the key implementation areas to highlight are as follows:

- The z-coordinate position of the particle was utilized to identify if the particle was within the bed load or suspended load regime.
- The flow velocity and turbulence properties (k and ω) utilized in the suspended load calculations was obtained from the cell the particle was located in.
- The particle properties utilized in the calculations (e.g. C_D) were identified based on the particle identity/classification.

While the suspended load model was incorporated into the interDymSediFoam solver, the validation of the suspended load model was performed in Matlab due to the large number of time-step required to obtain the statistical distribution of the sediment.

When modeling the behaviour of the suspended load, there are 2 key components that have to be considered . They are:

- The lateral motion of the particles
- The suspended load distribution.

For the lateral motion of the particle, the main driving force on the sediment is the velocity profile of the fluid at the location of the particle. Based on the DEM force balance on the particle, as described in Equation 5.1, the trajectory of the sediment will be determined, and the new position of the particle will be updated. The DEM force balance on a sediment particle was evaluated earlier utilizing the settling velocity simulation. It was shown that utilizing the force balance between the drag forces and gravitational force on the sediment, the predicted settling velocity agreed well with experimental data.

For the suspended load distribution however, utilizing a RANS turbulence model to predict the sediment distribution profile isn't trivial. This is because as discussed earlier, by utilizing a RANS turbulence model, the fluctuating vertical velocities which maintain the particles in suspension are no longer available due to the Reynolds averaging. As such, a suspended load model was proposed in this work to redetermined the fluctuating vertical velocities utilizing the turbulence kinetic energy (k) and the specific rate of dissipation (ω) as described earlier.

The theoretical distribution of the suspended load within a water body is that of a Vanoni distribution given as,

$$c = c_b \left(\frac{h - z}{z} \frac{b}{h - b} \right)^{\frac{w_s}{\kappa U_f}} \quad (5.18)$$

where h is the water depth, w_s is the settling velocity, κ is the von Kármán constant, U_f is the friction velocity and c_b is the suspended sediment concentration at a reference distance b from the bed. Based on Equation 5.18, it can be seen that the Vanoni distribution is a function of the Rouse parameter (R_o) given as,

$$R_o = \frac{w_s}{\kappa U_f} \quad (5.19)$$

Hence, the Rouse parameter is primarily defined by the sediment size, assuming the flow and turbulence conditions are consistent.

Based on the proposed suspended load model, five simulation cases (Case 6 - 10) were performed in this work. The key variable between each simulation case was the sediment sizes, which corresponds to a range of Rouse numbers. The simulation results of the suspended load distribution were benchmarked against the theoretical Vanoni distribution. All the suspended load simulations were performed in Matlab, whereby the turbulence kinetic energy (k) and the specific rate of dissipation (ω) profile were kept constant, as presented earlier.

Utilizing Case 6 as an example, the results of the suspended load model was evaluated in greater detail. For simulation Case 6, a relatively large sediment size of 1.25mm was utilized, which translate to a high Rouse number of 1.93. In the simulation, the suspended load particle was transverse along the z-direction (along the water depth) over 500,000 time-steps based on the suspended load model proposed earlier. The position of the particle at each time-step was recorded over time. A sample of the particle's position over a time period of 25,000 time-steps is shown in Figure 5.12 for Case 6 ($R_o = 1.93, d = 1.25mm$). From Figure 5.12, it can be seen that the sediment particle spend a significant amount of time within the lower region of the simulation domain, which will be as expected of such a large particle size.

Figure 5.12: The sample of the particle position over a time period of 25,000 sec.

Based on the particle's positions over the total time-steps, the normalized particle concentration along the water depth is shown in Figure 5.13. The theoretical Vanoni distribution for the corresponding Rouse number was likewise plotted in Figure 5.13 to benchmark the simulation results. The normalization of the particle concentration is obtained by dividing the total number of particle occurrence at each location by the total number of particle occurrence at the reference position b from the bed. In this work, the reference position $b = 2d$ was utilized.

Figure 5.13: Suspended load distribution for Rouse number of 1.93

The correlation between the suspended load model simulation results for Case 6 and the corresponding Vanoni distribution plot agreed well with each other as seen in Figure 5.13. The good correlation between the proposed suspended load model and the Vanoni distribution is attributed to the approach taken in deriving the suspended load model as presented as follows.

In the suspended load model, the turbulence kinetic energy (k) is utilized to provide the magnitude of the fluctuating velocity due to the turbulence at that location. In addition, the specific rate of dissipation (ω) provides the time-scale needed for the particle to move out of the current eddy it is in. Hence, assuming that the sediment particle behaves in a similar way to a water particle, both the k and ω will provide sufficient information to determine the particle's motion from eddy to eddy. However, for a sediment particle with a significant volume and mass, the response of the sediment for a given k and ω will be different depending on the sediment properties. To account for the response of the particle, the coefficient of drag is an ideal parameter which will incorporate the inertia of the particle due to the turbulence and velocity fluctuation it is experiencing. In addition, it was identified in this work that a tuning coefficient (C_{sedi}) of $C_{sedi} = 2.1$ was required for the simulation results to agree well with the theoretical distribution.

The simulation Case 6 was for that of a high Rouse number. To validate the proposed suspended load model further, a range of simulations (Case 7 - 10) for both high and low Rouse numbers ($R_o = 1.2, 0.95, 0.795, 0.181$) was performed. The simulation results together with the Vanoni distribution are plotted in Figure 5.14 - 5.16 as follows.

Figure 5.14: Suspended load distribution for Rouse number of 1.20

Figure 5.15: Suspended load distribution for Rouse number of 0.95

Figure 5.16: Suspended load distribution for Rouse number of 0.795

The simulation results showed good correlation with the Vanoni distribution for both high Rouse number (Case 7, $R_o = 1.2$) as well as for low Rouse number (Case 9, $R_o = 0.795$). The range of Rouse number ($R_o = 1.93 - 0.79$) used in this study was typical of that utilized in most experimental works. Such general good agreement between the proposed suspended load model and Vanoni distribution for the range of Rouse number can be attributed to the model's firm grounding on the physical parameters (k , ω , C_D) of the flow and sediment properties.

To determine the generality of the suspended load model, an extremely low Rouse number simulation of 0.181 (Case 10) was performed, whereby the sediment distribution is almost uniform within the flow profile. The results of this simulation is plotted in Figure 5.17. The simulation result of Case 10 shows a general uniform distribution along the water depth, which is in reasonable agreement with that of the Vanoni distribution. While the discrepancy between the simulated results and Vanoni distribution is greater for this extremely low rouse number case as compared to the earlier simulations, this can be attributed to the limitation of the top and bed boundary conditions applied in the Matlab model, whereby particle are reflected along these boundaries.

Figure 5.17: Suspended load distribution for Rouse number of 0.181

The development and validation of the suspended load model presented in this section were performed in Matlab. As mentioned earlier, the suspended load model was likewise implemented into the `interDymSediFoam` solver in OpenFOAM for completeness of this work. The interface (reference height b) between the bed load and suspended load regime was that of a reflective boundary, whereby particles crossing this interface will be reflective back into the transport regime it was originally in. This makes both the bed load and suspended load regimes mass conservative within the simulation domain. Such a reflective boundary interface between the bed load and suspended load model is a reasonable assumption, especially since the simulation domain size is relatively small and where there is minimal interaction between the transportation regimes. It is proposed that in future works, to improve the model further, this reflective interface condition between the two regimes can be further developed to allow the bed load and suspended load regimes to interact with each other, hence allowing suspended load particles to be deposited into the bed load regime and likewise bed load particle to be injected into the suspended load regime. Subsequently, such a model can be further validated by performing a range of clear-water and live-bed simulation studies.

5.5 Conclusion

In this chapter, a RANS-DEM sediment transport model is proposed. The purpose of developing the Lagrangian-base sediment model is to act as a cornerstone work to ultimately develop a Lagrangian-base scour model in the near future. Such a particle-based scour model can provide greater insight into the particle behaviour and mechanisms during the scour process. For Lagrangian-based models, the computational time requirement is typically high, which can make the application of such particle models into a scour model unfeasible. Hence, to reduce the computational time requirement, a computationally less expensive $k - \omega$ (SST) RANS turbulence model is utilized to obtain the flow field. In addition, a thought-experiment was performed to translate the sediment motion and behaviour into mesh morphology using tracer-particles, hence limiting the number of particles utilized in each simulation model.

The proposed RANS-DEM sediment transport model comprises of a few key elements which are given as follows:

- $k - \omega$ (SST) RANS turbulence model to obtain the flow field.
- DEM force balance between the drag and gravitational forces to obtain the particle trajectory.
- Bed load model, which utilizes a similar approach to that proposed by Engelund and Fredsøe (1976) but applied to a Lagrangian framework.
- Suspended load distribution along the water depth obtained using k and ω turbulence properties and a mixing length theory
- Bed morphology obtained by zoning the bed boundary and based on the tracer-particle motion/behavior over Δt , the bed boundary is deformed accordingly

Based on the proposed RANS-DEM sediment transport model, an OpenFOAM solver called the interDymSediFoam was developed and validated in 4 parts. Initially, to validate the implementation of the DEM force balance, a particle settling velocity simulation was performed. It was shown that the simulated settling velocity agreed well with theoretical calculations. Subsequently, the simulation of the bed load model was performed and likewise the simulation results obtained agreed well with that calculated theoretically from Engelund and Fredsøe (1976) equation. Applying the validated bed load model into the proposed bed morphological model, a thought-experiment was performed for the case of scour at a spillway. In the initial

simulation, the particles were prevented from leaving the system to evaluate the erosion and deposition capabilities of the model. The resultant bed morphology resembled that of a longshore groyne field which has a scour mechanism similar to that of the simulation. When repeating the bed morphological simulation for a realistic case whereby the sediments are allowed to leave the system, the scour mechanism observed in the simulation was in agreement with that observed by Breusers (1966, 1967).

For the suspended load model, while it was incorporated into the `interDymSediFoam` solver, the validation of the suspended load model was performed in Matlab due to the large number of time-steps required. Based on the simulations of the suspended load model done in Matlab, it was shown that the proposed suspended load model could reproduce the Vanoni distribution for a full spectrum of Rouse numbers. It is proposed that in future works, the interface between the bed load and suspended load models in the `interDymSediFoam` solver be developed further to allow interaction between the two transportation regimes.

In relation to scour modeling, the primary value of a Lagrangian-based sediment transport model as compared to an Eulerian-based model is that the motion/behaviour of individual particles is captured in a Lagrangian model. Evaluating such particle behaviours can bring new insight into the scour mechanism which is unavailable using Eulerian-based models and difficult to capture in experimental studies. Such models will be especially valuable when modeling scour around complex foundations, where understanding the sediment motion and behaviours around such foundations are more challenging. While there are drawbacks of a Lagrangian-based sediment transport model, such as the higher computational cost and the increased complexity of the model, when adopted to gain insight into the scour mechanism, such models can provide significant additional value as compared to a Eulerian-based sediment transport model. Hence, the RANS-DEM sediment transport model presented here provides the cornerstone for future developments towards a Lagrangian-based scour model. With additional validation studies and improvements, it is envisioned that such a particle-based sediment model will be able to be applied to complex scour cases in the future.

Chapter 6

Scour Model for Industrial Application

6.1 Introduction

The full numerical scour model developed in the earlier chapter has been shown to model the scour process well. While such models have provided much insight into the scour depth evolution and scour mechanism, the computational time required for such a model is significantly high. This results in such models being difficult to implement in practical engineering applications, which typically have a limited time constraint. In this chapter, based on the experience gained from developing a full numerical scour model, a simplified numerical scour model is proposed for industrial application. This model capitalizes on the bed parameters evolution during the scour process and an optimization function to estimate the equilibrium scour depth around a structure within a limited number of iteration. Such a model could potentially fill the gap within the industry for a practical general purpose scour assessment tool.

6.2 Theory

The scour around an offshore structure is primarily due to the changes in the sediment mobility within its vicinity. These changes in sediment mobility are due to the increase in bed shear stress caused by the constriction and amplification of the flow and the increase in turbulence around the structure. As such, these are the key areas to be modeled well in order to predict the equilibrium scour depth.

In a full numerical scour model, as that presented in chapter 4, the approach is typically to (1) model the turbulence flow around the structure, (2) determine stress amplification and Shields parameter on the bed and (3) utilizing these bed parameters as inputs into the sediment transport equations to solve the sediment transport and morphological evolution of the bed. This approach provides a significant amount of detail into the scour process. However, the computational time required is significantly high.

In this work, a hybrid numerical scour modeling approach is proposed to reduce the computational requirement for each simulation. The initial part of the approach is quite similar to the process described above, i.e. first, the turbulence flow around the structure is calculated, then the shear stress amplification and Shields parameters on the bed are determined. Subsequently, instead of determining the sediment transport behaviour and bed morphology, an objective function and mesh optimizer will be used to guide the bed morphology to an equilibrium scour depth of a pre-defined shape. This modified approach reduces the computational requirements of the scour simulation model by estimating the equilibrium scour depth within a limited numbers of iterations. The process flow of the proposed hybrid numerical scour model is shown in Figure 6.1.

Figure 6.1: The process flow of the proposed hybrid numerical scour model.

6.2.1 Turbulence Modeling

To model the turbulence of the flow, a Reynolds Averaged Navier-Stokes approach was utilized together with the $k - \omega$ (SST) turbulence model (Menter, 1994) as the closure model for the Boussinesq approximation. As identified in chapter 3, the $k - \omega$ (SST) model is the superior two-equation turbulence model for bluff body flows, which capitalizes on the accuracy of the $k - \omega$ model within the near-wall region and the $k - \varepsilon$ model in the far-field region. The $k - \omega$ (SST) model has been shown to be able to model a wide range of flow profiles with increased accuracy and is typically used by numerous industries due to its accuracy and reasonable computation requirements. As such, the confidence in utilizing such a model to simulate the turbulence flow around complex structures (e.g. Jacket foundations) is greater, especially within the industry which has significant experience in utilizing it. Hence, the $k - \omega$ (SST) model is used in this work to model the turbulence of the flow. Details of the $k - \omega$ (SST) is described in full in chapter 3 of this dissertation.

6.2.2 Bed Parameter Calculations

Based on the flow determined by the turbulence model, the bed shear stress, Shields parameter, critical Shields parameter and bed slope correction factors are calculated. These parameters are the sediment transport drivers which ultimately results in the scour hole being formed. The approach taken to determine these bed parameters are identical to that utilized for the full numerical scour model presented in chapter 4 earlier. These bed parameter equations are presented as follows for easier referencing and completeness of this chapter.

Bed shear stress

The bed/wall shear stress (τ_{bed}) calculated is determined based on the equation,

$$\tau_{bed} = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0} \quad (6.1)$$

where μ is the dynamic viscosity, u is the flow velocity and y is the distance from the wall surface.

Shields parameter

From the bed shear stress calculated, the Shields parameter (θ) is subsequently determined as,

$$\theta = \frac{\tau_{bed}}{(\rho_s - \rho)gd} \quad (6.2)$$

where ρ_s and ρ are the density of the sediment and fluid respectively, g is the acceleration due to gravity and d is the diameter of the particle.

Critical Shields parameter

The critical Shields parameter (θ_{cr0}) is determined based on the equation proposed by Soulsby and Whitehouse (1997) as follows,

$$\theta_{cr0} = \frac{0.24}{d_*} + 0.055 [1 - \exp(-0.02d_*)] \quad (6.3)$$

where d_* is the dimensionless particle diameter given as,

$$d_* = \left(\frac{\Delta g}{v^2} \right)^{1/2} \quad (6.4)$$

where v is the kinematic viscosity and Δ is $(\rho_s - \rho)/\rho$.

Bed slope correction factor

To account for variation in the bed slope, a bed slope correction factor was calculated using the equation proposed by Roulund et al. (2005) given as follow:

$$\theta_{cr} = \theta_{cr0} \left(\cos \beta \sqrt{1 - \frac{\sin^2 \alpha \tan^2 \beta}{\mu_s^2}} - \frac{\cos \alpha \sin \beta}{\mu_s} \right) \quad (6.5)$$

where θ_{cr0} is the critical Shields parameter for a flat bed, μ_s is the static friction coefficient, α is the angle between the flow velocity vector and direction of the steepest bed slope and β is the angle of bed slope.

6.2.3 Objective Function and Optimizer

Instead of solving a sediment transport equation as done in a full numerical scour model presented in chapter 4, it is proposed that the equilibrium scour depth be predicted using an optimization algorithm. Using the turbulent flow field together with the calculated Shields parameter and the bed slope corrected critical Shields parameter, an objective function (X_{object}) is implemented for the scour model to minimize. An additional constraint on the angle of repose is included to cater for sand-sliding when the bed angle becomes too large. In this work, a Torczon algorithm (Torczon, 1991) is used for the optimization. With the objective function and the constraint on the model, the bed morphology is deformed in a pre-defined direction as the simulation converges to an optimal/equilibrium scour depth using this optimization algorithm.

Objective function

The objective function (X_{object}) used for the simulation is formulated such that the localized Shields stress around the structure is optimal just below the critical Shields parameter given as follows,

$$X_{objective} = |\theta - \theta_{cr}| \quad (6.6)$$

where X_{object} is the objective function, θ is the Shields parameter and θ_{cr} is the critical Shields parameter corrected for bed slope effect.

Angle of Repose

An angle of repose (ϕ_{repose}) constraint was imposed onto the model to maintain the physical behaviour of the sand-slide when the bed angle becomes too steep. The empirical formula by Lim and Cheng (1998), similar to that utilized in chapter 4, was applied to determine the angle of repose as follow,

$$\tan(\phi_{repose}) = 0.457 \left(\frac{\sqrt{gd_{50}}d_{50}}{v} \right)^{0.05} \left(\frac{\rho_s - \rho}{\rho_s} \right)^{0.106} S_p^{-0.8} \sigma_g^{0.12} \quad (6.7)$$

where d_{50} is the mean particle diameter, S_p is the shape factor and σ_g is the sediment geometric standard deviation.

In this work, natural sand of $d_{50} = 0.96mm$ was used, which has an angle of repose of approximately 36° calculated from Equation 6.7. When the angle of repose is exceeded, the objective function (X_{object}) is set to an arbitrarily high value (a value of 0.5 was chosen in this work) so as to prevent convergence of the model into that state.

Torczon Optimizer

For the Torczon optimizer algorithm (Torczon, 1991), it is a direct search method optimization technique to determine the maxima and minima of a system. The working of the Torczon algorithm is illustrated in Figure 6.2. Given a design space where each vertex X_1 , X_2 and X_3 represent the initial objective function evaluation, the algorithm will attempt to find the next best functional value through 3 possible trial steps: (1) Rotation step, (2) Expansion step and (3) Contraction step. With reference to Figure 6.2, assuming vertex X_1 results in the best functional value among vertex X_1 , X_2 and X_3 , the algorithm will start with the rotation step to determine if a better functional value is available, which is given as X_{rot1} and X_{rot2} . If so, an expansion step (X_{exp1} and X_{exp2}) is subsequently performed. If not, a contraction step (X_{con1} and X_{con2}) is performed and accepted. Details of the Torczon optimization algorithm is described fully by Torczon (1991).

Figure 6.2: The visual illustration of the Torczon algorithm (Torczon, 1991) in determining the minimum functional value of the objective function.

6.3 Model Setup

In this chapter, the aim was to develop a scour model for industrial application. To allow for easier adoption by the industry, a commercial software was used as it is generally more popular among practicing engineers as compared to the more complex open-source and in-house codes. In particular, ANSYS-FLUENT was utilized in this work as it is a popular commercial CFD software used by many engineers worldwide. This software has inbuilt capabilities including various turbulence models and mesh morphological models for mesh motion/deformation which are needed in the current simulation. In addition, FLUENT provides a User-Defined Function (UDF) which allow users to write subroutines in the C language to provide customization to the simulations being performed.

In this work, three key models within FLUENT are of primary interest. They are:

- Turbulence models (RANS turbulence models)
- Morphology optimizer models
- User-Define Function (UDF)

As discussed earlier, the concept behind this work is to model the flow around the marine structure using the turbulence model in FLUENT. Once the flow field is obtained, the bed stresses, Shields parameters and bed slope correction factors are calculated via UDF functions which are written in C. Based on these calculations, the objective function is defined and the morphological optimizer model in FLUENT will determine an optimal way the bed boundary will be deformed (in accordance with a pre-defined mesh motion in the direction of the scour hole) and converged to the final equilibrium scour hole. The morphology optimizer in FLUENT provides a fast and easy approach to deform the mesh.

The modeling performed in this chapter is presented in three parts. First, the flow amplification around the monopile structure and within the scour hole is evaluated in greater detail without any attempt to model the scouring itself. The CFD results are compared with the flow experiments by Melville and Raudkivi (1977) to gain confidence in the chosen turbulence model. In the second and third part, the experimental conditions tested by Melville and Chiew (1999) were used for comparison with the simulation results of the proposed scouring model. The second part starts from an initially flat bed, and the bed boundary/scour hole is then manually deformed in intervals of $0.1D$ ($D = 0.1, 0.2, \dots, 1.3D$) to evaluate the bed shear stress amplification evolution around the structure as the scour hole develops towards the equilibrium scour condition. Finally, the Torczon optimizer algorithm (Torczon, 1991) is implemented and simulations were performed to evaluate the model's ability to converge to the equilibrium scour depth condition from the initial flat bed condition.

Mesh geometrical dimensions

For all the simulations performed in this chapter, a hexahedral block mesh with a domain dimension similar to that adopted in chapter 4 was utilized. In the x-y plane, the geometrical dimension was $25D$ by $20D$. The location of the cylinder was positioned $10D$ from the upstream inlet boundary and $15D$ from the downstream outlet. The side boundaries were $10D$ from the center of the cylinder, deemed far enough to have minimal influence on the fluid flow within the region of interest. In the z-direction, a dimension of $3D$ was used. Readers are referred to chapter 4 for more details of the mesh setup, which is identical to that adopted in this chapter.

Boundary conditions

In the first part, the 3 sets of flow fields profiles obtained by Melville and Raudkivi (1977) was used, consisting of an initially flat bed condition, an intermediate scour hole condition, and an equilibrium scour hole condition. As the purpose was to evaluate the simulated flow field and amplification against the measurement data, the sediment size is not relevant in this part. The details of the simulation model setup are presented in Table 6.1.

Table 6.1: Model setup for the flow validation simulation around the structure and within the scour hole

Simulation Name	Flow Velocity (m/s)	Pile Diameter (mm)	Scour Depth (mm)	Boundary Conditions			
				Inlet	Outlet	Top & Sides	Bottom & Cyl.
Flat bed	0.250	50.8	0	Velocity	Outflow	Sym	Wall
Intermediate	0.250	50.8	47.1	Velocity	Outflow	Sym	Wall
Equilibrium	0.250	50.8	61.3	Velocity	Outflow	Sym	Wall

In the subsequent two parts, the simulation model was setup based on experimental conditions (Run numbers N1, N7, and N9) by Melville and Chiew (1999). These runs were selected as they had identical sediment size and flow velocity. The details of the simulation model are presented in Table 6.2 (note the cases selected in this work was identical to that utilized in chapter 4).

Table 6.2: Model setup for the bed shear stress evolution and mesh optimization simulations.

Simulation Name	Flow Velocity (m/s)	Pile Diameter (mm)	Sediment Size (mm)	Boundary Conditions			
				Inlet	Outlet	Top & Sides	Bottom & Cyl.
N1	0.231	70	0.96	Velocity	Outflow	Sym	Wall
N7	0.231	50	0.96	Velocity	Outflow	Sym	Wall
N9	0.231	38	0.96	Velocity	Outflow	Sym	Wall

Numerical scheme

For all the parts of the simulations performed, a SIMPLE algorithm (Patankar and Spalding, 1972) was used for the velocity-pressure coupling and a second order up-wind scheme (Barth and Jespersen, 1989) was used for the spatial discretization.

Mesh sensitivity

A mesh sensitivity study was performed for the first part of the simulation to ensure the results obtained were mesh size independent. Using the initial flat bed and equilibrium scour depth runs, the drag coefficient (C_d) and pressure coefficient (C_{pbase}) downstream of the cylinder were evaluated. The mesh independence was obtained when the difference between the calculated parameters of the different meshes were less than 5%, as shown in Table 6.3 . Based on the mesh sensitivity evaluation, a mesh size of 178020 elements (Mesh 2) was used.

Table 6.3: Mesh sensitivity study for a flat bed and equilibrium scour hole configuration.

Mesh Name	Mesh Size (Element)	% Diff	Flat bed				Equilibrium Scour Hole			
			C_d	% Diff	C_{pbase}	% Diff	C_d	% Diff	C_{pbase}	% Diff
Mesh 1	120708	47.5%	0.225	8.0%	16.91	22.5%	0.283	7.7%	16.25	3.9%
Mesh 2	178020	43.1%	0.207	2.8%	21.83	4.9%	0.261	1.9%	16.91	1.1%
Mesh 3	254800	-	0.213	-	22.96	-	0.266	-	17.11	-

6.4 Results

6.4.1 Flow Amplification Validation

The main force driving the sediment motion during the scour process is the flow amplification and turbulence around a structure. This occurs as a result of the formation of the horseshoe vortex and flow constriction around the pile as discussed earlier. Similar to the validation performed in chapter 3, the experimental analysis and data presented by Melville and Raudkivi (1977) was utilized. As compared to the work presented in chapter 3, in this section, more focus was placed on the validation of the flow amplification around the monopile during the scour process for the flat bed condition, intermediate scour depth, and the final equilibrium scour depth conditions.

Initial flat bed

Figure 6.3: Simulation results and experimental data from Melville and Raudkivi (1977) of the upstream flow field profile (a,b) and the normalized velocity profile around the pile (c,d) for a flat bed.

The simulation profile and normalized flow velocity (flow amplification) for an initially flat bed are shown in Figure 6.3. Based on the simulation results obtained from the $k - \omega$ (SST) turbulence model, the flow field agrees well with the measured flow by Melville and Raudkivi (1977). Similar to that described in chapter 3, upstream of the pile, the vertical pressure gradient along the cylinder results in a downward flow which causes the formation of a horseshoe vortex at the toe of the pile (Figure 6.3a). With the horseshoe vortex and the fluid flowing past the cylinder, a flow constriction occurs at approximately 45° angle to the pile, resulting in the flow velocity acceleration (Figure 6.3c) and a local amplification of the bed shear stress at that location. The maximum flow amplification for both the simulation and experiment was approximately 1.30, hence showing good correlation between the two results. Although there is a slight discrepancy of the location of the maximum flow amplification, it must be noted that the flow around the cylinder is highly turbulent and difficult to measure experimentally. Richardson and Panchang (1998) reported similar discrepancy in the location of the maximum flow amplification between their simulation results using the $k - \varepsilon$ (RNG) turbulence model and the experimental results by Melville and Raudkivi (1977).

The simulations show the ability of the $k - \omega$ (SST) model to capture the flow amplification well, which is one of the key flow features in the scour process. The combined horseshoe vortex together with the flow amplification are the main driving force for the sediment transport around the pile and the development of a scour hole over time. This is consistent with that described by Roulund et al. (2005).

Intermediate and equilibrium scour hole

Figure 6.4: Simulation results and experimental data from Melville and Raudkivi (1977) of the upstream flow field profile (a,b) and the normalized velocity profile around the pile (c,d) for an intermediate scour hole.

For the flow within an intermediate and equilibrium scour hole, good agreement between the simulated and experimental results were likewise observed. Figure 6.4 shows the results for an intermediate scour hole flow profile and normalized flow velocity (flow amplification). Similar to the flat bed flow condition and that described in chapter 3, the pressure gradient in front of the pile together with the downward sloping bed gradient results in a horseshoe vortex formation upstream of the monopile (Figure 6.4a). For the normalized flow velocity, the $k - \omega$ (SST) model indicates an increase in flow velocity from the edge of the scour hole to the toe/edge of the pile, which is similar to that observed experimentally by Melville and Raudkivi (1977). These simulations validate the ability of the $k - \omega$ (SST) model to capture the flow amplification around a monopile.

6.4.2 Scour Hole Evolution

The formation of the horseshoe vortex together with the constriction and acceleration of the flow at the side of the pile results in a significant bed shear stress amplification around the pile. This increases the sediment transport around the pile to eventually form a scour hole. In the present simulation performed for a flat bed condition, the maximum bed shear stress amplification factor was approximately 4-5 as shown in Figure 6.5. This is in agreement with that reported by Hjorth (1975), where he found the amplification factor to be 4-6 times larger than that of an undisturbed flow. In the work by Hjorth (1975), the primary variable utilized in his study was the water depth-pile diameter ratio (h/D). For $h/D = 5$, Hjorth (1975) obtained an amplification of approximately 4, which is close to that obtained in the current simulation. Based on this agreement with experimental data, such an amplification factor obtained from the simulation can be applied for industrial application when determining the shear stress amplification around a structure with a similar configuration as that presented in the simulation.

Figure 6.5: Amplification factor around a pile for a flat bed configuration similar to that performed experimentally by Melville and Raudkivi (1977).

In most previous works (Melville and Raudkivi, 1977; Roulund et al., 2005), the stress amplification analysis is usually done for 2 cases: the flat bed and equilibrium scour hole conditions. Analysis of flow configurations in-between these two extreme conditions is rare. The amplification factor will evolve as the bed morphology changes towards the equilibrium scour hole condition. This evolution of the amplification factor is important especially for clear-water scour, whereby the equilibrium scour depth is reached when the shear stress amplification factor falls to a value such that the Shields parameter is equal or below that of the critical Shields parameter, and consequently net sediment transport stops.

In this section, the evolution of the Shields parameter amplification for an idealized scour hole development is analysed. This analysis is performed using the FLUENT morphology module which smoothens the mesh based on the movement/displacement of selected key points defined by the user. For this work, the node directly adjacent to the pile surface was deformed in intervals of $0.1D$ in the direction of the scour hole up to a final scour depth of $1.3D$. At each interval, a steady-state flow simulation using the $k - \omega$ (SST) turbulence model was performed and the maximum amplification around the pile was determined. The results provide insight into the evolution of the Shields parameter amplification as the scour hole develops. The experimental data selected from Melville and Chiew (1999) for this work are given in Table 6.4. This table shows that the experimental results are generally consistent with the equation proposed by Sumer et al. (1992b) whereby the equilibrium scour depth is approximately $1.3D$.

Table 6.4: Equilibrium scour hole obtained by Melville and Chiew (1999) for run N1, N7 and N9.

Run Number	Pile Diameter, D (mm)	Experiment Scour Depth
N1	70	$1.34D$
N7	50	$1.20D$
N9	38	$1.18D$

Figure 6.6: Simulation results of the evolution of the maximum normalized Shields parameter with increasing scour depth.

As described above, the simulations were carried out with the scour depth increasing at $0.1D$ intervals for the three cases in Table 6.4. The results of the normalized Shields parameter plotted against the normalized scour depth is shown in Figure 6.6. The normalization of the Shields parameter was based on the critical Shields parameter obtained from Equation 6.3. The scour depth was normalized against the pile diameter. It is not surprising to see a downward slope whereby the normalized Shields parameter decreases with increasing scour depth. It was interesting to observe that when the normalized Shields parameter reached a value of 1 (beyond which sediment transport motion should stop), the scour depth were within the range of $0.9D$ - $1.1D$ for the three cases (Figure 6.6).

Initial assessments give the perception that the simulation results of the equilibrium scour depth are slightly shallower than the experimental data (Melville and Chiew, 1999) and empirical formulation which estimates scour depth at $1.3D$ (Sumer et al., 1992b). However, on detailed assessment, it was noted that such discrepancy can be attributed to the vertical height of the simulation domain and the water depth utilized by Melville and Chiew (1999). In the simulation model, a vertical height of $3D$ was used, which limits the size of the horseshoe vortex being formed. Hence given this height limitation, the equilibrium scour depth obtained from em-

pirical formulation can be estimated by utilizing the equation presented by Sumer et al. (2001) given as,

$$\frac{S}{S_0} = 1 - \exp\left(-0.55\frac{h}{D}\right) \quad (6.8)$$

where S_0 is the equilibrium scour depth of infinitely tall piles and h is the pile height. Based on Equation 6.8, the estimated equilibrium scour depth for the cases was 1.05D, which agrees well with that obtained in the current simulation results. This influence of the pile height was likewise observed in the experimental data by Melville and Chiew (1999). For example in the experimental case of N7, the water depth utilized by Melville and Chiew (1999) was 200mm. Applying this water depth to Equation 6.8, this results in a theoretical equilibrium scour depth of 1.15D which likewise agrees well with that obtained experimentally of 1.2D by Melville and Chiew (1999).

While it may not be significant in the current model, it is worth mentioning a possible influence turbulence may have in obtaining a shallower equilibrium scour depth in the current approach. In full scour simulations, the bed morphology changes are determined based on the sediment transport equation. In the present simulation, the sediment transport is omitted and only the sediment driving forces in terms of the Shields parameter are analysed. Hence, the analysis does not account for the influence of turbulence on the sediment motion. Dixen et al. (2013) found that if turbulence is not considered in the sediment transport model of live bed scour, the simulation scour depth is generally shallower. Similar observations was reported by Roulund et al. (2005). Hence, by analysing only the sediment driving forces without considering the turbulence effects on the sediment motion, a shallower scour hole can be expected. Additional work is needed to gain further insight into the quantitative influence of turbulence on the sediment transport to possibly calibrate such analysis in the future.

Based on the analysis above, the potential of determining the equilibrium scour depth by following the evolution of the maximum normalized Shields parameter holds up. It should be noted that while the model used was an idealized symmetrical scour hole, the results obtained should be representative of a realistic scour hole when averaged over time, especially under tidal conditions which typically have symmetrical shapes in the direction of the tidal flow (Jensen et al., 2006).

6.4.3 Scour Depth Optimization

The work done in the earlier section showed the potential for a pathway to determine the equilibrium scour depth based on the evaluation of flow/bed parameters derived from the RANS turbulence simulations. For these earlier simulations, the scour hole evolution was pre-defined in intervals of $0.1D$ up to the equilibrium scour depth. While this has provided a detailed development towards the maximum scour depth, the utilization of an optimization algorithm should shorten the pathway (hence computational time) to evolve from an initial flat bed to its final equilibrium scour depth. In FLUENT 14, five optimization algorithms are available for such simulations; Compass optimizer (Kolda et al., 2003), Simplex optimizer (McKinnon, 1998), Powell optimizer (Press et al., 2007), Rosenbrock optimizer (Rosenbrock, 1960) and Torczon optimizer (Torczon, 1991). In this work, the focus will be on the Torczon optimizer algorithm (Torczon, 1991) described earlier.

Table 6.5: Simulation results of the predicted scour depth utilizing a Torczon optimizer (Torczon, 1991).

Run Number	Pile Diameter (mm)	Experiment Scour Depth (mm)	Simulated Scour Depth (mm)	No. of Iteration
N1	70	93.8	25.1	10
N7	50	60.0	61.9	10
N9	38	44.8	40.7	11

The predicted scour depth based on the optimization simulations together with experimental scour depths from Melville and Chiew (1999) are shown in Table 6.5. For simulations N7 and N9, the final predicted scour depth agrees reasonable (less than 10% deviation) with experimental data. However, for the larger pile (N1), deviation of the simulation result from the experimental data is observed as will be explained in the subsequent paragraphs. To provide further insight into the mechanism of the simulation process as it converges to the optimal scour depth, simulation N9 and N1 are analysed in greater detail.

Figure 6.7: Scour depth evolution of simulation N9 which utilizes a Torczon optimizer (Torczon, 1991).

Figure 6.8: Objective function evolution of simulation N9 which utilizes a Torczon optimizer (Torczon, 1991).

The scour depth and objective function evolution of simulation N9 are shown in Figures 6.7 and 6.8. Starting from a flat bed condition (Iteration 0), the system initially imposes larger perturbations to the scour depth to identify a gross scour depth of the system. This gross depth is achieved within approximately 3-4 iterations, where the scour depth is approximately 48.3 mm (Iteration 3 in Figure 6.7) and the objective function is approximately minimum (Iteration 3-4 in Figure 6.8). Subsequently, the perturbations of the scour depth are reduced as the system attempts to fine tune its equilibrium scour depth search, which is within the region of approximately 36mm - 48mm (see Figure 6.7). This ultimately results in the convergence to a scour depth of 40.7 mm after 11 iterations(see Table 6.5 and Figure 6.7) and this corresponds to the objective function minimum (Iteration 11 in Figure 6.8). A similar mechanism of the simulation system was observed in simulation N7 whereby the scour depth was initially perturbed over large values. Once an approximate equilibrium depth is achieved, the system likewise attempted to fine tune the scour depth search to finally reach a scour depth of 61.9mm (see Table 6.5).

While simulation N7 and N9 was shown to have good correlation with experimental observation, it was less the case for simulation N1. For simulation N1, the deviation in simulation results was 370% shallower than that observed in experiments and highlighted a possible limitation of such an optimization model. In the case of simulation N1, the pile diameter is the largest amount the other cases (N7 and N9), resulting in the highest Reynolds number for simulation N1($Re = 19500$) as compared to simulation N7 ($Re = 12500$) and N9 ($Re = 9500$). It has been shown in earlier works (Roulund et al., 2005) that as Reynolds number increases, the maximum stress amplification around the pile decreases. Hence, it is suggested that this results in the Shields parameter (or the objective function in the case of this optimization system) being less sensitive for larger piles and could explain the poorer performance of simulation N1 as follows.

Figure 6.9: Scour depth evolution of simulation N1 which utilizes a Torczon optimizer (Torczon, 1991).

Figure 6.10: Objective function evolution of simulation N1 which utilizes a Torczon optimizer (Torczon, 1991).

For simulation N1, the evolution of the scour depth and objective function is shown in Figure 6.9 and 6.10. Similar to simulation N7 and N9, for simulation N1, the optimization system attempts to evolve the scour hole to an approximate equilibrium depth initially. However, the less sensitive objective function/Shields parameter results in more iteration (6-7 iterations) to find an approximate equilibrium depth (Iteration 6 in Figure 6.9). Such increased number of iterations is an indication of the lack of convergence of the objective function. The smaller response (Figure 6.10) of the objective function in simulation N1 results in the system being less resistance to minor numerical errors which led the system to find the wrong initial pathway to an approximate equilibrium depth of 24mm(Iteration 6 in Figure 6.9), approximately 3 times smaller than that observed in experiments. The subsequent scour depth evolution (Iteration 7-10) was confined within the proximity of this depth and the system ultimately converged to a scour depth of 25.1mm, which deviates 370% from experimental observation. Hence, simulation N1 shows the potential limitation of the Torczon algorithm(Torczon, 1991).

While simulation N1 showed the pitfalls of this optimization approach, the optimistic results of simulation N7 and N9 showed the potential of such a scour model. For future developments, it is proposed that other optimization algorithm, including more complex decision-making tools like neural networks, could be implemented into FLUENT or other similar codes like OpenFOAM to overcome the current model limitations. The work presented here can act as a framework for future developments in this direction.

6.5 Industrial Applications

During the course of this work, numerous opportunities for personal communication with practising civil and geotechnical engineers within this industry have provided much insight into current engineering practices and the potential of applying numerical advancement into current scour analysis tools. As mentioned earlier, currently the industry relies mainly on empirical formulation and experience when tackling this issue on scouring. This includes the reliance on empirical based guidelines, such as DNV-OS-J101 (DNV, 2010), and obtaining Bathymetry data around similar structures within the vicinity of interest. While these approaches are reasonable for standard shaped structures, significant uncertainties arise for structures where less experience is available (e.g. sub-sea machinery, tidal turbines, large foundations, novel foundation designs). For these structures, whereby experience does not exist and where the structural configuration does not match any empirical model currently available, high safety margins are necessary to mitigate the risk of scour. This may be uneconomical for offshore wind and tidal turbine farms, where a large number of installations are typically required.

While there has been a drive towards the development of advanced full numerical scour models in recent times, adopting such models into industrial application is non-trivial in the short-run. This is due to the high computation cost for each simulation and the uncertainties in the performance of these models in complex environments. With the continual advancement of these models and computational capabilities, the long-term potential to adopt these models into industrial use seems possible. However, a significant amount of work has to be done before such a goal is realized.

Within the shorter term, the potential of using numerical models to provide insight into flow parameters, amplification factors, and flow profiles seem strong. Such numerical models will provide significant insight into the flow, which can be subsequently applied to empirical models or current scour analysis approaches. In this work, the potential of using numerical simulations to derive key scour parameters, such as the flow and shear stress amplification factors, are shown. Capitalizing on FLUENT ability to deform the mesh quickly, the shear stress evolution of the surrounding seabed as it converges to its equilibrium scour depth can be performed quickly. This can provide insight into the flow profile and corresponding scour around a structure during its operational life. In addition, there were efforts in this

work to use an optimization algorithm to derive the estimated equilibrium scour depth rapidly (by minimizing the number of iteration for each simulation). However, more work is needed in enabling the optimization model to increase consistency of the results it derives.

The data obtained from such simplified numerical models can be applied to current scour analysis tools and empirical formulations to determine the potential of scour. When applied to seabed mobility, Whitehouse (1998) provided a process flow of evaluating sediment mobility based on wave, tidal current, sediment properties and structure configuration as reproduced in Figure 6.11. For a structure configuration/installation, Whitehouse (1998) proposed the use of a stress amplification factor (M) as the key parameter to determine sediment mobility. He suggested using either empirical or numerical approaches to determine M . However, engineers tend to rely mainly on the empirically derived M values for different structural configurations (Hjorth, 1975; Sumer and Fredsøe, 1992; Baker, 1979; Hebsgaard et al., 1994), which may result in ambiguity for less standard structures. For the latter, and when ambiguity arises in the M value as a result of experimental variation, the potential of using numerical modeling to obtain M and provide insight into the flow profile around the structure is crucial.

When evaluating the scour depth around a structure, Whitehouse (1998) proposed the use of a generalized approach proposed by Breusers et al. (1977) as follows,

$$\frac{S}{D} = f_1 \left(\frac{\bar{U}}{\bar{U}_c} \right) \left[2.0 \tanh \left(\frac{h}{D} \right) \right] f_2(shape) f_3 \left(\alpha_{angle}, \frac{l}{b} \right) \quad (6.9)$$

where f_1 relates to the flow, f_2 is related to the shape factor and f_3 relates to the pier and angle of attack (α_{angle}). In this empirical formulation, f_2 considers the influence of the shape on the amplification of the flow and where numerical modeling is able to provide greater insight, especially for complex structures.

The approach of using simplified numerical models to enhance current scour models can potentially help to advance scour prediction significantly in the short-run. This is especially so as the industry has been using similar numerical models on other applications and does have the confidence in such modeling tools. In addition, merging empirical scour tools with numerical models allows engineers to leverage on the experience they have using the empirical approaches while still gaining insight into the flow physics and scour mechanism based on the numerical models.

Figure 6.11: The sedimentary mobility flow chart presented by Whitehouse (1998).

6.6 Conclusion

Currently, scour analysis performed by practicing geotechnical engineers are typically based on empirical equations, experience and incorporating high safety margins to mitigate risk. While over designing for scour was feasible in conventional oil and gas structures and bridge installations, this is less true for offshore wind farms which have large number of foundations and is typically working on a tighter budget. Hence, there is a need for more accurate scour prediction tools that will provide the industry with greater confidence of their installation while still keeping cost at a minimum. With the rapid development of computational capabilities, there has been an interest in developing numerical scour models for offshore structures. While applying a full numerical scour model (as presented in chapter 4) into engineering practice is still at present impractical, using numerical methods to gain better insight into the turbulent flow around the structure and resultant stress amplifications has great potential in advancing scour assessment tools further.

In this section, a commercial CFD software called ANSYS-FLUENT is utilized which is capable of modeling the turbulent flow, deform the mesh smoothly and has user-defined features which enable certain degrees of customization. By deforming the scour hole mesh manually at $0.1D$ intervals ($D = 0.1, 0.2, \dots, 1.3D$), it was demonstrated that the Shields parameter in the region around a monopile typically reaches the critical Shields parameter at the scour depth of around $0.9D - 1.1D$. This predicted scour depth agrees well with experimental work done by Melville and Chiew (1999) and empirical equations by Sumer et al. (1992b), especially when factoring in the effect of the vertical height of the domain and water depth.

With this clear evolution of the Shields parameter towards an equilibrium scour depth, attempts were made to use an optimizer function in FLUENT to predict the final scour depth around a structure. Based on the Torczon optimization simulation performed, the results have shown that there is potential of using an optimization algorithm to determine the equilibrium scour depth around a structure. However, an outlying simulation, N1, highlighted a potential limitation of such an optimization model. As observed in simulation N1, when the respond of the objective function is reduced, the optimization system can result in taking a wrong pathway, which would finally lead to an equilibrium scour depth result which deviates significantly from experimental observation. It is proposed that future works should be aimed

at identifying additional optimization models which can improve the robustness of this model further.

The present study focused on predicting numerically the maximum scour depth of a uniform monopile. The results showed that the concept of using hydrodynamics and a mesh deformation module could predict scour depth with reasonable agreement with experiments. The reliability of the model in its current form however may not be sufficiently high for immediate industrial adoption due to the limitations of the optimization model as discussed earlier. Hence, moving forward, more work needs to be done in implementing and testing additional optimization tools to improve the reliability of this proposed scour model further. In addition, this concept has to be applied to other structural configurations, such as a jacket structure and a gravity base foundation, to evaluate the robustness and reliability of the model before adopting it for industrial applications.

Chapter 7

Conclusions

The scouring around a foundation is a complex process which encompasses a variety of physical concepts ranging from turbulence flow around the foundation to the behaviour and transportation of the sediments in the surrounding environment. Such scour processes are typically analysed empirically due to its complex nature. However, with the advancement of computational capabilities, the possibility of modeling such scour processes numerically is coming increasingly feasible. Such a numerical scour model could complement earlier experimental works to provide greater insight into the scour mechanism. In addition, such models could provide scour prediction estimates for structural configurations which lack empirical data or experience (e.g. new foundations design).

7.1 Thesis Summary

In this work, a first principle approach to model the scour process around a foundation utilizing numerical methods was proposed to be developed. To achieve this objective, the current work was conducted in 3 stages which focuses on (1) the turbulence modeling, (2) developing a full numerical scour model and (3) developing a Lagrangian-based sediment transport model. In addition, to provide a practical scour assessment tool for the industry (which requires low computational time), a simplified numerical scour model utilizing an optimization model was likewise proposed. The overview of the work done in this dissertation is presented in Figure 7.1.

Figure 7.1: The overview of the work done in the current thesis.

Turbulence model

The flow profile around a structure is the key driving force which results in sediments getting transported away from the foundation and the formation of a scour hole. As such, in the initial stage of the project, the abilities of the one- and two-equation RANS turbulence model to predict the flow past a bluff body (cylinder) was evaluated. It was identified that for such flow profiles, the two-equation $k - \omega$ (SST) turbulence model was superior to the other RANS closure models. This was primarily due to the ability of the $k - \omega$ (SST) to model the flow separation, which subsequently corresponded to the other flow parameters (C_D , C_{Lrms} , St) being modeled well.

In this early stage of the work, it was anticipated that the computational requirements for a scour model would be high. Hence, to bring down the computational time, the possibility of tuning a cost efficient one-equation Spalart-Allmaras model was proposed. It was identified that by tuning the rotational correction coefficient (C_{prod}) to 1.0 and 3.5 for two-dimensional and three-dimensional flows respectively, the Spalart-Allmaras model was able to capture the flow around a cylindrical bluff body reasonably well.

Subsequently, the $k - \omega$ (SST) and Spalart-Allmaras model with $C_{prod} = 3.5$ was applied to a scour hole simulation and the results were benchmarked against experimental data by Melville and Raudkivi (1977). It was observed that both the $k - \omega$ (SST) and Spalart-Allmaras model with $C_{prod} = 3.5$ had reasonable agreement with experimental data (Melville and Raudkivi, 1977). Moving forward in the development of the numerical scour model, both the $k - \omega$ (SST) and the Spalart-Allmaras turbulence model was utilized to model the turbulent flow field. By applying both turbulence models, the ability of utilizing the Spalart-Allmaras model for such complex scour simulations can be evaluated better.

Numerical scour model

As mentioned earlier, based on the turbulence model study, the $k - \omega$ (SST) and Spalart-Allmaras model with $C_{prod} = 3.5$ was applied to the development of a numerical scour model. In this numerical scour model, the turbulent flow was modeled using the $k - \omega$ (SST) and Spalart-Allmaras model, the sediment transport was modeled using a bed load equation proposed by Engelund and Fredsøe (1976) and the

bed morphology was determined using the Exner equation. This approach taken to model the scour process was implemented into an OpenFOAM solver termed the interDymScourFoam. The simulation results for the clear-water scour cases for both the $k - \omega$ (SST) and Spalart-Allmaras model showed good correlation with the scour depth-time development equation proposed by Sumer et al. (1992a). It was interesting to note that the simulation model was able to reproduce the fine details of the scour initiation mechanism, whereby the scour propagates upstream before moving downstream. Such a mechanism is typically omitted in experimental observations due to its rapid occurring nature and this highlights the advantage of such a numerical scour model.

The focus of this work was on the scour initiation process/mechanism. Based on this initiation process, the simulation model scour depth-time development and mechanism correlated well with empirical models and experimental observations, validating the model. One of the limitations of the interDymScourFoam solver was its inability to perform parallelization in its simulation. As a result, the equilibrium scour depth was not obtained due to the limited computational resources available. It is proposed that in future works, the parallelization issue of the interDymScourFoam solver be tackled.

RANS-DEM sediment transport model

In the earlier developed numerical scour model, the sediments were transported based on a bed load transport model (Engelund and Fredsøe, 1976) implemented on an Eulerian framework. While such an approach models bed morphology well, the sediment particle behaviour during the scour process is loss. Hence, to provide foundational work for a future full Lagrangian-based scour model, a RANS-DEM sediment transport model was proposed and an interDynSediFoam solver was developed in this work. In this proposed model, a RANS turbulence model was utilized to model the flow and a DEM force balance between the drag and gravitational force was used to determine each sediment particle trajectory. Since a RANS turbulence model was utilized, when modeling the bed load and suspended load, slight modifications for each transportation regime was proposed. For the bed load model, the flow velocity/force (from the fluid on the particle) is obtained from the friction velocity calculated at the bed boundary which is subsequently mapped into the in-

ternal bed load region. Based on the friction velocity and the probability of bed load motion, the mean bed load velocity ($U_{b(PEF)}$) was obtained and was shown to have good correlation with that calculated using Engelund and Fredsøe (1976) theoretical equation. For the suspended load, the key feature of this regime was the sediment distribution within the water depth (Vanoni distribution). To model this distribution, the fluctuating velocities on the sediment particles are redetermined using the k and ω turbulence properties and a mixing length theory. Based on the proposed suspended load model, it was demonstrated that the model was able to reproduce the theoretical Vanoni distribution for both high and low Rouse numbers.

Based on the bed load model and sediment behaviour, a thought-experiment was performed in an attempt to translate the bed load sediment behaviour/motion to bed morphology. This bed morphological model proposed was obtained by allowing tracer-particles to move within and between zones on the bed boundary over a morphology time-step. Based on the initial and final position of these bed load particles, the bed boundary is deformed and the process is repeated till an equilibrium morphology is reached. The bed morphology model was evaluated based on two test cases, (1) a spillway where particles are not allowed to leave the domain and (2) a spillway where particles are allowed to leave the domain. For both simulations, the bed morphing mechanism had reasonable agreement with field and experimental observations. However, additional work is needed for this morphological model in regards to the scour depth achieved. The model proposed in this work will act as the cornerstone for future developments to eventually derive a full particle-based scour model in the future.

Industrial scour model

The numerical scour model developed earlier (interDymScourFoam) provides much details into the scour mechanism. However, the computational requirements for such models can be significant, which makes it difficult to be implemented in practical engineering cases where time constraints can be significant. Hence, a simplified numerical scour model utilizing the commercial software called FLUENT was developed. It was demonstrated that by evaluating the Shields parameter around the pile as the scour depth increases, the normalize Shields parameter reaches approximately 1 when the equilibrium scour depth is obtained. Hence, utilizing this concept and

a morphology optimizer (Torczon optimization), simulations were performed to determine the ability of such an optimization model to predict the equilibrium scour depth around a monopile within a limited time frame. Based on the simulation results, two test cases obtained excellent agreement between the achieved equilibrium scour depth and that observed in experiments. However, for one case, the estimated scour depth deviated significantly from experimental observations. It is suggested that the utilization of alternative optimization algorithm could overcome the limitation of the current proposed model. With an improved and more reliable optimization algorithm, such a numerical scour model could be readily adopted by the industry as a practical scour assessment tool. This will be especially valuable for new foundational structures, where limited data and experience are available.

7.2 Significant Contributions

Modeling the complex scouring process is a multi-disciplinary problem which requires a combination of turbulence modeling, sediment modeling and numerical methods. In this work, advancement had to be made in each of these disciplines in order to realize the objective of obtaining a first principle scour model. These advancements made in each aspect are described as follows.

Turbulence model

In previous studies on turbulence modeling around a bluff body, these works were typically limited to lower Reynolds numbers. Understanding the ability of such turbulence models to simulate higher Reynolds number flows are more critical in recent times as offshore structures are coming increasingly bigger (offshore wind turbine foundations up to 8m in diameter). In this work, it was demonstrated that for high Reynolds number flow around bluff bodies, the $k-\omega$ (SST) was the superior RANS turbulence model due to its ability to model flow separation. Such insight into the performance of the $k-\omega$ (SST) turbulence model will give engineers confidence when applying such a model on large offshore structures.

For large bluff bodies, the computational requirements for such a flow simulation can be significant, even when utilizing a two-equation RANS turbulence model. Hence, it was proposed in this work that a cost efficient one-equation Spalart-Allmaras model was able to simulate the flow around a bluff body by tuning the rotational correction coefficient (C_{prod}). It was demonstrated that by applying the $C_{prod} = 1.0$ and 3.5 for two-dimensional and three-dimensional flows respectively, such a one-equation turbulence model could be applied for bluff body flows. Such a cost efficient turbulence modeling tool will provide engineers with more modeling options when faced with a significant time constraint.

Numerical scour model

In this work, an OpenFOAM solver called the interDymScourFoam solver was developed for scour modeling around offshore structures. This solver was developed on top of a pre-existing interDyMFoam solver, which is a multiphase flow solver that has dynamic meshing capabilities. It was shown that the scour simulation obtained from the interDymScourFoam solver could reproduce the fine details of the scour initiation mechanism well, whereby the scour hole propagates upstream before moving downstream. Likewise, the scour depth-time development obtained from the simulation agreed well with the empirical equation proposed by Sumer et al. (1992a). The interDymScourFoam solver developed is one of the limited number of scour models (Roulund et al., 2005; Baykal et al., 2015; Stahlmann, 2013; Zhao et al., 2010) which have been developed in recent times.

Based on the findings of the turbulence model study performed earlier, both the $k - \omega$ (SST) and the Spalart-Allmaras turbulence model with $C_{prod}=3.5$ were applied to the interDymScourFoam solver. For the Spalart-Allmaras model, this is the first time such a one-equation turbulence model is applied to such a complex bluff body simulation. It was encouraging to note that the performance of the Spalart-Allmaras turbulence model was on par with the $k - \omega$ (SST) model for the scour simulations. With the current good performance of the Spalart-Allmaras model in simulating such a complex scour process, future scour models could apply such a cost efficient turbulence models to reduce the computational time requirements of such a simulation.

RANS-DEM sediment transport model

It is envisioned that in the near future, the sediment transport in a numerical scour model can be modeled at a particle-scale level, whereby the behaviour of individual sediment particles during the scour process can be determined. Hence a RANS-DEM sediment transport theory was proposed and the interDymSediFoam solver was developed in this work to act as the cornerstone for future developments.

The RANS-DEM sediment transport model successfully modeled the particle behaviour of both the bed load and suspended load using a RANS turbulence model. The bed load model was adapted from an approach proposed by Engelund and Fredsøe (1976), but applied to a Lagrangian framework. For the suspended load model, the particle fluctuating behaviour and distribution was redetermined utilizing the k and ω turbulence properties and a mixing length theory. Both the bed load and suspended load simulations showed good agreement with theoretical equations. Based on the behaviour of the sediment particles, a bed morphological model was proposed which utilizes tracer-particle movement along the bed boundary to determine changes in the bed height. It was shown that based on this thought-experiment, such a morphological model could reproduce scour mechanism for a spillway case study. Hence, with the proposed RANS-DEM sediment transport model which utilizes a RANS turbulence model and a limited number of particles, the computational time is kept low. This provides for a low-cost particle-based sediment transport model which could be applied to future scour models.

Industrial scour model

The full scour simulation model provides much detail into the scour process but results in high computational time making it difficult to be applied in engineering practice. Hence, in this work, a simplified numerical scour model was proposed which could be used as a scour assessment tool by the industry. It was demonstrated that the evolution of the normalized Shields parameter converges to 1 as the scour depth reaches its equilibrium depth. Hence, utilizing the normalized Shields parameter as an objective function, it was shown that a Torczon optimization model could estimate the equilibrium scour depth within a limited number of iteration.

While there are limitations to the Torczon optimization model, it is suggested that other optimization models could be utilized in future works. Such simplified numerical scour models will be significantly beneficial to engineers when evaluating scour around new foundational design, where there is limited or no data available and where experience in these designs are likewise limited.

Industrial impact

The development and research into numerical scour modeling, as that done in this work, will have significant impact on the offshore industry. This is because such tools can provide engineers with more cost efficient approaches (non-experimental approach) to obtain accurate scour assessments on foundational designs early in the design stage, hence reducing the need for subsequent costly re-works. In addition, such numerical models can provide insight into sediment behaviour/bed morphology changes during the scour process, which will enhance the engineer's capabilities in making improved scour assessment analysis. Obtaining these improved scour assessments can lead to significant cost savings, especially for sectors such as the offshore wind industry which requires the installation of a large number of foundations. While there is still a significant amount of work that has to be done before such numerical scour models are ready for industrial application, the work in this thesis will contribute to its incremental development towards this direction.

7.3 Future Works

The current work done has contributed much insights into various aspects of scour modeling (e.g. turbulence modeling, sediment transport). This has led to the development of two OpenFOAM solvers (interDymScourFoam and interDymSediFoam) and one simplified numerical scour assessment tool. While this work has laid the foundation for a first principle scour model, additional work has to be done before such a model is fully realized.

Numerical scour model

The interDymScourFoam solver has been shown to have the capability to model clear-water scour well. However, two main limitations of the solver/model are its inability to be parallelized and the mesh instability for live-bed scour. It is critical that these issues have to be addressed through improvements made to the interDymScourFoam solver to ensure the generality and practically (shorter computational time) of such a model.

In addition, in the current work, the focus was on scour due to uniform current only, with no wave and tidal effects. Such wave and tidal effects can influence the ultimate scour depth around a structure (Sumer et al., 1992b) . There are currently few works on numerically modeling scour due to waves and tidal effects (Fuhrman et al., 2014). This is primarily due to the difficulties in modeling the free surface for the waves and the flow velocity variability for tidal flows. Addressing such influences in future scour models will advance numerical scour modeling significantly as such influences are typically experienced by most offshore structures installed currently.

Particle modeling

The RANS-DEM sediment transport model proposed in this work provides the foundational work needed in the development of a Lagrangian-based scour model. The OpenFOAM solver developed for this work (interDymSediFoam) incorporated the bed load, suspended load and bed morphological model. However, only the bed load and bed morphological model were validated using the solver while the suspended load was validated using Matlab. Hence, developing the interDymSediFoam solver further, it is suggested that the interaction of the suspended load model and the bed load model be developed, and a full live-bed scour validation simulation be performed.

For a spillway scour, this work has shown that the model developed was able to reproduce the scouring mechanism for this relatively simple case. However, additional work is needed to develop the morphological model future for it to obtain better scour depth predictions. In addition, it is proposed that moving forward, more challenging cases, such as scour at a sluice gate and around abutments, be modeled before applying it to a complex case of scour around a foundation.

Industrial scour assessment tool

The simplified numerical scour model developed in FLUENT has provided a roadmap of an approach to develop numerical scour assessment tools for the industry. It has been shown that with the Torczon optimization model, the equilibrium scour depth can be estimated for the majority of the cases. However, limitations to the model do exist and it is suggested that alternative optimization models (e.g. neural networks) could be utilized to improve the reliability of such an approach further. In addition, to gain more insight into the proposed model, simulations on other foundation configurations (e.g. Jacket structure) utilizing this simplified scour model could be done to evaluate the model performance for more complex structures.

The landscape of the contributions of this work and the proposed future works are shown in Figure 7.2. While the current work done has advanced the progression of numerical scour modeling, there is still a significant amount of additional work that is required before a full first principle scour model is ultimately realized.

Figure 7.2: The landscape of the thesis contribution and the proposed future works.

7.4 Final Comments

With the advancement in computational capabilities, developing numerical models for complex processes, such as scouring, is increasingly coming feasible. Furthermore, from an industrial perspective, numerical simulations are likewise increasingly coming a common tool used in engineering design and assessments. These progressions have motivated this work in the development of numerical scour models for offshore structures. Based on this work, a full numerical scour model and a simplified scour model have been developed. In addition, a particle-based sediment transport model has been proposed for future implementation into more complex scour models. These proposed sediment transport and scour models act as the foundational works for future advance scour models to be built upon. With the rapidly increasing development of numerical modeling, it is the author's opinion that such numerical scour models will be a customary tool when assessing scour around offshore structures in the future. With the contribution from this work, as well as future projects on numerical scour modeling, such a vision for a robust numerical scour model being commonly applied to engineering assessments could be realized within the next decade.

References

- Achenbach, E. (1968). Distribution of Local Pressure and Skin Friction Around a Circular Cylinder in Cross-Flow up to 5×10^6 . *Journal of Fluid Mechanics*, 34(4):625–639.
- Achenbach, E. (1971). Influence of Surface Roughness on the Cross-Flow Around a Circular Cylinder. *Journal of Fluid Mechanics*, 46(2):321–335.
- Alam, M. S. and Cheng, L. (2010). A Parallel Three-Dimensional Scour Model to Predict Flow and Scour Below a Submarine Pipeline. *Central European Journal of Physics*, 8(4):604–619.
- Allmaras, S. R. and Johnson, F. T. (2012). Modifications and Clarifications for the Implementation of the Spalart-Allmaras Turbulence Model. In *Proceeding of the 7th International Conference on Computational Fluid Dynamics*, Kohala Coast, Hawaii.
- ANSYS (2011). ANSYS-FLUENT User’s Guide. Technical report, ANSYS Inc.
- Baker, C. J. (1979). The Laminar Horseshoe Vortex. *Journal of Fluid Mechanics*, 95(2):347–367.
- Barth, T. J. and Jespersen, D. C. (1989). The Design and Application of Upwind Schemes on Unstructured Meshes. In *Proceeding of the 27th Aerospace Sciences Meeting*, Reno, Nevada.
- Bateni, S. M., Borghei, S. M., and Jeng, D. S. (2007). Neural Network and Neuro-Fuzzy Assessments for Scour Depth Around Bridge Piers. *Engineering Applications of Artificial Intelligence*, 20(3):401–414.

- Baykal, C., Sumer, B. M., Fuhrman, D. R., Jacobsen, N. G., and Fredsøe, J. (2015). Numerical Investigation of Flow and Scour Around a Vertical Circular Cylinder. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, 373(2033).
- Behr, M., Hastreiter, D., Mittal, S., and Tezduyar, T. E. (1995). Incompressible Flow Past a Circular Cylinder: Dependence of the Computed Flow Field on the Location of the Lateral Boundaries. *Computer Methods in Applied Mechanics and Engineering*, 123:309–316.
- Breusers, H. N. C. (1966). Conformity and Time Scale in Two-Dimensional Local Scour. In *Proceeding of the Symposium on Model and Prototype Conformity*, Poona, India.
- Breusers, H. N. C. (1967). Time Scale of Two-Dimensional Local Scour. In *Proceeding of the 12th Congress of the International Association of Hydraulic Research*, Fort Collins, Colorado.
- Breusers, H. N. C., Nicollet, G., and Shen, H. W. (1977). Local Scour Around Cylindrical Piers. *Journal of Hydraulic Research*, 15(3):211–252.
- Breusers, H. N. C. and Raudkivi, A. J. (1991). *Scouring: Hydraulic Structures Design Manual Series*, volume 2. CRC Press.
- Brørs, B. (1999). Numerical Modeling of Flow and Scour at Pipelines. *Journal of Hydraulic Engineering*, 125(5):511–523.
- Byrne, B. W. and Houlsby, G. T. (2003). Foundations for Offshore Wind Turbines. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, 361(1813):2909–2930.
- Catalano, P., Wang, M., Iaccarino, G., and Moin, P. (2003). Numerical Simulation of the Flow Around a Circular Cylinder at High Reynolds Numbers. *International Journal of Heat and Fluid Flow*, 24(4):463–469.
- Cebeci, T. and Bradshaw, P. (1977). *Momentum Transfer in Boundary Layers*. Hemisphere Publishing Corporation, Washington DC.

- Cheng, N. S. (1997). Simplified Settling Velocity Formula for Sediment Particle. *Journal of Hydraulic Engineering*, 123(2):149–152.
- Chopard, B., Masselot, A., and Dupuis, A. (2000). A Lattice Gas Model for Erosion and Particles Transport in a Fluid. *Computer Physics Communications*, 129(1):167–176.
- Dacles-Mariani, J., Zilliac, G. G., Chow, J. S., and Bradshaw, P. (1995). Numerical/Experimental Study of a Wingtip Vortex in the Near Field. *AIAA Journal*, 33(9):1561–1568.
- Dargahi, B. (1989). The Turbulent Flow Field Around a Circular Cylinder. *Experiments in Fluids*, 8(1):1–12.
- De-Hauteclocque, G., Dix, J., Lambkin, D., and Turnock, S. (2007). Flow and Likely Scour Around Three-Dimensional Seabed Structures Evaluated using RANS CFD. Technical Report Ship Science Report 144, University of Southampton.
- DHI (2012). Predicting Scour in Offshore Wind Turbines. <http://www.dhigroup.com/media/Publications/CaseStories/08/Predicting-scour-in-offshore-wind-turbines-now-a-breeze-DHI-case-story.ashx>, Last accessed on 15 Aug 2015.
- Dixen, M., Lohmann, I., and Christensen, E. (2012). Method To Predict Long Time Span of Scour Around Offshore Wind Turbine Foundations. In *Proceedings of the 33rd Conference on Coastal Engineering*, Santander, Spain.
- Dixen, M., Sumer, B. M., and Fredsøe, J. (2013). Numerical and Experimental Investigation of Flow and Scour Around a Half-Buried Sphere. *Coastal Engineering*, 73:84–105.
- DNV (2010). Design of Offshore Wind Turbine Structures. Technical Report DNV-OS-J101, Det Norske Veritas.
- Dong-Energy (2015). First Foundation Successfully Installed. <http://www.dongenergy.com/en/media/newsroom/news/articles/first-foundation-successfully-installed1>, Last accessed on 15 Aug 2015.

- Draper, L. (1991). Wave Climate Atlas of the British Isles. Technical Report OTH 89-303, HMSO, London, United Kingdom.
- Dupuis, A. and Chopard, B. (2002). Lattice Gas Modeling of Scour Formation Under Submarine Pipelines. *Journal of Computational Physics*, 178(1):161–174.
- Elghobashi, S. E. and Abou-Arab, T. W. (1983). A Two-Equation Turbulence Model for Two-Phase Flows. *Physics of Fluids*, 26(4):931–938.
- Engelund, F. and Fredsøe, J. (1976). A Sediment Transport Model for Straight Alluvial Channels. *Nordic Hydrology*, 7(5):293–306.
- Escarameia, M. (1998). Laboratory Investigation of Scour Around Large Structures in Tidal Waves. In *Proceeding of 3rd International Conference on Hydrosience and Engineering*, Berlin, Germany.
- Escauriaza, C. and Sotiropoulos, F. (2011). Lagrangian Model of Bed-Load Transport in Turbulent Junction Flows. *Journal of Fluid Mechanics*, 666:36–76.
- Fingersh, L. J., Hand, M. M., and Laxson, A. S. (2006). Wind Turbine Design Cost and Scaling Model. Technical Report TP-500-40566, National Renewable Energy Laboratory.
- Fredsøe, J. and Deigaard, R. (1992). *Mechanics of Coastal Sediment Transport*, volume 3. World Scientific.
- Fuhrman, D. R., Baykal, C., Sumer, B. M., Jacobsen, N. G., and Fredsøe, J. (2014). Numerical Simulation of Wave-Induced Scour and Backfilling Processes Beneath Submarine Pipelines. *Coastal Engineering*, 94:10–22.
- Fuhrman, D. R., Sumer, B. M., and Fredsøe, J. (2011). Roughness-Induced Streaming in Turbulent Wave Boundary Layers. *Journal of Geophysical Research*, 116(C10).
- Heald, J., McEwan, I., and Tait, S. (2004). Sediment Transport Over a Flat Bed in a Unidirectional Flow: Simulations and Validation. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, 362(1822):1973–1986.

- Hebsgaard, M., Ennemark, F., Spangenberg, S., Fredsøe, J., and Gravesen, H. (1994). Scour Model Tests with Bridge Piers. *International Association of Navigation Congresses, PIANC*, Bulletin No. 82:84–92.
- Hjorth, P. (1975). *Study on the Nature of Local Scour*. Bulletin Series A, Issue 46, Department of Water Resources, Lund Institute of Technology, Lund, Sweden.
- Hossein Kazeminezhad, M., Yeganeh-Bakhtiary, A., Etemad-Shahidi, A., and Baas, J. (2011). Two-Phase Simulation of Wave-Induced Tunnel Scour Beneath Marine Pipelines. *Journal of Hydraulic Engineering*, 138(6):517–529.
- Issa, R. I. (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting. *J. Comput. Phys.*, 62(1):40–65.
- James, W. D., Paris, S. W., and Malcolm, G. N. (1980). Study of Viscous Cross-flow Effects on Circular Cylinders at High Reynolds Numbers. *AIAA Journal*, 18(9):1066–1072.
- Jemcov, A. and Maruszewski, J. P. (2009). Shape Optimization Based on Downhill Simplex Optimizer and Free-Form Deformation in General Purpose CFD Code. In *17th Annual Conference of CFD Society of Canada*, Ontario, Canada.
- Jensen, M. S., Larsen, B. J., Frigaard, P., De Vos, L., Christensen, E. D., Hansen, E. A., Solberg, T., Hjertager, B. H., and Bove, S. (2006). Offshore Wind Turbines Situated in Areas with Strong Currents. Technical report, Offshore Center Denmark.
- Jones, G. W., Cincotta, J. J., and Walker, R. W. (1969). Aerodynamic Forces on a Stationary and Oscillating Circular Cylinder at High Reynolds Numbers. Technical Report R-300, National Aeronautics and Space Administration.
- Jones, J. S. and Sheppard, D. M. (2000). Scour at Wide Bridge Piers. In *Proceedings of the Joint Conference on Water Resources Engineering and Water Resources Planning and Management*, Minneapolis, Minnesota.
- Kader, B. (1981). Temperature and concentration profiles in fully turbulent boundary layers. *International Journal of Heat and Mass Transfer*, 24(9):1541–1544.

- Katsui, H. and Toue, T. (1992). Bottom Shear Stress in Coexistent Field of Superimposed Waves and Current and Scouring Around a Large-Scale Circular Cylinder. *Costal Engineering in Japan*, 35(1):93–110.
- Kidanemariam, A. G. and Uhlmann, M. (2014). Direct Numerical Simulation of Pattern Formation in Subaqueous Sediment. *Journal of Fluid Mechanics*, 750:R2.
- Kolda, T. G., Lewis, R. M., and Torczon, V. (2003). Optimization by Direct Search: New Perspectives on Some Classical and Modern Methods. *SIAM Review*, 45(3):385–482.
- Lagasse, P. F., Clopper, P. E., Zevenbergen, L. W., and Girard, L. G. (2007). Countermeasures to Protect Bridge Piers from Scour. Technical Report NCHRP Report 593, Transportation Research Board.
- Launder, B. E. and Spalding, D. B. (1972). *Lectures in Mathematical Models of Turbulence*. Academic Press, London, England.
- Launder, B. E. and Spalding, D. B. (1974). The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, 3(2):269–289.
- Liang, D., Cheng, L., and Li, F. (2005). Numerical Modeling of Flow and Scour Below a Pipeline in Currents: Part II. Scour Simulation. *Coastal Engineering*, 52(1):43–62.
- Lim, S. Y. and Cheng, N. S. (1998). Prediction of Live-Bed Scour at Bridge Abutments. *Journal of Hydraulic Engineering*, 124(6):635–638.
- London-Array (2011). London Array - The World’s Largest Offshore Wind Farm. http://www.londonarray.com/downloads/london_array_brochure.pdf, Last accessed on 13 Aug 2015.
- Loth, E. (2000). Numerical Approaches for Motion of Dispersed Particles, Droplets and Bubbles. *Progress in Energy and Combustion Science*, 26(3):161–223.
- Mao, Y. (1987). The Interaction Between a Pipeline and an Erodible Bed. Technical Report Series Paper 39, Technical University of Denmark.

- McKinnon, K. I. M. (1998). Convergence of the Nelder–Mead Simplex Method to a Nonstationary Point. *SIAM Journal on Optimization*, 9(1):148–158.
- McNeilan, T. and Smith, K. R. (2011). Seabed Scour Consideration for Offshore Wind Development on the Atlantic OCS. Technical Report 656, Fugro Atlantic, Norfolk, Virginia.
- Melling, G., Dix, J., Turnock, S., and Whitehouse, R. (2011). CFD-Based Methods for Numerical Modelling of Scour. In *Proceeding of Numerical Towing Tank Symposium*, Poole, Great Britain.
- Melville, B. W. and Chiew, Y. M. (1999). Time Scale for Local Scour at Bridge Piers. *Journal of Hydraulic Engineering*, 125(1):59–65.
- Melville, B. W. and Coleman, S. E. (2000). *Bridge Scour*. Water Resources Publications.
- Melville, B. W. and Lim, S. Y. (2013). Scour Caused by 2D Horizontal Jets. *Journal of Hydraulic Engineering*, 140(2):149–155.
- Melville, B. W. and Raudkivi, A. J. (1977). Flow Characteristics in Local Scour at Bridge Piers. *Journal of Hydraulic Research*, 15(4):373–380.
- Menter, F. R. (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA Journal*, 32(8):1598–1605.
- Noormets, R., Ernstsens, V., Bartholom, A., Flemming, B., and Hebbeln, D. (2003). *Local Scour in a Tidal Environment: A Case Study from the Otzumer Balje Tidal Inlet, Southern North Sea - Preliminary Results*, pages 1134–1136. International Association for Hydro-Environment Engineering and Research.
- Olsen, N. R. B. and Kjellesvig, H. M. (1998). Three-Dimensional Numerical Flow Modeling for Estimation of Maximum Local Scour Depth. *Journal of Hydraulic Research*, 36(4):579–590.
- Olsen, N. R. B. and Melaaen, M. C. (1993). Three-Dimensional Calculation of Scour Around Cylinders. *Journal of Hydraulic Engineering*, 119(9):1048–1054.

- Ong, M. C., Utnes, T., Holmedal, L. E., Myrhaug, D., and Pettersen, B. (2009). Numerical Simulation of Flow Around a Smooth Circular Cylinder at Very High Reynolds Numbers. *Marine Structures*, 22(2):142–153.
- Ouriemi, M., Aussillous, P., and Guazzelli, E. (2009). Sediment Dynamics. Part 2. Dune Formation in Pipe Flow. *Journal of Fluid Mechanics*, 636:321–336.
- Patankar, S. V. and Spalding, D. B. (1972). A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows. *International Journal of Heat and Mass Transfer*, 15(10):1787–1806.
- Press, W. H., Teukolsky, S. A., Vetterling, W. T., and Flannery, B. P. (2007). *Numerical Recipes 3rd Edition: The Art of Scientific Computing*. Cambridge University Press, New York, USA.
- Rance, P. J. (1980). The Potential for Scour Around Large Objects, In: Scour Prevention Techniques Around Offshore Structures. In *Proceeding of a One Day Seminar, Society for Underwater Technology*, London, United Kingdom.
- Richardson, E. V. and Davis, S. R. (2001). Evaluating Scour at Bridges. Technical Report FHWA-HIF-12-003, U.S. Department of Transport Federal Highway Administration.
- Richardson, J. E. and Panchang, V. G. (1998). Three-Dimensional Simulation of Scour-Inducing Flow at Bridge Piers. *Journal of Hydraulic Engineering*, 124(5):530–540.
- Rosenbrock, H. H. (1960). An Automatic Method for Finding the Greatest or Least Value of a Function. *The Computer Journal*, 3(3):175–184.
- Roshko, A. (1961). Experiments on the Flow Past a Circular Cylinder at Very High Reynolds Number. *Journal of Fluid Mechanics*, 10(3):345–356.
- Roulund, A., Sumer, B. M., Fredsøe, J., and Michelsen, J. (2005). Numerical and Experimental Investigation of Flow and Scour Around a Circular Pile. *Journal of Fluid Mechanics*, 534:351–401.

- Rudolph, D., Raaijmakers, T. C., and Sonnevile, B. D. (2009). Challenges and Recent Advances in Offshore Scour Modelling. In *Proceedings of the 12th International Conference: The Jack-Up Platform Design, Construction and Operations*, London, United Kingdom.
- Rumsey, C. L., Smith, B. R., and Huang, G. P. (2010). Description of a Website Resource for Turbulence Modeling, Verification and Validation. In *Proceeding of the 40th AIAA Fluid Dynamics Conference*, Chicago, Illinois.
- Schewe, G. (1983). On the Force Fluctuations Acting on a Circular Cylinder in Crossflow from Subcritical up to Transcritical Reynolds Numbers. *Journal of Fluid Mechanics*, 133:265–285.
- Schmeeckle, M. W. and Nelson, J. M. (2003). Direct Numerical Simulation of Bed-load Transport Using a Local, Dynamic Boundary Condition. *Sedimentology*, 50(2):279–301.
- Schmidt, L. V. (1966). Fluctuating Force Measurements Upon a Circular Cylinder at Reynolds Number up to 5×10^6 . Technical Report ID 19660022955, National Aeronautics and Space Administration.
- Sederberg, T. W. and Parry, S. R. (1986). Free-form deformation of solid geometric models. *ACM SIGGRAPH computer graphics*, 20(4):151–160.
- Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J. (1994). A New k-epsilon Eddy Viscosity Model for High Reynolds Number Turbulent Flows: Model Development and Validation. *Computers Fluids*, 24(3):227–238.
- Shih, W. C. L., Wang, C., Coles, D., and Roshko, A. (1993). Experiments on Flow Past Rough Circular Cylinders at Large Reynolds Numbers. *Journal of Wind Engineering and Industrial Aerodynamics*, 49(1):351–368.
- Smirnov, P. E. and Menter, F. R. (2009). Sensitization of the SST Turbulence Model to Rotation and Curvature by Applying the Spalart–Shur Correction Term. *Journal of Turbomachinery*, 131(4):041010.
- Soulsby, R. L. and Whitehouse, R. (1997). Threshold of Sediment Motion in Coastal Environments. In *Proceedings of the 13th Australasian Coastal and Ocean En-*

gineering Conference and the 6th Australasian Port and Harbour Conference, Christchurch, New Zealand.

- Spalart, P. R. and Allmaras, S. R. (1994). A One-Equation Turbulence Model for Aerodynamic Flows. *La Recherche Aerospatiale*, (1):5–21.
- Spalart, P. R. and Shur, M. (1997). On the Sensitization of Turbulence Models to Rotation and Curvature. *Aerospace Science and Technology*, 1(5):297–302.
- Squires, K. D., Krishnan, V., and Forsythe, J. R. (2008). Prediction of the Flow Over a Circular Cylinder at High Reynolds Number using Detached-Eddy Simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(10):1528–1536.
- Stahlmann, A. (2013). Numerical and Experimental Modeling of Scour at Foundation Structures for Offshore Wind Turbines. In *Proceedings of the 23th International Offshore and Polar Engineering Conference*, Anchorage, Alaska.
- Sumer, B. M. (2007). Mathematical Modelling of Scour: A Review. *Journal of Hydraulic Research*, 45(6):723–735.
- Sumer, B. M. (2014a). A Review of Recent Advances in Numerical Modelling of Local Scour Problems. In *Proceeding of the 7th International Conference on Scour and Erosion*, Perth, Australia.
- Sumer, B. M. (2014b). Flow–Structure–Seabed Interactions in Coastal and Marine Environments. *Journal of Hydraulic Research*, 52(1):1–13.
- Sumer, B. M., Christiansen, N., and Fredsøe, J. (1992a). Time Scale of Scour Around a Vertical Pile. In *Proceeding of the 2nd International Offshore and Polar Engineering Conference*, San Francisco, California.
- Sumer, B. M. and Fredsøe, J. (1992). A Review of Wave/Current-Induced Scour Around Pipelines. In *Proceedings of the 23rd Conference on Coastal Engineering*, Venice, Italy.
- Sumer, B. M. and Fredsøe, J. (1997). *Hydrodynamics Around Cylindrical Structures*, volume 12. World Scientific.

- Sumer, B. M. and Fredsøe, J. (2002). *The Mechanics of Scour in the Marine Environment*, volume 17. World Scientific.
- Sumer, B. M., Fredsøe, J., and Christiansen, N. (1992b). Scour Around Vertical Pile in Waves. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 118(1):15–31.
- Sumer, B. M., Petersen, T. U., Locatelli, L., Fredsøe, J., Musumeci, R. E., and Foti, E. (2012). Backfilling of a Scour Hole Around a Pile in Waves and Current. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 139(1):9–23.
- Sumer, B. M., Whitehouse, R. J., and Tørum, A. (2001). Scour around coastal structures: A summary of recent research. *Coastal Engineering*, 44(2):153–190.
- Torczon, V. (1991). On the Convergence of the Multidirectional Search Algorithm. *SIAM Journal on Optimization*, 1(1):123–145.
- Torsethaugen, K. (1975). Lokalerosjon Ved Store Konstruksjoner. Modellforsøk. Technical Report STF60-A75055, Norwegian Hydrotechnical Laboratory, Trondheim, Norway. - In Norwegian.
- Tseng, M. H., Yen, C. L., and Song, C. (2000). Computation of Three-Dimensional Flow Around Square and Circular Piers. *International Journal for Numerical Methods in Fluids*, 34(3):207–227.
- Unal, U. O. and Goren, O. (2006). Vortex Shedding from a Circular Cylinder at High Reynolds Number. In *Proceedings of the 11th International Congress of the International Maritime Association of the Mediterranean*, Lisbon, Portugal.
- Wardhana, K. and Hadipriono, F. (2003). Analysis of Recent Bridge Failures in the United States. *Journal of Performance of Constructed Facilities*, 17(3):144–150.
- Warren, I. and Bach, H. (1992). MIKE 21:A Modelling System for Estuaries, Coastal Waters and Seas. *Environmental Software*, 7(4):229–240.
- Whitehouse, R. (1998). *Scour at Marine Structures: A Manual for Practical Applications*. Thomas Telford Publications.

- Whitehouse, R., Harris, J., and Rees, J. (2008). Dynamics of Scour Pits and Scour Protection - Synthesis Report and Recommendations. Technical report, UK Department of Energy and Climate Change.
- Wilcox, D. C. (1988). Reassessment of the Scale-Determining Equation for Advanced Turbulence Models. *AIAA Journal*, 26(11):1299–1310.
- Yakhot, V. and Orszag, S. A. (1986). Renormalization Group Analysis of Turbulence. I. Basic Theory. *Journal of Scientific Computing*, 1(1):3–51.
- Yeganeh-Bakhtiary, A., Kazeminezhad, M. H., Etemad-Shahidi, A., Baas, J. H., and Cheng, L. (2011). Euler–Euler Two-Phase Flow Simulation of Tunnel Erosion Beneath Marine Pipelines. *Applied Ocean Research*, 33(2):137–146.
- Zhang, X. T., Li, Z. Y., Fu, S. X., Ong, M. C., and Chen, Y. (2014). Study of the flow around a cylinder from the subcritical to supercritical regimes. *Ocean Systems Engineering*, 4(3):1–17.
- Zhao, M. and Cheng, L. (2008). Numerical Modeling of Local Scour Below a Piggy-back Pipeline in Currents. *Journal of Hydraulic Engineering*, 134(10):1452–1463.
- Zhao, M., Cheng, L., and Zang, Z. (2010). Experimental and Numerical Investigation of Local Scour Around a Submerged Vertical Circular Cylinder in Steady Currents. *Coastal Engineering*, 57(8):709–721.
- Zhao, Z. and Fernando, H. J. S. (2007). Numerical Simulation of Scour Around Pipelines Using an Euler–Euler Coupled Two-Phase Model. *Environmental Fluid Mechanics*, 7(2):121–142.
- Zounemat-Kermani, M., Beheshti, A. A., Ataie-Ashtiani, B., and Sabbagh-Yazdi, S. R. (2009). Estimation of Current-Induced Scour Depth Around Pile Groups Using Neural Network and Adaptive Neuro-Fuzzy Inference System. *Applied Soft Computing*, 9(2):746–755.