The Influence of Trailing Edge Shape on Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

by

Nu Rhahida Arini

Thesis for the degree of Doctor of Philosophy

January 2018
UNIVERSITY OF SOUTHAMPTON

ABSTRACT

FACULTY OF ENGINEERING AND ENVIRONMENT
Civil, Maritime and Environmental Engineering and Sciences Unit

Doctor of Philosophy

THE INFLUENCE OF TRAILING EDGE SHAPE ON FLUID STRUCTURE INTERACTION OF A VERTICAL AXIS TIDAL TURBINE BLADE

by Nu Rhahida Arini

The depletion of fossil fuel and the increase of fuel consumption globally create an increased demand for the use of renewable energy. Vertical axis tidal turbines are a promising renewable energy device which needs to be improved. One problem appears in its operation is the structural instability and noise coming from the vibration of the long slender vertical axis blades. The vibration is a result of fluid structure interaction between turbine blades and the unsteady tidal current. This interaction of the tides and the blade generates vortical features which can excite the turbine blades to vibrate and causes a tonal noise known as singing. The aim of this work is to predict the blade response and locked-in condition by controlling the vortex shedding. The vortex is controlled by modifying blade’s trailing edge shape. The modifications include truncated, sharp and rounded trailing edge shapes. The response is modeled by vibrations using a spring damper system. A 2D numerical model of a vertical axis tidal turbine blade is developed to resolve the vibration using OpenFOAM 2.2. The blade has 0.75 m chord length and \(3.07 \times 10^6\) Re. The model employs the equivalence incoming velocity method which represents the actual unsteady tidal current by time varying velocity magnitude and angle of attack of the model incoming flow. The problem is examined by observing the force applied to a static blade, and a rotating three bladed vertical axis turbine primarily. This is to confirm that the mesh topology and selected boundary conditions are sufficient and robust to resolve the blade response model. The locked-in condition is clarified by the blade main frequencies, pressure distribution, displacement, and force coefficients. In addition to the reference trailing edge, three different trailing edge shapes were studied. From the results it can be seen that the response is sensitive to pitching motion, high blade initial angle of attack, high tidal velocity and low spring and damping constant blade material. The results also show that the blunt (conventional truncated) foil has the largest ability to control the turbine blade response which is demonstrated by the smallest amplitude and the least frequent turbine blade’s vibration. For all three trailing edge shapes, along with a more limited investigation of an asymmetric trailing
edge all are shown to be able to shift the frequency of the resonant response. This will allow the designer to study the likely behaviour of their design. Overall, the developed methodology using a two-dimensional, three degree of freedom solution of the unsteady CFD around the foil is shown to provide useful insight to the tidal turbine designer at a reasonable computational cost.
To Dad

To Mum, Alec, Jasmine and Luwi
## Contents

Declaration of Authorship ........................................ xix
Acknowledgements ................................................. xxi
Nomenclature ......................................................... xxiii

1 Introduction ...................................................... 1
   1.1 Motivation .................................................. 1
   1.2 Background ................................................ 3
   1.3 Aim and Objectives ........................................ 6
   1.4 Research Methodology ..................................... 7
   1.5 Outline ..................................................... 10

2 Literature Review ............................................... 13
   2.1 A Vertical Axis Tidal Turbine ............................. 13
      2.1.1 Blade Shape and Blade Thickness ...................... 14
      2.1.1.1 Modified Blade .................................... 16
      2.1.2 Aspect Ratio .......................................... 17
      2.1.3 Number of Blades ..................................... 17
      2.1.4 Tip Speed Ratio ....................................... 19
      2.1.5 Solidity ............................................... 21
      2.1.6 Turbine Power and Coefficient of Performance ....... 22
      2.1.7 Reynolds Number ....................................... 23
      2.2 A Basis Vertical Axis Tidal Turbine Design ............ 24
      2.2.1 Periodic Inflow Equivalence Model .................... 26
      2.3 A 2D CFD Model For A Vertical Axis Tidal Turbine Using OpenFOAM ........................................ 28
         2.3.1 Developing The Domain Grid in Preprocessing Process ........................................ 30
         2.3.2 Solving Mathematical Models in Processing Stage ........................................ 32
            2.3.2.1 Solver Algorithm ................................ 32
            2.3.2.2 Turbulence Model ................................ 33
            2.3.2.3 Interpolation ..................................... 34
            2.3.2.4 Boundary Condition .............................. 35
      2.4 Dynamic Mesh in OpenFOAM ................................ 37
      2.5 A Three Bladed Vertical Axis Tidal Turbine .......... 39
      2.6 Introduction to FSI on a Vertical Axis Tidal Turbine Blade ........................................ 41
      2.7 Vortex Shedding around A Foil ............................ 44
      2.8 Prediction of Fluid Induced Vibration ................... 49
         2.8.1 Frequency Lock-in .................................... 51
# CONTENTS

2.9 Introduction to Dynamic Response .............................................. 53
2.10 A Vertical Axis Tidal Turbine Blade Response Model ......................... 56
   2.10.1 Phase Averaged Method .............................................. 59
   2.10.2 Power Spectrum Density using Fourier Transform ...................... 61
   2.10.3 Natural Frequency .................................................. 62
   2.10.4 A Spring Damper System for A Blade Response Model ................. 63
2.11 Modified Foil Response ..................................................... 70
2.12 Summary ................................................................. 72

3 Development of the CFD Methodology for Static and Dynamic Foils 75
   3.1 Static Foil Performance .................................................. 75
      3.1.1 Force Coefficients ................................................ 77
      3.1.2 Static Foil Fluid Regime ......................................... 80
   3.2 A 2D Single Foil CFD Model ........................................... 83
      3.2.1 Topology .......................................................... 84
          3.2.1.1 C-structured Grid ........................................ 84
          3.2.1.2 Rectangular Grid ......................................... 86
      3.3 Mesh Independence Study and Validation ................................ 88
          3.3.1 C-structured Grid ............................................ 88
          3.3.2 Rectangular Grid ............................................ 89
   3.4 Modified Foils Performance ............................................... 93
      3.4.1 Blunt Profile .................................................... 93
      3.4.2 Sharp and Rounded Profiles ..................................... 95
   3.5 Modified Foil Models .................................................... 97
      3.5.1 C-structured Grid ................................................ 97
      3.5.2 Rectangular Grid ............................................... 97
          3.5.2.1 Blunt Foil in Rectangular Grid .......................... 97
          3.5.2.2 Sharp Foil in Rectangular Grid ......................... 99
          3.5.2.3 Rounded Foil in Rectangular Grid ....................... 100
   3.6 Flow Regime on A Dynamic Foil ......................................... 104
      3.6.1 Pitch Motion (Oscillating Foil) ................................ 105
      3.6.2 Heave Motion (Plunging Foil) .................................. 106
      3.6.3 Pitch and Heave Motion (Pitching and Plunging Foil) .............. 108
   3.7 An Oscillating Foil Model .............................................. 109
   3.8 A Rotating Three Bladed Vertical Axis Tidal Turbine Model ............... 112
      3.8.1 Result ........................................................ 115
   3.9 Summary ............................................................. 117

4 Fluid Structure Interaction of a Foil with Steady Incoming Flow 119
   4.1 Foil Response Model Results .......................................... 121
      4.1.1 Calculation Routine of A Response Model ......................... 121
      4.1.2 Inlet Fluid Velocity Magnitude Variation ......................... 128
      4.1.3 Initial Angle of Attack Variation ................................ 134
      4.1.4 Number of Response Motions .................................... 142
      4.1.5 Stiffness Constant Variation ................................... 150
      4.1.6 Damping Constant ............................................... 155
   4.2 Modified Trailing Edge Foil Response .................................. 157
4.2.1 Blunt Foil ........................................ 159
4.2.2 Rounded Foil .................................... 162
4.2.3 Sharp Foil ....................................... 164
4.2.4 An Asymmetric Foil for Singing Reduction .... 167
4.3 Summary ........................................... 170

5 Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade 175
5.1 Vertical Axis Tidal Turbine Response Representative .... 176
  5.1.1 TSR variation ................................... 181
  5.1.2 Re variation .................................... 183
  5.1.3 Lift to Drag Ratio ................................ 184
5.2 Response of a Vertical Axis Tidal Turbine Using Original Blade .... 185
  5.2.1 Unsteady Angle of Attack and Velocity of the Incoming Flow ... 186
  5.2.2 Response Result .................................. 187
  5.2.3 Time step Independence Study ....................... 193
5.3 Response of A Vertical Axis Tidal Turbine Using Modified Blade ... 195
  5.3.1 Blunt Blade ..................................... 195
  5.3.2 Rounded Blade ................................... 199
  5.3.3 Sharp Blade ..................................... 202
5.4 Summary ........................................... 209

6 Conclusions and Recommendation for Future Work 211
6.1 Conclusion .......................................... 211
6.2 Future Work ........................................ 214

A OpenFOAM codes 217
  A.1 U for Vertical Axis Turbine Blade Response ............ 217
  A.2 pointDisplacement for Vertical Axis Tidal Turbine Response ... 218

References 221
# List of Figures

1.2 Oil production and consumption in Indonesia (Indonesian gov, 2016) ........... 4
2.1 Instantaneous power coefficient of different types of NACA on tidal turbine blades (Gosselin et al., 2013) ................................................................. 15
2.2 The modified foil in perspective views: a. blunt foil, b. round foil, c. sharp foil ................................................................................................................... 16
2.3 Influence of aspect ratio on $Re$ and rpm of tidal turbine (Brusca et al., 2014) ........................................................................................................... 18
2.4 Effect of number of blades in turbines (Gosselin et al., 2013) ................. 19
2.5 Influence of tip speed ratio to power coefficient at numbers of different solidity (Brusca et al., 2014) ............................................................... 20
2.6 Theoretical instantaneous angle of attack in a Darrieus turbine for different $\lambda$ (Gosselin et al., 2013) ................................................................. 20
2.7 CFD simulation of three blade turbines with different solidities (Gosselin et al., 2013) ................................................................. 22
2.8 Coefficient of performance of turbine as a function of tip speed ratio (Abdel Aleem et al., 2014) ................................................................. 23
2.9 Illustration of: a. Angle of attack in vertical axis tidal turbine revolution (Danao et al., 2012) b. Time dependent rotational resultant velocity (relative wind) variation works on a vertical axis turbine blade (Nemec, 2001), c. Range of angle of attack on a blade during one rotation ............ 27
2.10 Log of the wall region (Cradle, MSC Software Company, 2016) ............. 37
2.11 a. Simulation domain with 14 subdivision domain. b. Single foil mesh (Gretton et al., 2009) ................................................................. 40
2.12 VATT simulation domain (Lanzafame et al. 2014) .................................. 41
2.13 Vortex on NACA 0012 .............................................................................. 42
2.14 The independence of Vortex Street with Reynolds number (Blevins, 1990) 44
2.15 Flow pattern at zero angle of attack foil .................................................. 45
2.16 Vortex in pure plunge (left) and pure pitch Motion (right) of NACA 0012 (Khalid et al., 2014) ................................................................. 47
2.17 Pressure distribution associated with vortex shedding stages (Lee and Su, 2015) ................................................................. 48
2.18 Strouhal number as a function of Reynolds number in a cylinder (Blevins, 1990) ................................................................. 49
2.19 Lock-in frequency of NACA 0012 at $40^\circ$ angle of attack: a. vortex shedding, b. foil vibration (Besem et al., 2014) ........................................ 52
2.20 Lock-in frequency band of the case in 2.19 (Besem et al., 2014) ............ 53
2.21 a. Deformation on turbine blade at 3.5 rad/s, b. velocity profile (Khalid et al., 2013b) .......................................................... 55
2.22 Predicted stress and deflection of a turbine blade (Young et al., 2010) .......................................................... 56
2.23 A response model of NACA 0012 using spring damper system (Rana et al., 2009) ......................................................... 57
2.24 Fluctuated original signal (f), a periodic signal and phase averaged signal in phase averaged method (Hussain and Reynolds, 1969) ....... 59
2.25 Signal decomposition: a. original, b. foil-induced signal, c. vortex signal (Jung and Park, 2005) ......................................................... 60
2.26 Response model as a vibrational system with spring and damper components .......................................................... 64
3.1 Simulation result of fixed angle of attack (Mittal and Saxena, 2002) a. lift coefficient, b. drag coefficient ............................... 80
3.2 Pressure and vortex contour at various fixed angle of attack for steady state solution (Mittal and Saxena, 2002) ......................... 81
3.3 Lift and drag coefficient. Solid line is increasing angle and broken line is decreasing angle (Mittal and Saxena, 2002) ............... 82
3.4 a. CI signal, b. vortex behind trailing edge (Gioria et al., 2009) ........ 83
3.5 a. C-structured grid, b. Mesh around an original foil in c-structured grid 85
3.6 a. Domain grid of original foil, b. Mesh around normal trailing edge 87
3.7 Lift coefficient at four different c-structured refinement grids .... 88
3.8 Drag coefficient at four different c-structured refinement grids .... 89
3.9 Lift coefficient compared to experimental and numerical data .... 89
3.10 Drag coefficient compared to experimental and numerical data 90
3.11 Four grids for mesh independence study in rectangular grid: a. 8600 cells, b. 10472, c. 11838 cells, d. 12711 cells .......................... 90
3.12 Drag coefficient at four different refinement grids ................... 91
3.13 Validation of lift coefficient ................................................. 91
3.14 Validation of drag coefficient ............................................. 92
3.15 Two methods of trailing edge modification (Murcia and Pinilla, 2011) 94
3.16 Vortex profile behind blunt trailing edge with two pulse behind trailing edge (Krentel and Nitsche, 2013) ......................... 96
3.17 The modified foil constructed in C-structured: a. blunt foil, b. round foil, c. sharp foil .............................................. 98
3.18 Lift coefficient comparison between original and modified foils .... 98
3.19 Drag coefficient comparison between original and modified foils 99
3.20 Mesh around blunt trailing edge ......................................... 99
3.21 Mesh around sharp trailing edge ....................................... 100
3.22 Mesh around rounded trailing edge ................................... 100
3.23 Lift coefficient of four trailing edge shapes ......................... 101
3.24 Drag coefficient for trailing edge modifications .................. 102
3.25 Two fluid pulse behind blunt trailing edge at zero angle of attack 103
3.26 Hysteresis on $C_l$ and $C_d$ (Corrêa et al., 2014) .................... 106
3.27 Vortices in trailing edge of a foil as the function of foil angle of attack (Frederich et al., 2009) ............................... 107
3.28 Lift coefficient of the oscillating foil model .......................... 110
3.29 Velocity component in oscillating foil, velocity (m/s) vs simulation time step .............................................. 110
3.30 Fluid flow regime of the oscillating foil: a. 15°, b. 25° (upstroke), c. 15° (downstroke), d. 5° (downstroke), e. 15° (upstroke) .................................................. 111
3.31 2D vertical axis tidal turbine domain ................................................................. 113
3.32 2D Rotor domain with a circle interface between the rotor and stator domain ................................................................. 114
3.33 Lift and drag coefficients in one turbine revolution with TSR of 1.25 and fluid velocity of 1.4 m/s, shown in azimuth angle of 30° decrement (12=30°, 11=60°, etc) .................................................. 116

4.1 Flow chart for calculating FSI response of a vertical axis tidal turbine blade after response model mesh is validated .................................................. 120
4.2 Two points (pink mark) where the velocity signal is sampled .................................. 122
4.3 Reference (input) and original (output) velocity signal for the standard foil model .................................................. 122
4.4 Phase averaged signal taken from the reference (input) for the standard foil model, shown along with the input and output signal .................................................. 123
4.5 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the original standard operating foil model .................................................. 124
4.6 Default condition of original foil fluid flow regime during one cycle of vibration, at t/T: a. 0.1, b. 0.6, c. 1 .................................................. 125
4.7 Pressure distribution on the original foil in default operation at minimum and maximum extent of vibration .................................................. 125
4.8 Wake profile at x/c = 0.216 at t/T =0.1, 0.6, 1 .................................................. 126
4.9 Displacement vs time step at x/c= 0.216 for original foil response in the standard model. pointDisplacement (magnitude) corresponds to magnitude of displacement, pointDisplacement (0) equals to x direction displacement, pointDisplacement (1) to y direction .................................................. 127
4.10 Force coefficients for the standard operation original foil model .................................. 128
4.11 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the fluid velocity of 6 m/s model .................................................. 129
4.12 Original foil fluid flow regime at 6 m/s over one cycle of vibration, at t/T: a. 0.071 b. 0.286, c. 0.5 d. 1 .................................................. 130
4.13 Pressure distribution in the case of high velocity response, corresponds to Figure 4.12 .................................................. 131
4.14 Displacement vs time step at x/c= 0.49 for original foil response in the high velocity model (6 m/s). pointDisplacement (magnitude) corresponds to magnitude displacement, pointDisplacement (0) equals to x direction displacement, pointDisplacement (1) equals to y direction .................................................. 132
4.15 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the fluid velocity of 2 m/s model .................................................. 133
4.16 Lift coefficient for the case of velocity variation .................................................. 134
4.17 Drag coefficient for the case of velocity variation .................................................. 134
4.18 Moment coefficient for the case of velocity variation .................................................. 135
4.19 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the 8° initial angle of attack model .................................................. 136
4.20 Fluid flow regime at 8° initial angle of attack, at t/T: a. 0.04, b. 0.22, c. 0.35, d. 0.57, e. 1 .................................................. 137
4.21 Pressure distribution in the case for 8° initial angle of attack, at t/T: a. 0.04, b. 0.22, c. 0.35, d. 0.57, e. 1 .................................................. 138
4.22 Displacement in y direction vs time step at x/c= 0.297 for 8° initial angle of attack ................................................................. 138
4.23 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the 4° initial angle of attack model ................................................. 139
4.24 Displacement in y direction vs time step at x/c= 0.297 for 4° initial angle of attack ................................................................. 140
4.25 Lift coefficient in the case of initial angle of attack ................................................................. 140
4.26 Drag coefficient in the case of initial angle of attack ................................................................. 141
4.27 Moment coefficient in the case of initial angle of attack ................................................................. 142
4.28 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch heave surge model ................................................. 143
4.29 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch surge model ................................................. 144
4.30 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch model ................................................................. 145
4.31 Fluid flow regime at pitch response, at t/T: a. 0.03, b. 0.67, c. 1 ................................................................. 146
4.32 Pressure distribution in pitch response, at t/T: 0.03. 0.67, 1 ................................................................. 147
4.33 Lift coefficient in number of response mode variation ................................................................. 148
4.34 Drag coefficient in number of response mode variation ................................................................. 148
4.35 Moment coefficient in number of response mode variation ................................................................. 149
4.36 a. Reference, original and phase averaged velocity signal for higher stiffness, b. frequency domain for higher stiffness ................................................................. 152
4.37 Fluid flow regime at high stiffness model, at t/T: a. 0.04, b. 0.4, c. 0.6, d. 1 ................................................................. 153
4.38 Pressure distribution in high stiffness response, at t/T: 0.04, 0.4, 0.6, 1 ................................................................. 153
4.39 Reference, original and phase averaged velocity signal for lower stiffness material, b. frequency domain for lower stiffness ................................................................. 154
4.40 Lift coefficient in stiffness variation ................................................................. 155
4.41 Drag coefficient in stiffness variation ................................................................. 155
4.42 Moment coefficient in stiffness variation ................................................................. 156
4.43 a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the over damped model ................................................................. 157
4.44 Pressure distribution for high stiffness response, at t/T: 0.04, 0.4, 0.6, 1 ................................................................. 158
4.45 Lift coefficient for damping variation ................................................................. 158
4.46 Drag coefficient for damping variation ................................................................. 159
4.47 Moment coefficient for damping variation ................................................................. 159
4.48 Reference, original and phase averaged velocity signal for lower stiffness material, b. frequency domain for lower stiffness ................................................................. 160
4.49 Displacement in y direction on the blunt foil response model ................................................................. 161
4.50 Vorticity contour behind the blunt foil at t/T=: a. 0.067, b. 0.08, c. 0.2, d. 0.28, 0.76, e. 0.84, f. 1 ................................................................. 162
4.51 Pressure distribution from blunt foil response model at t/T: 0.08, 0.28 ................................................................. 163
4.52 a. Reference, original and phase averaged velocity signal for rounded foil model, b. frequency response for rounded foil model ................................................................. 164
4.53 Fluid flow regime in rounded foil model, at t/T: a. 0.04, b. 0.22, c. 0.35, d. 0.57, e.1 ................................................................. 165
4.54 Pressure distribution for the rounded foil model at t/T: 0.04, 0.22, 0.35, 0.57, 1 ................................................................. 166
4.55 a. Reference, original and phase averaged velocity signal for sharp foil model, b. frequency response for the sharp foil model ........................................... 167
4.56 Fluid flow regime at sharp foil model, at t/T: a. 0.06, b. 0.28, c. 1 .......................... 168
4.57 Pressure distribution for the sharp foil model at t/T: 0.04, 0.22, 0.35, 0.57, 1168
4.58 Drag coefficient for modified foil variations ...................................................... 169
4.59 Lift coefficient for modified foil variations ....................................................... 169
4.60 Moment coefficient for modified foil variations .............................................. 170
4.61 Fluid flow regime on an asymmetric foil at t/T: a. 0.2, b. 0.4, c. 0.6, d. 0.8, e. 0.99 ................................................................. 171
4.62 Asymmetric Foil: a. Model, b. Mesh ................................................................. 171

5.1 a. $C_d$ from Almohammadi et al. 2015. Only sharp trailing edge data is considered in b. $C_d$ from Almohammadi et al.’s model (in orange, taken from Figure 5.1(a)) and the static response model (blue) .............................. 178
5.2 a. $C_l$ from Almohammadi et al. Only sharp T.E data is considered in validation b. $C_l$ from the response model ...................................................... 179
5.3 Lift coefficient contour of various different symmetric blade thickness (Sheldahl and Klimas, 1981) ................................................................. 180
5.4 Drag coefficient of the static response models with TSR variation and Re of $3.07 \times 10^6$ ................................................................. 182
5.5 Lift coefficient of the static response models with TSR variation and and Re of $3.07 \times 10^6$ ................................................................. 182
5.6 Drag coefficient from models with $Re$ variation and TSR = 5.1 .......................... 183
5.7 Lift coefficient from models with $Re$ variation and TSR = 5.1 .......................... 183
5.8 Lift drag ratio of a vertical axis tidal turbine using original blade in standard operation ................................................................. 185
5.9 Angle of attack during one turbine rotation ...................................................... 186
5.10 Resultant velocity during one turbine rotation .................................................. 187
5.11 a. Reference, original and phase averaged velocity signal of a vertical axis tidal turbine blade model, b. frequency response of original blade model ................................................................. 188
5.12 Fluid flow regime during a rotation of a vertical axis tidal turbine using original blade model. From $0^\circ$, shown every $30^\circ$ azimuth angle clockwise. U magnitude in m/s ................................................................. 189
5.13 Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: $0^\circ$, $30^\circ$, $60^\circ$, $90^\circ$ ................................................................. 190
5.14 Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: $90^\circ$, $120^\circ$, $150^\circ$, $180^\circ$ ................................................................. 191
5.15 Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: $180^\circ$, $210^\circ$, $240^\circ$, $270^\circ$ ................................................................. 191
5.16 Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: $270^\circ$, $300^\circ$, $330^\circ$, $360^\circ$ ................................................................. 192
5.17 Time history displacement for the original blade response associated with selected azimuth angles, measured at a point of a quarter chord length ................................................. 193
5.18 Displacement of quarter chord length point at Courant Numbers of: a. 0.9, b. 0.7, c. 0.6, d. 0.5, e. 0.4 ................................................................. 194
5.19 a. Reference, original and phase averaged velocity signals for a vertical axis tidal turbine with a blunt blade model, b. frequency response for a blunt vertical axis tidal turbine model ................................................................. 196
5.20 Fluid flow regime over a cycle of a vertical axis tidal turbine model using blunt blade .......................................................... 197
5.21 Pressure distribution of vertical axis tidal turbine model with blunt blade at azimuth angle of: 0°, 90°, 180°, 270° .......................................................... 198
5.22 Time history displacement for the blunt blade response associated with selected azimuth angles, taken at point located at one quarter chord length 198
5.23 a. Reference, original and phase averaged velocity signals for a vertical axis tidal turbine with a rounded blade model, b. frequency response for a blunt vertical axis tidal turbine model ........................................... 199
5.24 Fluid flow regime for one cycle of a vertical axis tidal turbine model using a rounded blade .......................................................... 200
5.25 Pressure distribution for vertical axis tidal turbine model with a rounded blade at azimuth angle of: 0°, 90°, 180°, 270° .......................................................... 202
5.26 Time history displacement for the blunt blade response with selected azimuth angle positions sampled at a position of one quarter chord length . 202
5.27 a. Reference, original and phase averaged velocity signals for a vertical axis tidal turbine with a sharp blade geometry, b. frequency response for a sharp blade vertical axis tidal turbine model ........................................... 203
5.28 Fluid flow regime for one a cycle of a vertical axis tidal turbine model using a sharp blade profile .......................................................... 204
5.29 Pressure distribution for vertical axis tidal turbine model with a sharp blade at azimuth angle of: 0°, 90°, 180°, 270° .......................................................... 205
5.30 Time history displacement for the blunt blade response with selected azimuth angles denoted, measured at a position one quarter chord length . 206
5.31 Lift coefficient for a vertical axis tidal turbine with modified blade profiles .......................................................... 206
5.32 Drag coefficient for a vertical axis tidal turbine with modified blade profiles .......................................................... 207
5.33 Moment coefficient for a vertical axis tidal turbine with modified blade profiles .......................................................... 208
List of Tables

2.1 \textit{Re} and azimuth angle associated with maximum and minimum resultant velocity and angle of attack ........................................ 24
2.2 Blade and fluid properties .................................................. 25
2.3 Standard Setting for the foil response model .......................... 64
2.4 Variation of simulation parameters ........................................ 69
3.1 Mesh static for original foil .................................................. 88
3.2 Lift and drag coefficients in two different y+ values ................ 93
3.3 Mesh static for blunt foil .................................................... 99
3.4 Mesh static for sharp foil ................................................... 100
3.5 Mesh static for blunt foil ................................................... 100
3.6 Oscillatory motion properties .............................................. 109
3.7 Mesh static for rounded foil ............................................... 113
4.1 Dominant frequency and vibration amplitude in initial angle of attack variation ............................................................... 139
4.2 Natural frequency of each model in stiffness variation .......... 150
4.3 Maximum Heave Displacement and vortex shedding frequencies for all models with the associated natural frequency at each foil geometry. (P): a resonance to pitch natural frequency, (H): a resonance to heave natural frequency ........................................ 172
4.4 Maximum displacement and vortex shedding frequency for the original and modified foils with the associated natural frequency. (P): a resonance to pitch natural frequency ........................................ 173
5.1 Design parameters used in the original and modified blade response model 175
5.2 Turbine design parameters of original blade model and Almohammadi et al.’s model ............................................................ 176
5.3 CFD parameters of static blade response model and Almohammadi et al.’s model ............................................................ 177
5.4 TSR and \textit{Re} variation for the static response model validation .... 181
5.5 Vortex shedding frequency for a vertical axis tidal turbine with modified blade geometries with the associated natural frequency. (H): lock-in by heave natural frequency ........................................ 209
5.6 Total displacement for four vertical turbine blades profiles ..... 210
Declaration of Authorship

I, Nu Rhahida Arini, declare that the thesis entitled *The Influence of Trailing Edge Shape on Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade* and the work presented in the thesis are both my own, and have been generated by me as the result of my own original research. I confirm that:

- this work was done wholly or mainly while in candidature for a research degree at this University;
- where any part of this thesis has previously been submitted for a degree or any other qualification at this University or any other institution, this has been clearly stated;
- where I have consulted the published work of others, this is always clearly attributed;
- where I have quoted from the work of others, the source is always given. With the exception of such quotations, this thesis is entirely my own work;
- I have acknowledged all main sources of help;
- where the thesis is based on work done by myself jointly with others, I have made clear exactly what was done by others and what I have contributed myself;
- parts of this work have been published as: Arini et al. (2016a) and Arini et al. (2016b)

Signed:..........................................................................................................................

Date:...........................................................................................................................
Acknowledgements

I would like to acknowledge a number of people who contributed, encouraged and gave their support and love to make me believe that I could go through all these years of my PhD.

I would like to thank my first and second supervisors: Prof. Stephen R. Turnock and Dr. Mingyi Tan for all their dedication to guide and enlighten my study. Thank you for your supervision. It has directed me how to manage a good research project and how to be a good researcher. I would also like to thank my examiners: Dr. Tahsin Tezdogan from Department of Naval Architecture, Ocean and Marine Engineering at University of Strathclyde and Dr. Kamal Djijeli from Computational & Engineering Design Group at University of Southampton. Their valuable feedback during the viva helped to make my thesis solid and improved.

My gratitude is always sent to my parents and my children. Thank you Dad. Although you cannot see me with this degree, your vision always encouraged me to stand when I was feeling down. I dedicated my success to you. Thank you Mom, your caring and prayers helped me through all the difficult moments. Thank you kids, your patience and love influenced me to always have laughter and joy in every single day. I could not make it through this work without you. To all my big family back home, your support and love has always been part of my journey here.

My thanks also goes to my best friends Dr. Dina Shona Laila from Coventry University for her support and to Robbie for all the discussions. To all friends in FSI group at University of Southampton, thanks Guys. To all my friends from Indonesian community in Southampton, thank you for the lovely friendship and moments we shared together.

Last but certainly the biggest thanks to God for bringing me here, for making my dream come true and for giving the most beautiful life I could ever imagine. This all because of You, Your help is always there

"And when My servants ask you, [O Muhammad], concerning Me - indeed I am near. I respond to the invocation of the supplicant when he calls upon Me. So let them respond to Me [by obedience] and believe in Me that they may be [rightly] guided” Al Baqarah 186
## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area (m$^2$)</td>
</tr>
<tr>
<td>$c$</td>
<td>Chord length (m)</td>
</tr>
<tr>
<td>$C$</td>
<td>Damping constant</td>
</tr>
<tr>
<td>$C_c$</td>
<td>Critical damping</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Drag coefficient</td>
</tr>
<tr>
<td>$C_{d0}$</td>
<td>Minimum drag coefficient</td>
</tr>
<tr>
<td>$C_l$</td>
<td>Lift coefficient</td>
</tr>
<tr>
<td>$C_m$</td>
<td>Moment coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure coefficient</td>
</tr>
<tr>
<td>$C_t$</td>
<td>Transverse damping constant</td>
</tr>
<tr>
<td>$dP$</td>
<td>Gradient pressure (Pa)</td>
</tr>
<tr>
<td>$dx$</td>
<td>Length difference (m)</td>
</tr>
<tr>
<td>$f_s$</td>
<td>Vortex shedding frequency (Hz)</td>
</tr>
<tr>
<td>$F_D$</td>
<td>Drag Force (N)</td>
</tr>
<tr>
<td>$F_L$</td>
<td>Lift Force (N)</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravitational force (m/s$^2$)</td>
</tr>
<tr>
<td>$G$</td>
<td>Shear modulus (GPa)</td>
</tr>
<tr>
<td>$h$</td>
<td>Plunging amplitude (m)</td>
</tr>
<tr>
<td>$I_{zz}$</td>
<td>Moment of Inertia (m$^4$)</td>
</tr>
<tr>
<td>$l$</td>
<td>Spanwise length (m)</td>
</tr>
<tr>
<td>$L$</td>
<td>Object length (m)</td>
</tr>
<tr>
<td>$K_I$</td>
<td>Stiffness matrix for second order vibration equation</td>
</tr>
<tr>
<td>$K_t$</td>
<td>Transverse stiffness constant (N/m)</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass (kg)</td>
</tr>
<tr>
<td>$m_i$</td>
<td>Mass in each segment (kg)</td>
</tr>
<tr>
<td>$M$</td>
<td>Moment Force (Nm)</td>
</tr>
<tr>
<td>$M_i$</td>
<td>Mass matrix for the second order vibration equation</td>
</tr>
<tr>
<td>$N_b$</td>
<td>Number of turbine blade</td>
</tr>
<tr>
<td>$p$</td>
<td>Static pressure (Pa)</td>
</tr>
<tr>
<td>$P$</td>
<td>Fluid Pressure (Pa)</td>
</tr>
<tr>
<td>$P$</td>
<td>Freestream pressure (Pa)</td>
</tr>
<tr>
<td>$P_0$</td>
<td>Static pressure (Pa)</td>
</tr>
</tbody>
</table>
\( P_m \) Real turbine power (W)
\( P_w \) Theoretical turbine power from kinetic energy of tides (W)
\( R \) Distance (m)
\( R \) Turbine radius (m)
\( Re \) Reynolds number
\( s \) Spanwise length (m)
\( St \) Strouhal number
\( t \) Time (s)
\( t_x \) Maximum foil thickness fraction, expressed by the last two digit of NACA type
\( u \) Fluid velocity (m/s)
\( u^+ \) Dimensionless velocity
\( u_t \) Friction velocity (m/s)
\( v \) Velocity (m/s)
\( V \) Volume (m^3)
\( V_{\infty} \) Freestream velocity
\( x \) Displacement matrix for the second order vibration equation
\( X \) Displacement amplitude (m)
\( \bar{X} \) Centre of mass coordinate (m)
\( X_i \) Centre of mass in each segment coordinate (m)
\( x_t \) x coordinate for NACA profile (m)
\( y_t \) y coordinate for NACA profile (m)

\( \tan\delta \) Damping capacity
\( \mu \) Dynamic viscosity (N.s/m²)
\( \alpha \) Angle of attack (° or rad)
\( \alpha_{0d} \) Angle of minimum \( C_d \) (° or rad)
\( \alpha_{0l} \) Angle of attack of which lift coefficient is zero (° or rad)
\( \alpha_{L/D_{\text{max}}} \) Angle of attack where lift or drag coefficient are maximum (° or rad)
\( \zeta \) Damping ratio
\( \theta \) Azimuth angle (°)
\( \lambda \) Tip speed ratio
\( \lambda_i \) Eigen value
\( \rho \) Fluid density (kg/m³)
\( \rho_s \) Foil density (kg/m³)
\( \sigma \) Solidity
\( \tau \) Wall shear stress
\( \omega \) Angular velocity (rad/min)
\( \Omega \) Non-dimensional parameter of frequency
\( \omega_n \) Natural frequency (rad/min)
\( \omega_r \) Rotational natural frequency (rad/min)
\( \omega_t \) Transverse natural frequency (rad/min)
\( \nu \) Kinematic viscosity (m²/s)
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Full Form</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>2(two) Dimension</td>
</tr>
<tr>
<td>3D</td>
<td>3(three) Dimension</td>
</tr>
<tr>
<td>ADM</td>
<td>Adomian Decomposition Method</td>
</tr>
<tr>
<td>AR</td>
<td>Aspect Ratio</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CS</td>
<td>Control Surface</td>
</tr>
<tr>
<td>CV</td>
<td>Control Volume</td>
</tr>
<tr>
<td>DMA</td>
<td>Dynamic Mechanical Analysis</td>
</tr>
<tr>
<td>DTI</td>
<td>Direct Time Integration</td>
</tr>
<tr>
<td>EEPIS</td>
<td>Electronic Engineering Polytechnic Institute of Surabaya</td>
</tr>
<tr>
<td>FFT</td>
<td>Fast Fourier Transform</td>
</tr>
<tr>
<td>FSI</td>
<td>Fluid Structure Interaction</td>
</tr>
<tr>
<td>FT</td>
<td>Fourier Transform</td>
</tr>
<tr>
<td>LEV</td>
<td>Leading Edge Vortex</td>
</tr>
<tr>
<td>LSA</td>
<td>Linear Stability Analysis</td>
</tr>
<tr>
<td>NACA</td>
<td>the National Advisory Committee for Aeronautics</td>
</tr>
<tr>
<td>NRE</td>
<td>New and Renewable Energy</td>
</tr>
<tr>
<td>PSD</td>
<td>Power Spectrum Density</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge</td>
</tr>
<tr>
<td>TSR</td>
<td>Tip Speed Ratio</td>
</tr>
<tr>
<td>VATT</td>
<td>Vertical Axis Tidal Turbine</td>
</tr>
<tr>
<td>VIV</td>
<td>Vortex Induced Vibration</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Motivation

Energy consumption is increasing every year and is becoming a crucial issue due to the depletion of conventional energy resources. One way to resolve the problem is to harness renewable energy coming from ocean tidal currents. Unlike wind energy technology which has been thoroughly developed, tidal energy technology still needs to be investigated in more detail. Currently some studies and experiments in the area of tidal energy are being conducted enthusiastically as part of renewable energy development. Tidal energy gained from motions of tides is converted into the mechanical energy by means of a turbine. An appropriate turbine design is required to capture the kinetic energy from tides more efficiently. According to IRENA (2014), tidal technology is classified onto three categories. First category is to harvest the tidal energy using a barrage by means of a turbine. The energy comes from the height of the high and low tides. Second category is tidal stream technology. The example of this category is to harvest the tidal energy using a barrage type and which produce less pollutants. There are two different types of tidal turbine constructed and operated as shown in Figures 1.1(a) and 1.1(b). Those typical tidal turbines have been designed and installed for industrial or research purposes.

The type of turbine which has been considered in many regions globally is a vertical axis turbine. A vertical axis tidal turbine as seen in Figure 1.1(b) has some benefits such as having simpler structure and does not require an additional device for directing the turbine towards the tidal stream direction. The simple construction of a vertical axis

turbine comes from the generator position which can be placed above water level. The vertical axis tidal turbine is also flexible to adjust the incoming tidal direction so that no directional problem exists for different diurnal flow change (Khalid et al., 2013a). However its engineering design including hydrodynamic force prediction, structure sustainability and environmental effect need to be further understood. Those aspects are strongly related to turbine response to incoming fluid flow during operation which is the main concern in this work.

The vertical axis tidal turbine is named according to how it rotates in such a way that the axis of rotation is vertical and perpendicular to incoming fluid. In this work the Darrieus type turbine, as illustrated in Figure 1.1(b), is designed, modified and its interaction with the tides is investigated. As a turbine blade contacts with unsteady incoming fluid, the turbine starts to rotate and converts tidal kinetic energy to mechanical energy. Under dynamic fluid loading exerted during operation, a vertical axis tidal turbine structure becomes critical to vibrational or buckling failure because its long slender shape (high
aspect ratio) contacts with unsteady fluid loading along its span. The turbine blade supports the structure which is held either at the top and bottom or at the middle of the blade span and creates a dynamic bending moment and deflection on spanwise length as it is in contact with the fluid. The dynamic bending moment can be a source of fatigue problems which may potentially harm the turbine structure.

The unsteady fluid loading from turbulent ocean tides is not the only source of dynamic loading experienced by the turbine. The varying angle of attack exists during turbine rotation and the wake or vorticity from the leading blade makes the dynamic load more unpredictable and severe. Dynamic fluid loads excite the turbine structure to vibrate and induce blade’s elastic deformation (turbine flexures). The vibration can induce a noise which is termed as ”singing”. This singing which is typically a clear harmonic sound is undesirable (Carlton, 2011). The potential for damage comes from deformations leading to fatigue which accompany singing caused by the vibration. This situation becomes more serious because the vibration varies with the depth at the location where turbine is installed. The vibration is not only location-dependent but also time-dependent which contributes to more harmful and destructive situation for a turbine operation. Thus the vibration as an interaction effect between the tides and turbine blades rotation during operation is an important aspect to study. Most of the studies which have been done attempt to minimize the risk of dynamic fluid loading deformations, fatigue failure and singing phenomena which can prolong a turbine design life.

The fluid structure interaction (FSI) between a vertical axis tidal turbine blade and the tide which generates vibrations is the main focus in this work. The FSI study purpose is to understand how the blade turbine responds during the interaction. The expected novelty of this project is to identify the fluid structure interactions of a vertical axis tidal turbine blade in one turbine’s revolution and the prediction of lock-in on a the turbine’s operation. Additional objectives are also addressed in this thesis which include assessing how the turbine blade response changes in different fluid operational conditions and assessing how the response can be controlled by modifying the blade trailing edge shape. The modified blades alter the fluid flow behavior and minimize the wake and vorticity generation to overcome the vibration effect.

1.2 Background

A vertical axis tidal turbine is a device to convert kinetic energy from tides to mechanical energy by its rotation. The relatively new concept of tidal turbine adopts basic theory of wind turbine designs which is more mature as it evolved over a longer time period. However, the difference between the tidal current behavior and wind should be taken into account carefully when designing and modifying a tidal turbines. Tidal behaviour is more predicted because it is affected by periodic phases of the moon and it also has
much larger density. Potential places to install vertical axis tidal turbine are near coastal areas which have shallow depths of ocean. The installation of tidal turbines is beneficial for a country surrounded by ocean or has long coast lines like Indonesia.

Historically Indonesia has abundant fossil fuel reserves and was a natural energy exporting country. This country depends mainly on fossil fuel deposits for energy supply including for their electrical power plants. The situation has started to change since approximately 15 years ago (see Figure 1.2) when the consumption exceeded the capacity of fossil fuel energy productions. This situation has become worse every year with the consumption rising and oil production and supply decreasing.

![Figure 1.2: Oil production and consumption in Indonesia (Indonesian gov, 2016)](image)

Indonesia’s geographical condition which comprises more than 17,000 islands, is also an important issue for the transport, supply and distribution of electricity across the country. In particular regions where some remote islands are located, inadequacy of electricity supply happens because energy resources are not provided nor is possible to deliver. In this situation renewable energy from the ocean is believed to be one solution to power the islands continuously and maintain a low cost budget. Realizing all threats and opportunities condition, since 2015 Indonesian government started to focus on New and Renewable Energy (NRE) technology and launched its long term vision and policy for national development of energy through the ministry of energy and mineral resources roadmap (Indonesia Ministry of Energy and Mineral Resources, 2016). The policy is based on the predicted future population in 2025 which will reach 285 million and will cause high energy demand. The policy sets an increasing target from 6% of NRE in 2014 to 23% by 2025 while reducing oil supply dependency from 41% to 25% by the same year. This target needs a strategy and has been proposed by the government to support research, development and the application of renewable energy technologies. Indonesian government further drives and encourages all including industry and education, to act
and contribute to Indonesia NRE to accelerate the target and overcome the problem of Indonesian electricity supply.

To support government policy, the Energy Generation System Department in Electronic Engineering Polytechnic Institute of Surabaya (EEPIS) for the past seven years has conducted a renewable energy project which was funded by the Indonesian Ministry of Research. It is a long term project which aims to help the Indonesian Government to supply electricity in remote coastal area in Indonesia by developing and installing vertical axis tidal turbines. Preliminary goals of the project are to study and design a vertical axis tidal turbine which is suitable to be installed on Indonesia eastern coast. This project is expected to contribute in solving Indonesia’s energy deficiency and electricity distribution problems.

A preliminary study of a three bladed vertical axis tidal turbine was investigated numerically using a CFD commercial software called ANSYS FLUENT. The main goal of the project was to find the power extracted by turbines in tidal velocity ranges from 1 to 3 m/s. In that project, a 2D CFD model had been developed in the hybrid grid domain and the result showed the velocity distribution profile around the turbine. Although the project produced a significant fluid velocity distribution prediction for a vertical axis turbine to be used for Indonesian ocean, some aspects have not been evaluated which are of importance in the turbine design and its performance. These aspects of turbine and fluid interaction which leads to vibrations and singing phenomenon need to be explored thoroughly in order to establish the behavior of fluid flow for less-vibrated turbine performance and for longer operational life. Motivated by this situation, the study is continued by this work to understand the interaction between the tides and a vertical axis tidal turbine blade and to predict the blade responses in a particular fluid flow regime behavior due to the interaction.

This work was commenced as a result of the initial Indonesian renewable energy project. The work intents to understand the FSI physical behavior on the rigid vertical axis tidal turbine structure. The FSI turbine behaviour includes how the blade trailing edge profile influences turbine performance which have not been explored in the initial Indonesian renewable energy study.

Fluid structure interaction analysis between a vertical axis tidal turbine blade response is the focus of this work. Although the topic has been of interest in recent decades, the interaction of the turbine’s structure and the tides is not fully understood and the effect of the interaction such as turbine vibrations remain unresolved. The current discussions about a vertical axis tidal turbine mainly focus on the turbine performance such as the studies done by Almohammadi et al. (2015) and MacPhee and Beyene (2015). They developed a 2D CFD model of a vertical axis turbine and observed the lift and drag coefficients to identify the turbine performance. Almohammadi et al. (2015) also suggested turbine blade modifications for improving the turbine performance.
However their work did not provide the fluid flow analysis in detail and showed only three azimuth angles. The blade interaction with the fluid which produce vibrations is not yet understood therefore it will be highlighted in this work.

The vibration occurs as a result of the blade and tides interaction. The interaction introduces a large magnitude load from the unsteadiness and turbulence of the tides to turbine’s structure and is required to be controlled. To understand the control process requires the investigation of the fluid regime which is determined from the pressure distribution on the upper (outer) and lower (inner) blade’s surfaces during a turbine revolution. The process is expected to identify the turbine lock-in condition during various operational conditions and its crucial positions over turbine rotation where the lock-in condition occurs. The lock-in condition will be observed from the main frequency of the turbine fluid regime (vorticity) and will be verified by the blade displacement. To obtain a constructive investigation, aim and objectives are set as explained in Section 1.3.

1.3 Aim and Objectives

The aim of the study is to predict a vertical axis tidal turbine blade response caused by its interaction with the tides. The response is modeled as vibrations and further controlled by modification of blade trailing edge shape. The investigation is accomplished by developing a 2D numerical vibration model which helps to understand the interaction between the tidal current and a vertical axis tidal turbine blade. The model simulates tidal fluid flow and how the blade responds to it. The analysis intends to overcome the major problem associated with the existence of vibration such as singing effect as the turbine installation vibrates. Understanding this interaction is achieved by the following proposed objectives:

- Generating a 2D CFD model simulation in OpenFOAM software using unsteady solver to accommodate the turbulence of the tidal current. The model utilizes nine static angles of attack applied to an original NACA 0012 and also to modified foils, and an oscillating NACA 0012 using dynamic mesh utility.

- Developing 2D static angle of attack models with two types of mesh on which the lift and drag coefficients are validated to ascertain that the grid topology is appropriate. Lift and drag coefficients are validated with published experimental research (Abbott and Von Doenhoff, 1959) and by comparison with numerical work (Eleni et al., 2012). The validated mesh is duplicated and utilized for an oscillating foil model which is validated using numerical result (Frederich et al., 2009). Prior to the validation, the mesh independence study is carried out.
Chapter 1 Introduction

- Developing a 2D rotational three bladed vertical axis tidal turbine model which is validated using published numerical results. This confirms that the model is valid and following this the vertical axis tidal turbine design is implemented. The result verifies the 2D vertical axis tidal turbine blade response model.

- Developing a 2D foil response model which is analysed in a range of constant parameters. These range of parameters represent the various operational parameters experienced by the turbine blade. The model employs steady incoming flow entering the domain model. The results determine the operational parameter to be applied in the blade response model. The results are recorded in fifteen seconds. However the result during the last twelve seconds is considered where the transient in early stage of a simulation has been passed.

- Developing a 2D vertical axis tidal turbine blade response model. The model represents the turbine rotation using periodic inflow equivalence method in which the turbine rotation is modeled by a time varying angle of attack with a constant magnitude inlet velocity entering the model domain. This allows the blade motion to model only the FSI response and neglect the rotational vertical axis turbine motion.

- Understanding how vertical axis tidal turbine blades response is controlled by applying three different blade trailing edge shapes. Time dependent lift and drag forces as well as moment forces on those different trailing edge shapes are observed to identify how the blades response changes and how the lock-in condition is affected.

1.4 Research Methodology

The objective explained in Section 1.3 will be achieved using a method to solve the FSI response of a vertical axis tidal turbine blade. For a vertical axis tidal turbine, the interaction between the structure and the fluid becomes a critical factor when the flow is unsteady. The analysis includes forces that act on the structure and the deformation caused by the forces. For a tidal turbine analysis, fluid-structure interaction is necessarily reviewed to ascertain that the turbine response under a certain condition (i.e. one particular Reynolds number) does not produce strong vortex shedding frequency which introduces lock-in to the vertical axis tidal turbine’s structure.

Lock-in (also called synchronization) is a phenomenon when the vortex shedding frequency is shifted to structure’s natural frequency (Blevins, 1990). It is one of the keypoints in this work and will be explained in Subsection 2.8.1. A vertical axis tidal turbine experiences a lock-in when the turbine blade response forces the vortex to shed near the blade’s natural frequency. The turbine lock-in condition is observed from its
vortex shedding frequency and is verified by the blade’s natural frequency which is detailed in Subsection 2.10.3. The vortex shedding frequency is obtained using the Phase Averaged and Power Spectrum Density (PSD) methods which will be discussed in Subsections 2.10.1 and 2.10.2 respectively. The Phase Averaged method returns a vortex shedding velocity from a velocity of reference and output points which is recorded for twelve seconds of simulation time. The vortex shedding velocity is further converted to frequency domain vorticity to identify main frequencies using the PSD method.

Lock-in increases and intensifies the turbine vibrations which can cause a destruction to the turbine thus it should be controlled. Controlling the turbine vibration during its operation will achieve a favorable tidal turbine design due to its structural stability. Vertical axis tidal turbine blade vibrations can also be detected through noise which is known as a *singing*, coming from the structure. The singing is reduced by limiting the vibration which is generated by vorticity. The vibration can also affect the turbine stability. One effective method for controlling vorticity is to modify the turbine blade trailing edge shape. This will also be a key point in this work. A challenging investigation to obtain a comprehensive understanding of fluid-structure interaction response influenced by the fluid flow on a vertical axis tidal turbine and the response affecting the fluid flow behind trailing edge is undertaken.

The study begins with by modelling a single foil at nine different static angles of attack and developing the model domain using two different grids. Both are analysed using an unsteady incompressible simulation solver in OpenFOAM, named pimpleFoam. Each grid is run at nine fixed angles of attack maintaining the same simulation parameters including the turbulence model (k-ω SST), residual control, solution solvers, etc. Lift and drag coefficients are recorded during fifteen seconds actual time and validated against published experimental and numerical results. This stage is performed to establish the grid which the best agrees with the validation data. This grid is adopted in the next stage. The unsteady results are expected to yield information on how the fluid regime behaves under various angles of attack and to observe the vorticity which may appear as a result of flow separation.

Following the fixed angle of attack unsteady simulation, an oscillating foil is modelled and analysed using the pimpleDyMFOam solver. Lift and drag coefficients and the fluid flow are observed as in the static foil models. This procedure is implemented on three modified trailing edge foils. The modifications include blunt, rounded and sharp edges. The modified foil results are later compared to original foil to determine the influence of trailing edge modification on lift and drag forces.

Based on the static and oscillating foil prediction, A 2D model of a three bladed rotational vertical axis tidal turbine is developed. The mesh is generated using Pointwise, imported into OpenFOAM and modelled using the pimpleDyMFOam solver. The model is developed to determine the boundary condition applying in the response model.
A foil response model is developed in a rectangular domain and simulated using pimpleDyMFoam solver. The model studies five parameters variation to represent an actual 3D case to be feasibly implemented in a 2D model. The parameters includes incoming fluid velocity variation, initial angle of attack variation, number of response orientation, stiffness and damping coefficient. The mesh is generated using the snappyHexMesh utility and the response is modelled by free vibrations using the sixDoFRigidBodyDisplacement boundary condition type applied on the foil body. From the study a standar operation which contains spesific CFD properties is determined. The typical CFD properties are also applied in the modified foil models and the results are compared to the original foil result. The comparison shows the influence of trailing edge to the foil response and how the modified foils are equivalent to the various operations experienced by a foil. This stage also determines the suitable turbine operation for typical turbine structure. This standard operation parameters are also applied in the vertical axis tidal turbine blade response.

The vertical axis tidal turbine with original and modified blades response model are modelled by taking one out of three blades and assumes that the blade experiences the same forces as the other two. The models are executed using standard operation which has been clarified in a foil response model with steady incoming flow. Vortex shedding frequencies are determined using The Phase Averaged and PSD methods. Vorticity is justified by pressure distribution contour. Lock-in is also confirmed by the displacement at the output point. In a lock-in condition, a turbine structure drives the vortices to shed in a frequency close to turbine blade’s natural frequency. This condition strengthens the blade vibrations which can be observed from the blade displacement. Lock-in induced vibration is indicated by a high amplitude displacement. Lift, drag and moment coefficients are recorded in fifteen seconds to examine the turbine’s performance.

All models run using k-ω SST turbulence model and utilizes NACA 0012 which has 0.75 m chord length before truncation. The blade modification is achieved by truncating the blade trailing edge for three different shapes. These are blunt, sharp and rounded. These modifications are introduced to control the vertical axis turbine vibrations effectively. The vertical axis tidal turbine original blade response model has the Reynolds number of 3.07x10^6 based on the resultant velocity magnitude which is produced from freestream velocity and angular turbine velocity. Courant number of 0.9 is applied for all models which creates non uniform time step during simulation.

All the vertical axis tidal turbine designs and CFD procedures will be detailed in this thesis. This thesis will cover all important aspects and breaks down into some chapters as outlined in Section 1.5.
1.5 Outline

This thesis is divided into 6 chapters to detail all works done during the study. The 6 chapters comprise the explanation and discussion of background theories behind a vertical axis tidal turbine design, fluid induced vibration including vortex shedding on a turbine blade, generating CFD response model and discussion of the results. The chapters order are as follows:

Chapter 1: Introduction

This chapter introduces motivations and general backgrounds which inspire the project. The background section also explains the preliminary study commencing this work. Aim and objectives are highlighted further to describe a concise overview of the project’s target. The thesis outline is detailed within each chapter to provide a brief description of the work which has been done.

Chapter 2: Literature Review

The theory for designing a vertical axis tidal turbine and for modelling the vertical axis tidal turbine are highlighted in this chapter. Parameters which are taken into account in the design process are physically described. Considerations for determining those parameters are also observed and confirmed by published literatures to support the decision making. This chapter also examines the concept of developing a 2D CFD model for a vertical axis tidal turbine. It explains two topologies generation (c-structured grid and rectangular grid), simulation set up and some design parameters which are applied to the model. CFD background theory will be discussed briefly as it is not the main focus of this project.

Fluid induced vibration which dominates the fluid structured interaction between the vertical axis tidal turbine structure and the tides is also discussed. The discussion covers the concept on modelling the fluid induced vibration and the generation of vortex shedding on foil which is influenced by pressure distribution over the blade surfaces. The effects of vorticity to the forces and vibration are also introduced. The fluid induced vibration is analysed to identify the resonance occurrence on the blade.

Chapter 3: Development of the CFD Methodology for Static and Dynamic Foils

This chapter discusses the stages to model the FSI of a vertical axis tidal turbine. It covers the fundamental theory behind the trailing edge vortex regime of a static and dynamic single foils and how to model it. Mesh independence study and validation in the static and dynamic single foils are also discussed. The dynamic mesh includes the oscillating foil. The validated mesh is further applied to modified trailing edge shapes model. The modified foil results are compared to the result of the original foil.
Chapter 4: Fluid Structure Interaction of a Foil with Steady Incoming Flow

This chapter discusses all models’ results including five different operational parameter variations. These include velocity, initial angle of attack, number of orientation response, stiffness constant, and damping constant. The vertical axis tidal turbine blade response model in a time dependent incoming velocity is further examined. The model is applied to the modified trailing edge profile blade and compared to the original blade. The method is expected to find the blade modification to reduce the fluid induced vibration on a vertical axis tidal turbine.

Chapter 5: Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

The fluid structure response in a vertical axis tidal turbine which is the focus of this work is detailed in this chapter. The response of interaction by means of vibrations is discussed and analysed using the vortex shedding and the blade displacement observations. The vortex shedding observation is verified by the pressure distribution over the blade’s surfaces. From the observation, the lock-in phenomenon which occurs in the blade can be justified.

Chapter 6: Conclusions and Recommendation for Future Work

This chapter summarizes the results achieved in this thesis and suggests a plan for future work.

The motivation and background of this work, together with the aim and objectives have been outlined. The thesis structure addresses a step by step understanding from a static foil point of view to a blade response phenomenon. At the end, the influence of trailing edge shape to a vertical axis blade response will be discovered and revealed.
Chapter 2

Literature Review

This chapter discusses the background theory and state of the art underlying this work from a vertical axis tidal turbine design, the CFD stages for developing a three bladed vertical axis tidal turbine model to the FSI method applied in a design of vertical axis tidal turbine blades. General aspects of a vertical axis turbine are also implemented and selected to construct a solid turbine with good performance.

2.1 A Vertical Axis Tidal Turbine

A tidal turbine is a device which converts tidal kinetic energy to mechanical energy by its rotation. Commonly a tidal turbine is coupled to a generator shaft so that the turbine rotation drives an electrical generator to rotate and generate electricity. There are two types of tidal turbine installation and operation classified by the way a tidal turbine rotates and by the way its axis of rotation is oriented. These two types, as illustrated in Figure 1.1(b), are horizontal and vertical axis tidal turbines. Horizontal or vertical refers to how the axis of rotation is positioned with respect to the incoming tidal flow. For a vertical axis tidal turbine, the turbine axis of rotation is vertical and perpendicular to the incoming fluid flow. This position gives some benefits when designing and installing a tidal turbine according to its location and environment. Therefore a vertical axis turbine is chosen for this project as it is more advantageous than a horizontal axis turbine. The advantage mainly comes for its compact vertical orientated structure. The vertical structure does not need a device to point the turbine to the tides direction which results in less parts required in the structure. In this case a vertical axis turbine can be constructed lighter and smaller which reduces the complexity when installing. A vertical axis tidal turbine size also reduces the operation maintenance and is more robust as all electrical and mechanical components can be placed above the tidal surface. This different construction between horizontal and vertical axis turbine also impacts the turbulence in both fluid regimes. There are also other advantageous aspect found in a
vertical axis over a horizontal axis tidal turbine for example as stated by Eriksson et al. (2008) and Riegler (2003).

In general there are two distinct types of vertical axis turbine; these are Savonius and Darrieus types as illustrated in Figures 1.1b. A Savonius turbine uses a drag force generated from the interaction with fluid to rotate the turbine rotor and extract the power. Meanwhile a Darrieus turbine which was named after a French aeronautical engineer who invented the first vertical turbine is driven by a lift force to harvest the mechanical energy. A Darrieus turbine has symmetric straight bladed rotor construction and lift force driven rotation which makes it more flexible and gains more power. The smaller size of vertical axis tidal turbine also makes it more feasible to transport to even extreme areas like a secluded island hence it can be installed easier. A careful design is required to adjust a vertical axis turbine to fit with its environmental condition during operation and to have a structure stability to stand for the fluid condition in that environment. Some vertical axis turbine aspects, as discussed in the next subsections, are reviewed in the design process to ensure a high performance and effective turbine operation.

2.1.1 Blade Shape and Blade Thickness

Many different types of foils have been developed for the past decade for different purposes. Those foils are classified according to its shape by two classes. These are symmetric and asymmetric foils. For a symmetric foil, the upper and lower surfaces have identical distance normal to the foil chord line. One example of symmetric foil is the foil from NACA class. On the other hand, for a asymmetric foil the distance normal to the foil chord line is different by creating camber in either surface. There are a number of works which have been done for choosing and optimizing symmetric and asymmetric foils for various purposes, for example studies by Xiao and Liao (2009), Ashraf et al. (2011), Islam et al. (2007), Sun (2012), and Ouyang et al. (2007). Most of blade shape studies rely on lift and drag forces to decide the profile which will be used.

In aerodynamic point of view, symmetric foil has some advantages such as aerodynamic center can be considered constant and does not move as the angle of attack changes. Lift and drag forces can be more easily observed at aerodynamic center which takes place at quarter of chord length behind leading edge. Aerodynamic center in quarter of chord length is theoretically and experimentally proven to have constant moment independently of angle of attack, also center of pressure and lift force. Thus moment at aerodynamic center is considered zero and that point is served as the center of blade rotation.

From construction and manufacture point of view, the blade production is the most challenging (Payne et al., 2017) in the whole vertical axis tidal turbine fabrication.
Therefore for practical purposes, a straight blade symmetric foil is preferably used in a vertical axis tidal turbine. Based on these basic reasons, NACA 0012 is selected to be used in the Darrieus vertical axis tidal turbine blade in this present work. Goett and Bullivant (1939) also found in their full scale experiment of NACA 0009, 0012, and 0018 that lift coefficient at $Re$ of three million was maximum on NACA 0012 (Table 1 page 4 of his paper). Therefore NACA 0012 is preferable for three bladed vertical turbine in this work. The four digit symmetric NACA family and its offset is calculated using the relation:

$$y_t = 5t_x \left[ 0.2969 \sqrt{\frac{x_t}{c}} - 0.1260 \left( \frac{x_t}{c} \right) - 0.3516 \left( \frac{x_t}{c} \right)^2 + 0.2843 \left( \frac{x_t}{c} \right)^3 - 0.1015 \left( \frac{x_t}{c} \right)^4 \right]$$

(2.1)

$y_t$ is blade y axis coordinate, $x_t$ is blade x axis coordinate, and $t_x$ is the maximum blade thickness fraction which is expressed by the last two digits of NACA type.

For four digits NACA symmetric blade, blade thickness is expressed with the last two digits of its code which means percentage of the blade chord length. The first two digit indicate the maximum and location of camber.

Gosselin et al. (2013) has studied the effect of blade thickness on vertical axis turbine performance by running simulations for four different NACA foils. He modeled his turbine using NACA 0012, 0015, 0020, 0025 at azimuth angle from 0° to 360° as depicted in Figure 2.1.

![Figure 2.1: Instantaneous power coefficient of different types of NACA on tidal turbine blades (Gosselin et al., 2013)](image)

Their result showed that the maximum power coefficient occurred at NACA 0012 and 0015. They also found that at that power coefficient peak, TSR of NACA 0015 is lower.
Chapter 2 Literature Review

than NACA 0012. Lower TSR induces unsteadiness in the fluid flow as explained in Subsection 2.1.4. However at azimuth angle higher than 90°, the performance of NACA 0012 drastically decreases because of the lower lift force gained by the turbine. This situation can be improved by applying blade treatment such as modifying blade profile.

2.1.1.1 Modified Blade

Modified foil is created to improve a vertical axis tidal turbine performance. The performance of a vertical axis tidal turbine is strongly affected by turbine thickness as shown in Figure 2.1. To improve the turbine performance, the NACA 0012 blade requires to have a treatment. The treatment will increase its lift force by increasing the blade thickness therefore it is designed to be truncated in the trailing edge section. In this work there are three different modifications suggested. Those are blunt, rounded and sharp profile as shown in Figures 2.2(a), 2.2(b), and 2.2(c) respectively. The influence of these trailing edge profiles in the fluid regime to increase a vertical axis turbine performance will be detailed in Section 3.4.

![Figure 2.2: The modified foil in perspective views: a. blunt foil, b. round foil, c. sharp foil](image)

The blade modification also creates a side effect on a blade structure during turbine operation. The trailing edge truncation generates singing because the fluid flow at the trailing edge is more likely to separate. The modified blade structure drives the vortex shedding which generates fluid induced vibration. This typical vibration creates a singing which also needs a further blade treatment to control it.
2.1.2 Aspect Ratio

The aspect ratio of turbine blade is defined as the ratio of the blade chord to spanwise length which is expressed in the relation:

\[ AR = \frac{c}{s} \]  

(2.2)

\( AR \) is turbine aspect ratio, \( c \) is turbine chord length and \( s \) is turbine spanwise length.

The aspect ratio of a turbine is of importance as it impacts the turbine structure stability. A vertical axis turbine is considered to have a high aspect ratio when the span length exceeds three longer its chord length. This makes it a slender structure with small width in its cross section area. A slender structure risks deflection with the largest deflection presents in the middle part of the blade’s spanwise length and cause failure when pressure forces exceed its fracture limit. When a vertical axis turbine blade experiences a time varying or dynamic load from unsteady fluid, the deflection exhibits similar time varying displacement and sometimes has an oscillatory vibration pattern which increases turbine fatigue failure. Thus in dynamic loading, the structure is loaded more intensely and should be investigated as a dynamic structure with a dynamic analysis under consideration.

Brusca et al. (2014) numerically investigated the correlation between aspect ratio to Reynold number and vertical axis turbine performance (power coefficient). They defined the aspect ratio as ratio between blade the span length and turbine radius and applied it in a symmetric straight-bladed vertical axis turbine. They investigated the correlation between aspect ratio and \( Re \), power coefficient, and Tip Speed Ratio (TSR). Their results show that for constant \( Re \) and power coefficient, higher TSR leads to smaller aspect ratio and on the other hand at constant \( Re \) and TSR, smaller aspect ratio occurs at higher \( C_p \). Another condition was also found that more power could be extracted if the aspect ratio of vertical axis turbine is higher and operates at higher Reynolds number. At an aspect ratio of less than one, The Reynolds number changes significantly as the size of turbine rotor changes as depicted in Figure 2.3.

Gosselin et al. (2013) also investigated the correlation between aspect ratio and turbine coefficient of performance. He modeled his two turbines with two different aspect ratio which are 7 and 15. He found that turbine with aspect ratios of 15 had higher turbine coefficient of performance reaching 26%. In this case, the higher aspect ratio results in the higher coefficient performance.

2.1.3 Number of Blades

There are few types of symmetric straight bladed vertical axis tidal turbine according to its number of blades utilized in the turbine arrangement operated. It was found
that number of blades used in the turbine induces blade contact surface with tides and contributes to power generation obtained. The higher number provides more surface area to extract power from tides, but a consideration should also be taken for torque applied at an initial starting condition. The more number of blades used require the more initial torque to rotate the rotor to start an operation and consequently decreases net power produced by a turbine. An optimum balance condition between both parameters becomes a concern in a turbine design.

The relation between number of blades, tip speed ratio and power produced in vertical axis turbine is studied by Gosselin et al. (2013) as shown in Figure 2.4. Gosselin et al. (2013) simulated three different numbers of blade assembled in the turbine which were 3, 4 and 9. In general greater number of blade being used causes the turbine to rotate smoother with less fluctuation and impacts. However adding more blades means increasing the solidity and leads to reduce turbine efficiency. Moreover with more blades to be attached, turbine has more surface to contact with fluid and produces more drag force at which can slow down the rotation and reduce power production. The larger blade’s surface interacts with fluid, the stronger vibrations occur on the blade as the fluids introduce more response to turbine structure.

An optimum condition should be determined which allows the turbine to extract as high power as possible but on the other hand also have less drawbacks from turbine efficiency and vibrations. For symmetric turbine configurations simulated by Gosselin et al. (2013)(Figure 2.4), it is seen that 3 blades employed with solidity of 0.1829 has the highest maximum power coefficient which reaches approximately 0.48 corresponding to aspect ratio of 16.04. The result from Gosselin et al. (2013) and Brusca et al. (2014) show that higher blade aspect ratio gained higher turbine coefficient of performance. It
also revealed that the optimum TSR value is shifted with increasing aspect ratio thus for TSR equals to 5.1, the aspect ratio is approximated to be 25.

### 2.1.4 Tip Speed Ratio

Tip speed ratio (TSR) is defined as the ratio between tangential velocity working on a turbine blade and free stream velocity, as shown in Equation (2.3). This is a design parameter which compares turbine rotational velocity to actual fluid velocity

\[
\lambda = \frac{\omega R}{V_\infty}
\]

where \(\lambda\) is tip speed ratio, \(\omega\) is turbine angular velocity, \(R\) is turbine rotor radius, and \(V_\infty\) is fluid free stream velocity (the tidal velocity).

Brusca et al. (2014) observed the correlation between tip speed ratio, solidity and coefficient of performance. These parameters are crucial for a turbine design as they influence the power extracted by a turbine. Their results show that at a Reynold number between \(1.6 \times 10^6\) to \(5 \times 10^6\), maximum power can be gained at solidity of 0.3-0.4 as indicated in Figure 2.5 which shows an example flow at \(Re\) of \(1 \times 10^6\). They simulate four different \(Re\) flows and the results show similarity at which maximum power coefficient was obtained at solidity of around 0.3 and TSR approaching 3.

TSR is also used to define the theoretical instantaneous angle of attack on a vertical axis tidal turbine blade as proposed by Gosselin et al. (2013) as shown in Equation 2.4. Their analysis used ANSYS FLUENT to diagnose the effect of solidity, number of blades, \(Re\), blade pitch angle of attack and blade thickness on a symmetric three bladed vertical axis turbine efficiency.

Figure 2.4: Effect of number of blades in turbines (Gosselin et al., 2013)
\[ \alpha = \arctan \left( \frac{1}{\lambda \sin \theta} + \frac{1}{\tan \theta} \right) + \theta - \frac{\pi}{2} \] (2.4)

\( \alpha \) is the blade angle of attack, \( \lambda \) is tip speed ratio, \( \theta \) is turbine azimuth angle.

The result is useful to predict the initial angle of attack at all azimuth angles or locations during one cycle of rotation. Figure 2.6 shows the initial angle of attack for various TSR over one cycle of rotation. It can be observed that a lower TSR gives a higher angle of attack at any position of a turbine rotation and the angle of attack influences fluid behavior across a turbine blade. A high angle of attack tends to produce unsteadiness and chaotic flow due to heavy vortex shedding. An angle of attack higher than 17° causes stall phenomenon which yields drawbacks such as high drag force and blade flutter. Accordingly, TSR consideration is essential in the design process of a turbine as it influences the fluid behavior across a turbine blade.

Figure 2.5: Influence of tip speed ratio to power coefficient at numbers of different solidity (Brusca et al., 2014)

Figure 2.6: Theoretical instantaneous angle of attack in a Darrieus turbine for different \( \lambda \) (Gosselin et al., 2013)
From Figure 2.6 it also can be seen that low tip speed ratio should be prevented as it results in high incident angle of attack and gives low turbine performance because of unsteadiness in fluid behavior.

A high TSR will be chosen in this design to avoid unsteady fluid flow behavior caused by stall phenomenon at a high angle of attack. The optimum tip speed ratio (λ) suggested by Gosselin was between 4 and 6 with the highest power extraction found at a tip speed ratio approximately 5 (Figure 2.6). At that λ, the angle of attack never reached dynamic angle of attack (around angle of 11°) which means the operation never have a stall condition. Therefore λ = 5.1 is chosen for this research.

### 2.1.5 Solidity

Turbine solidity is defined as the ratio of the area swept by turbine blade to the total area through which incoming fluid pass. Turbine solidity which is calculated using Equation (2.5) depends on number of blades, turbine diameter and size of the blades thus the solidity reflects the size and weight of a turbine.

\[
\sigma = \frac{N_b c}{2\pi R}
\]  

(2.5)

σ is solidity, \(N_b\) is number of turbine blades, \(c\) is the blade chord length and \(R\) is turbine radius.

High solidity is associated with high blade surface area which means the turbine is arranged with a larger number of blades, longer spanwise length or longer chord length. This design parameter should be taken into account with careful consideration as it has a big impact on turbine torque. High solidity requires high torque in accordance with higher mass to carry on rotation and lower drive force as less fluid is in contact with the turbine. Gosselin et al. (2013) numerically studied the interactions between TSR variations to gain power in a range of solidity as shown in Figure 2.7. At the low \(\lambda\), the stall occurs in the operation hence lessen the extracted power significantly and consequently the turbine efficiency as well. At high \(\lambda\), although the stall is not found but the power extracted is found less than at optimum range thus it is not suggested. There is an optimum solidity value to compromise these risks along with the situations behind the influence of solidity to other turbine properties.

Gosselin et al.’s result (2013) showed power coefficient is strongly affected by solidity TSR. For the same tip speed ratio, increasing turbine solidity generates more power to extract but the trend does not appear at the low \(\lambda\). The case is valid for TSR to its optimum condition which varies from one to another solidity. In general for lower solidity, a turbine should have higher TSR to obtain maximum power extracted for
instance in the condition of maximum gained power, TSR of 0.1372 solidity is higher than solidity of 0.5486. Low solidity also gives low vertical axis turbine self-starter.

2.1.6 Turbine Power and Coefficient of Performance

Turbine output power is fundamentally kinetic energy extracted from its rotation. It is predicted from kinetic energy of tides and derived from velocity of tides passing through turbine. However the real output power of turbine is of tide kinetic energy fraction. The fraction of turbine power from kinetic energy resources is so called coefficient of performance. By the definition, a real power turbine can be calculated from $C_p$ as in Equation (2.6).

$$ P_m = C_p P_w = C_p \left( \frac{1}{2} \rho AV_\infty^3 \right) $$

$P_m$ is the real turbine power, $C_p$ is pressure coefficient, $P_w$ is turbine theoretical power, $\rho$ is fluid density, and $V_\infty$ is the tidal velocity.

Coefficient of performance is not constant and for Darrieus turbine, it is strongly related and varied with tides speed, angular speed of turbine and angle of attack. For horizontal turbine, Albert Betz suggested that no turbine could not gain power more than 59.3% which means the turbine coefficient of performance cannot exceed 59.3% and called as Betz limit. Although he investigated the concept on a wind turbine, but it is also reliable and can be an early prediction for a horizontal tidal turbine. For vertical axis tidal turbine, Abdel Aleem et al. (2014) conducted an experiment to find tidal turbine $C_p$ and depicted in Figure 2.8.

Li (2014) defined and derived mathematically power coefficient (coefficient of performance) of ducted turbine and compared the result with traditional definition (non-ducted tidal turbine). He found ducted turbine coefficient of performance was smaller...
than power coefficient calculated from traditional equation. Thus for ducted tidal turbine, the shape and size of ducting system should be taken into account.

### 2.1.7 Reynolds Number

Reynolds number which was named after Osborne Reynolds, a scientist who socialized the concept in 19th century, is a dimensionless quantity defined the ratio of inertia forces to viscous forces of a fluid. The quantity also describes the behavior of fluid flow regime and resolve the turbulence, laminar or transition condition of fluid. Each condition carries different characteristic of flow and impact to a process of energy transfer. $Re$ is calculated as:

$$ Re = \frac{\rho L V_\infty}{\mu} = \frac{L V_\infty}{\nu} $$  \hspace{1cm} (2.7)

$Re$ is Reynolds number, $\rho$ is the fluid density, $L$ is characteristic length in this case is the blade chord length, $V_\infty$ is the tidal fluid velocity, $\mu$ is the fluid dynamic viscosity, and $\nu$ is the fluid kinematic viscosity.

The calculation of $Re$ in this work based on blade chord length for L and fluid mechanical properties listed in Table 2.2 which is found to be $3.07 \times 10^6$. However for single foil steady and unsteady simulation at fixed angle of attack and oscillating foil, the incoming fluid velocity is set to be maximum resultant fluid velocity magnitude calculated from tidal velocity and angular velocity. The resultant velocity is time varying parameter so the magnitude is different from time to time as illustrated in Figure 2.9(b). The resultant velocity magnitude reaches maximum when free stream velocity parallel to tangential velocity at zero azimuth angle and reach minimum when both velocities magnitude are opposite to each other at $180^\circ$ azimuth angle. The location where velocity magnitude
appears in extreme condition apparently independent from TSR, solidity and turbine aspect ratio. The maximum and minimum angle of attack associated with azimuth angle of turbine and resultant velocity with its corresponding $Re$ are listed in Table 2.1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Re</th>
<th>Azimuth (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max resultant velocity (m/s)</td>
<td>4.0016</td>
<td>$3.07 \times 10^6$</td>
<td></td>
</tr>
<tr>
<td>Min resultant velocity (m/s)</td>
<td>2.69</td>
<td>$2.06 \times 10^6$</td>
<td></td>
</tr>
<tr>
<td>Max angle of attack</td>
<td>11.31</td>
<td></td>
<td>102.41</td>
</tr>
<tr>
<td>Min angle of attack</td>
<td>-11.31</td>
<td></td>
<td>259.06</td>
</tr>
</tbody>
</table>

For single foil model, constant inlet fluid velocity and associated $Re$ is taken from resultant velocity rather than the free stream velocity so to provide prediction for lift and drag from actual vertical axis tidal turbine condition. Although the resultant velocity all locations in a circle of one rotation varies, the choice of maximum velocity in the simulation setting up is reasonable as to detect the phenomenon in the extreme condition. The fluid behavior is strongly influence by $Re$ and angle of attack of turbine blades and expected to be revealed, early predicted and for validation purposes in single foil models. The angle of attack is assorted into nine variations and the fluid flow regime behavior observed further as well as lift and drag coefficient at all those nine angles. The fluid behavior possibility will be detailed in Chapter 3.

### 2.2 A Basis Vertical Axis Tidal Turbine Design

A vertical axis tidal turbine should be designed properly to capture kinetic energy of tides with favorable performance or operates with high efficiency. The design parameters which have already explained in previous sections determine the turbine performance as to obtain maximum power extracted from total kinetic energy of tides interacts with turbine blades. The performance is indicated by turbine coefficient of performance which means the amount of power can be harnessed from total power swept by turbine blades as explained in Subsection 2.1.6. Coefficient of performance also means a turbine efficiency which reveals how much a turbine converts tidal energy into turbine rotation. Unlike the horizontal axis turbine of which maximum efficiency can be predicted from Betz’s law as discussed in Subsection 2.1.6, regardless its tides condition, for vertical axis turbine maximum coefficient of performance is still a new concept. A brief discussion has been stated earlier and found that the maximum coefficient of performance value reaches 0.45. However the value is not constant and varies with tip speed ratio, the turbine installation (ducted or not) and the turbine arrangement (tidal turbine farm). Other essential parameters such as aspect ratio, TSR, solidity, number of blades and thickness of blades are of importance so to determine and take into account carefully.
According to Gosselin et al. (2013), due to the benefit of narrow thickness of NACA foil for the application of vertical axis turbine, NACA 0012 is chosen for turbine blades in the present work with a chord length of 0.75 m. The turbine aspect ratio and tip speed ratio are designed to be 5.1 and 0.1829 respectively as numerically proven by Gosselin et al. (2013) and Brusca et al. (2014). This dictates the value of rotor radius from the governing Equations Equation 2.3 and (2.4). All parametric design and fluid mechanical properties are listed in Table 2.2.

### Table 2.2: Blade and fluid properties

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord (m)</td>
<td>0.75</td>
</tr>
<tr>
<td>TSR</td>
<td>5.1</td>
</tr>
<tr>
<td>Solidity</td>
<td>0.1829</td>
</tr>
<tr>
<td>Blade aspect ratio</td>
<td>25</td>
</tr>
<tr>
<td>Turbine aspect ratio</td>
<td>0.383</td>
</tr>
<tr>
<td>Kinematic viscosity (m²/s)</td>
<td>1.002e-6</td>
</tr>
<tr>
<td>Dynamic viscosity (kg/m.s)</td>
<td>1.03e-3</td>
</tr>
<tr>
<td>Density</td>
<td>1025</td>
</tr>
</tbody>
</table>

This work is a study case to be implemented in Indonesia Ocean to overcome the country’s issues to supply resources of electricity and distribution across the nation as discussed in Section 1.2. The environmental properties take Indonesian Ocean character with maximum tidal velocity of 0.65 m/s (Tillinger, 2011). Only the strongest interaction which is likely induced by the maximum tidal velocity is identified in this work. The maximum tidal velocity is accounted, however in the single foil model, the fluid velocity will be varied in order to understand how the interaction change in different tidal velocity.

The tidal velocity is assumed to be constant and independent to ocean depth variation during simulation which imply to have a constant $Re$ (discussed in the Subsection 2.1.7). When the tides flow and in contact with turbine blades, the incoming velocity acted on the blades in the 2D CFD model are a resultant velocity obtained from the free stream tidal velocity and tangential velocity as illustrated in Figure 2.9(a). This resultant incoming velocity has time varying magnitude and angle of attack on turbine rotation. The angular velocity is calculated using Equation (2.3) and found to be 1.7 rad/s. Time varying angle of attack of one cycle is calculated using Equation (2.4) with the TSR as discussed in Subsection 2.1.4. All essential design parameters are given in Table 2.2.

A design of optimization process can investigate all design parameters to select the best parameter combination. However the basis design with its specific properties is chosen based on the numerical and experimental reason as discussed in Section 2.1
2.2.1 Periodic Inflow Equivalence Model

The Periodic Inflow Equivalence Model is utilized in this work based on the time varying velocity and angle of attack working on the blade in turbine revolutions. A velocity vector working on a vertical axis tidal turbine blade in a revolution is depicted in Figure 2.9(a). It is a resultant velocity vector ($W$) which is returned from freestream velocity ($U$) and turbine tangential velocity ($V$). The resultant velocity acts as an incoming fluid velocity in 2D vertical axis tidal turbine blade response model. The angle between ($W$) and ($V$) in this model becomes the blade angle of attack. The magnitude of the inflow velocity and the angle of attack is time-dependently varied according to Equation (2.4) as shown in Figure 2.9(b).

A blade starts on the top of a revolution as shown in Figure 2.9(b). In that figure turbine tangential velocity is represented by blade speed (B) which has the same magnitude but opposite direction as ($V$). When azimuth angle is zero and tangential velocity coincides with the free stream velocity, the angle of attack acts on the blade in this location is zero thus resultant velocity reaches maximum. As the blade moves forward on the revolution and the azimuth angle increases, the angle of attack also increases and turns the resultant velocity magnitude lower until the blade reaches approximately $90^\circ$ azimuth angle. From that point onward until it reaches $180^\circ$ azimuth angle, the angle of attack is decreasing until it is found zero at $180^\circ$. During rotation, the angle of attack is a function of TSR as expressed in Equation (2.4). The angle of attack range lies within a certain positive and negative degree of angle as illustrated in Figure 2.9(c). The angle of attack magnitude history plot corresponding to azimuth angle for three different TSR are drawn in Figure 2.6.

The time varying angle of attack and velocity magnitude during a turbine revolution is effectively inflow velocity works on a blade in its rotation. The equivalence incoming velocity method takes this time varying inflow magnitude and angle of attack to replace the actual turbine revolution in the 2D CFD blade response domain. By using Equivalence Incoming Velocity Method, the blade can be located fixed without rotational movement in the domain. By applying a spring damper system on the blade, the blade motion solely represents a blade response as a result of the interaction with the tides. This FSI approach contributes for small simulation time. It is provided mainly by much smaller domain developed because only one blade is modeled. The OpenFOAM code expressing the Equivalence Incoming Velocity method is shown in Appendix A.1.

In this work a vertical axis tidal turbine is designed using a symmetric arrangement of three NACA 0012 blades. The symmetrical position of the three turbine blades is assumed to produce identical load applied on all blades during rotation. This allows developing the response model of a vertical axis turbine by applying a single blade in the domain. A single blade construction in a FSI response model can reduce complexity and execution time compared to a full three bladed vertical axis tidal turbine construction.
Figure 2.9: Illustration of: a. Angle of attack in vertical axis tidal turbine revolution (Danao et al., 2012) b. Time dependent rotational resultant velocity (relative wind) variation works on a vertical axis turbine blade (Nemec, 2001), c. Range of angle of attack on a blade during one rotation

The study begins by running the model using unsteady simulation with varied static angle of attack. It is continued by a turbine blade in a dynamic mesh for simulating a foil response with constant incoming velocity. The final target is achieved by modeling a single blade of vertical axis tidal turbine response with the equivalence inflow velocity.
approach to predict the vertical axis tidal turbine behavior as it interacts with the tides.

The unsteady fluid loading experienced by a turbine blade can easily generate an irregular and chaotic interaction which should be avoided in the operation of a turbine. The interaction can be controlled by replacing the original vertical axis tidal turbine blades with modified blades. In this work three modified blade profiles are proposed. These are blunt, rounded and sharp as seen in Figures 2.2(a), 2.2(b), and 2.2(c) respectively. All models are simulated with same design parameters and environmental properties as listed in Table 2.2. The Equivalence Incoming Velocity method is also applied in the vertical axis tidal turbine response model using modified blade.

2.3 A 2D CFD Model For A Vertical Axis Tidal Turbine Using OpenFOAM

The computational steps and background theory for modeling a vertical axis tidal turbine and a turbine blade will be detailed in this chapter. The computational description is based on the simulation procedure using OpenFOAM 2.2 in which all models are simulated. For the post-processing stage, all models are analyzed using Paraview 3.12.

OpenFOAM is an open source computational fluid dynamics (CFD) program which can be downloaded online (OpenFOAM foundation, 2017). OpenFOAM simulates a CFD case using the finite volume method which divides a domain into numerous cells and for each cell the Navier-Stokes and the mass integral equations (continuity equation) are applied. The Navier Stokes and the mass integral are written in Equations (2.8) and (2.9). OpenFOAM gives more power to predict the fluid dynamic problems than the CFD commercial package. OpenFOAM allows users to manipulate CFD parameters so that it can model various and flexible cases. The manipulation process is helpful and powerful aspect for an FSI approach for a vertical axis tidal turbine model due to dynamic inflow conditions and various turbine blade structures.

\[ \nabla \cdot \rho \mathbf{u} + \frac{\partial \rho}{\partial t} = 0 \quad (2.8) \]

\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \nabla) \mathbf{u} - \nu \nabla^2 \mathbf{u} = -\nabla p + g \quad (2.9) \]

The mathematical models of a fluid problem (Equations (2.8) and (2.9)) are a non-linear and coupled between the conservation term and inertia term thus the equations cannot be solved analytically. Therefore a numerical solution is required to resolve the model. CFD itself is a numerical method to approximately solve a fluid dynamics problem as the exact solution is difficult to obtain. However the approximation solution should return
acceptable fluid properties (Joel and Peric, 1999). The solution should be consistent, stable, converged, conserved, bounded, realizable and accurate. A consistent solution means that the error of the method tends to zero as the mesh or the time spacing tends to zero. The method also needs to produce a result where the value is not increasing during the calculation (stable) and should approach an exact solution (converged).

A numerical solution method is preceded by some stages/components which are performed in a sequential process. Initially one should construct mathematical models of the case. These mathematical models are a way to express a case in a constructively logical step for solving it in an intuitive manner. In most CFD problems, the mathematical models of a fluid are expressed by the Navier-stokes and the continuity equation. The varying value of the fluid at all points in the domain is resolved by approximating process. Once the mathematical models are constructed, the discretization method used should be determined. OpenFOAM implements the finite volume method which is based on the integral of a variable value over its boundary surface in a control volume. In the finite volume method, the surface integral is applied to the convective term and the volume integral is applied to diffusive term. The mathematical model and its discretization method are imposed on a numerical grid which is constructed within the domain, this can be two or three dimensional domain, depending on the focus of the investigation.

The surface integral of a variable at a boundary of a control volume is used to calculate the variable flux passing through the cell face. The flux is obtained using two levels of approximation. Firstly it is approximated from points on the cell face and secondly the cell face value is approximated from the nodal value at the centroid of the control volume. The value of the surface integral can be obtained from the mean value over the cell face. Since the value on the cell face is unknown, the value of the surface integral can be calculated using interpolation which is applied on the cell face from the value of the nodes nearby. The interpolation process will be explained in Subsection 2.3.2.3 discussing the processing process.

The Navier-Stokes equation (Equation (2.8)) is derived for a Newtonian Fluid in which the viscous stress is proportional to the angular deformation rate. The solution of the mathematical models, Equation (2.8) and 2.9, can be simplified when implemented for incompressible and inviscid flow. For incompressible flow, the compressibility term in the Navier-Stokes equation is cancelled and for inviscid flow the equation reduces to Euler’s equation.

The cells present in a plane region which has 4 boundary surfaces for 2D simulation and a volume region with 6 boundary surfaces for 3D. The unknown parameter in a convective and diffusive component of an integral equation must be solved for the entire control volume by means of calculating the net flux through the boundary volume. An integral equation calculates flux of a variable in the control volume with an approximation of two levels (Joel and Peric, 1999). The first level approximation calculates variable value
from one or more points from the cell face in the boundary surface. Cell values can be approximated in terms of value of the nodal centre thus it needs interpolation to get the values at the cell face.

The grid on which a variable value is interpolated and calculated can be organized in two different ways named structured or unstructured grid. A mesh is developed before the calculation of a variable begins (preprocessing step). The number of cells and grid topology are of importance to the CFD process because both strongly influence the simulation stability and the accuracy of the result. However a refined mesh or a large number of cells developed within a model does not guarantee to give an accurate result. But it is certain that a refined mesh model is executed longer.

Numerical methods for solving the problem of fluid dynamics can be complex and consume a lot of time due to the number of equations and iterations required for solving the model. The requirement for a high specification computer is sometimes necessary to acquire an accurate and time efficient calculation. The success of a numerical method depends on discretization (meshing) and choosing the correct scheme of interpolation to solve the equations. OpenFOAM operates in three stages including preprocessing, solving and post processing. Each stage which plays an important role in developing stable simulations and delivering accurate results will be discussed in the following section.

2.3.1 Developing The Domain Grid in Preprocessing Process

In the preprocessing stage, OpenFOAM creates a grid topology by discretizing the domain into cells on which the fluid equations are applied and solved. Each cell represents a finite control volume in the simulation. The meshing is generated by the blockmesh utility (BlockMesh command). In the blockmesh utility, by default the domain is decomposed into a hexahedral cells which are arranged with 8 vertices and 12 edges. The edge lines are straight by default although they can be set to curved lines by stating specified entries such as arc or spline in the blockmesh dictionary (blockMeshDict) (OpenFOAM foundation, 2017). blockMeshDict is located in the polymesh folder under the constant folder. The blockMeshDict is also equipped with a grading cells entry so that the mesh can be refined to expect better accuracy in results. The blockmesh utility is suitable for simple geometry objects and it arranges cells into a well-organized structured grid.

Two grid topologies are used in to construct the domain of a vertical axis tidal turbine blade model. These are c-structured grid and rectangular grid. After the domain is constructed, the object being studied is placed in the domain and the domain grid can be refined. C-structured grid is refined by setting the domain points while the rectangular domain utilizes snappyHexMesh utility. The method by which the snappyHexMesh utility works is to overwrite and refined the former grid which has been constructed in the blockMesh process employing four steps. First stage is to refine the mesh around
the object and this can be done by adding boundary layer and adjusting the level of the refinement in the grid between the boundary layer and the far grid. The number of boundary layers and the expanded height of each layer (expansion ratio) is defined in snappyHexMeshDict. The expansion ratio defined for a layer height in terms of the underneath layer whereas level of refinement determines the number of division assigned to each grid.

The second stage is to remove the unused cell to obtain a castellated mesh. A castellated mesh is achieved by removing the unused mesh from the object which intersect with the domain grid. In this stage a new object is produced by changing the object boundary line from the original construction. The object boundary line smoothness strongly depends on the level of refinement chosen in the previous stage. The higher the level of refinement, the smoother the new boundary line of the object being generated. However one should be aware that a very refined grid creates a larger number of cells thus it takes more time to apply the mathematical models on the cells and generate a solution. Beyond a certain level of refinement, the result remains the same thus it is recommended to carry out a mesh independence process prior to running the exact model. The third stage is to remove unused cells as results of cell refinement. The fourth stage is snapping the mesh onto the surface. This creates the line which defines the object surface. Finally the snappyHexMesh utility adds layers around the surface object. Following the domain grid and snappyHexMesh process, the grid quality needs to be examined and checked using the checkMesh utility. The purpose of a quality examination is to ascertain that a good mesh is being used for the simulation. The mesh should be refined and well-ordered to resolve the fluid flow and vortex behavior and also designed to run with sufficient computation effort. One of the advantage of snappyHexMesh is its capability to following a sharply curved body surface and refine a complex shape in a straightforward manner.

In the checkMesh utility, besides the mesh statistics, topology of the mesh and geometry of the cells are assessed. The mesh statistics reveal the number of components composing the grid, e.g number of faces, cells and points. In the topology and geometry assessment, the aspect ratio, non-orthogonality and skewness are displayed. The aspect ratio is explained in Subsection 2.1.2. Non orthogonality is a parameter to express the non-parallel of a face cell vector. An ideal cell is built with all line segments perpendicular to each other and in that case the parallel faces have a parallel face vector to each other. This is not always possible especially if the cells are near an object as the cell follows the shape of the object curvature and perpendicularity is rarely obtained. Skewness is a mesh distortion parameter which indicates the non-perpendicularity of a face related to its adjacent face.

In this work, models of NACA 0012 foil including the modified foils are constructed in Matlab R2016a and SolidWorks 2014. In Matlab the normal and modified foil shape is constructed based on NACA equation and its modification scenario. The foil is then imported to SolidWorks for creating the modifications and developing the 3D shape.
Finally the construction is saved into STL format before being attached in the snappy-HexMesh dictionary (snappyHexMeshDict). In the snappyHexMeshDict, the refinement process uses level 5 refinement and adds seven layers around the foil surface. The mesh and snapping process is done following the grid construction in this work.

After the grid domain is established, the mathematical models are calculated on the designated control volume grid. For finite control volume discretization, the value of fluid properties are applied in nodal center (the centroid of the CV). The value of the fluid properties being calculated in that cell are the flux value and are approximated using two stages as mentioned earlier. The process to calculate the fluid properties value is detailed in next subsection.

2.3.2 Solving Mathematical Models in Processing Stage

The mesh which has been generated is then assigned to the mathematical models. The models are simulated using a solver provided by OpenFOAM. A steady state and incompressible fluid utilizes the simpleFoam solver to solve the mathematical models. A vertical axis tidal turbine response case is an unsteady and incompressible case. Such a case is simulated using pimpleFoam solver and pimpleDyMFoam solver for the dynamic case. The solver solves the problems based on an algorithm which is controlled by some numerical schemes and interpolations scheme in fvScheme dictionary within the System folder. The interpolation scheme will discussed in Subsection 2.3.2.3.

2.3.2.1 Solver Algorithm

simpleFoam which is a steady state solver in OpenFOAM employed for incompressible turbulent flow, solves the Navier-stokes equation based on the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm. The algorithm procedure begins by setting initial condition in the domain including flow velocity, turbulent kinetic energy and inlet pressure. Other parameters such as density and dynamic viscosity remain constant during the calculation. In the next step, the solver solves the velocity field via the momentum equation for all cells and the mass flux is then calculated. The result is substituted to find the pressure in the equation. The resulting pressure is then fed back to the mass equation and is used to find the velocity value. All parameters from the first calculation are applied at the boundary conditions and then the results are fed back into the calculation process until the solution converges. The converged criteria depends on the difference of the initial value of all the parameters and the values obtained from the previous result (residual factor). All residual factors need to meet convergence criteria simultaneously to provide a converged simulation.

Another algorithm used in this work is PISO algorithm. PISO algorithm is built for unsteady simulations and is used for the pimpleFoam and pimpleDyMFoam solver in
OpenFOAM. It is similar to the SIMPLE algorithm but without applying the under relaxation factor. The correction of pressure and velocity field is performed several times, therefore the unsteady simulation normally produces more accurate result than SIMPLE algorithm. The simulation stability depends on the Courant number which is interpreted as the velocity of fluid travels through a cell length for a definite time.

2.3.2.2 Turbulence Model

As mentioned in Chapter 2, the characteristics of fluid behavior are determined from the fluid Reynolds number. At $Re$ of above $5 \times 10^5$, the fluid flow regime is considered as turbulent. In the turbulent flow, all fluid properties are transferred inconstantly. All fluid properties in the turbulent flow are decomposed into mean and fluctuating components. The unsteadiness and the fluctuating component in the turbulence makes the quantity of the flow properties (e.g. velocity, pressure, momentum, etc) are transported irregularly thus the value is difficult to calculate precisely. Thus the result is approximated and calculated by the implemented turbulence model. A turbulence model is defined as a concept to approximate the mean flow properties and predicts the influence of the fluctuating flow to the overall flow. The unsteady and fluctuating parts of the turbulent flow is caused by complex interactions within the fluid as a result of viscous and inertial behavior of fluid particles. The unsteady interaction creates an irregular and random flow. A turbulent flow is also characterized by diffusivity and dissipation of the fluid particles.

There are several turbulence models accepted for engineering analysis. They are RANS-based (Reynolds Averaged Navier-Stokes), LES (Large Eddy Simulation) and DNS (Direct Numerical Simulation) turbulence models. RANS-based turbulence model which is used for a wide range of engineering problems, is classified by zero, one and two equations. The classification indicates the number of equation being solved. In this work, $k-\omega$ SST turbulence model from the two equations class of RANS model is used.

The RANS model solves the Navier-Stokes equation based on the Boussinesq hypothesis and the Reynolds Stress Models for stress in fluid. In 1877 Boussinesq proposed that the mean rate of fluid deformation can be linked to Reynolds stress. The method by which the quantity is transported is strongly related to the deformation (turbulent flow). A new quantity was proposed for solving the model which was named turbulent viscosity. The concept of turbulent viscosity is similar to viscosity in the common fluid analysis. Another quantity which is also significant for turbulent flow is turbulent kinetic energy. It is the mean of the kinetic energy in a turbulent flow and is associated with turbulent eddies. The term is important because it influences the energy dissipated in the turbulent flow and the transportation of any turbulent quantity. The $k-\omega$ SST turbulence model was firstly introduced by Menter (1993; 1994). The two equations calculated in this model are the turbulent kinetic energy equation ($k$) and a time scale parameter
associated with turbulence ($\omega$). $k$-$\omega$ SST is applied in all cases in this work following other works which is previously successful to solve their cases using this turbulence model.

For the case of a static angle of attack Eleni et al. (2012) built a model of a NACA 0012 at Reynolds number of $3 \times 10^6$ and analyzed the model using sparatAlmaras, $k$-$\omega$ SST, and $k$-epsilon turbulence models. The results were validated using experiments from Abbott (1959). The lift and drag coefficient from his simulation result showed that the simulation using $k$-$\omega$ SST were in best agreement with Abbot’s experimental result. Eleni also performed a mesh independence investigation and found that 80,000 cells were the minimum number to reach mesh independent solution. Frederich et al. (2009) was also simulated his oscillating foil using $k$-$\omega$ SST and validated his model to McAlister’s experiment (McAlister et al., 1982). He found that the result using $k$-$\omega$ turbulence model agreed with McAlister’s experiment. $k$-$\omega$ SST turbulence model is also applied by Ducoin and Young (2013) for his hydrostatic response model which is validated by his own experiment. He developed the foil response model using a spring damping system. His result showed that $k$-$\omega$ model had a good agreement with experiment and can be implemented for an FSI model.

### 2.3.2.3 Interpolation

As mentioned earlier, a surface integral takes an interpolation into account for approximating a flow quantity value. Interpolation is used to approximately determine an unknown value between two or more definite value. Since it is an approximation, the accuracy of an interpolation scheme is always a concern thus a term is used, called order of interpolation. The higher the order, the more precise the approximation.

There are four interpolation schemes commonly used in CFD, these are zeroth, first, second and higher order. The order denotes the degree of the polynomial interpolation function. In zero order interpolation, the value is approximated by mean value of the function while the first order interpolation uses a linear function for the approximating. A linear interpolation scheme is an example of first order interpolation. In second order interpolation such as upwind scheme, an unknown value is approximated using a quadratic function. This scheme is similar to higher order interpolation which accounts for a three degree function to generate the unknown value. QUICK and cubic are two examples of higher order interpolation schemes. In the short history and application of CFD, a higher order interpolation scheme is believed to be more complicated, less robust, slower to converge, and require higher memory than lower order interpolation. However Wang et al. (2013b) found some advantages in the implementation of higher order schemes in CFD. They investigated and compared higher order interpolation versus lower order interpolation scheme in terms of the simulation error. They found that
for a smooth geometry and a smooth function, higher order interpolation scheme demonstrated better accuracy although the CPU running time is longer. In this work a cubic interpolation scheme is used for approximating the surface integral.

A cubic scheme is a higher order differencing scheme using a third degree function which has four unknowns thus four equations are needed for its solution. These equations require four conditions to solve which can either be satisfied from the value of the function and its derivative at two points closest to the interpolated point or from the value of four different points close to the interpolated point. This scheme produces a quite accurate value in this case as the foil and the blade is considered to be a smooth geometry (Wang et al., 2013b).

2.3.2.4 Boundary Condition

In CFD, this is a condition which satisfy the boundary of a region when resolving a set of differential equations on the region. When using the control volume method, the region is a face/surface of a cell or an object and the differential equations to be solved are usually momentum and continuity equation.

In OpenFOAM there are many boundary condition features which is defined at the cell face. Boundary conditions should be set to all cell face which are inlet, outlet, front, back, bottom, top and the object. Some standard boundary conditions in OpenFOAM which is used in this work are empty, fixedValue, calculated, inletOutlet, and zeroGradient. Other boundary conditions can be found in the OpenFOAM user guide documentation (OpenFOAM foundation, 2017)

empty boundary condition (BC) is assigned at back and front face of the domain for all properties in the zero folder polymesh, k, U (velocity), p (pressure), (ω) and nut (kinematic turbulent viscosity). This type of boundary condition provides an empty condition which means there is no solution in the direction perpendicular to the surface. This BC normally is used to reduce dimensional analysis in CFD. In this work, it is applied to develop 2D model of 3D turbine case. In this case OpenFOAM neglects the space between front and back region which corresponds to the spanwise length of a foil. Thus the spanwise length in z direction is negligible and the model is considered as a 2D case.

fixedValue is a BC chosen for a constraint value that does not change during simulation. This boundary condition is designated for the no slip condition on the velocity field at the surface of the foil. inletOutlet is specified on the outlet patch with assigned inlet condition and is also used for velocity at the outlet cell face. At the outlet cell faces, the velocity is calculated and requires information from the inlet. zeroGradient boundary condition applies constant value at the patch at which it is assigned. This type of boundary condition is used to specify the pressure in the inlet. Other type of
boundary conditions are described briefly in the OpenFOAM user guide (OpenFOAM foundation, 2017).

At the surface of a foil, turbulence model functions such as $\omega$, $k$, and $\nu$ should be defined. All these functions will calculate the turbulence properties at each cell. The boundary condition for the wall also determines the fluid condition in the boundary layer including the viscous effect. In near wall region, viscous effect become stronger and influences the behavior of fluid flow pass through it. To overcome this problem in CFD normally the mesh near wall region is refined. However this method cause another problem such as increasing execution time thus it can slow down the simulation. Another way which will be introduced to resolve this problem, is the use of Wall Function Theory. A new term is introduced, called $y+$. $y+$ is a parameter denotes the condition of the first layer from the wall in Wall Function Theory.

The idea of wall function theory is to assign the near wall region with a value which describes the near wall layer condition according to its distance from the wall, shear stress and viscosity using $y+$ ($y+$) value. Coles (1956) explained the implementation of wall function in a turbulent boundary layer. $y+$ can also be interpreted as the distance from the boundary layer’s first cell to the wall. Wall function requires the first cell type to be in the log layer region (see Figure 2.10) in order to implement Law of the Wall which declares that the average velocity of a turbulent flow at a certain point, which can be regarded as $y+$, is proportional to the logarithm of the distance from that point. The Law of the Wall is calculated from Equations (2.10) - (2.13).

$$u+ = \frac{1}{\kappa} ln(y+) + C^+$$  \hspace{1cm} (2.10)

where

$$u+ = \frac{u}{u_r}$$  \hspace{1cm} (2.11)

$$u_\tau au = \sqrt{\frac{\tau}{\rho}}$$  \hspace{1cm} (2.12)

$$y+ = \frac{yu_\tau}{v}$$  \hspace{1cm} (2.13)

The Law of the Wall concept is valid for high $Re$ and applicable for $y+$ values above 30. In the region where $y+$ is lower than 30 the correlation between the first cell distance from the wall and the velocity follows a different pattern as shown in Figure 2.10. Below $y+$ value of 30, the layer can be categorized as a buffer layer (approximately $5 < y+ < 30$).
or as a viscous sublayer (y+ below 5). In a region which has y+ below 30, the Law
of the Wall cannot be implemented thus Equation (2.14) is used.

\[ u^+ = y^+ \tag{2.14} \]

![Figure 2.10: Log of the wall region (Cradle, MSC Software Company, 2016)](image)

In this work the y+ value is more than 30 so the first cell near the foil wall is in
the log-law region. The boundary condition on the turbulence model properties are
determined from wall function theory for example kqRWallFunction is chosen for k
boundary condition. Other boundary condition assigned are nutLowReWallFunction
for the kinematic turbulent viscosity and omegaWallFunction for omega.

### 2.4 Dynamic Mesh in OpenFOAM

In a CFD model a mesh plays an important role in the process to obtain a good result.
Dynamic mesh refers to moving cells in a CFD model. It is an essensial feature in a
FSI model to represent a response of the interaction. In a vertical axis tidal turbine
blade response case, the blade is vibrating which requires a flexible moving mesh to
accomodate the blade vibration.

Dynamic mesh features in OpenFOAM are handled by a solver with suffix DyMFOam. In
this work, the solver used is pimpleDyMFOam which is applied to an incompressible and
inviscid flow using the PIMPLE algorithm. There are two methods (classes) of dynamic
mesh manipulation provided in OpenFOAM. First method of dynamic mesh is named
dynamicFvMesh. It refers to a mesh whose cell shape is changing during the simulation.
In the second method, named topoChargerFvMesh, the mesh size and shape remain
the same but the mesh topology changes. For each class OpenFOAM provides several
sub-classes of manipulation mesh method to define the desired motion in the model.
Chapter 2 Literature Review

The class and sub-class mesh manipulation is defined in the dynamicMeshDict file under the constant folder. An example of subclasses mesh manipulation under the class of dynamicFvMesh are dynamicMotionSolverFvMesh, solidBodyMotionFvMesh, etc. In this work all dynamic mesh models including 2D vertical axis tidal turbine, oscillating foil, the foil and blade response model assign dynamicFvMesh with the subclass of solidBodyMotionFvMesh for 2D vertical axis tidal turbine model and dynamicMotionSolverFvMesh for oscillating foil and foil response. In the dynamicMotionSolverFvMesh, the mesh point’s motions are solved using diffusivity model in which the boundary condition of the wall is applied. The solidBodyMotionFvMesh used by oscillating foil and foil response prescribes a the mesh motion by a motion function defined in pointDisplacement dictionary in the 0 folder. 0 folder specifies simulation parameter at initial condition.

In the dynamicMeshDict file, one has also to define the solver for resolving mesh manipulation method. There are four different solvers for dynamicFvMesh by OpenFOAM, these includes displacementLaplacian, velocityLaplacian, LaplaceFaceDecomposition, and SBRStress. In this work displacementLaplacian is chosen to solve the dynamic mesh model. In the displacementLaplacian solver the motion is determined by the equation of Laplacian diffusivity and the prescribed motion boundary condition defined in the pointDisplacement dictionary. The pointDisplacement dictionary basically is additional set of equations needed in order to resolve the dynamic mesh mathematical model when a mesh requires an equation to define the motion.

In an oscillating foil and foil response model, the time dependent motion is prescribed at the foil boundary in the pointDisplacement. OpenFOAM provides the prescribed motion features to set up the motion of an oscillating foil. In this work angularOscillatingDisplacement is assigned to model Frederick’s foil oscillations (Frederich et al., 2009). angularOscillatingDisplacement states the axis of motion, the point of origin, the initial angle, amplitude and angular velocity to define the foil’s oscillatory motion. The oscillating foil patch position at each time step is resolved based on the input from the motion feature file. Different from the oscillating foil, for a foil response where the motion is influenced from the interaction with a fluid, the fluid mathematical model coupled with the dynamic motion equation are solved together. The dynamic motion in this case is controlled by a set of vibration equations which model the motion response of the foil. Similar to the oscillating foil model, the response motion in the foil response model is also defined in the pointDisplacement file in the 0 folder. The typical features for vibration-modelled response to be used is sixDoFrigidBodyDisplacement.

An additional sub-dictionary in the pointDisplacement file other than sixDoFrigidBodyDisplacement are the constraint and the restraint sub-dictionary. In constraint sub-dictionary, the vibration motion is assessed in terms of the direction and axis of vibration whereas in the restraint sub-dictionary the spring and damper system is applied. The dictionary of sixDoFrigidBodyDisplacement requires the foil mass, centre of mass, moment of inertia and density of the fluid to be declared. Mass is predicted
from the density of the foil material which is designed as a composite material. However
the mass is not purely obtained from the mass of the foil alone since it operates in the
water so the added mass should be considered. The added mass is a virtual mass of
an object and fluids around the object when the body moves in the fluid. In general a
moving body mass in the fluid is bigger than in air since the body inertial force in fluids
is greater due to higher drag force.

The centre of mass and moment of inertial are calculated based on the foil’s shape. The
moment of inertia is examined in 2D with respect to z axis. It is an expression for the
capability of an object to rotate and is calculated using Equation (2.15).

\[
I_{zz} = \int r^2 dm
\]  
(2.15)

All these simulation properties are applied to the foil response and vertical axis tidal
turbine model.

2.5 A Three Bladed Vertical Axis Tidal Turbine

Vertical axis turbine designs are experiencing a progressively technological advance.
The advance is not just because it is employed in various models and configurations
but also the power transmission and energy harnessing ability is increasing [Khan et al.
(2009), Bhutta et al. (2012)]. The designs intent is to minimize operation failure and
suit ability to the environmental condition where it is installed. Growing computer
technology creates a large contribution to the fast-growing turbine designs and helps
researchers to accelerate their observations with accurate results. Complex phenomenon
can be more easily discovered and visualized with less costly methods using CFD which
is more preferential to perform than actual experiments. Some researchers developed
a tidal turbine model to gain a better understanding the physical phenomenon behind
its rotation such as models developed by Gretton et al. (2009) and Lanzafame et al.
(2014).

Gretton et al. (2009) developed a model of a vertical axis tidal turbine and validated it
with some Sheldahl and Klimas (1981). He started his studies by modelling a single fixed
angle of attack foil and continued with a foil in an oscillating motion and finally built a
model of a three bladed NACA 0012 vertical axis tidal turbine. He used an O-structured
grid for the tidal turbine model and performed it with k-\omega SST turbulence model and
found the tidal turbine performance as function of tip speed ratio and azimuth angle as
seen in Figure 2.11. This work is also started with the analyzing of single and oscillating
foil prior to a three bladed vertical axis tidal turbine and the foil response model.
Lanzafame et al. (2014) developed a 2D CFD model of vertical axis tidal turbine using a transition turbulence model. The model employed a hybrid mesh which was comprised of a structured grid in the foil vicinity and an unstructured grid in the far field region. The simulation was validated using his own experiment and both approaches showed good agreement. The far field region was a rectangular fixed domain in which the rotating turbine domain was allocated. Some boundary conditions (BC) were defined including inlet, outlet, symmetry for the top and bottom parts, wall and interface BC. The interface was the contact surface between the fixed domain and rotating turbine domain. The model as shown in Figure 2.12, is later adopted with some modifications for this research to predict the vertical axis tidal turbine trailing edge vortex behavior and will be detailed in the following chapter. Gosselin (2013) studied the relations between some parameters for designing vertical axis turbines using k-ω SST turbulence model. The parameters included solidity, number of blades, Re, pitch angle (fixed and
variable), and blade thickness. The study was done to obtain the best aerodynamic configuration. He analyzed and compared three different turbulence models, these include Spalart Almaras, k-ω SST, and transition SST and he found that k-ω SST was the most robust and gave most consistent result. He also derived an equation for calculating an instantaneous angle of attack with the assumption of uniform free stream velocity as in Equation (2.4).

![VATT simulation domain](image)

**Figure 2.12: VATT simulation domain (Lanzafame et al. 2014)**

In this chapter fluid structure interaction (FSI) which is the main topic of this work will also be discussed. FSI is a branch of science which discuss an interaction of a deformable structure with its surrounding fluids. The surrounding fluids produce a loading and deforms or move the structure. The phenomena is also found in the operation of a vertical axis tidal turbine which will be discussed in this chapter.

### 2.6 Introduction to FSI on a Vertical Axis Tidal Turbine Blade

In the fluid structure interaction of the tides and the blades, the fluid loading is unsteady and generates a vibration as a response of the interaction. The vibration is an important issue in a structure under dynamic loading. In the fluid structure interaction, during operation an immersed structure experiences a dynamic fluid loading which mainly originates from the unsteadiness of the fluid flow. The dynamic loading of the fluid can cause irregular fluctuations and an unpredictable loading. Such a load induces unpredictable vibrations which can be harmful to the structure. However by understanding the behavior of the fluid flow regime around a structure, the vibration problem is avoidable.

A typical structure immersed in a shallow depth of water such as a vertical axis tidal turbine is subjected to a dynamic loading mainly coming from fluid velocity variations.
The varying fluid velocity sweeps passing the turbine blades and generates a lift force which is beneficial to rotate the turbine. However at the same time, the fluid flow also introduces vibrations which raise stability problems on the turbine blades and the structure, in addition to some other issues such as fatigue failure and structure lifetime reduction. The instability appears as the effect of vortex shedding or wake generation on the blade’s surface in the downstream flow region. The vortex and wake interact with the blade and the interaction causes the blade to vibrate. The vortex itself is understood as a mechanism of the changing in the fluid pressure when the fluid hits the blades leading edge and suddenly the pressure is raised from free stream to stagnant pressure. It can be dictated from a pressure distribution plot over both foil’s surfaces. The stagnant pressure on the front of a foil reduces velocity in the lowermost of the boundary layer near the foil surface. When the reducing velocity is very low, a reverse fluid velocity and a big velocity discrepancy between layers occurs. The reversed fluid particle on the lower layer is rolled up by the above particle forming a vortex and flow downward to the fluid flow regime as shown in Figure 2.13. The vortex shedding around foil will be detailed more in Section 2.7.

![Figure 2.13: Vortex on NACA 0012](image)

The mechanism of vortex generation is understood to happen when the fluid collides at the stagnation point of the leading edge and the pressure alters from stream to stagnant pressure. The high pressure at the stagnant point forces the fluid particle to move downward along the foil surface and creates layers within the fluid flow. The layers exist as the effect of velocity difference in the fluid in the direction of perpendicular of incoming flow. The velocity variation exists because of the fluid viscous effect. The innermost layer which is located at the nearest part of the foil surface, has the lowest velocity because of the highest friction stress which occurs between the fluid and the surface. In contrast, the outermost layer flows fastest as it lays next to free stream flow and has the least friction stress. In such way, the flow has a velocity profile which is gradually increased from the foil surface to the upstream and creates a velocity boundary layer on the surface of the foil. The velocity variation within the boundary layer becomes higher as the fluid flows further along the surface and produces the highest velocity on the
uppermost layer. This configuration rolls the fluid particles in the bottom layers when flowing down and generates vorticity which is shed towards the trailing edge region as seen in Figure 2.13. The emerging vortex from the velocity layers can be estimated and detected from the Reynolds number.

Many researches focused their studies on how the vortex was induced by Reynolds number since the early of the century such as studied by Blevins (1990). He started his study with a simple and common structure such as a cylinder. A correlation between vortex shedding and $Re$ on a cylinder has been proposed by Lienhard (1966) (cited in Blevins, 1990) as shown in Figure 2.14. Figure 2.14 displays the vortex street is being shed at the near wake of a single cylinder at various $Re$.

In Figure 2.14, it is shown that below $Re$ of 5 the flow is regular and appears to be laminar without vortex creation. As $Re$ is increased to 40, the fluid remains laminar but the vortex generation is detected. Vortices are formed on two opposite direction pulses at the back of the cylinder. The generated vortices are also laminar as indicated by the two well organized regime of vortex shedding from the trailing edge. From $Re$ equal to 40 up 90, regular laminar staggered vortices are shed alternatively opposing each other (vortex street). From then, at higher $Re$, the wake becomes unstable and causes the vortices to break (Friehe, 1980. Cited in Blevins 1990) which induces turbulent flow. In this condition the flow is categorized in the transition period which is identified by turbulent fluid flow regime but with a laminar boundary layer on the structure (Roshko, 2012. Cited in Blevins, 1990). For much higher Re, for instance $Re = 10^6$, fluid flow becomes fully turbulent which is characterized by an irregular and unsteady fluid flow regime. In the transition phase and turbulent flow, the vortex becomes more chaotic and disordered, and the fluid boundary layer is very irregular.

The irregularity caused by the vortex excites a vibration (vortex induced vibration) on the structure as discussed by Blevins (1990). He found that the vortex induced vibration in the subsonic flow is created by oscillating surface pressure. The dynamic pressure load causes the structure to vibrate and to incur a potentially serious defect on the structure. Therefore the understanding of how the vortex is generated on the foil during the operation of a tidal turbine is of importance to predict the vibration and prevent the turbine from severe damage.

The actual FSI response is a three dimensional physical phenomena working on a vertical axis turbine blade. The response is dictated by fluid loading from all blade’s parts. This is very complex to understand and solve in a 3D CFD model. The validity data is also very rare and not easy to obtain in an experiment. Therefore a 3D case of a blade FSI response can be modeled by representing the third dimension by stiffness and damping properties. The damping and stiffness constant is taken to represent a part of the blade structure b choosing appropriate the spring constants. However the detail of the spring constants related to the represented blade part is beyond the focus of this work. The
aim of this research is to understand the FSI of fluid structure interaction of a vertical axis tidal turbine blade as discussed in Section 1.3. The FSI model investigates the vorticity changes and a blade response in a selected set of the blade properties.

2.7 Vortex Shedding around A Foil

A foil is a critical device utilized for hydro and aero dynamic purposes. The vortices around a foil trailing edge are considered as a source of vibration on a turbine blade and affect the overall turbine structure. In general, the mechanism of vortex generation on a foil is similar to the aforementioned phenomenon of vortex development on a cylinder in Figure 2.14, only that the separation is delayed because of the trailing edge foil geometry. A foil has a smooth sloping trailing edge which avoids the effect of a bluff body at the back of a structure. Consequently the fluid is attached to the foil surface longer thus separation and the development of vortices can be delayed.
Vortices on a foil are also influenced by $Re$ and angle of attack, as previously explained. For the effect of $Re$, the profile of the vortices on a foil is similar to the patterns shown in Figure 2.14. In high $Re$ situation (in millions) where the vortex shedding forms alternating staggered vortices (vortex street), an opposing pressures give a dynamic loading to the foil and has more potential risk to damage of the structure. The vortex generated on a foil also depends on the angle of attack in which the foil operates. The vortex is shed stronger when the foil’s angle of attack is higher due to high pressure gradient in the direction of the fluid flow as mentioned earlier. The vortices which were generated on a fixed angle of attack foil were investigated by many researchers. At zero angle of attack, the fluid around foil is regular and no vortices are found as seen in Figure 2.15.

![Figure 2.15: Flow pattern at zero angle of attack foil](image)

When the angle of attack is increased from zero degree, the boundary layer is laminar and no vortices appear in the fluid stream until the angle of attack reaches approximately 17° (Mittal and Saxena, 2002). At angle of attack greater than this, the vortex generation starts from the leading edge of a foil and travels streamwise along the foil surface. The higher the angle experienced by a foil, the bigger the vortices which are produced at the leading edge. The flow begins to separate as the vortex detached the foil surface and the foil is likely to experience a stall in this stage.

In fluid dynamic science a foil shape is known to delay fluid separation. Its shape is also designed to deliver higher lift and handle the fluid detachment as the angle of attack rises. However the characteristics of inviscid flow which creates layers on the foil surface cannot be avoided and establish some disadvantages in the fluid flow in general. The drawback is associated with vortex generation near the leading edge which is related to Reynolds number as explained previously. The higher the $Re$, the stronger vortex which is shed to the downstream flow near the trailing edge and leads to stronger vibration. A strong vibration creates a serious problem when the vibration frequency is close to the vortex shedding frequency. The condition shifts vortex shedding frequency close to natural frequency and the lock-in frequency will occur. Bishop and Hassan
Chapter 2 Literature Review

(1964) defined lock-in frequency as a synchronization between shedding and natural frequency which increase the strength of shedding frequency. In the case of a structural vibration at a frequency near shedding frequency, lock-in phenomenon allows shedding frequency to shift the frequency of vibration. Lock-in frequency can be dangerous for a marine structure because the effect is mutually dependent, a vortex becomes stronger by the vibration and this causes the structure to vibrate harder than its oscillation limit. Accordingly the structure might vibrate beyond its fatigue limit. Therefore much research in the area of vortex induced vibration focuses on determining and exploring lock-in frequency more intensively such as that conducted by Besem et al. (2014).

Vortices on NACA 0012 at $Re$ ranges from $10^4$ to $10^5$ was also investigated experimentally by Jung and Park (2005). He found that the range of vortex shedding frequencies at a fixed angle of attack foil was higher than that of an oscillating foil. In his static angle of attack foil experiment, the vortex shedding frequency itself was decreased when the angle of attack was increased. He also found that for an oscillating foil case, the vortex shedding frequency was decreased when the reduced frequency was increased.

Vortices can be also shed from other foil motions such as plunging or the combination of plunging and pitching. However the vortex shedding phenomenon behind the combination motion is not yet well understood. Some researchers study the similarity of vortex caused by pure plunging and pure pitching to have a better concept of combination motion. For a plunging vibrating foil, a study was performed by Chandravanshi et al. (2010) which has been explained in Subsection 3.4.2. He numerically investigated flow behind a foil which was in a plunging motion. NACA 0012 was modeled at $Re$ of $10^4$ and at various reduced frequencies which were 0.393, 0.785, 1.178. For a higher amplitude vibration, the drag coefficient was found to be lower whereas the thrust coefficient was found to be higher. His result also showed that from the wake pattern and thrust coefficient, reduced frequency and plunge velocity is important factors for plunging motion.

Khalid et al. (2014) numerically investigated the equivalence of plunging and pitching motion of NACA 0012 regarding its vortex shedding, lift and thrust coefficient. His result can be useful for predicting the vortex shedding phenomenon in plunging motion via pitching motion result inspected from their kinematic similarity. In his study, Khalid took aerodynamic performance into account to establish equivalence between plunging and pitching motion. Two methods were implemented, these were the effective angle of attack and the equal-St approach. From his model, he found that for predetermined $C_t$, the equal-St approach predicted pitching motion very close to plunging motion but failed for $C_l$. In the $C_l$ result, the pitching motion was found to be higher for both methods and hysteresis was also present. For the vortex shedding contour prediction, the equal-St approach was also found to be similar as shown in Figure 2.16.
Figure 2.16: Vortex in pure plunge (left) and pure pitch Motion (right) of NACA 0012 (Khalid et al., 2014)

The prediction of vortex shedding in the plunging vibration can be determined from pitching motion by applying the equal-St approach. The equal-St approach employ plunge-equivalent pitching amplitude as written in Equation (2.16).

\[
\alpha = \arctan\left( \frac{h}{0.75c} \right)
\]

(2.16)

\(\alpha\) is the blade angle of attack, \(h\) is the blade heave motion amplitude, and \(c\) is the blade chord length.

The vortex shedding stage can be recognized from the pressure distribution profile as observed by Lee and Su (2015). They conducted an experiment to investigate the relationship between the pressure distributions and flow pattern on NACA 0012 in pure pitching, pure heaving and pitch heaving motions. They characterized the vortex shedding stage by the profile of pressure distribution over the foil surface. The sequence of vortex shedding stage can be classified into initiation and growth of leading edge vortex (LEV), the spillage of LEV, and the LEV-induced massive separation. These stages are shown in Figure 2.17.

The initiation of LEV is recognized from a dynamic shoot (peak) on the pressure distribution contour plot, marked by LSB (Least Significant Bit) in Figure 2.17a and 2.17b.
A pressure distribution peak exhibits a high pressure gradient on the direction of the flow on that region as expressed in Equation (2.17).

\[
\frac{dP}{dx} > 0 \quad (2.17)
\]

\(dP\) is pressure difference, \(dx\) is distance

The high gradient indicates high pressure at the point where the vortex begins to form. The high pressure reduces the fluid velocity near the foil surface and produce high fluid velocity discrepancy between the innermost layer and its layer above. The fluid in the layer above rolls up the fluid particle in the lower layer and vortex is generated.

The LEV grows and flows downstream which can be distinguished from the receding of LEV-induced pressure on the contour plot as depicted in Figure 2.17c-2.17e. Then the LEV begins to spill from the surface and finally the massive vortex detaches from the trailing edge. The sequence of spilling process up until the vortex detachment is illustrated in Figure 2.17f-2.17h. Vortex shedding determines the fluid flow regime around foil. It also affects the foil response such as foil vibration behavior which is detailed in Section 2.9.
2.8 Prediction of Fluid Induced Vibration

Flow induced vibration can be defined as a vibration on a structure immersed in a fluid flow caused by fluid vortex because of vortex irregularity and disorder (Vortex induced vibration). Vortex induced vibration can be very harmful for structure stability thus needs to be predicted and controlled in order to develop structure sustainability. Once the vortex shedding is predicted the vibration likely becomes controlled and the stability issue can be resolved.

The vortex is shed almost the same form for all structure regardless the structure geometry. The vortex is shed with a frequency which impacts vibration frequency. A dimensionless parameter, labelled as Strouhal number (St), is discovered to determine the vortex frequency. Strouhal number, named after the a Czech physicist who found that an Aeolian tones were related to wind speed and wire thickness, is a dimensionless parameter which describes the relation between the predominant frequency of vortex shedding and fluid stream velocity as shown in the Equation (2.18).

\[
S_t = \frac{f_s L}{V_\infty} \tag{2.18}
\]

\(S_t\) is Strouhal number, \(f_s\) vibration frequency, \(L\) length of vibrating object, and \(V_\infty\) fluid velocity.

For a cylinder, strouhal number is a function of Reynolds number as depicted in Figure 2.18. In hydro or aero application, the correlation between these dimensionless parameter attracted some researchers to explore further such as Yarushevych’s experiment (Yarusevych et al., 2009).

![Figure 2.18: Strouhal number as a function of Reynolds number in a cylinder (Blevins, 1990)](image)
Yarusevych et al. (2009) conducted an experiment of a foil for $Re$ between $5.5 \times 10^3$ and $2.1 \times 10^5$ at three different angles of attack which were $0^\circ$, $5^\circ$, and $10^\circ$. His experiment showed that the rolls-up vortices frequency in a separated layer was proportional to $(Re)^n$ which $n$ was a constant parameter from 0.9 to 1.9 depending on the angle of attack. He also found that the wake frequency depended on $Re$. He proposed the Strouhal number for a foil were laying in the value between 0.45 and 0.5 in the case of his experiment. The vortex shedding frequency was also investigated by Blevins (1990) and he introduced the Strouhal number for a foil based on the width of boundary layers ($D$) approximately 0.2.

In most cases, vortex is not shed at a single distinct frequency but rather it sheds at a range of frequencies (narrow band). When vortex is shed at the same frequency as its shedding frequency calculated from St in the Formula 11, the lift and drag force magnitude begin to oscillate during the time operation. However in this condition, it was found that the drag and lift force oscillation are different. Lift force oscillates at the same frequency as vortex shedding frequency whereas drag force fluctuates at twice vortex shedding frequency. Vortex shedding exhibits a force on the structure and mutually the structure also impose a force which appears as a structure vibration, on the fluid. The vibration induced by vortex can also produce a sound effect (singing) in the fluid when the vortex sheds at the vortex shedding frequency.

A structure vibration frequency near vortex shedding frequency induce a big effect to vortex shedding such as increasing the strength of the vortices and cause the vortex shedding frequency to alter to frequency of vibration and those two frequencies become synchronized to each other (lock-in frequency). In the lock-in frequency, vibration of the structure mainly controls the vortex frequency. For a very big amplitude, the vibration shifts the vortex shedding frequency as much as 40% from normal condition (Blevins, 1990).

Whenever the fluid velocity increases or decreases and bring the vortex shedding frequency at or near natural frequency, the vortex shedding frequency is being synchronized and is locked-in by the natural frequency. In this case both frequencies forms a resonance. In a resonance the vortex shedding is strengthen and transfer energy to the structure which creates a high amplitude vibration. The lock-in frequency can be dangerous situation as it makes the vibration suddenly stronger and uncontrolled and creates a severe failure on a fluid structure. Thus in designing a fluid structure, one should maintain the fluid sheds vortices at far different frequency from the structural natural frequency. Moreover at a particular vortex shedding frequency, the boundary layer and the fluid flow over a foil are very unique thus contribute to drag and lift changes. Once the vortex shedding is determined the vibration is predicted and further lift and drag force can also be estimated.
Vortex shedding behind trailing edge on fixed angle of attack and oscillating foil was explained in detail in this chapter and will be continued how those trailing edge vortex affect structural behavior on a blade in the following chapter.

2.8.1 Frequency Lock-in

Frequency Lock-in is a phenomena when the fluid response frequency is forced to shed nearly to a structure’s natural frequency. The phenomena is strongly related to VIV and identified by high amplitude vibrations. In the FSI vertical axis tidal turbine problem, the blade vibration generates vortex which has similar frequency to the blade’s natural frequency. Lock-in was investigated formerly by Bishop and Hassan (1964). They conducted an experiment of a circular cylinder which was forced to move in a flowing fluid and examined the lift and drag coefficient in different forces exitacition and velocity flows. They found a lock-in condition is in a range of fluid frequency.

Besem et al. (2014) numerically predicted lock-in and structural natural frequency of NACA 0012 at 40° initial angle of attack and validated his own numerical model with his own experiment. His simulation results were in agreement with his experiment and he found that lock-in frequency in a flat plate including an foil at high angle of attack are the same regardless its shape. The pressure contour of the vortex shedding period related to the foil position is shown in Figures 2.19(a) and 2.19(b) respectively. At T2, the vortex shedding detaches from the trailing edge and cause the pressure at the region on suction surface decreases. Beyond that point, at T3 and T4 where the angle of attack is higher, the vortex becomes stronger and flows back to the leading edge. The reverse vortex creates lower pressure at leading edge but increases trailing edge pressure.

His experiment also showed that during the foil motion, lock-in frequency does not exist at a single distinct frequency but it propagates as a frequency band and becomes wider as the amplitude of vibration becomes higher. The wider lock-in frequency band at higher vibration amplitude is a consequence of the shedding frequency which is excited more easily by a high amplitude rather than a low one. Lock-in frequency band from Besem et al. (2014) experiment and simulation is depicted in Figure 2.20.

In the computational work, Blackburn and Henderson (1996) succeded to model a lock-in phenomena in his numerical investigation. The result showed an evidence that lock-in condition could also be predicted numerically. Blackburn and Henderson (1996) developed a model of a cylindrical object and observed a lock-in behavior when the cylinder was forced by cros-flow oscillation. The model reproduced a VIV response on the cylinder and showed the lock-in behavior which was demonstrated by the changing fluid regime. Lock-in in circular cylindrical vortex-induced vibration was also investigated numerically by Mittal et al. (2016). They developed a model of circular cylindrical in square domain and identified the lock-in phenomena using Direct Time Integration
Figure 2.19: Lock-in frequency of NACA 0012 at 40° angle of attack: a. vortex shedding, b. foil vibration (Besem et al., 2014)

(DTI) and Linear Stability Analysis (LSA). The result agreed with the experiment from Khalak and Williamson (1999). The lock-in phenomena occurred at a range of frequency and identified by a high amplitude oscillations.

The influence of trailing edge modification on lock-in phenomena has been studied by Ausoni et al. (2007). They conducted an experiment of a truncated foil and found that lock-in phenomena occurred at the blade eigen frequency (natural frequency). The other modified trailing edge profiles regardless the modification profiles are also attempted to have a lock-in physical phenomena.
The blade’s vibration is a dynamic response as a result of FSI which is the turbine tides interaction in this case. Modeling the blade dynamic response is significant for identifying the lock-in phenomena in the turbine operation. This will be discussed in next section starting from the background theory to the response model construction.

2.9 Introduction to Dynamic Response

In general practice, a structural response is assessed in order to understand the behavior of a structure when it is subjected to an external force. The force mainly comes from its surroundings including fluid such as wind or tides. This situation can be harmful when the force is unsteady. The unsteady loading imposes an unsteady response on a structure, which is commonly known as a dynamic response. The dynamic response is distinguished from a static response regarding the fluctuation of the load acting on a structure. A structural dynamic response of high fluctuated loading is a fundamental source of some severe structural problems including fatigue failure. In a hydrodynamic case in which an elastic structure such as tidal turbine operates, investigation is performed on fluid structure interaction. The investigation is proposed to overcome the complexity of a fluid structure interaction problem. In this thesis the fluid structure interaction analysis covers the vertical axis turbine blade response due to foil vibration which is generated by vortex shedding. Therefore, the foil response is modeled by a vibration system which interacts with typical vortex shedding due to foil trailing edge modification.

Dynamic response is a major topic of study in fluid structure interaction analysis. There are two approaches to analyze this interaction to determine how the structure behaves under dynamic fluid loading. The methods are named one way and two way solution. The one way approach solves the interaction problem by applying converged fluid loading
to a structural dynamic response solution sequentially. The problem decomposes two cases which are solved by separated methods. The converged fluid case is solved for instance by developing a model and simulating the model using CFD method which is actually a very complex approach since many procedures must be considered rigorously. Similar to the fluid solution, the dynamic structural case requires one to build a detailed design by applying structural methods such as beam theory. In contrast to one way approach, two way approach works to solve the fluid and structural case simultaneously. Both aforementioned methods, CFD and the beam theory, can be conducted in two way method with an iterative manner. The background theory governed and the model developed for solving a blade response problem is explained in the next section.

As the vortex is shed downstream on an oscillation or a static angle foil, the vibration cannot be prevented from happening during the operation. This indicates that vortex shedding controls a structure response including a vertical axis tidal turbine blade response. In the operation of a vertical axis tidal turbine, the structural vibration is subjected to the turbine blades response interacted with the tides. The interaction of the tides and turbine blades can be amplified when the vortex shedding frequency coincides with the blade natural frequency as described earlier. In a three bladed vertical axis tidal turbine, the vortex induced vibration is a complex problem because the difficulties to model the behavior of the turbulent tidal turbine fluid flow regime. In fact vortex shedding is not the sole factor which causes turbulence of the fluid flow in the vicinity of a turbine. The other significant aspects which impact the turbulence on a turbine blade are flow behavior from the front of the blade and the unsteadiness of tidal velocity. Nevertheless the problem can be simplified by assuming a constant tidal velocity passing the turbine blades and using a small aspect ratio in the design the turbine geometry. With a small aspect ratio chosen, a tidal turbine has a big diameter to operate and accordingly the turbulence from the front of the blade can be neglected. A response of such turbine has been observed by some researchers.

The response of vertical axis tidal turbine blades have been investigated numerically for example studies conducted by Young et al. (2010), Khalid et al. (2013b) and Banks et al. (2013). Khalid et al. (2013b) numerically studied a coupled method for the dynamic response of a four bladed vertical axis tidal turbine with unsteady incoming flow and validated his work with experiment. He used two way FSI methods which solved hydrodynamic and structural problems iteratively and he later found that his simulation gave an accurate result with the experiment. His simulation result showed the velocity profile, the prominent stress and deformation on each of the blades spanwise, as shown in Figure 2.21(b). Although his simulation gave good result in predicting an accurate turbine blade response, the behavior of the vortex generation and the time-dependent structural response due to vortex behavior was not clearly investigated.

From Figure 2.21(b), it can be seen that the velocity profile is not detail and cannot show the vortex shedding from the blade surface. It also cannot predict the vortex...
Figure 2.21: a. Deformation on turbine blade at 3.5 rad/s, b. velocity profile (Khalid et al., 2013b)

shedding frequency to identify whether the lock-in condition appears on the turbine or not. Figure 2.21(a) shows that the maximum stress is located at the spanwise midpoint. This point is adopted for investigation in this work when developing the 2D model of a vertical turbine blade to obtain the most critical response during turbine operation.

Young et al. (2010) proposed a numerical method by combining boundary element
and finite methods and validated it with Bahaj’s experiment (Bahaj et al., 2006). The method inspected the thrust coefficient, power coefficient and stress and deflection distribution as a response of fluid loading on tidal turbine. Stress and deformation on a blade during one rotation is depicted in Figure 2.22.

![Figure 2.22: Predicted stress and deflection of a turbine blade (Young et al., 2010)](image_url)

In Figure 2.22, the predicted stress and the displacement amplitude of a turbine blade is oscillating and over a cycle of rotation the blade experiences the first mode of vibration. Hence the vibration system in this model is of first mode.

Investigations on the structural response of a vertical axis turbine blade requires an essential dynamic response as part of a comprehensive process in designing a turbine. The purpose of the study is to develop a good understanding of a vertical axis turbine behavior to achieve a good performance and longer life time. However there is not much data and information provided to design the turbine with actual vortex shedding condition. By knowing the vortex condition, the way to control it, and its effect on the elastic structure of a turbine, a designer can predict the dynamic response of a turbine blade through a rigorous study of fluid structure interaction model which is explained in the next section.

### 2.10 A Vertical Axis Tidal Turbine Blade Response Model

The response of a foil has been modelled by some researchers such as Akcabay et al. (2014; 2014), Ducoin et al. (2009; 2013), and Rana et al. (2009). The foil response is modelled by a vibration system with spring and damping components. Besem et al. (2014) developed a model and conducted an experiment for understanding the vortex on an oscillating foil. Part of his results have been explained in Subsection 2.8.1. He also proposed a model for a foil response with a two spring system attached in the trailing edge. The angle of attack of the incoming fluid flow was varied to 50°. The supports which were attached at the trailing edge allowed the foil to oscillate by a maximum of
5°. His results show that the amplitude at lock-in frequency reached four times the unlocked frequency.

A vibration system with spring and damping component has been used to model a foil response as proposed by Akcabay et al. (2014; 2014). They suggested a model for representing a flexible hydrofoil response using spring damper system. Their blade was allowed to move in pitch and heave direction in respect to incoming fluid. Their result show a good agreement with Ducoin’s result (Ducoin et al., 2013; 2009) also applied a vibration method with spring damper system to model a hydrofoil response generated by cavity.

Rana et al. (2009) also developed a response model of NACA 0012 using a vibration system with vertical spring and damper component for representing heave response (transversal vibration) and spiral spring and damper component for modeling pitch response (torsional vibration). His model is depicted in Figure 2.23.

![Figure 2.23: A response model of NACA 0012 using spring damper system (Rana et al., 2009)](image)

Rana used his model to study flutter characteristic in 2D and generated pressure distribution, time history of heave and pitch motion. He placed the spring and damper component at a quarter of chord length which is a similar geometry adopted in this work. At that point the moment produced by the total force is constant regardless of the angle of attack applied to the foil. He also assumed the damping ratio to be 0.01 which is similar to damping ratio adopted in this work.

In this works the vertical axis tidal turbine blade response is modelled by a vibrational system with a spring and damping system. One spring and damping component performs one mode of response motion. The difference between the foil response and the blade response model is that the incoming fluid velocity in the foil response model is constant. In the blade response model, the equivalent incoming velocity is applied. This method is to imitate the turbine rotation and is applied to incoming fluids velocity of the model.
domain. The equivalent incoming velocity is calculated using Equation (2.4) and inserted into the inlet velocity boundary condition.

For 2D analysis, the maximum number of motions experienced by the foil are three, these are pitch, heave and surge. In this case there are three spring damper components with each component performing each response motion. The response model is simulated using OpenFOAM 2.2 applying sixDoFRigidBodyDisplacement dictionary in pointDisplacement file located in 0 time directory folder. pointDisplacement expression is written in Appendix A.2. Two way methods is utilized in the OpenFOAM routine which solves fluid problems using PIMPLE algorithm and body displacement problems using vibration system equations. The equation for three degrees of freedom vibration model of a blade response are shown in Equation (2.19)-2.21.

\[
m \ddot{y} + C_t \dot{y} + K_t y = F_L
\]  \hspace{1cm} (2.19)

\[
m \ddot{x} + C_t \dot{x} + K_t x = F_D
\]  \hspace{1cm} (2.20)

\[
I \ddot{\theta} + C_r \dot{\theta} + K_r \theta = M
\]  \hspace{1cm} (2.21)

m and I denote the blade mass and the blade moment of inertia respectively. \(y, \dot{y}, \ddot{y}\) are displacement, velocity and acceleration in heave performing transversal vibration in y direction, while \(x, \dot{x}, \ddot{x}\) have same the definition but for surge representing transversal vibration in x direction. The same superscript symbols apply to \(\theta\) representing displacement, velocity, and acceleration in pitch motion for torsional vibration. C and K are damping and stiffness coefficients respectively with t and r subscripts denoting transversal and torsional vibration. \(F_L\) and \(F_D\) is lift force and drag force respectively which acts on the blade whereas \(M\) is moment force. In the two way method, the results from the fluid problem are inserted in the body displacement functions using those second order vibration equations. From that routine the properties of fluid and the result of vibration in each cell are known including the velocity, pressure, the foil displacement after the process in each time step finished. The pressure distribution over the foil surface in each time step will be generated along with time history force coefficients such as drag, lift and moment. The fluid flow regime profile will also be displayed to identify the vortex generation in each case. The velocity at all cells in the domain can be extracted in a time history plot. From the time varying velocity the vortex shedding can be determined using phase averaged method which will be detailed in the next session.
2.10.1 Phase Averaged Method

The phase averaged method was discovered and first introduced by Hussain and Reynolds (1969; 1971). It is a method for extracting a fluctuating signal of a coherent turbulent structure from an original signal. The phase averaged method calculates the fluctuating signal component by subtracting original signal from a periodic reference signal. The graph in Figure 2.24 shows an illustration of the phase averaged method procedure.

![Fluctuated original signal (f), a periodic signal and phase averaged signal in phase averaged method (Hussain and Reynolds, 1969)](image)

Any fluctuating signal including a vortex shedding frequency can be obtained using the phase averaged method. The original signal which is extracted from any coherent turbulent signal is basically decomposed from a fluctuating component ($f'$) and a periodic component of the coherent signal which expressed by a phase averaged signal. Thus by knowing a phase averaged component and subtracting from the original signal, the fluctuating component of any coherent signal can be determined. The phase averaged component itself is the sum of a stochastic component of the corresponding coherent turbulent structure($\tilde{f}$) and a time averaged component ($\bar{f}$).

The Phase Averaged Method was applied by Jung and Park (2005) to their experimental data to find the vortex shedding frequency. They conducted an experiment on an oscillating foil at a constant velocity incoming flow and measured the velocity of fluids at 0.1c behind trailing edge using PIV. This probe distance is adopted in this work. The induced velocity on the foil was taken as the reference signal. They reported that the vortex shedding frequency signal can be obtained by subtracting the phase averaged velocity signal to the actual signal of the foil motion. All velocity signals from their experiment are given in Figure 2.25. From that graph, it is found that the original signal in actual condition (Figure 2.25a) is obtained from a decomposition of vortex signal (Figure 2.25c) and foil induced velocity signal (Figure 2.25b). Since the signal of oscillating foil
(Figure 2.25b) is identified from the Phase Averaged method, the foil-induced vortex signal can be extracted as seen in Figure 2.25c. Detail for the Phase Averaged method (Hussain and Reynolds; Hussain and Reynolds, 1969; 1971) used in their experiment is explained in Subsection 2.10.1. Jung and Park (2005) method is adopted in this work to find vortex shedding frequency.

Ostermann et al. (2015) also applied the Phase Averaged Method to their experimental study to extract fluctuating components in their naturally oscillating flow field. They used the Hilbert Transform to find a phase averaged signal from a reference signal and applied Power Spectrum Density (PSD) method to find the vortex shedding frequency. They also suggested that the reference signal should be a signal with a purely periodic character and should not be affected by the fluctuating component. Bourgeois et al. (2013a; 2013b) also applied the Phase Averaged method to his model and found that the method yielded a good result in predicting the vortex shedding frequency. Perrin et al. (2006b; 2006a) implemented the Phase Averaged method into his experiment and
found that the method gave a good prediction for obtaining vortex shedding frequency. In this work, the Phase Averaged method is employed to determine the vortex shedding frequency. The original signal is taken from the velocity history plot in a cell which is 0.1 m behind trailing edge (Jung and Park, 2005). At this point, the velocity signal is expected to be dominated by the vortex shedding behavior. The reference signal is obtained from a cell near the leading edge which demonstrates a purely periodic vibration motion (Ostermann et al., 2015). The reference signal is then converted to a phase averaged signal using the Hilbert transform and is subtracted from the original signal to obtain the vortex shedding velocity in the time domain. To extract the vortex shedding velocity in frequency domain from time domain data, the PSD method is used. Telgarsky (2013) discussed the PSD method for identifying dominant frequency of a system in his paper and used matlab code to extract the time domain data.

The vortex shedding frequency acquired from the Phase Averaged method and the Power Spectrum Density (PSD) method is then compared to the natural frequency of the vertical axis tidal turbine blade which is discussed in Subsection 2.10.3. The PSD method will be covered in Section 2.10.2. The comparison is made to examine if the lock-in condition exists on the blade structure. The amplified vibration of the blade is detrimental to the vertical axis tidal turbine and should be avoided. Thus to prevent the lock-in condition, the turbine designer requires data about the vortex shedding frequency and the vertical tidal turbine blade natural frequency.

### 2.10.2 Power Spectrum Density using Fourier Transform

Power Spectrum Density is a method to identify the strength variation (power) of a signal as a function of frequency. It employs Fourier Transform method to convert the parameter time domain into frequency domain. In this work, the PSD converts the time domain vortex shedding velocity into frequency domain in unit of power per frequency (W/Hz). The PSD result shows in what frequencies the vortex shedding energy (power per frequency) are strong and weak. The strong power signal in PSD plot indicates the frequency where the vortex is shed from the blade’s surfaces. The method predicts the critical (main) frequency at which the vorticity is generated.

The PSD method employs the FT method to convert the parameter domain. FT takes a time-based signal and decompose it into a frequency-based. The time-based signal is extracted into some components which decompose the original signal. The number of decomposed signals (signal sampling frequency) can be determined from nyquist frequency of signal sampling frequency. In this work, the PSD method applied in all response models are calculated using Fast Fourier Transform (FFT) algorithm in Matlab R2016a code. To speed up the FFT algorithm calculation, one can imply next larger power of two to define frequency sampling. However from Nyquist Theorem, the number of sample which can be estimate do not exceed half of sampling rate.
In this work sampling rate is taken from the velocity data which is recorded for twelve second steady state (omit the first three seconds of transient data from fifteen seconds in total). The data interval is not uniform as it is obtained from a constant courant number applied all over the simulation. The variety of data interval is interpolated first to decompose a uniform data interval prior to the FT calculation.

### 2.10.3 Natural Frequency

The natural frequency ($\omega$) is a mechanical property of an undamped structures associated with its free vibration condition when the material is initially displaced. From Equation (2.19), (2.20), (2.21) and when the foil model is undamped, only the mass and stiffness component remain and all other right hand side terms are zero. The equations becomes second order harmonic differential equations and the solutions are satisfied by finding a matrix for $\omega$ as an eigenvalue.

For a two degrees of freedom, heave and pitch response, Equation (2.19) and 2.21 accounts for a free vibration case by neglecting the damping constant and the damping term. Thus the forces on the right hand side in Equation (2.19) and 2.21 are zero. In matrix form the undamped free vibration can be written as in Equation (2.22).

\[
\begin{bmatrix}
  m & 0 \\
  0 & I
\end{bmatrix} \frac{d^2}{dt^2} \begin{bmatrix}
y \\
\theta
\end{bmatrix} + \begin{bmatrix}
k_t & 0 \\
0 & k_\theta
\end{bmatrix} \begin{bmatrix}
y \\
\theta
\end{bmatrix} = \begin{bmatrix}
0 \\
0
\end{bmatrix}
\]

(2.22)

In general, the matrix form of Equation (2.22) can be simplified to :

\[
M_i \frac{d^2 x}{dt^2} + K_i x = 0
\]

(2.23)

The system to which Equation (2.23) is applied is assumed to be a harmonic system. A harmonic system has a solution form of $X \sin \omega_n t$ with $X$ the amplitude of the displacement. By substituting the solution into the Equation (2.23), the vibration motion equation becomes:

\[
\omega^2_n M_i X = K_i X
\]

(2.24)

Generalized eigenvector ($u_i$) and eigenvalue ($\lambda_i$) terms can be employed to solve this problem by substituting the terms in the Equation (2.24) so that the Equation (2.24) becomes $\lambda_i M_i u = K_i u$. The natural frequency of the system ($\omega_n$) is the solution which satisfies: $\omega_i = \sqrt{\lambda_i}$

In simple vibration, when only one degree of freedom involved on a system, the transversal or the torsional natural frequency can be calculated directly from the data of the
material stiffness constant, the structure mass and the moment inertia using Equation (2.25) and 2.26.

\[ \omega = \sqrt{\frac{K}{m}} \]  
\[ \omega_r = \sqrt{\frac{K_r}{I}} \]  
\[ (2.25) \]
\[ (2.26) \]

\( \omega \) denotes natural frequency, \( t \) performing transversal vibration and \( r \) torsional vibration. \( K \) and \( m \) are material stiffness and blade mass respectively.

From the material properties data of a blade, the material stiffness is obtained for transversal and torsional vibration and inserted to Equation (2.19) and 2.21 to find the structure natural frequency. Davies et al. (2013) conducted an experiment for tidal turbine blade and found static flexural stiffness of blades using polymer composite was approximately 1000 N/m. For the torsional vibration, material stiffness was found to be 200 Nm/radians which will be discussed in detail in Subsection 2.10.4.

The environmental condition should also be considered when examining a structural natural frequency. The condition of the tide in where a vertical turbine structure is installed, gives forces on the turbine blades. As a result the blade response is affected and varies according to the tidal variation. For a structure immersed in liquid such as water, a term called added mass should be taken into account when examining structural natural frequency. Ghassemi and Yari (2011) and Da Lozzo et al. (2012) studied numerically added mass and developed a model for their object. Ghassemi and Yari (2011) considered added mass and included it in his turbine propeller model. His result showed that a factor of 1.74 should be implemented when a water turbine is being designed. An added mass factor is taken and inserted into Equation 2.19 through 2.21 and found that transversal and torsional natural frequency are 0.9425 Hz and 1.4758 Hz respectively.

2.10.4 A Spring Damper System for A Blade Response Model

Dynamic mesh is handled for solving the fluid structure instruction problem of a vertical axis tidal turbine blade. Jasak and Tuković (2010) reviewed fluid structure interaction problems handled by dynamic mesh in OpenFOAM. The use of OpenFOAM for solving FSI problems is recommended as the physic solvers for solving the fluid condition and the structure equation are coupled and accessible in the OpenFOAM library. The use is relatively simple within only a single executable run. This is equipped by robust mesh refinement process such as snappyHexMesh for running in dynamic mesh. The good quality meshing is introduced by three stages which can reduce the number of cells constructed in the grid. This will maintain the numerical stability and precision result
with taking less computational effort and time consuming. The snappyHexMesh process is also open for some improvement and combined with other algorithm process to make the process more predicted and avoid unnecessity trial error (Fabritius and Tabor, 2016)

The response model is developed to understand the interaction between a fluid and a foil or a vertical axis tidal turbine blade. This interaction or the foil response is modelled by a vibration motion on a system of spring and damper as shown in Figure 2.26.

![Figure 2.26: Response model as a vibrational system with spring and damper components](image)

In Figure 2.26, the response is modelled by two degrees of freedom, these are pitch and heave as in the standard model from Table 2.3. The standard model is a default foil response model in which the fluid and foil mechanical properties will be varied to understand the behaviour of the fluid flow regime and the foil response. The variations simulated in the response model as listed in Table 2.4.

Table 2.3: Standard Setting for the foil response model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial velocity (m/s)</td>
<td>4</td>
</tr>
<tr>
<td>Initial angle of attack (°)</td>
<td>0</td>
</tr>
<tr>
<td>Orientation</td>
<td>Pitch, heave</td>
</tr>
<tr>
<td>Stiffness (N/m): heave, pitch</td>
<td>1000, 200</td>
</tr>
<tr>
<td>damping (N.s/m)</td>
<td>2</td>
</tr>
</tbody>
</table>

The damper and spring system are virtually attached to the foil point of rotation (dynamic centre) which is one quarter of chord length from the leading edge. The interaction is represented by a vibration of the spring damper system when the foil is in contact with the fluid. In the standard model (Table 2.3) there are two spring and damper systems which perform pitch and heave responses. The pitch and heave responses are modelled by torsional and transversal vibration. In general the number of spring damper system is equal the number of degree of freedom. For two degree of freedom model such as in the standard model, there are two spring damper systems which are oriented to perform heave and pitch response orientation. Heave and pitch response are caused by the transversal and torsional vibration respectively. The torsional and a transversal vibration acting on the blade are assumed in the first mode of force vibration and performed at quasi steady. The simulation uses a fully coupled FSI method which resolves the fluid
field and structure motion equations simultaneously as developed by Chen et al. (2004). The governing equation for transversal and torsional response are shown in Equation (2.19) and 2.21 respectively. The damping and stiffness coefficients are determined based on the blade material which is assumed to be manufactured from a polymer composite material.

The transversal stiffness constant (in heave direction) listed in Table 2.3 is approximated from Davies’ experimental result (shown in his paper in Figure 7, Polymer line, in his paper) (Davies et al., 2013). His experiment tested a tidal turbine blade which was manufactured from a polymer material subject to flexure load. The polymer material for his tidal turbine was manufactured from a polyurethane casting and strengthened by composite reinforcement from glass fibres (chopped strand mat or quasi-unidirectional) to increase the polymer stiffness properties. The composite material was then impregnated with epoxy resin. The usage of polyurethane material for tidal turbines is also recommended by Klaassen (2015). He suggested a polyurethane coating for a composite material which can protect a blade from cavitation and pressure changing during operation. In this work, the turbine blade is assumed to be manufactured from sixty percent e-glass and forty percent of epoxy by volume (74.71% fiber and 25.29% resin by weight). The density of the material is taken from a composite tool website (net composite) using a volume fraction of sixty percent e-glass fiber and forty percent epoxy resin. The fiber and resin have density of 2.56 g/cm$^3$ and 1.3 g/cm$^3$ respectively. The typical combined material has the density of 2.06 g/cm$^3$. The e-glass which is stronger than epoxy is added in larger quantity to the material mixture to strengthen the blade. The blade is then coated with a polyurethane substance to protect the surface from cavitation and pressure changing damage.

The torsional stiffness constant is obtained from the material transversal stiffness constant with the assumption that the material is isotropic. In an isotropic material, the mechanical properties do not depend on a body’s orientation thus the mechanical properties are identical in all directions. However the stiffness constant depends on the mass and the moment of inertia of the body. The stiffness constant in a particular direction can be approximated from the transversal stiffness constant using the ratio of mass to moment inertia. The foil mass is calculated using the material density and the foil volume. The mass of the model with a 0.12 m spanwise length is found to be 16.73 kg. The moment inertia which is calculated using Equation (2.15) in respect to pitch motion is 3.75 kgm$^2$. From the isotropic approach using the mass to moment of inertia ratio, the torsional stiffness is found to be 224.15 Nm/rad and is rounded to 200 Nm/rad in this work as listed in Table 2.3. The torsional stiffness constant calculated from the isotropic material approach is verified with the torsional stiffness constant calculated using the ADM method investigated by Tabassian (2013)

The torsional vibration stiffness coefficient obtained from Tabassian’s model (Tabassian, 2013) approach is approximated from the natural frequency of a shaft having a circle
cross section area and shear modulus of the turbine blade material. He also proposed a torsional analysis to determine a natural frequency for shafts with different supports. One of Tabbassian’s beam models is concentrated mass shaft supported with springs at both ends. The general formula for Tabassian’s model is written in Equation (2.27).

\[ \Omega = \omega_n \frac{l}{C_s} \]  

(2.27)

with

\[ C_s^2 = \frac{G}{\rho_s} \]  

(2.28)

Jankowski (2012) suggested the shear modulus of a composite material (listed in Table 2 in his paper) for his numerical model of buckling vibration of a composite column. Chung (2001) reviewed materials for vibration damping and observed in detail the shear modulus of a vibrating material. Shear modulus approximated using ADM method in Equation (2.28) is taken from Chung’s result (2001). In his paper Chung (2001) reviewed vibration damping from several types of material including polymers and composites. He compared those representative materials in Table 1 of his paper in three different parameters. These are \( \tan \delta \), storage modulus and loss modulus. These are parameters used in Dynamic Mechanical Analysis (DMA) to express the stress response of a viscoelastic material when a dynamic load is applied on the material. \( \tan \delta \) is a parameter to measure damping capacity. A viscoelastic material exhibits deformation in between a viscous fluid and an elastic material. When a force is applied, a viscous fluid deforms according to Newtonian stress behaviour which is proportional to the shear-strain rate while elastic material deformations are proportional to stress strain.

The storage modulus is an applied force which is stored within the molecules of the material and is dissipated to return to the material to its initial condition before the force is applied. Thus it is an elastic solid like behaviour. The loss modulus is created when the applied force is higher and dissipated to break the molecules bond of the material. The material is being deformed (flow) and said to have a viscous like behaviour. To increase the vibration damping ability, the material should have high \( \tan \delta \) and/or high shear modulus. Both quantities influence the loss modulus. The shear modulus in Equation (2.28) is taken from the loss modulus of epoxy listed in Table 1 in Chung’s paper. The loss modulus of epoxy is found to be 0.11 GPa.

The non-dimensional parameters of frequency from Tabbassian’s model for a constrained beam with both ends supported with a spring in Equation (2.27) is 0.808675703. The length of turbine blade (\( l \)) in Equation (2.28) which is calculated from the blade aspect ratio and the blade chord length in Table 2.2 is found to be 18.75 m. The shear modulus from Chung (2001), the material density, the length of turbine and the non-dimensional
parameter of frequency are then substituted into Equation (2.27) and 2.28 to predict the natural frequency. The natural frequency approximated using this method and the data of moment of inertia is then inserted into Equation 2.26 to calculate the torsional stiffness constant. The torsional stiffness constant obtained using this method is found to be 278.42 Nm/rad which is close to the torsional stiffness constant obtained from the isotropic approach. The small discrepancy of torsional stiffness coefficient between the ADM method and isotropic approach is likely because in the ADM method the observed shaft has a circular cross sectional area.

Another advantage in the use of composite materials is the ability to absorb vibrational energy. Kumar and T (2015) conducted an experiment to determine the damping characteristics of hybrid polymer matrix composites. He found that the damping ratio directly proportional to natural frequency (Table III of his paper). The damping ratio of the model in this work is found to be 0.0074 with a critical damping value in heave direction is 258.69 Ns/m. The heave critical damping is calculated using Equation (2.29).

\[ C_c = 2\sqrt{mk} \] (2.29)

A new term is introduced, called damping ratio, which defines the ratio of an actual damping value of a system to its system’s critical damping value. Damping ratio determines the damped condition of a system during vibration. The system is over damped and returns to steady state without vibrating if the damping ratio value is larger than 1. On the other hand if the damping ratio is less than 1, the system is considered to be under damped condition. In the under damped condition, a system experiences oscillations which decay in an exponential manner prior to the steady state condition being reached. The damping ratio is calculated from Equation (2.30):

\[ \zeta = \frac{C}{C_c} \] (2.30)

Using Equation (2.30), the damping coefficient in heave direction is calculated to be 1.91 Ns/m and is rounded to 2 Ns/m. The method for determining heave damping coefficient is also applied to the torsional damping coefficient calculation. The damping ratio from Kumar and T (2015) is employed. The torsional critical damping calculated using Equation (2.29) is rounded to be 50 Ns/m and inserted into Equation (2.30) to determine the torsional damping coefficient. The torsional damping coefficient is found and rounded to be 0.2 Ns/m.

The damping value in this model is varied based on over damped and under damped condition. The under damped system is a condition of a vibrational system when the damping ratio is less than. In contrast the over damped system happens when its
damping ratio is more than one. In this work the over damped model has the damping constant value of 300 Ns/m.

The importance of damping for response model was studied by Veilleux and Dumas (2013). They applied a damping factor to combined pitch and heave motion on a NACA 0012 model. They developed the model and validated it with their own experiments. Within a range of heave stiffness their model with damping effect applied was found to give closer result to the experiment than a case without damping. In this work, a damping factor is also implemented based in Veilleux and Dumas (2013) result and applied on the response model using spring and damper system.

The vibration in OpenFOAM is modelled in the dynamic mesh to allow for moving object during vibration. The simulation of vibrating object is possible due to moving points in the mesh which is incorporated using the pointDisplacement dictionary. The pointDisplacement dictionary is implemented as an added file in the zero time directory and it contains boundary fields which are necessary to describe the motion of blade vibrations. The pointDisplacement dictionary requires fifteen inputs to define the blade response when force is applied including mass, centre of mass, moment of inertia, orientation, constraint and restraint of the spring damper system etc. The mass is determined from the added mass of the system according to De La Torre et al. (2013) study. They investigated the effect of added mass on a hydrofoil and they found three modes of added mass constant associated with the foil vibration orientation. The orientation mode are similar to the first bending, second bending and torsion vibration. In this work the second bending and torsion mode is taken and the value from De La Torre’s results are found to be maximum of 2.56 and 1.74 respectively. The mass is obtained using data of material density and the foil volume. The volume is calculated by integrating foil area and multiply with spanwise length. Centre of mass is calculated using Equation (2.31) and moment inertia is obtained by Equation (2.15).

\[
\bar{X} = \sum X_i m_i / \sum m_i
\]  

(2.31)

The response condition also needs to be defined in the pointDisplacement file in the boundaryField entry. For modelling vibrations of the blade response, the patch field point locations are determined by sixDoFRigidBodyDisplacement boundary condition in the boundaryField entry.

The incoming fluid velocity magnitude is defined in the zero folder and substituted by the fluid resultant velocity. The resultant velocity is calculated from the tidal velocity (Tillinger, 2011) and the turbine angular velocity for the foil response model. The velocity is kept steady during the simulation. At this stage, standard model parameter are selected as listed in Table 2.3. Following on from this, the parameter in Table 2.3
are varied as shown in Table 2.4 for each model to investigate the response of the foil at different conditions.

The blade response model is developed using unsteady incoming velocity entering the domain. The unsteady incoming velocity represents the time varying inlet tidal velocity experienced by a blade due to time varying angle of attack and the velocity magnitude. The velocity magnitude variation is happened because of the resultant velocity magnitude variation acting on a blade. The turbine blade response is modelled separated from the turbine rotating motion and as a result the motion working on the blade is a vibration response only. This strategy allows to reduce complexity for modelling two different motions which does not have the same reference axis in one object.

Unsteady incoming fluid flow is assigned on the inlet of CFD domain. The unsteadiness includes a time dependent angle of attack and velocity magnitude. Gosselin et al. (2013) reported a method to find angle of attack for one rotation of vertical axis turbine as in Equation (2.4) whereas the magnitude of fluid velocity is calculated from resultant steady incoming flow and angular velocity of the turbine. Complete design parameters of the turbine is given in Table 3.7.

For a steady incoming flow simulation, some operational conditional are varied in order to predict the response under various conditions and the likely lock-in frequency of turbine response. The streamwise fluid flow regime behind trailing edge is further analyzed to confirm the lock-in phenomenon as it is strongly detected by the vorticity appearing in the flow regime. The varying simulation parameters are velocity magnitude, angle of attack, number of degree response, stiffness coefficient and damping coefficient as listed in Table 2.4. For each model when one simulation property is varied, the other simulation properties remain the same as listed in Table 2.3. The model is run for fifteen seconds simulation time for all cases and the plot for lift, drag and moment coefficient are shown and discussed. Each force consists of total forces of pressure and viscous force component in assigned direction. The Results of both models will be detailed and discussed in the next subsection and followed with modified blade model, results and discussions.

Table 2.4: Variation of simulation parameters

<table>
<thead>
<tr>
<th>Properties</th>
<th>Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity (m/s)</td>
<td>2</td>
</tr>
<tr>
<td>Orientation Pitch</td>
<td>Pitch heave</td>
</tr>
<tr>
<td>Angle of attack (°)</td>
<td>0</td>
</tr>
<tr>
<td>Stiffness (N/m)</td>
<td>500</td>
</tr>
<tr>
<td>Damping (N.s/m)</td>
<td>2</td>
</tr>
</tbody>
</table>

The vertical axis tidal turbine blade is allowed to move in heave and pitch motion with respect to the incoming tidal velocity which induce transversal and torsional vibration. The vibration is modelled with two spring damper systems attached to the turbine blade.
to allow transversal and rotational vibration. The model is simulated using OpenFOAM 2.2 employing sixDoFRigidBody in the pointDisplacement dictionary to handle the blade motion and is run using pimpleDyMFoam solver OpenFOAM foundation, 2017). The spring and damper are attached at the blade dynamic centre point which is located at a quarter of the chord length from leading edge. The blade shape is NACA 0012 and has a chord length of 0.75 m.

The response of a vertical axis tidal turbine blade is also generated using the same simulation properties as detailed in Table 2.3. The difference between these two models is that in the vertical axis tidal turbine blade response model the incoming velocity is not constant since it is equivalent to a periodic turbine revolution. Hence the motion of the blade solely represents the vertical axis tidal turbine blade response and interaction with the tides. This method is also applied to the modified blades which include the blunt, rounded and sharp profile.

### 2.11 Modified Foil Response

The original shape of the foil was intended to reduce some fluid dynamic drawbacks on a bluff body such as high drag force and separation. However the nature of vortex shedding on original foil is still remained and influenced fluid pressure over the entire foil surface. The surface pressure becomes oscillated (Blevins, 1990) and generates vibrations on the foil which influence the turbine performance. It was noticed at early decade that modifying the foil trailing edge shape can increase lift force or reduce drag force and is also expected to reduce vortex shedding. Since then some experiments and numerical studies were conducted to investigate the fluid flow over modified foil trailing edge shape including work by Ramjee et al. (1986), Thompson and Whitelaw (1988), El-Gammal et al. (2010) and simulation by Standish et al. (2003), Gomez and Pinilla (2006), and Murcia and Pinilla (2011). Some of their results have been introduced in Section 3.4.

Ramjee et al. (1986) developed a simulation of NACA 0012 with trailing edge modifications and his numerical results were validated by own experiments. He truncated the trailing edge by 5%, 10%, 15%, and 20% of chord length and found that the modifications influenced the lift force. He stated that by increasing the bluntness, the maximum lift coefficient increased to the point when 15% of chord length was being cut. Beyond that bluntness configuration, the lift coefficient was decreased. The truncation was also found to affect the foil lift and drag ratio in the same result as the lift force is affected. Lift and drag ratio was improved when 15% of truncation was applied and decreased if any further length was being cut off. These results are adopted in this work for determining the cut off length for modifying the foil model.
Thompson and Whitelaw (1988) set up experiments to provide further understanding of the flow phenomenon over a foil with trailing edge modifications. He used three different foil modifications in his experiments including sharp, blunt and inclined it to 14° and 17° angles of attack. He found that the flow was affected downstream from the foil trailing edge to 4.5% of chord length. This affected region existed in all of his foils regardless its trailing edge shape. A recirculation flow was observed on the blunt and round foil at 0.04% chord extended region at the near wake behind the trailing edge. The base pressure for all trailing edge shapes was equal to the average suction pressure regardless the trailing edge shape. Gomez and Pinilla (2006) generated a simulation of a truncated asymmetric NACA foil. He found the most important effect of trailing edge truncation was to increase the angle of attack at which the value of $C_l/C_d$ was maximum. He proposed a mathematical relation to calculate the increasing angle as:

$$\alpha_{L/D_{\text{max}}} = \alpha_{0L} + \sqrt{(\alpha_{0l} - \alpha_{0d})^2 + \frac{C_{D0}}{\mu}}$$  \hspace{1cm} (2.32)

The primary focus of this study on a modified foil is to provide the fundamental theory of foil performance due to the behaviour of forces as an effect of the fluid flow over the modified foil surface. Following the early stages the investigation of a modified foil is further applied to turbine blades to improve the turbine performance. Standish et al. (2003) developed a model for a modified blade to be used in a low speed turbine. He truncated an asymmetrical foil trailing edge by 5% and 10% of chord length. His simulation revealed that, aside from obtaining turbine structural benefits, larger truncation of the turbine blades produced higher maximum lift force and improve turbine performance.

In the study of fluid force on modified foil, El-Gammal et al. (2010) conducted an experiment to understand drag force behaviour on a foil. He decomposed the measurement from its two components which were pressure and skin friction drag. The pressure drag over the suction and pressure surface exhibited the same behaviour in which it decreased drastically from the stagnant point. Beyond that point the pressure drag slightly raised, and then reached maximum near trailing edge. Meanwhile the skin friction drag increased significantly from the stagnant point and reached its peak value at approximately 0.01 and 0.02 of $x$ on the suction and pressure surface respectively. The increasing skin friction drag occurred due to the generation of boundary layer as the fluid particle accelerated beyond the stagnation point. The pressure drag dominated the measured total drag force and contributed to 60%.

The modified foil response will also be modelled by a vibrational system with a spring damper system for each degree of freedom response motion. It is built from the same type of material and hence spring and damping constant are equal to the original blade’s stiffness and damping constant. The trailing edge shape modification on the tidal turbine
blade is predicted to control the behaviour of fluid including the vortex generation. The pressure will be affected and eventually the foil response from the interaction with the fluid will also be changed. The modifications are expected to weaken the vortex shedding and reduce the vibration response of the vertical axis tidal turbine blade.

The investigation of the vortex shedding can be feasibly predicted by a computational fluid dynamic program such as OpenFOAM. However, the simulation requires a high performance computing and consumes lots of time to finish as a consequence of the complexity of the fluid problem and governing equations which must be resolved. Hence developing a well-ordered and organized topology for the foil contributes to the ease of simulation. Well-ordered topology is provided by organizing segments of the CFD domain and limiting the number of cells to a sufficient resolution as the case requires. The knowledge and strategy used in developing the vertical axis tidal turbine as well as descriptions of the simulation properties will be detailed in the next chapter.

2.12 Summary

The background theory behind the vertical axis tidal turbine design and the FSI method applied in a vertical axis tidal turbine blade were explained in this chapter. The theories underlie the selection of the vertical axis tidal turbine design parameters as listed in Table 2.2. The selection of blade type, aspect ratio, number of blades, tip speed ratio and solidity are discussed and determines the nominate vertical axis tidal turbine. The turbine employs three straight bladed original NACA 0012 with 0.75 chord length and $Re$ of $3.07 \times 10^6$ based on inflow velocity condition. The modified blades are also introduced to replace the original blade for controlling the blade response. The three modifications are assigned at the blade's trailing edge section. These are blunt, rounded and sharp profiles.

Equivalence Incoming Velocity method which is applied in the basis vertical axis tidal turbine is detailed. This method allows turbine revolution to be modeled by the time varying velocity and angle of attack entering the domain of the 2D CFD response model. This method reduces the complexity and domain mesh which influences to cut off the running time. The 2D CFD procedures employed in the model are also examined. These procedures are applied in the three bladed vertical axis tidal turbine model. This model represents the actual turbine revolution with constant incoming velocity and analyze in 2D manner. The result from published papers which are adopted in this work are also discussed. These procedures underpin the developing of CFD methodology for the static and dynamic foil model which is the main topic in Chapter 3.

Chapter 2 also discussed the physics of a fluid structure interaction response on a blade and analyze it. The discussion of the FSI method began with the characteristic of the interaction which was denoted by the existence of the vortex shedding. The vortex
generates a fluid induced vibration which is signified by the lock-in phenomenon. The lock-in condition is determined using the Phase Averaged and Fourier transform methods. The methods are applied in the fluid velocity of points in the leading edge and 0.1c from trailing edge. The calculation result shows main frequencies of vortex shedding velocity on the trailing edge region. The lock-in phenomenon happen when one of the main frequencies coincides with one of the blade’s natural frequencies.

The vertical axis tidal turbine model which was constructed using the three blades requires long time to run as will be detailed in the next chapter, Chapter 4. Therefore a single blade model is selected to identify the fluid structure interaction problem on a vertical axis tidal turbine using the Equivalence Incoming Velocity method. This strategy anticipates the unreasonable time period and allows the blade to model the response motion only and will be explained in Chapter 5.
Chapter 3

Development of the CFD Methodology for Static and Dynamic Foils

The research procedure to obtain the FSI analysis of an original vertical axis tidal turbine blade is discussed in this chapter. The procedure is also applied in a vertical axis tidal turbine using modified foils. The procedure starts with the mesh independence study for all models. The model’s result is verified with an experimental and two published numerical results. The validated mesh further is implemented to modified foil model for static and dynamic cases. The static case includes nine angles of attack models and the dynamic case employs an oscillating foil.

3.1 Static Foil Performance

Characteristics of a single foil in contact with a fluid flow has been of interest for the past two decades. The subject has been studied intensively and has been motivated by growing needs of air transportations and vast demands of better performance renewable energy turbines across the world. Since that time some researchers have numerically and experimentally explored and analysed the fluid flow behavior passing a single foil to yield a better understanding of the flow around a turbine. For numerical investigations, the rapid progress in computer technology helps a computational fluid dynamics model to gain precise prediction and to reduce the model’s simulation time. Similar to the numerical process, a high specification computer is also needed for an experimental investigation. Accordingly the experiment is being improved by advancing the measurement process and data acquisition with the use of a super computer. The fast growing of computer technology also helps to develop a single foil CFD model applicable and more accurately predicted. The simulation can be an estimation reference to decide or select
experimental sets up to anticipate a big number of experiments. Based on the CFD prediction, the experiments are directed to only work on the essential unpredictable phenomenon which is needed to be understood and proven. In this way a project can be reduced to a minimum cost and a less time consuming process. On the other hand, an experimental result can also be a tool for researchers for validating their numerical results before carry out the next cases or developing new models.

An experimental study of a static NACA 0012 foil to identify its fluid flow regime had been conducted by Abbott and Von Doenhoff (1959). He performed experiments for some single symmetric foils including NACA 0012 at high Reynolds numbers of $3 \times 10^6$, $6 \times 10^6$ and $9 \times 10^6$. He showed the lift coefficient plot from -24$^\circ$ to 32$^\circ$ angle of attack and the relation between drag coefficient to lift coefficient within this range. For numerical studies, some researchers have modeled and simulated a static single foil such as Eleni et al. (2012) and Mittal and Saxena (2002). For dynamic foils, numerical models have been simulated by Martinat et al. (2008) and Frederich et al. (2009). They developed models for pitching or oscillating foils which will be adopted in this work.

In this work, a single foil is modeled in two states which are stationary state with nine static/fixed angles of attack and an unsteady state including oscillating foil model. The fixed angle of attack model is run using PIMPLE algorithm in pimpleFoam solver and validated using experimental result from Abbott and Von Doenhoff (1959) and numerical result from Eleni et al. (2012) and Mittal and Saxena (2002). For the unsteady foil, a prediction had been done by McCorskey (1982) and was considered as one of the pioneers in unsteady foil investigation. His motivations behind a study of a unsteady foil was to reduce vibrations which required the magnitude of phase or time lag prediction in the unsteady fluids dynamic loading. He proposed a model for an oscillating foil in unsteady flow and also observed lift, drag, and moment coefficients and pressure distribution. The wake behavior was also detailed in his paper. He also found that a hysteresis significantly involved in forces and moment result.

In this chapter, the flow regime on single foil models both in stationary and unsteady conditions will be explained according to experimental and numerical results. The employed static angles are at low angles of attack based on the range of angle of attack existing on the vertical axis tidal turbine blade design during operation. The range of the turbine blade pitch angle is from $11.31^\circ$ to $-11.31^\circ$ (in Table 2.1). Nine static angles of attack are selected from $-8^\circ$ to $8^\circ$ to identify the fluid behaviour in various pitching conditions. For the unsteady foil model, a forced motion is applied to the foil to form an oscillating motion according to Frederich’s model (Frederich et al., 2009). The model’s fluid flow profile is validated using Frederich’s (2009) and Martinat’s (Martinat et al., 2008) result. The lift and drag coefficients of this model is recorded during fifty seconds and compared to Frederich et al.’s result. The validated model further is utilized for developing a vertical axis tidal turbine blade response model in both constant and
unsteady inlet velocities. The unsteadiness is modeled by a time dependent inflow velocity magnitude and angle of attack. Those variations replace the vertical axis turbine’s rotation.

A NACA 0012 foil is selected in the design of this vertical axis tidal turbine blade over asymmetric foil and other NACA types subjected to advantages as it has been detailed in Section 2.1.1. Lift force is focused because the lift is a driven force to rotate a vertical axis turbine. The vertical axis tidal turbine is also designed to enhance lift force to improve the extracted power. However the understanding of fluid behavior across a turbine blade and how the blade response to fluid influences the turbine performance including lift force is not well known. Therefore this works mainly discuss the blade response and how fluid-induced vibration affects the turbine performance (lift, drag and moment coefficients).

### 3.1.1 Force Coefficients

Force coefficients are observed in this work including lift, drag, moment and pressure coefficients. The coefficients history over fifteen seconds are recorded and plotted on time domain graphs. However only the lift and drag coefficients are taken for validation process in static and oscillating foils cases. The lift coefficient is an indicator for a turbine operation which reflects energy extractions to rotate the turbine. Lift coefficient is obtained from the lift force acting on a blade and calculated using Equation (3.1)

\[
C_l = \frac{2F_l}{\rho V_\infty^2 A}
\]

\(C_l\) is lift coefficient, \(F_l\) is lift force, \(\rho\) is fluid density, and \(V_\infty\) is the tidal velocity.

Different from the lift force, the drag force indicates a viscous force acting on a blade. A viscous force is a resistance of fluids to a foil’s motion (drag) and is working on the foil’s surface. The drag coefficient is directed parallel to the fluid flow and opposite to direction the foil’s movement. Drag force are also expressed as normalized values of the drag coefficient and calculated using Equation (3.2).

\[
C_d = \frac{2F_d}{\rho V_\infty^2 A}
\]

\(C_d\) is drag coefficient, \(F_d\) is drag force, \(\rho\) is the fluid density, and \(V_\infty\) the tidal velocity.

The moment coefficient drives the blade to have a pitch motion which is also used for rotating a turbine. The moment coefficient is defined as a dimensionless parameter produced when pitching moment force working on a foil and calculated using Equation (3.3).
\[ C_m = \frac{M}{pA_c} \]  

(3.3)

\(C_m\) is moment coefficient, \(M\) is moment force, \(p\) is static pressure.

Another force coefficient to observe is the pressure coefficient which denotes a pressure at a point relative to the pressure of freestream and defined as in Equation (3.4).

\[ C_p = \frac{p - p_\infty}{p_0 - p_\infty} = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2} \]  

(3.4)

\(C_p\) is pressure coefficient, \(p\) is static pressure, \(p_\infty\) is freestream pressure, \(p_0\) is atmospheric pressure \(\rho_\infty\) is fluid density, and \(V_\infty\) the tidal velocity.

It is a dimensionless parameter for expressing a distribution pressure throughout a flow field over a foil's surface. The pressure distribution plot can predict the fluid flow regime condition over a surface including the prediction of vortex shedding stage as described by Lee and Su (2015) and will be detailed in 2.7.

A NACA 0012 blade with 0.75 m chord length at \(3.07 \times 10^6\) Reynolds number is modeled. The offset of the foil is calculated using NACA equation as shown in Equation (2.1). The lift, drag and moment forces on the NACA blade due to interaction with the fluid are recorded. As the fluid passing the blade’s upper and lower surfaces, the fluid exerts pressure and viscous forces on the surfaces which contributes to the lift and drag force generation. However pressure and viscous forces are exerted on different direction hence affecting in different manners to turbine construction. Pressure forces act perpendicularly to blade surface with the convention positive forces acting toward the blade surface. For zero angle of attack, pressure distribution on lower and upper surface are equal, producing balance forces in the upper and lower surface and leading to zero lift force.

At non-zero static angle of attack, the blade pressure distribution between lower and upper surface is not equal due to velocity differences of the fluid on both surfaces. The high velocity fluid on the upper surface creates a low pressure which is directed outward. In contrast, on the lower surface the lower velocity magnitude generates higher pressure than the upper surface pressure. All forces cumulatively supply lift force to the blade. In an analysis of forces, lift force is commonly expressed by its normalized or dimensionless value, called lift coefficient \((C_l)\) as calculated in Equation (3.1). The dimensionless parameter is preferred in the analysis as it is independent regardless the size of the blade and the fluid velocity.

In the turbine operation, lift forces are expected to be high since high lift induces high turbine power. On the contrary, drag forces decrease the power of a turbine as it resists the turbine in its rotation. Therefore in the design the turbine is expected to allow as low drag force as possible during the operation of turbine. As it can be seen in
Abbott and Von Doenhoff (1959), Eleni et al. (2012), and Mittal and Saxena (2002), higher angle of attack at same Re can produce higher lift force and higher drag force. However at an angle of about 17° and higher, stall phenomenon exists which is marked by the fluid separation and vortex shedding. In this stage, the fluid flow regime has high resistance to its motion and accordingly it has a very high drag coefficient and a low lift coefficient which a turbine designer should be avoided. In this project, the maximum and minimum angle of attack during the vertical axis tidal turbine operation which has been explained in Subsection 2.1.4 and shown in Table 2.1, are still below limited angle for vortex shedding and separation occurrence. Therefore the turbine design is recommended to operate but its performance needs to be predicted and improved. It is also inferred that the choice of the turbine properties which was detailed in Chapter 2 including turbine tips speed ratio, solidity and aspect ratio are important.

Another disadvantage of the fluid separation and vortex shedding is to cause unsteadiness in the fluid flow regime. The unsteadiness introduces vibrations typically on the vertical axis tidal turbine blades and structure in general. The higher the angle of attack, the stronger the separation and the vortex shedding on the blade surface and this will eventually produce more vibrations than at lower angle. Vibrations cause issues in the operation of a turbine such as harmful fatigue failure and serious noise if it is not controlled. Therefore this work proposes a way to control a blade response and reduce its vibration by controlling the fluid flow regime in terms of its wake and vortex generation by observing its lift, drag and moment history results.

There are several types of NACA foil which are commonly used for renewable energy turbines. Sheldahl and Klimas (1981) tested NACA 0009, 0012, 0015, 0018, 0021, and 0025 for the use of vertical axis wind turbine to identify the aerodynamic characteristics over 180° angle of attack. Their experiments revealed that for a wider foil (higher number of NACA) with a similar lift coefficient, the drag coefficient was higher. Hence NACA 0012 is a more suitable foil regarding its small drag force to be used for the turbine design compared to the other wider foils. Goett and Bullivant (1939) conducted a full scale wind tunnel experiment of NACA 0009, 0012, and 0018 to examine those three NACA foils lift coefficient. Their experiments showed that at Reynolds number of $3 \times 10^6$, NACA 0012 has highest lift coefficient in comparison to others. Therefore considering the aerodynamics characteristic of both the above experiments, NACA 0012 is chosen for the vertical axis turbine design in this project.

When a lift force is introduced on a turbine blade, a torque is also generated to rotate the turbine in the same direction as the generated force. The initiation turbine torque is of importance in the design of turbine regarding the power to overcome the drag to rotate turbine initially. Douak and Aouachria (2015) found that a vertical axis turbine is characterized by a low starting torque. The low starting torque is beneficial for a turbine because a light power is needed to rotate turbine from stationary. Although the
required torque is low, a sufficient initial lift forces is still required to obtain adequate torque to overcome turbine viscous condition.

3.1.2 Static Foil Fluid Regime

Static angle of attack on a foil has been studied numerically by Mittal and Saxena (2002), and Eleni et al. (2012), and experimented by Abbott and Von Doenhoff (1959). Abbott and Von Doenhoff (1959) tested symmetric airfoils including NACA 0012 at three different Reynolds number which were $3 \times 10^6$, $6 \times 10^6$, and $9 \times 10^6$, and obtained lift coefficient for an angle of attack from $-24^\circ$ until $32^\circ$. He also plotted correlation of drag coefficient to lift coefficient at the same angle ranges. His result were of significance and used by many researchers to validate their numerical work including by Eleni et al. (2012). Mittal and Saxena (2002) observed the hysteresis and characteristic of flow past stationary NACA 0012 at $10^6$ Reynolds and the results are showed in Figure 3.1.

\begin{figure}[h]
\centering
\begin{subfigure}{0.45\textwidth}
\centering
\includegraphics[width=\textwidth]{figure3.1a.png}
\caption{}
\end{subfigure}
\begin{subfigure}{0.45\textwidth}
\centering
\includegraphics[width=\textwidth]{figure3.1b.png}
\caption{}
\end{subfigure}
\caption{Simulation result of fixed angle of attack (Mittal and Saxena, 2002) a. lift coefficient, b. drag coefficient}
\end{figure}

Mittal and Saxena (2002) changed the angle of attack by progressively increasing and decreasing angles during simulation. They developed a numerical domain which was decomposed from hybrid mesh of 37,896 cells in total. The unstructured grid which
provides flexibility to handle complex geometries was generated in the far region from the foil and a structured grid is provided around foil. The angle of attack was simulated at 0°, 5°, 10°, and 15° and they recorded lift and drag coefficient at zero to twenty degree angle of attack as shown in Figure 3.1. They also detected a dynamic stall at an angle of attack around 17° and a point of separation which started at 5° angle of attack. Thus after 5° angle of attack drag coefficient increases. Their experiment showed that at angle 17°, dynamic stall happens. Stall is a phenomenon when lift coefficient is suddenly drop and drag coefficient significantly increases because of the existence of the fluid separation and vortex shedding. On a blade with upward motion, the separation and vortex shedding take place on the blade upper surface. The point of separation is constantly moving backward towards trailing edge when angle of attack is increased.

From Mittal and Sexena’s simulation (Mittal and Saxena, 2002), it can also be seen that at a low angle of attack, starting from zero angle, the lift coefficient increases with increasing angle. Fluid flow regime was also smooth without vortex generation until the angle of 15°. At 15° a weak vortex was detected within the region behind the trailing edge. Stall began at 17° and afterwards the fluid behaved irregularly with the stronger generation of vortex for small increment of increasing angle of attack. The Von Karman vortex was found at 18.5° and at 19° and the separation had become stronger at the leading edge. At this point, the downstream flow exhibited a strongly chaotic behavior on the upper surface and in the fluid behind trailing edge. The fluid behavior from Mittal’s simulation can be seen in Figure 3.2.

![Figure 3.2: Pressure and vortex contour at various fixed angle of attack for steady state solution (Mittal and Saxena, 2002)](image)

From Mittal and Sexena’s numerical results (Mittal and Saxena, 2002), they also found out that lift and drag coefficient behaved differently on the decreasing and increasing angle because of hysteresis. Drag coefficient on decreasing angle of attack had higher amplitude but lower frequency fluctuation than increasing angle whereas for lift coefficient the opposite was found. Lift coefficient is found to have higher amplitude and higher frequency at increasing angle of attack. This phenomenon can also be seen in the lift and drag coefficient around 17°-19° as seen in Figure 3.3.
Figure 3.3: Lift and drag coefficient. Solid line is increasing angle and broken line is decreasing angle (Mittal and Saxena, 2002)

The effect of hysteresis was primarily noticed by McCorskey (1982). In his paper, he mentioned that the force coefficients for a static angle of attack were not the same as the upstroke and downstroke motion and hence caused a delay to stall for dynamic foil. The influence of hysteresis in dynamic stall was further reviewed by Carr (1988). He mentioned the relation between dynamic stall with some parameters including pitch axis location, mean angle, amplitude, Mach number, etc. He also found that stall was strongly affected by foil oscillation amplitude.

Eleni et al. (2012) developed a model of NACA 0012 and run with three different turbulence models at five different high Reynolds number from $1 \times 10^6$ until $5 \times 10^6$. He also tested four different numbers of cells and found a sufficient number of cells to be proposed for a single blade simulation which is 80,000 quadrilateral cells. Lift and drag coefficient were recorded for angle of attack from -12° until 20° and validated his result using experiments from Abbott and Von Doenhoff (1959). He found for low angle of attack (less than 10°), the lift coefficient of his simulation results agreed Abbott and Von Doenhoff’s experiments (1959). He showed pressure and velocity contour of 3°, 9° and 16° angle, and fluid was found to be smooth and regular in all of them. Three different turbulent models he implemented were Sparat Almaras, k-ε and k-ω SST and he came to a result that k-ω is the most accurate turbulence model amongst those three.
Chapter 3 Development of the CFD Methodology for Static and Dynamic Foils

The fluid behavior at a static high angle which is at 20°, was also investigated numerically by Gioria et al. (2009). They found the vorticity pattern and the fluctuated lift forces experienced by the blade as depicted in Figure 3.4. The vortex and separation as shown in Figure 3.4b are responsible for force coefficient fluctuation which become stronger at angle closer to stall angle.

![Figure 3.4: a. Cl signal, b. vortex behind trailing edge (Gioria et al., 2009)](image)

Lee and Gerontakos (2004) conducted an experiment on NACA 0012 at \( Re = 1.35 \times 10^5 \) for angles of attack from 0° to 20°. Their results show that for lift and pressure coefficient at increasing angles, the stall exists at approximately 15° angle of attack. They also found that the location of fluid separation moved forward with increasing angle which is similar to the numerical results from Mittal and Saxena (2002) and Eleni et al. (2012).

### 3.2 A 2D Single Foil CFD Model

A 2D CFD model of static single foil has been of interest recently. A 2D model is commonly chosen for modeling a CFD due to less computational efforts and execution times compared to 3D CFD model. Thus a 2D approach is selected in this work.

The topology of a foil model can be constructed in the two different grids. These are C grid and O grid. Lutton (1989) studied the C and O grid mesh generation for NACA 0012 foil use and found that the C grid has some advantages over the O grid. One of the advantages of C grid is constructed easier specially when the grid is generated in the trailing edge area. This good capability affect to the numerical stability and thus the result when the trailing edge regime is a concern. Khare et al. (2009) also studied grid generation for CFD application. His result agreed with Lutton’s result that C grid can capture the wake flow in the trailing edge region better than O grid. This is obtained by the better grid quality at the trailing edge constructed using C grid.
Eleni et al. (2012) built a model of NACA 0012 foil at Reynolds number of $3 \times 10^6$ in C-structured grid. He simulated the model using sparatAlmaras, k-ω SST, and k-ε turbulence model and validated the results with experiments from Abbott and Von Doenhoff (1959). The lift and drag coefficients from his simulation result showed that simulation using k-ω SST gave the best agreement to Abbott and Von Doenhoff’s experimental result. Eleni et al. also performed mesh independence simulation and found out that for C-structured grid, 80,000 cells was adequate to reach mesh independent solution. Lidlte et al. (2016) also used a c-structured grid to develop his foil model. His model is adopted in this work.

### 3.2.1 Topology

The fluid structure response of a vertical axis tidal turbine blade model is generated using two topologies which are C-Structured and rectangular grids. In both grids, the blade is located in the middle of vertical wise of the domain. Both topology are also applied for the three static modified trailing edge blades. Both topologies are discussed in the next sub section.

#### 3.2.1.1 C-structured Grid

For C-structured grid the blade is located in the center with the leading edge facing the curve of the C shape as shown Figure 3.5(a). C-structured grid domain is divided into nine parts and each part has self-adjusted mesh by setting the number of points on the connectors. There are seven boundary fields developed including inlet, outlet, front, back, top, bottom and the foil surface. The fluid enter the domain through the inlet, pass the inside mesh and the blade surface and finally exits from outlet. In C-structured grid, the inlet region is the C curve which is half circle with diameter length is the same as the width of the domain. The foil which has 0.75 chord length, is placed in the center of the grid which is 12.75c away from inlet.

This topology can be controlled by adjusting the number of points and expansion ratio along the connectors which separate parts in the domain. The adjustable connector locates in the front part of foil, upper and lower foil surface, mesh in the vertical direction, and mesh in the downstream behind the trailing edge. The mesh utility has also the ability to control boundary layer over the foil surface in terms of the number, expansion ratio and total layers height. At the juncture of foil upper and lower connector on the trailing edge, a gap is existed inevitably and impact the solution of fluid flow thus an argument added to set number of layer to be placed. This argument is essential for the blunt foil simulation as the gap is bigger produced by trailing edge truncation. The original NACA 0012 foil in C-structured grid is depicted in Figure 3.5(a) and the mesh around the original NACA 0012 foil in c-structured is shown in Figure 3.5(b).
All cells developed are structured grid which has quadrilateral shape for two dimensional model. Mesh independence also becomes a concern as to ascertain the simulation runs efficiently by obtaining high fidelity result but less time consuming. Independence foil mesh study is examined based on data from Eleni (2010) by adjusting the number of points and expansion ratio along each adjusted connector. The adjustment is checked by checkMesh utility which check the quality of mesh such as aspect ratio, skewness and non-orthogonality. The purpose of quality examination is to ascertain good quality mesh being used in the simulation which is refined and well-ordered to resolve the phenomena and vortex behaviour but also adequate to run with sufficient computation effort. Mesh independence is indicated by the number of cells prominently and validated by the lift coefficient from some references. The configuration with least number of cell with the least lift coefficient error beyond which the simulation cannot produce less error, is accepted to be the default configuration. In that case the cell refinement (more cells added) does not influence simulation result.

Boundary layer is developed along the upper and lower surface of the foil which can be set up from number of layers and expansion ratio on both surface independently. Number
of layers in this boundary is arranged from the leading edge towards the trailing edge portion in back field. Cell arrangement in vertical direction is controlled for all cells in whole domain with the expansion ratio goes from the foil surface outward. The front part of foil is constructed by an angle drawn from the foil surface towards the front field which can be controlled from its number and expansion point lies in the connector.

3.2.1.2 Rectangular Grid

Different from c-structured grid, in rectangular grid the blade position is closer to domain inlet and has longer space behind trailing edge to capture wake and vorticity after passing the blade. The rectangular grid has 8.3c length and 5.9c width for normal foil and 9.8c length and 6.7c width for modified foils. The blade is located 1.2c length from inlet and in the middle of its width. The mesh has seven boundary layers around blade, three different level grids with 6 cells for adjustment between levels. Cell expansion ratio is 1.3 with width layers between 0.25 and 0.7. The spanwise length of the foil and the domain was 0.01 which is not to be a concern as the simulation is analyzed in 2D. Therefore front and back fields, and top and bottom fields have symmetric configuration.

A single foil model is developed prior to modelling the vertical axis tidal turbine and the blades response. A NACA 0012 foil with 0.75 m chord length is assembled using Matlab. For the three modified foils, a fifteen percent trailing edge truncation (Ramjee, 1988) reduces the length to 0.6375 m. The trailing edge modification are generated using SolidWork and converted into STL file format before they are attached using the snappyHexMesh dictionary in OpenFOAM. The trailing edge region for different trailing edge are depicted in Figure 2.2.

A 2D CFD rectangular domain for analyzing the foils is then created using in the blockMesh dictionary. The domain is discretized with a structured grid which has quadrilateral shape for the 2D model. The rectangular domain has a length of 8.3c and a width of 5.9c for the original foil and a length of 9.8c and a width of 6.7c for the modified foils. snappyHexMesh refinement is controlled in the snappyHexMesh dictionary (snappyHexMeshDict) within the system folder. The foil is located in the centre of the domain at a distance of 1.6c from inlet boundary. In the boundary file, the inlet, outlet, top and bottom field are defined as patch, the front and back boundaries are set as empty to accommodate a 2D model simulation and analysis. The embedded foil profile is specified as wall boundary in the boundary file which is located in polyMesh folder within the constant folder.

Seven boundary layers are specified around the foil surface with level 5 refinement. Three different grid resolutions are set with 6 adjustment cells between levels in each grid. Adjustment cell is described as number of layers built up between different levels. In these models, there are six layers generated between cell layers. Cell expansion ratio
is set as 1.3 with a range of layer width between 0.25 and 0.7. The spanwise length of the foil and also the domain is 0.05 m. This does not affect the model as analysed in 2D. The front and back fields of the domain have symmetric configurations and their boundary conditions are set in 'empty' type. This mesh topology of original foil is also applied for the three modified trailing edge ones. The grid of original foil at zero angle of attack and the mesh near its trailing edge can be seen in Figure 3.6.

In the simulation, the static angle of attack is varied from 8°, 6°, 4°, 2°, 0°, -2°, -4°, -6°, and -8°. The nine static angles of attack foil are located in the aforementioned domain grid. A mesh validation process is undertaken and will be discussed in Subsection 3.3. The mesh properties including the number of points, faces and cells in the domain are listed in Table 3.1. This domain grid is also used in the oscillating foil and the foil response model.

The mesh is checked by running the checkMesh utility which examines the quality of mesh by inspecting the aspect ratio and non-orthogonality. From the checking process, the maximum aspect ratio and average mesh orthogonality is found to be 6.861 and 4.762.
Chapter 3 Development of the CFD Methodology for Static and Dynamic Foils

Table 3.1: Mesh static for original foil

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points</td>
<td>21,968</td>
</tr>
<tr>
<td>Faces</td>
<td>42,400</td>
</tr>
<tr>
<td>Cells</td>
<td>10,472</td>
</tr>
</tbody>
</table>

respectively which are acceptable for an OpenFOAM case. The boundary layer mesh is developed around the foil surface and seven boundary layers are set in this model. The expansion ratio for the layer is 1.3 with uppermost layer thickness and minimum thickness in each layer are set to be 0.7 cm and 0.25 cm respectively. The generation of the modified foil domain will be explained in the following section. The fluid velocity passing through the inlet patch is 4 m/s (Tillinger, 2011) which yields a $Re$ of $3 \times 10^6$. The other fluid properties are listed in Table 2.2.

3.3 Mesh Independence Study and Validation

3.3.1 C-structured Grid

Mesh independence study for lift coefficient constructed in c-structured grid shown in Figures 3.7 and 3.8. Four models are developed in four different number of cells which are 182,202 cells (grid 1), 418,202 cells (grid 2), 781,122 cells (grid 3) and 1,543,924 cells (grid 4).

![Figure 3.7: Lift coefficient at four different c-structured refinement grids](image)

From Figures 3.7 and 3.8 it can be seen that grids 2 and 3 are close to Abbott and Von Doenhoff’s data (1959). The grid 2 is implemented for the single foil model because
it has less number of cells so to reduce executing time. The model constructed in c-structure grid 2 further is compared with experimental and numerical data as shown in Figures 3.9 and 3.10

Figure 3.9: Lift coefficient compared to experimental and numerical data

3.3.2 Rectangular Grid

Mesh independence study is applied to four different refinement grids. The four different grids have 8600 cells, 10472 cells, 11838 cells and 12711 cells.

The study compares the drag force at 8° static angle of attack as shown in Figure 3.12.

From Figure 3.12, it can be seen that Cd value is drastically reduced at the second refinement grid with 10472 cells and closer to Abbott’s experimental data (1959). From
the second refinement, the more refined grid does not influence the Cd value. The execution time of higher refinement grid is also affected. The grid with 21329 cells is simulated nine times longer than the first refinement. Based on the result of the mesh independence study, the second refinement with 10472 cells is utilized in this work. The force coefficients of the model with chosen grid refinement is further validated using the reference data. The validation of lift and drag coefficient to the reference data are shown

Figure 3.10: Drag coefficient compared to experimental and numerical data

Figure 3.11: Four grids for mesh independence study in rectangular grid: a. 8600 cells, b. 10472 , c. 11838 cells, d. 12711 cells
in Figures 3.13 and 3.14 respectively.

The validation process is performed to static angle of attack model at nine angles of attack. The lift and drag coefficients at nine static angles of attack are compared to experimental results from Abbott and Von Doenhoff (1959) and numerical results from Eleni et al. (2012) and Mittal and Saxena (2002). The models are simulated in an unsteady state using the PIMPLE algorithm. Plots of the lift coefficients in the steady and unsteady condition along with reference data are shown in Figure 3.13 while the drag coefficient plot is depicted in Figure 3.14.

As can be seen from Figure 3.13, the model using rectangular grid agrees with experimental data from Abbott with the error of less than 8.5% except for the 4° angle model. Compared to a numerical result, the rectangular model result from the 2° angle do not
satisfy Eleni’s data very well. The error from the model exceeds 19.35%. For the drag coefficient the result of the rectangular grid together with the reference data are depicted in Figure 3.14.

![Figure 3.14: Validation of drag coefficient](image)

Figure 3.14 shows that the model constructed in the rectangular grid over-predicts the experimental and numerical reference data. The discrepancy is caused by the mesh resolution which likely needs to be refined. The refined mesh is required in order to resolve the viscous condition for the drag force in the boundary layers. As explained in Subsection 2.3.2.4, the viscous effect in the boundary layer region can be solved satisfactorily when the yplus value is less than 1. However in some situations a larger yplus value still acceptable to model the viscous effect using the log law region function. The log law region requires that yplus is more than 30 in order to resolve the viscous effect in the boundary layer using a selected turbulence model. The log law region which is adopted in this work, applies the wall function equation as shown in Equations (2.10) to (2.13). The wall function is one option to solve turbulence model equation without requiring a refinement process. This is very helpful for a dynamic mesh model which will be discussed in 3.7. A dynamic mesh is restricted to a small number of cells to avoid time consuming simulations (although a small number is subjective and could vary between one model and another). Thus the number of cells and the result precision of a dynamic mesh simulation should be optimized.

Another models are developed at higher angle of attack with two different number of boundary layers around the foil. The different boundary layers are developed to obtain different y+ values. This is to identify the effect of y+ to the rectangular model. The identification is proposed to solve the discrepancy of drag coefficient as shown in Figure 3.14. The case is simulated at 10° angle of attack with y+ value of 34 and 30. The lift and drag coefficients for the two y+ values are shown in Table 3.2.
Table 3.2: Lift and drag coefficients in two different y+ values

<table>
<thead>
<tr>
<th>y+</th>
<th>Lift coefficient</th>
<th>Drag coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>34</td>
<td>0.984</td>
<td>0.036</td>
</tr>
<tr>
<td>30</td>
<td>1.02</td>
<td>0.029</td>
</tr>
</tbody>
</table>

Table 3.2 shows that lower y+ value gives significant change to drag coefficient but not lift coefficient. In this model the lower y+ is obtained by reducing the boundary layer thickness and the number of cells remain the same. Thus the simulation time does not increase. However the cell thickness can only be reduced until a certain level before the mesh shape is highly disrupted and the simulation becomes crashed.

3.4 Modified Foils Performance

Normal NACA 0012 which is used in this work starts to cause separation at an angle of 5° (Mittal and Saxena, 2002). The separation tends to strengthen the drag force since the fluid friction becomes higher. To some extent, at higher angle of attack vortex at trailing edge is also formed together with separation of the fluids. These vortices increases the drag force more, weaken the lift force and generate fluid-induced vibration. These detrimental effects experienced by a vertical axis tidal turbine blade can be reduced by minimizing the irregularity in the fluid flow regime by controlling the separation and vortex shedding. A method to control the trailing edge fluid behaviour is to modify the trailing edge shape. Three different trailing edge modifications are proposed in this work to smoothen the trailing edge fluid flow regime to intentionally eliminate drag force, enhance lift force and reduce fluid-induced vibration. The trailing edge shape modifications which are proposed to be applied to this vertical axis tidal turbine blade are blunt, sharp and rounded.

A study of trailing edge modification has been conducted, since fifties and has become a topic of interest in the last decade. To improve airfoil or hydrofoil performance various trailing edge shape has been tested and modelled to ensure the foil performance improvement. Thompson and Whitelaw (1988) studied blunt, round, and sharp trailing edges to understand the characteristics of boundary layers in the suction surface. He found that the near wake caused by the trailing edge modification was influencing on a very small to region within 4.5% of chord length behind trailing edge.

3.4.1 Blunt Profile

Creating a blunt trailing edge is one way to modify a foil to obtain better performance as proven by Smith and Schaefer (1950), Ramjee et al. (1986), Gomez and Pinilla (2006), and Murcia and Pinilla (2011). The method by which a blunt trailing edge can increase
lift force and lift to drag ratio was also investigated by El-Gammal et al. (2010) and Standish et al. (2003).

In 1950 Smith and Schaefer (Smith and Schaefer, 1950) built an experiment rig for testing a NACA 0012 with three trailing edge modifications by cutting from its trailing edge 1.5, 4, and 12.5 percent of its chord length. Lift and drag coefficients were recorded. Their result proved that increasing the cutting thickness gave insignificant change to the maximum lift coefficient. Unlike the lift coefficient, the maximum drag coefficient increased progressively for a thicker foil. Ramjee et al. (1986) numerically studied the effect of trailing edge truncation of NACA 0012 and validated with his experiment. He simulated four trailing edge truncation and tested the foil at angle of attack from 0° to 12°. He found that the maximum enhancement of lift coefficient occurred when fifteen percent of chord length of NACA 0012 trailing edge was cut off. Although lift coefficient increased with fifteen percent truncation, the stall phenomenon appeared earlier at angle of attack of 14°. This truncated geometry is applied in this work for all three trailing edge blade modifications.

Standish et al. (2003) studied blunt foils for wind turbine applications to improve the lift force. His results show that by truncating the wind turbine blade, not only was lift force increased but also the lift curve slope. Further a truncation method was numerically investigated by Murcia and Pinilla (2011). Two modifications methods on blunt foil were investigated which included cutting off and added thickness as applied to NACA 4421. Those two methods are illustrated in Figure 3.15.

Figure 3.15: Two methods of trailing edge modification (Murcia and Pinilla, 2011)
Cutting off method removes a rear portion of a foil and rescales the foil from its original chord length. With this method the thickness of a foil is larger after truncation but the chord length remains the same, yielding a constant $Re$ before and after cutting. In contrast to the cutting off method which removes rear segments of the foil, in the added thickness method a symmetrical portion is added along the baseline of a foil starting at a point between $x/c = 0.3$ and 0.5 (Figure 3.15). Compared to original shape prior to truncation, each method gives higher maximum lift coefficient, delayed stall, but drag coefficient increases. However a smaller drag coefficient was found in the cutting off method than the added thickness method. This condition is caused by pressure distribution in the trailing edge which created base drag. In both methods, lift coefficient was increased, although the blade also experienced higher drag coefficient. Ratio of lift to drag should be considered in this case to ensure that it is not smaller than the normal NACA 0012 before modification. Murcia and Pinilla (2011) also compared ratio of lift and drag between the original foil and the two methods of truncations. He found that at an angle between $-4^\circ$ to $6^\circ$ angle of attack, the foil truncated with added thickness method has a higher ratio. For an angle higher than $6^\circ$, a hysteresis was observed thus at this condition the maximum ratio is less than the cutting off method maximum ratio. The cutting off is implemented in this work because the lower drag force is produced using this method.

Gomez and Pinilla (2006) investigated aerodynamic characteristics of thirty nine NACA profile modifications. His results show that the performance of those NACA modifications with the same thickness is similar regardless the type of NACA. He also found that a thicker NACA profiles have lower ratio of lift to drag. The thicker foil also influence to the occurrence of maximum $C_l/C_d$ value which happens at higher angle of attack.

The wake behind a blunt trailing edge was experimented by Krentel and Nitsche (2013) to capture the near wake of a blunt foil which was cut at 80% of its chord length. The alternating and periodic vortex formation at the base was also been investigated. Two pulse recirculation vortices were generated at the base as depicted in Figure 3.16. At position 1, a vortex emanates and alternates with a vortex at position 2. However at position 2, the vortex was found to be weaker with the Strouhal number twice than position 1 and $180^\circ$ phase shifted.

### 3.4.2 Sharp and Rounded Profiles

Another two trailing edge shapes are proposed and studied in this work for controlling vertical axis turbine blade vibration namely sharp and rounded. A sharp trailing edge is also shown to improve blade performance in studies by Thompson and Whitelaw (1988). His experiments examined fluid flow over three different trailing edge shapes. He compared fluid flow behaviour downstream of blunt, sharp and round trailing edge foils. His result showed that recirculation flow was present at 0.04% chord length behind
trailing edge into the near wake in the case of the blunt and round foil. In contrast to the blunt and rounded foil, in the sharp edge foil the recirculation is not found in downstream flow so accordingly the fluid remained attached on the upper surface of the sharp foil. However the difference of fluid flow for blunt and sharp was not obvious on the downstream region at 4.5% of chord length.

Similar to blunt and sharp edge, rounded trailing edge are also considered by turbine designers for reducing drag forces as studied by Kahn et al. (2008). He studied several trailing edge shapes including rounded and found that a rounded trailing edge shape could reduce base drag. Although modified foils are believed to improve turbine performance including controlling blade vibration, the concept itself is not fully understood. Therefore the flow regime behind a modified trailing edge will be observed in this work to control the fluid induced vibration. The vibration induced by fluid and the response of a vertical axis tidal turbine will be detailed in the next chapter.

As a vertical axis tidal turbine contacts with the incoming fluid, two types of response are incurred on turbine blades. The responses are in the form of turbine rotation and the local vibrations. Local vibrations associated with energy losses should be minimized by reducing the wake and vorticity generation. For a 2D analysis, the vibration of the turbine can originate from three motions, heave, surge and pitch. In this work, four different cases associated with those three basic motions are observed. The observations cover one, two and three degrees of freedom modes which can be generated during turbine operation. For one degree of freedom, only pitch response is modelled as it is assumed if only one direction of response is allowed, the turbine rotation induces a rotating motion (pitch) on the turbine blades. In a similar way for two degrees of freedom, heave pitch
and surge pitch are considered in this study (neglecting heave surge motion). The last case to model for the degree of response variation case is when all those three motions, pitch heave and surge, exist together. Vibration as a physical foil response, together with the foil response model will be explained further in the next section.

3.5 Modified Foil Models

Trailing edge modifications are employed to an original NACA 0012 to control vortex generation. The controlled vortex main frequency is driven by the modified blades to shed away from the blade’s natural frequencies and no lock-in occurs. The modified foil models are also developed in c-structured and rectangular grids.

3.5.1 C-structured Grid

Three modified trailing edge foils are constructed and modeled, these include blunt, sharp and rounded as shown in Figure 2.2. The trailing edge modification is achieved by truncating an original NACA 0012 foil by 15% as suggested by Ramjee et al. (1986). The truncation length was obtained from his experiment which shows that 15% truncation applied on a foil trailing edge had the best lift coefficient amongst four other truncation length, ranges from 5% to 20%. The truncation method adopts the Cutting-off method from Murcia and Pinilla (2011) which was explained in Subsection 3.4.1. Because of the truncation the chord length of modified foil is not same as the original foil which influences a change to the Re. The modified foil Re which is calculated using Equation (2.7) and is based on the cut-off chord length and incoming flow velocity is $2.54 \times 10^6$. Although the vortex regime is very sensitive to the change of Re, the original and modified foil Re are still considered as a high Re ($Re > 10^6$) which will generate similar effect to the fluid flow regime.

3.5.2 Rectangular Grid

The modified foil models are developed in the rectangular grid as explained in Subsection 3.2.1.2. The foil is modified to the profiles as explained in Subsections 3.4.1 and 3.4.2.

3.5.2.1 Blunt Foil in Rectangular Grid

The blunt foil is generated by truncating the original foil by 15%. For a static angle of attack model, the blunt foil is also modeled using nine different angles of attack. As mentioned previously, the model is constructed in SolidWork 2014, saved in an STL format and attached using the snappyHexMesh dictionary. The topology grid is similar
Figure 3.17: The modified foil constructed in C-structured: a. blunt foil, b. round foil, c. sharp foil

Figure 3.18: Lift coefficient comparison between original and modified foils

to original foil mesh topology which has been explained in Subsection 3.2.1.2. The zero angle of attack blunt foil mesh in trailing edge region is shown in Figure 3.20.

The fluid properties are the same as those in the original foil model which are listed in Table 2.2. The mesh statistics for blunt foil are found to be slightly different from the original foil due to the shape of trailing edge. The mesh statistics are listed in Table 3.3.
Figure 3.19: Drag coefficient comparison between original and modified foils

Figure 3.20: Mesh around blunt trailing edge

Table 3.3: Mesh static for blunt foil

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points</td>
<td>21,256</td>
</tr>
<tr>
<td>Faces</td>
<td>41,063</td>
</tr>
<tr>
<td>Cells</td>
<td>10,145</td>
</tr>
</tbody>
</table>

3.5.2.2 Sharp Foil in Rectangular Grid

The sharp foil is constructed by sharpening the upper and lower surface of the blunt foil into a $45^\circ$ angle from the foil chord line. With this shape, the fluid flow is expected to detach from the foil surface smoother than the blunt foil and so reducing the vorticity generation at the trailing edge. The foil and domain grid development process is similar to the blunt foil method by using SolidWorks 2014. The grid properties and fluid properties are the same as those used in the blunt foil. The mesh near the trailing edge region is depicted in Figure 3.21 and the mesh statistics are listed in Table 3.4.
3.5.2.3 Rounded Foil in Rectangular Grid

The rounded foil is achieved by truncating and rounding the trailing edge into a semicircular circle. The grid properties and the fluid properties used in the model is the same as in the blunt and sharp foil. The mesh around the trailing edge and the mesh statistics for the rounded foil are shown in Figure 3.22 and Table 3.5 respectively.

Table 3.4: Mesh static for sharp foil

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points</td>
<td>21,288</td>
</tr>
<tr>
<td>Faces</td>
<td>41,127</td>
</tr>
<tr>
<td>Cells</td>
<td>10,161</td>
</tr>
</tbody>
</table>

Table 3.5: Mesh static for rounded foil

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points</td>
<td>21,230</td>
</tr>
<tr>
<td>Faces</td>
<td>41,017</td>
</tr>
<tr>
<td>Cells</td>
<td>10,134</td>
</tr>
</tbody>
</table>
The modified foils are modelled with nine static angles of attack as in original foil model. The modified foils which include blunt, rounded and sharp trailing edge shapes are depicted in Figure 2.2. The models are analysed in unsteady simulations using the PIMPLE algorithm in the pimpleDyMFoam solver. Following the modified foil model development, all results are compared to the original foil results which was already validated using Abbott’s (1959) experiment and Mittal’s (2002) and Eleni’s (2012) numerical models. The comparisons give an early prediction of the foil performance improvement with the modified trailing edge geometry. The lift and drag coefficients for the four trailing edge models are depicted in Figure 3.23 and 3.30.

The single modified foil models including normal, blunt, sharp, and rounded NACA 0012 are analysed in an unsteady simulation using the pimpleFoam solver at nine different angles of attack and these are -8°, -6°, -4°, -2°, 0°, 2°, 4°, 6° and 8°. The residual factor is $1 \times 10^{-5}$ for all parameters including pressure, velocity, $k$ and omega. The lift coefficient for four different trailing edge shapes in various the angles of attack are shown in Figure 3.23.

![Figure 3.23: Lift coefficient of four trailing edge shapes](image)

From Figure 3.23, it can be seen that at all static angles the blunt foil has the highest lift coefficient while the sharp foil has the lowest lift coefficient, except at angle of 6° and -6°. At those angles, the mesh topology which is adopted from the original foil is less suitable for the sharp trailing edge. The sharp foil has a pointed end at the trailing edge which likely requires a more refined mesh topology and boundary layer construction. A refined mesh is expected to follow the sharp profile and capable to refine the edge more smoothly. Thus at 6° and -6° in the sharp trailing edge case, the lift coefficient appears to be over predicted. From Figure 3.23, it also found that a sharp and rounded foil has a similar lift coefficient which is lower than original and blunt foil.

It can also be seen from Figure 3.23, that the blunt foil has the highest lift coefficient magnitude. The manner in which a trailing edge is cut in the blunt foil modification affects the fluid passing over it and increases the fluid velocity. The increase is caused by
the sudden change of trailing edge profile. The flat surface of the trailing edge also creates a bluff body effect which induces a base drag pressure behind the trailing edge. The base drag produces a void space at the back of the foil which provides a higher suction pressure towards the downstream region and accelerates the fluid passing over it. The base drag also creates two fluid pulse recirculation areas as shown in Figure 3.25. The increasing velocity contributes to the changing of pressure distribution magnitude which reduces at the suction surface thus the lift force is likely to be higher than original foil. At a higher angle of attack, this effect becomes stronger and the largest discrepancies in each shape are found at 8° and -8°. This effect does not appear in the sharp and rounded foil which has a lower lift coefficient than the original foil. It seems that the sharp and rounded trailing edge profiles restrain the fluid flow and create a lower velocity and higher total pressure on the upper surface of foil which consequently lead to a lower lift coefficient. For the drag coefficient, the plot is shown in Figure 3.23.

![Figure 3.24: Drag coefficient for trailing edge modifications](image)

The blunt foil has the largest drag coefficient as seen in Figure 3.23. This has also been predicted by Smith and Schaefer (1950). The cutting thickness is a potential source for the rise in drag coefficient. The cutting thickness also may cause the decrease in lift force but only for truncation above 15% (Ramjee et al., 1986). The original foil still has the lowest drag coefficient because the shape of the trailing edge prevents the bluff body effect and delays a fluid detachment. The sharp and rounded edge foils lie between the original and the blunt shape which means that those edges can delay fluid detachment but not as much as the original foil. Another term which is also analysed is lift to drag ratio. The lift to drag ratio in a vertical axis turbine can be interpreted as the ratio of power extracted to a viscous force which resist the turbine from rotating. Thus the lift and drag ratio is an indicator of a turbine efficiency.

The highest lift to drag ratio for each angle of attack is found in original, blunt, sharp and rounded foil respectively. For the same trailing edge shape, the ratio increases with the increase of angle of attack. This increment is a result of higher lift force
enhancement rather than the increasing drag force. However at some point the ratio drop significantly as a result of the stall phenomenon occurrence in which the viscous effect becomes stronger, thus producing a higher drag force. The modified foil lift to drag ratio has the same trend as the original foil and can be estimated from their lift and drag coefficient line trends. From simulation result and validation, the chosen configuration and mesh topology are adequate and can be applied to the foil response and a vertical axis tidal turbine response model. This will be described in the following section. The lift coefficient produced on blunt foil at low static angle of attack prior to the occurrence of the stall angle is predicted to be higher than that for the original foil as a consequence of higher fluid velocity caused by the trailing edge profile.

The blunt surface influences the vorticity profile behind the trailing edge. Figure 3.25 visualizes vorticity streamlines for the blunt foil at zero angle of attack. It is shown in Figure 3.16 that behind the trailing edge there are two pulses of vortex recirculation as predicted by Krentel and Nitsche (2013). The recirculation is caused by the trailing edge truncation which creates a base drag which has been explained previously. Figure 3.16 also shows that on both foil surfaces, the fluid is still fully attached up to trailing edge which makes the fluid flow regime is laminar. Because of the flat truncated profile at trailing edge, the backward flow is only reconstructed at the recirculation region. The fluid does not flow back on the upper and lower surface and creates less vorticity formation. The recirculation pulses, no reverse flow and less also happen at an angle higher than $0^\circ$ angle of attack. Therefore the blunt foil lift coefficient is higher. However the drag ratio is also high due to the bluff body effect as a result of the truncated profile. The static angle and oscillating foil result is an early prediction for the development of a vertical axis tidal turbine response model which will be discussed in Chapter 4.

![Figure 3.25: Two fluid pulse behind blunt trailing edge at zero angle of attack](image-url)
3.6 Flow Regime on A Dynamic Foil

Unsteady foil behavior and flow passing over it was reviewed by McCorskey (1982). He was a spearheading the study and since then many experimental and numerical inspections have been done during the last decade to understand how unsteady foil behaves. The aim of unsteady foil investigations are mainly to overcome the problems occurring such as stall, vorticity or vibration. One method to induce unsteadiness in an experiment or CFD is by applying a prescribed force on a foil to excite oscillations or pitching with a specific reduced frequency. This unsteadiness induces vibration as can be observed in flutter and studies in the area of fluid-induced vibration. In this section, the unsteadiness discussion covers the possibilities of foil motion such as pitching, plunging and the combination of those motions which potentially induce vibrations. The fluid induced vibration is also identified and investigated in order to control the vibration in a structure.

Unsteadiness of a blade can also happen when the foil undergoes a high static angle of attack as explained previously. High angle of attack blade creates separation and vortex shedding on the upper surface of the foil for angle higher than 17°. Vorticity and wake in the fluid flow regime generate a stall and excites a flutter on the foil which creates foil unsteadiness. The separation and vorticity also produce high drag force and, to some extent reduce the lift force which result in the stall phenomenon. Both in static and dynamic foil, the separation is known as a fluid physical behaviour distinguished by bubbles detachment from the upper surface at leading edge flowing downstream towards trailing edge of unsteady foil. This phenomenon was further investigated by Lee and Gerontakos (2004). They conducted experiments with reduced frequency equals to 0.05, zero initial angle, and 7.5° amplitude and showed lift, drag and moment coefficient for static and oscillating foils. On the oscillating foil, their result revealed that hysteresis exists on lift and moment coefficient at almost all angle of attack for, but it happens to drag coefficient only at positive angle. The onset of transition from laminar to turbulent was also delayed when reduced frequency decreases.

Another experiment which investigated the behaviour of vortex shedding was observed by Jung and Park (2005). They conducted static and oscillating foil experiments at Re ranging from $10^4$ to $10^5$ to measure vortex shedding frequency. The foil oscillation had a reduced frequency varying from 0.1 to 0.4. Unlike the static foil, their results showed that vortex shedding frequency was found to vary with the phase of oscillations rather than angle of attack. They suggested a method to find a vortex shedding frequency from the time averaged velocity signal (actual signal) and the velocity of foil motion over 360° oscillation phase. The velocity signal was decomposed from three components: the time varying velocity signal, a phase averaged oscillatory foil signal and vortex shedding frequency. The phase averaged oscillatory foil signal is retrieved by applying phase averaging technique to a reference signal. The phase averaging methods to calculate the
fluctuating signal was introduced by Hussain and Reynolds (1969) and it was detailed in Chapter 2.

3.6.1 Pitch Motion (Oscillating Foil)

Flow over an oscillating foil has been tested by Berton et al. (2002), Lee and Gerontakos (2004), Jung and Park (2005), and numerically studied by Frederich et al. (2009), AntonSavioLewie (2014), and Chandravanshi et al. (2010). Through their rig using embedded laser doppler velocimeter, Berton et al. (2002) studied the physics of unsteady separated flows around an oscillating NACA 0012 with $Re$ of $10^5$ and $2 \times 10^5$, reduced frequency of 0.188 and amplitude of $6^\circ$. The fluid flow regime was shown at angle of attack of $10^\circ$ and $14^\circ$ fixed angle of attack, incidence down stroke and upstroke oscillation. It was clearly shown that separation occurred from the upper side of leading edge at upstroke $10^\circ$, however it did not appear at the steady and incidence downstroke case. At $14^\circ$ angle of attack, separation became stronger at incidence upstroke although separation did not exist at steady and downstroke case.

Oliveira and da Silveira Neto (2005) conducted a simulation of an oscillating NACA 0012 using immersed boundary method with virtual physical model. They had a rectangular grid with $8c$ width and $10c$ length. They placed the foil on a point of $2.7c$ behind inlet patch and in the vertical center of their domain. Their virtual physical model was used to simulate and visualize the fluid flow regime. They recorded the time history of $C_l$ and $C_d$ for pitching foil with mean angle of $15^\circ$ and amplitude of $10^\circ$. They investigated the influence of changing reduced frequency to the fluid flow. Their result with rectangular grid with aforementioned size can capture the vorticity and predict lift and drag coefficient. It also showed that by increasing the reduced frequency delayed the separation to higher incidence angle.

Pitching motion or an oscillating foil has been examined by Berton et al. (2002), Frederich et al. (2009), Martinat et al. (2008), Corrêa et al. (2014) and many more. Corrêa et al. (2014) numerically investigated $C_l$ and $C_d$ hysteresis on an oscillating foil as an effect of flow over a NACA 0012. Their simulation used two different turbulence model and they validated the result with experimental data from Berton et al. (2002) and simulations from Martinat et al. (2008) as depicted in Figure 3.26. Again hysteresis is found from the lift and drag coefficient plot. Beyond $12^\circ$ angle of attack for lift coefficient, $k-\epsilon$ and Spalart Almaras turbulence model over predicted the experiment result while $k-\omega$ model did not have a good prediction except at angle of attack around stall. For the drag coefficient all models gave similar prediction to experimental data as shown in Figure 3.26.

Martinat et al. (2008) built a 2D model for NACA 0012 at $Re$ of $10^5$ and $10^6$, using three different turbulence models ($k-\omega$, $k-\epsilon$, and Sparat Almaras). They validated lift and
drag coefficients with their own experiment. They concluded that advanced turbulence models like SST shows better results than classical URANS models for dynamic stall prediction and vorticity capture. Therefore $k-\omega$ SST is used in this model.

For high Re, the fluid flow prediction on an oscillating foil was also studied by Frederich et al. (2009). They investigated NACA 0012 numerically and developed a CFD model using a C-structured and O-structured grid with $Re$ of $1 \times 10^6$ using two different turbulence models which were RANS and DES for 2D flow. Their result visualized vorticity profile and separation as shown in Figure 3.27. At angle of 20°, the separation started at leading edge flows downstream and become stronger when the angle of attack was higher. The fluid flows abruptly and started to stall at the angle of 23°. At the decreasing angles, the fluid flow behaves similarly with hysteresis found in the flow as found in static angle. From their result, it can be seen that either RANS or DES is applicable for predicting flow behaviour and dynamic stall. Frederich et al. (2009) result will be used to validate the oscillating foil model.

3.6.2 Heave Motion (Plunging Foil)

A plunging foil is characterized by the lateral motion in the direction perpendicular to the chord line. The foil is moving vertically in respect the blade position in x-y plane and its motion is similar to heave vibration in the blade response model. Plunging motion of a foil has been experimentally investigated by Cleaver et al. 2009 at a post stall angle of attack of 15° with an amplitude of 2.5% to 20% of chord. From the results, they found that dramatic lift enhancement could be achieved with the maximum improvement.
Chapter 3 Development of the CFD Methodology for Static and Dynamic Foils

Figure 3.27: Vortices in trailing edge of a foil as the function of foil angle of attack (Frederich et al., 2009)

reached 300% over a static foil. The enhancement was due vorticity formation on the
trailing and leading edge regardless Reynolds number.

Young and S. Lai (2004) developed a 2D model of plunging NACA 0012 and visualize the wake behind the trailing edge. They found that wake generation was not affected by Mach number. The trailing edge influenced strong effects on the formation of wake structure but only had secondary effect on the lift and thrust force. The pressure distribution, as well as aerodynamic pressure are closely related to leading edge separation particularly when the separation becomes prominent. Chandravanshi et al. (2010) were conducted a simulation for plunging foil motion on NACA 0012 using rectangular grid. They simulated their model with variation of a parameter, labelled as \( kh \). \( kh \) is obtained from reduced frequency (k) and plunging amplitude (h). They found that for plunging foil, the \( kh \) parameter is more important than reduced frequency itself.

### 3.6.3 Pitch and Heave Motion (Pitching and Plunging Foil)

An experimental study combined with numerical approach to fluid structure interaction on a NACA 0012 blade was done by Veilleux and Dumas (2013). The blade numerical model was developed with self-sustained pitch and heave vibration and simulated using RANS turbulent model and laminar model in OpenFOAM. The experiment and numerical results are confronted with both theoretical pitch and heave natural frequency. From their experimental and numerical result, it can be shown that their blade oscillates at frequency close to the heave natural frequency. Therefore they tried to examine the influence of heave stiffness variation to the response while kept the pitch stiffness constant. The heave stiffness was varied by 1484 N/m and 800 N/m. The model resolved quite well with their experiment especially predicting the pitching motion in high heave stiffness and can be applied in high Re. This heave stiffness variation from Veilleux and Dumas (2013) is adopted in this work.

Yang et al. (2006) also worked numerically on the combination of pitching and heave motion on a foil. The instantaneous angle of attack on the pitch heave foil with phase difference of 90° is lower than either pure pitch or heave foil. Therefore lift and moment coefficients at the combination foil are lower than either static, pure pitching or pure plunging foil. Their result also showed a hysteresis in the lift and moment coefficient.

A foil undergoes pitch and heave motion with a phase difference of 0°, 90°, 180°, and 270° was tested by Lee and Su (2015). Their experiment worked on NACA 0012 and visualized fluid flows using a smoke detector. They also resolved pressure distributions of some instances at those four phase differences. Their result showed that at a phase difference of 0°, the instantaneous pressure coefficient of pure heave is similar to one of the pure pitch foil. Also at this phase difference, the pressure coefficient of pitch heave foil resembles either pure pitch or pure heave but it has higher magnitude. The leading edge vortex is formed when very low leading edge pressure gradually changes
and becomes almost zero at the trailing edge. The leading edge vortex begins to grow and spill from the foil’s surface. The growing vortex is characterized by a sloped pressure coefficient line whereas the spilling vortex is known by the flat line which is almost zero over the foil surface.

The dynamic foil model and analysis will also be applied to modified trailing edge shape similar to the case of static angle. The physical phenomenon and the background theory is detailed in the next section.

In the dynamic mesh which is employed for the oscillating foil and for the response model, the cell shape changes during the simulation. For the oscillating foil model, the changing cells are determined by a prescribed motion whereas in the response model the cell position is resolved from the fluid equation coupled with the vibration motion equation.

### 3.7 An Oscillating Foil Model

The topology and number of cells are very sensitive in dynamic mesh model and affectes the c-structured grid model execution time. The selected c-structured grid model is executed ten hours to model 0.1 second which is likely unreasonable. Therefore the rectangular grid is implemented to dynamic model including forced oscillation foil model and the vertical axis tidal turbine response model for original and modified blades. As mentioned in Section 3.3.2, the log law region is applied to accomodate the y+ value which is bigger than 30 in the rectangular grid.

The oscillating foil model is developed using rectangular grid used in the static angle of attack foil which has been explained in Section 3.2.1.2. The results are validated using the results from Frederich et al. (2009). The oscillatory motion is prescribed by the angularOscillatingDisplacement utility in the pointDisplacement file under the zero folder. The oscillation properties adopt Frederick’s model and are listed in Table 3.6. The dynamicMotionSolverFVMesh mesh manipulation is used in the oscillating model, thus the simulation requires displacementLaplacian solver as a mesh motion solver.

<table>
<thead>
<tr>
<th>Table 3.6: Oscillatory motion properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial angle (rad)</td>
</tr>
<tr>
<td>Amplitude (rad)</td>
</tr>
<tr>
<td>Angular velocity (rad/s)</td>
</tr>
<tr>
<td>Axis of rotation</td>
</tr>
</tbody>
</table>

A single oscillating foil is modelled in this work to obtain a prediction of the dynamic mesh performance and to ensure the capability of the model to solve a dynamic foil motion. The oscillating model is validated using simulation results from Frederich et al.
(2009) and Martinat et al. (2008) which have 0.1 reduced frequency and a mean incidence angle of 15° with an amplitude of 10°. The lift force result is given in Figure 3.28.

![Figure 3.28: Lift coefficient of the oscillating foil model](image)

At the beginning of the simulation unsteady flow occurs as shown by the fluctuating lift coefficient near 5° as shown in Figure 3.28. Beyond 8° angle of attack, the lift coefficient is stable although hysteresis exists on the foil. In upward motion the foil reaches a maximum lift coefficient value of 1.7 which is not observed in downward motion. This hysteresis was also observed by Martinat et al. (2008) and Corrêa et al. (2014) as shown in Figure 3.26. However the results do not give good agreement for angles of attack higher than 15.9°. Nevertheless the domain still can be applied in the response model since the highest angle of attack in the blade turbine response calculated from Equation (2.4) is 11.3°. The vortex shedding frequency which is the main focus of this work is influenced by the phase of oscillation or vibration rather than angle of attack itself (Jung and Park, 2005). For time history component of velocity are plotted in Figure 3.29.

![Figure 3.29: Velocity component in oscillating foil, velocity (m/s) vs simulation time step](image)
From Figure 3.29, it can be seen that the frequency of $U_x$ is twice that of $U_y$ due to the alternating wake in the fluid flow regime. The magnitude of $U_y$ is also found to be higher than $U_x$ which will cause higher lift force than drag force thus the advantage of lift force is exploited to drive the vertical axis tidal turbine rotor. The lift coefficient frequency also propagates with 0.5 drag coefficient frequency. The fluid forces on the blade are strongly influenced by fluid behavior. The fluid flow regime in oscillating blade at selected times are shown in Figure 3.30.

(a)  
(b)  

(c)  
(d)  

(e)  

Figure 3.30: Fluid flow regime of the oscillating foil: a. 15°, b. 25° (upstroke), c.15° (downstroke), d. 5° (downstroke), e. 15°(upstroke)

In Figure 3.30a a blade is at 15° angle of attack. A weak wake is detected on the upper part of the foil and generates a smooth fluid flow regime as also found in Frederick’s model (2009). From that point the foil oscillates upward reaching maximum angle of 25° when a very strong vortex is generated on the upper surface. The vortex then flows downstream towards the foil trailing edge. $U_x$ and $U_y$ magnitude becomes zero. The
blade moves downward from this point and reaches $15^\circ$ at the midpoint as depicted in Figure 3.30c. Although it is in the midpoint, i.e. at the same position as in Figure 3.30a, the wake pattern is stronger at this stage due to hysteresis and a higher lift coefficient is produced on the upward than downward motion. In Figure 3.30c, the blade has a stronger wake which cause the lift coefficient in downward motion to be less than the upward motion as can be seen in the lift coefficient history in Figure 3.28. The blade oscillates downward and reaches a minimum angle of $5^\circ$ as illustrated in Figure 3.30d. The wake does not occur in the fluid flow regime because the angle of attack is very low. At the end of the cycle, from the minimum position, the blade oscillates upward, back to its initial position at $15^\circ$ with the vortex generation found on the upper surface.

The fluid flow behavior sequence in Figure 3.30 agrees quite well with Frederich’s model (2009) in terms of the fluid flow pattern and stall occurrence prediction. The presence of vortex shedding can create an abruptly irregular and chaotic flow in which the stall phenomenon can be detected. This phenomenon is responsible for total pressure reduction, lift force drops and a significant increase in drag. The stall phenomenon occurs at $16^\circ$ as shown in Figure 3.28 where the lift force suddenly drops from 1.686 to 0.756. The prediction of the separation event also shows good agreement with Frederich’s (2009) simulation which begins after $15^\circ$ angle. With these validation, the mesh configuration is considered to be of good quality, the turbulence model is reliable, and can cope with vortex shedding phenomenon on the foil. Therefore the mesh topology is expected to handle the response problem for the original and modified foil. This is detailed in following section. The response models also use dynamic mesh and pimpleDyMFoam solver.

### 3.8 A Rotating Three Bladed Vertical Axis Tidal Turbine Model

A single unaltered NACA 0012 foil and its three different trailing edge modifications including blunt, sharp and rounded modelled at $Re$ of $3 \times 10^6$, using the k-\omega SST turbulence model will be explained in this chapter. The models are analyzed in a rectangular domain and the mesh is refined using snappyHexMesh utility. The foil is generated from NACA 0012 Equation (Equation (2.1)) using Matlab and extruded into 3D form using SolidWorks 2014. The models are simulated in OpenFOAM 2.2 using the simpleFoam solver for the static angle of attack foil in steady state and the pimpleFoam solver for the static angle of attack foil in unsteady state. The pimpleDyMFoam solver is used for dynamic mesh simulation in the oscillating foil, the foil response and vertical axis tidal turbine blade response models. The static angle of attack model is analysed using in nine different angles and validated according to Abbott and Von Doenhoff (1959) experimental results, in addition to Eleni’s (2012) and Mittal’s (2002) numerical result.
The validation compares the simulation results and investigating the error between these and the reference data. The result will be discussed in Chapter 4.

A vertical axis tidal turbine model is also constructed in this work using Pointwise V17.2R2. The model is then exported to OpenFOAM and simulated using pimpleDyM-Foam solver. The topological strategy for the vertical axis tidal turbine is to divide the model into two main parts, the stator and rotor. This strategy allows the turbine model to only have moving parts in the rotor region and to concentrate on that region by specifying a high resolution mesh. The rotor is the part where the three turbine blades are located whereas the stator is the stationary outer region of the blades. This strategy reduces number of cells being required in the model without losing precision in the result and also allows reduction in simulation time.

The three bladed vertical axis tidal turbine is modelled in this project by adopting the design from Lanzafame et al. (2014). The turbine design was previously employed in the vertical axis tidal turbine projects for Indonesian renewable energy project as discussed in Section 1.2. The turbine design parameters are taken from Gosselin’s simulation results (2013) and have previously been explained in Chapter 2. The turbine design parameters are listed in Table 3.7.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord (m)</td>
<td>0.75</td>
</tr>
<tr>
<td>TSR</td>
<td>5.1</td>
</tr>
<tr>
<td>Solidity</td>
<td>0.1829</td>
</tr>
<tr>
<td>Radius (m)</td>
<td>1.958</td>
</tr>
<tr>
<td>Blade aspect ratio</td>
<td>25</td>
</tr>
<tr>
<td>Turbine aspect ratio</td>
<td>0.383</td>
</tr>
</tbody>
</table>

The turbine and its surrounding mesh which are generated using pointwise V17.2R2 software is depicted in Figure 3.31.
The mesh is comprised of two main parts which are a stator mesh and a rotor mesh. The stator mesh represents the far region or the outer environment from the turbine. The rotor mesh simulating the rotating part of the turbine which model the symmetric three blades configuration. Each blade occupies a 120° symmetric region in respect to the rotor centre with expansion ratio in all of the interface connectors are 0.1 as seen in Figure 3.32. The interface connector is a circular line which connects turbine stator and rotor and is created using AMI (Arbitrary Mesh Interface) utility. This utility enables the turbine model to have combined adjacent domain configured from a stationary and a moving part. The stator or stationary part has coarser mesh than in the rotor region.

There are eight boundary fields in the whole domain including top, bottom, front, back, inlet, outlet, the rotor, and interface between rotor and stator. The fluid enters from the inlet boundary field which lies on the left hand side of the domain, flows through the stator, passes the rotor part and exits the domain at the outlet field.

Both the rotor and the stator mesh are generated using Pointwise V17.2R2 and imported into OpenFOAM. The mesh configuration uses a hybrid mesh which employs a combination of structured and unstructured grids. The structured grid is composed from an array of hexahedrons and creates a more organized configuration than the unstructured grid. Although the cell shape is quite different, the structured and unstructured grid which is applied to NACA 0012 almost produce the same result (Alakashi and Basuno, 2014). The grid type implementation depends on the domain or object complexity. The unstructured grid is more flexible to follow more complex domain and has a more flexible arrangement, thus allowing it to follow an irregular body shape. In the mesh generation stage, the unstructured grid is less time consuming to generate. In this work, both grids are implemented to gain these benefits. Moreover the CFD solution does not affect by the manner of both meshing process (Alakashi and Basuno, 2014).

The unstructured grid is generated in the stator region, where the fluid flow regime is not the main concern. The unstructured grid is also generated at the region within the
Chapter 3 Development of the CFD Methodology for Static and Dynamic Foils

rotor. Meanwhile the structured grid is applied at the boundary layers around each of the blades. This configuration allows the model to build the mesh in less time since only the crucial part is generated using a structured grid. Although in this work the foil wake inside the rotor is not the main focus as the trailing edge vortex, it potentially impacts the result when contacts with trailing edge vortex regime from the front blade. The unstructured grid is also chosen for the inside rotor mesh but it is generated with a higher resolution so that the foil near wake can be captured. However the refined grid generation in this region requires more simulation time. The element size of the stator unstructured grid gradually changes from the rotor edge (interface connection) towards the far field region. There are four division in the stator region in which the mesh is controlled by the setting either the number of the points in lines or the mesh expansion ratio. The four stator divisions are located at the upper left and right and the lower left and right region drawn from the rotor centre line. The cells at the upper left in stator region propagates to the domain top and inlet line with the expansion ratio of 0.4 and 0.3 respectively. The generated cells at the other three divisions are identical.

The three truncated blades are identical and the inside rotor mesh for the three blades are symmetric. The blade is formed by 720 points with expansion ratio of 0.001 at the leading edge and of $10^{-4}$ at the trailing edge. Twenty five boundary layers are drawn around the blade with the expansion ratio of 1 outward from the foil’s surface. At the trailing edge region the boundary layers are extruded with the same length as the foil’s chord. Twenty points is added at the truncated side of the blade to produce another boundary layers. The added boundary layer has the same expansion ratio as the leading edge boundary layer. The outermost boundary layer is connected to the interface connector by creating two straight lines on the upper left and right of the blade.

The three identical $120^\circ$ rotor regions are connected side by side creating a whole circular rotor mesh. Each connector line has 150 points and is expanded with ratio of 0.1 towards blade and 0.01 to the interface connector. The interface which is manipulated using topoSet is stitched to the stationary zone and defined in the topoSetDict file under the system folder. This utility sets the interface connector as a sliding line to create a rotating zone at the rotor region. In the sliding mesh such as in this turbine model, the cells shape remain the same during the simulation but the mesh topology changes. In contrast from the sliding mesh, the dynamic mesh cell shapes changes during the simulation. The dynamic mesh is used in the oscillating foil and the blade response model and will be detailed in the next section.

3.8.1 Result

A three bladed vertical axis turbine model is generated using a hybrid grid generated in Pointwise and imported into OpenFOAM. The model is simulated using AMI (Arbitrary Mesh Interface) utility. This utility connects two surfaces from different parts which
intersect each other. AMI stitches both surfaces together to achieve an interface surface on which both original boundary conditions are combined. The AMI utility is employed in the interface line between the rotor and stator which represent the boundary of the rotating turbine and its environment. The design parameters are taken from Gosselin’s (2013) simulation results. These includes essential parameters which have been listed in Table 2.2. The result is shown in Figure 3.33.

![Figure 3.33: Lift and drag coefficients in one turbine revolution with TSR of 1.25 and fluid velocity of 1.4 m/s, shown in azimuth angle of 30° decrement (12=30°, 11=60°, etc)](image)

In Figure 3.33 lift and drag forces during a turbine rotation are depicted. The values are shown in every 30° azimuth angle starting from 0° counterclockwise. From the result it can be predicted that in the first quadrant (azimuth angle of 0° to 90°) the turbine experiences higher lift coefficient. The high lift coefficient also occurs in the early stage of fourth quadrant (azimuth angle of 270°). The drag force is found high at the second quadrant when the turbine blade approximately reaches 120° azimuth angle. This prediction will be verified with the respond model of the three bladed vertical axis tidal turbine in Chapter 4. The foil response is generated in a standard model as shown in Table 2.3. The foil response is proposed to obtain an early prediction of the turbine blade response. An early prediction is achieved by further varying the standard model simulation parameters and the foil mechanical properties to represent variations of the environmental condition. These variations are shown in Table 2.4. The standard foil response model and its variation will be discussed in the next section.

The rotating three bladed vertical axis tidal turbine model was executed in Iridis, University of Southampton’s supercomputer facility. The facility has technical specification as follows (isolutions website, University of Southampton)

- 16*2.6 GHz cores per node
4 GB of memory per core
• 4 high-memory nodes with 32 cores and 256 GB of RAM
• 24 Intel Xeon Phi Accelerators (25 TFlops)
• 1.04 PB of storage with Parallel File System
• Infiniband network for interprocess communication

The vertical axis tidal turbine model is executed in iridis using 1 node with 16 processors. With typical topology and number of cells which the model has takes 24 hours to finish 0.007 second simulation time. The running time is not reasonable for processing the response model simulation. Therefore the single blade using rectangular domain and the Equivalence Incoming Velocity is utilized to resolve the vertical axis tidal turbine blade response. This will reduce the model complexity and number of cell and that will much reduce the model execution time. The response model using spring and damper system which is detailed in Subsection 2.10 and 2.10.4 are employed. The result of a foil response in various condition and a vertical axis tidal turbine using original and modified blade using the spring damper system are discussed in Chapter 4.

3.9 Summary

In Chapter 3 fluid flow regime passing the static and oscillating original foil are discussed. The foil performance parameters are first introduced including $C_m$, $C_p$, $C_l$, and $C_d$. The fluid flow condition is examined. Based on the physical phenomenon on the static foil, a 2D CFD model is developed. The model employs a single NACA 0012 foil with 0.75 chord length in two different grid including e-structured and rectangular grid. Both topologies have denser mesh around or behind the foil to solve the near wall viscous effect on the fluid and capture the vortex generation and separation. Mesh independence study is taken to ascertain the minimum mesh requirement to run the simulation in effective manner (precise result with the minimum running time). The static foil is simulated in nine angles of attack and the results are verified using one published experimental data and two numerical results.

Similar to the static foil investigation procedures, in the dynamic foil investigation, the fluid behaviour passing an oscillating foil is examined. The dynamic mesh is validated using a reference data to assure the mesh is robust to solve a dynamic foil. The dynamic foil describes a moving foil in either force applied motion (oscillation) or free motion such as in the response model. The fluid behaviour of a foil in pitch motion, heave motion, and combined pitch and heave motion are explained. The fluid behaviour around those force applied motion foil is significant as a basis justification for the free motion foil in the response model.
The fluid behavior passing modified foils are also discussed in this chapter. These profiles are selected based on the published researches which proved the modified blades capability to improve a turbine performance. The development of modified foil CFD models in the static and dynamic case are examined. The fluid behaviour in original and modified foils in the static and dynamic case will be a significant information to advice the development in the response model which is the main issue in next chapter, Chapter 4.
Chapter 4

Fluid Structure Interaction of a Foil with Steady Incoming Flow

The interaction between a foil and fluid is developed in this chapter. The interaction is modelled by a vibration response system using spring damping components. The model is then replicated for a vertical axis tidal turbine blade in order to understand the blade behavior during the turbine operation. The foil response and the vertical axis tidal turbine blade response models are developed using simulation parameters which have been tested for static angles of attack and oscillating cases. The lift, drag, moment, and pressure coefficients are recorded over fifteen seconds in each case. The velocity at two points which are near the leading edge and at 0.1c behind the trailing edge are monitored to obtain a purely periodic signal and the original fluctuating signal. These two velocity signal are analyzed using the Phase Averaged and PSD methods to determine the vortex shedding frequency. The vortex shedding frequency is further used to identify the existence of lock-in phenomenon.

The mesh topology is constructed and subjected to static angle of attack and used for the dynamic mesh model. There are nine angles of attack simulated for the static angle models. For the dynamic foil cases, an oscillating foil, a foil response in steady incoming flow and a vertical axis tidal turbine blade response are modelled. The static angle of attack models are analyzed using the PIMPLE algorithm (pimpleFoam solver). Both results are verified and validated prior to developing the dynamic mesh models. A good result validation ascertains the constructed mesh to be adequate to solve the problem and capture the behavior of the fluid flow regime. The validation process is performed by comparing lift and drag coefficients from nine static angles to experimental and numerical reference data. The dynamic mesh is also validated by comparing the oscillating foil results to reference numerical results.
Figure 4.1: Flow chart for calculating FSI response of a vertical axis tidal turbine blade after response model mesh is validated.
4.1 Foil Response Model Results

In this section, the result from a foil response model when it interacts with fluid will be discussed. The incoming fluid velocity is assumed to be constant and its velocity magnitude is taken from the maximum incoming velocity in the turbine blade response model. Firstly the foil response model is developed according to the standard operating properties shown in Table 2.3. The operating properties are then varied to observe the behavior of the foil and fluid interaction in different environmental conditions. The foil response is observed in the case of velocity variation, initial angle of attack variation, number of orientation variation, stiffness constant variation and damping constant variation.

The number of cells and mesh topology is a big challenge in dynamic mesh models thus simulation time is very sensitive and will be validated for the vertical axis tidal turbine response with original blade. The model is validated using Almohammadi et al.’s model (2015) and discussed in the next subsection.

4.1.1 Calculation Routine of A Response Model

The foil response in the standard model is obtained by fitting the fluid and mechanical properties as in Table 2.3. The model is calculated using the routine as shown in the calculation flow chart of Figure 4.1.

1. Select two nodes for velocity recording The two points as shown in Figure 4.2 are sampled to obtain the reference (input) and original (output) velocities to be used in the Phase Averaged method. The input signal is taken from leading edge point whereas the output signal is recorded from a point which is 0.08 m behind trailing edge. The distance is adopted from Jung and Park’s result (2005) which has been detailed in Subsection 2.10.1.

The vortex shedding velocity obtained from the Phase Averaged method is then inserted into the PSD method to find the vortex shedding frequency. The reference and original signal history are found in Figure 4.3. From this figure it can be seen that during the first three seconds the case is transient with an unsteady velocity pattern. After three seconds, the original velocity plot stabilizes and the steady condition appears from 3 seconds onward. The first three second is then omitted in the Phase Averaged process and the PSD methods as shown in Figure 4.4. The Phase Averaged and PSD method are adopted during that temporal period for this case and the rest of the models in this project.

2. Record velocity from the designated nodes
3. Phase averaging process to obtain vortex shedding frequency

Figure 4.4 illustrates the Phase Averaged signal (yellow line) which is obtained from original signal (blue line) using the Hilbert Transform method.

4. PSD method to obtain main vortex shedding frequencies

The vortex shedding velocity from the original foil in the standard model and its frequency domain power spectral plot is shown in Figures 4.5a and 4.5b respectively. The frequency domain plot also illustrates the foil natural frequency which is calculated from the foil stiffness and its mass using a second order vibration equation as discussed in Subsection 2.10.3. The heave and pitch motion natural frequencies were calculated and the values obtained are 0.9425 Hz and is 1.4758 Hz respectively. Both natural frequencies of the system are always plotted in frequency domain graph for all the response models to identify the resonance presence.
Figure 4.4: Phase averaged signal taken from the reference (input) for the standard foil model, shown along with the input and output signal

5. Compare the main vortex shedding frequencies with natural frequencies to identify the lock-in condition

From Figure 4.5b it can be seen that $f_h$ (natural frequency in heave direction) is lower than pitch natural frequency. This indicates the pitch response is dominant over the heave response as it will be proven later in the discussion for a number orientation response in Subsection 4.1.4. There are three dominant frequencies (peaks) obtained from the default case, these are 0.31 Hz, 2.34 Hz, and 4.76 Hz. All frequencies exist far from heave and pitch natural frequencies therefore the vibration is not lock-in or a resonance does not exist in the system. A resonance is detected by a strong vibration which can be observed from the time history of the object’s displacement. The displacement of the standard foil model is shown in Figure 4.9. The vibration (displacement) is driven by the fluid forces which are illustrated in Figure 4.10.

6. Verify the response condition (whether it is locked-in or not) The locked-in condition as a result of the blade interaction with the fluid is verified using three features. These are the captured vortex shedding profile in the near wake, pressure distribution over the foil’s surfaces and the foil’s displacement.

- Vortex shedding profile

Figure 4.6 shows one oscillation cycle for the original foil response in the standard model. At $t/T$ equal to 0.1, the foil is at the lowest position in the oscillation cycle. A small vortex is shed from the leading edge and moves toward trailing edge as the foil moves upward until reaches its maximum elevation at $t/T = 0.6$. At this position, the wake is attached and generates the highest lift force one oscillation cycle. The lift force is strongly influence by the pressure distribution over the foil’s surface as shown in Figure 4.7
Figure 4.5: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the original standard operating foil model which corresponds to fluid flow regime in Figure 4.6. From highest position, the foil moves downward to reach the lowest position at $t/T = 1$. The overall vibration is found to be weak. A small vorticity is observed to form in front of the foil (LEV, Leading Edge Vortex) during the vertical motion. The LEV which is constantly formed at all stages, is distinguished by a pressure peak and has been discussed in Section 2.7 (Figure 2.17).

- Pressure distribution
As predicted by Blevins (1990), the distribution pressure line oscillates on a surface where the vortex is shed. This is also happens on the standard foil model shown in Figure 4.7. Figure 4.7 displays the pressure distribution contour over the foil surface at $t/T$ equals to 0.1, 0.6 and 1 corresponding to Figures 4.6a, 4.6b and 4.6c respectively. The pressure fluctuates more
Chapter 4 Fluid Structure Interaction of a Foil with Steady Incoming Flow

Figure 4.6: Default condition of original foil fluid flow regime during one cycle of vibration, at $t/T$: a. 0.1, b. 0.6, c. 1

Figure 4.7: Pressure distribution on the original foil in default operation at minimum and maximum extent of vibration at the leading edge where the separation or vorticity begins to form (LEV). This means a separation or a vorticity is triggered by the highly fluctuating pressure. A fluctuating pressure is denoted by a pressure gradient in the direction of the flow (x direction). Hence it can be stated that the separation
or vorticity is generated in the high pressure gradient region which is denoted by a pressure peak in the pressure distribution profile. This result is in a good agreement with Lee and Su’s result (2015) as explained earlier in Section 2.7. The pressure distribution peak exists at $x/c = 0.261$ where the vortex begins to form as shown in Figure 4.8. The highly fluctuating pressure indicates the phenomenon of separation occurring in that location as depicted in Figure 4.8 (marked by a pink spot).

At the trailing edge the pressure distribution in Figure 4.7 captures an oscillating behavior which signifies a flow separation at that position. At $t/T = 0.6$, at the trailing edge of the suction surface, the fluid pressure is positive. This causes the foil to move downwards to its minimum position when $t/T = 1$. At this position, the vortex is shedding from the leading edge and flows towards the trailing edge as shown in Figure 4.8c.

In Figure 4.8c when the point is at $t/T=1$ the separation is strengthened. This is caused by the higher pressure similar to that at time at $t/T = 0.1$. The high pressure fluid particle tends to maintain higher potential energy than kinetic energy, thus in this situation the velocity is reduced. When the pressure gradient along the flow direction increases, an adverse pressure gradients exists which causes a reverse fluid flow. The reverse fluid flow creates a fluid separation and eddies starts to form. The fluctuating pressure induces fluctuating lift forces on the foil which cause it to vibrate as shown Figure 4.9. The foil motion at the leading edge and at the trailing edge and its dominant frequency has been explained and is shown in Figures 4.4 and 4.5.
From the plot of dominant frequency in Figure 4.5b, none of the dominant signal is close to heave or pitch natural frequency. This indicates that no resonance exists and only a small vibration induced on the foil as seen in the displacement plot in Figure 4.9.

- **Displacement**

In Figure 4.9, the displacement in the y direction is oscillates regularly with a peak to peak amplitude of 0.013 m. The vibration response of the foil is due to its interaction with the incoming fluid which is very regular and laminar. As a result, the foil experiences a weak vibration during the interaction. To understand the foil interaction with the incoming fluid comprehensively, the fluid parameters are varied as listed in Table 2.4. These parameters include incoming fluid velocity variation, initial angle of attack variation, number of response motion variation, stiffness constant variation and damping constant variation. This will be discussed in the following sections.

![Displacement vs time step at x/c= 0.216 for original foil response in the standard model.](image)

Figure 4.9: Displacement vs time step at x/c= 0.216 for original foil response in the standard model. pointDisplacement (magnitude) corresponds to magnitude of displacement, pointDisplacement (0) equals to x direction displacement, pointDisplacement (1) to y direction.

7. **Analyze the force coefficients**

Figure 4.10 shows the lift, drag, and moment coefficients for original foil in the standard model. The lift coefficient which has the highest magnitude drives the turbine to rotate and provides a power which can be extracted during operation. The fluid induced forces applied to the blade are regularly fluctuating which indicates the presence of vortex shedding as predicted by Gioria et al. (2009). He confirmed the vortex generation from the fluctuation in lift coefficient (Figure 3.4)
which is similar to fluctuations observed in Figure 4.10. The regular forces are reproduced by a fluctuating velocity signal with a small amplitude during the operation. The periodic forces are evidence that the foil and fluid interaction yields a weak vortex shedding. The interaction can also be recognized from the way the turbine vibrates as seen from the history of original foil response in Figure 4.6.

### 4.1.2 Inlet Fluid Velocity Magnitude Variation

The fluid velocity is a significant parameter for vertical axis tidal turbine operation. The direct contact of the fluid on the blade can damage the structure when the fluid flow becomes very chaotic and disruptive. The disruption is caused by a turbulent fluid flow regime when vorticity is formed and contacts with the blade. The vortex formation is of importance for a structure as the fluid loading becomes unsteady and the structure suffers from a fatigue problem. The situation is more severe when the fluid loading induces a response with same frequency as structure’s natural frequency. Both frequencies interact with each other, exhibiting resonance and further damage the structure. One aim of varying the operational parameters is to predict the resonant frequency so that this condition can be avoided. A range of velocity magnitudes are simulated as shown in Table 2.4. The Phase averaged and PSD methods are also applied to the cases. The reference, original and phase averaged velocities for 6 m/s and 2 m/s case are depicted in Figures 4.11a and 4.15a respectively. The vortex shedding frequencies at 6 m/s and 2 m/s incoming fluid velocity from the PSD method are shown in Figure 4.11b and 4.15b.

At 6 m/s, the fluid velocity fluctuates with high amplitude oscillations as shown in the original signal. The high amplitude fluctuating velocity indicates that the foil experiences a strong vibration as shown in the time history response images in Figure 4.12.
Figure 4.12 displays the sequence for a single vibration cycle induced on the foil when interacting with 6 m/s incoming tidal velocity. It can be seen that the fluid induced vibration on the foil is quite strong. This can be ascertained from the foil orientation and displacement. The detail displacement plots are shown in Figure 4.14.

![Graph showing velocity and displacement](image1)

(a)

![Graph showing frequency domain](image2)

(b)

**Figure 4.11:** a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the fluid velocity of 6 m/s model

In the case of the highly fluctuating velocity, it is important to analyze the velocity in the frequency domain to determine whether the lock-in phenomenon occurs. The frequency domain plot for the 6 m/s case is shown in Figure 4.11b. Although some high frequencies appeared and no single distinct dominant frequency is found, it should be noticed that the highest frequency is close to the pitch natural frequency. The highest frequency which is found to be 1.328 Hz, induces a resonance in the foil system. The resonance enhances the frequency of vortex shedding and influences fluid induced vibration on the foil. The resonant condition generates a high amplitude vibration response as shown in
Figure 4.12: Original foil fluid flow regime at 6 m/s over one cycle of vibration, at t/T: a. 0.071 b. 0.286, c. 0.5 d. 1

the time history motion plot Figure 4.14. The vibration is caused by a high fluid force applied to the foil. The forces in the velocity variation case are shown in Figure 4.16 - 4.18 and will be discussed later. Comparing the lock-in frequency in this case to the unlocked frequency in the standard model, it is shown that the lock-in vortex shedding frequency (1.328 Hz) exceeds four times the frequency in the shedding in standard model (0.31 Hz) as predicted by Besem et al. (2014).

Figure 4.12 shows the sequence of a revolution of the foil response when it interact with 6 m/s tidal flow. The foil is at its maximum position when velocity is at its minimum as shown in Figure 4.12a. From that position, the foil moves downward and reaches its midpoint (equilibrium) position at t/T = 0.286. From the equilibrium position the foil moves downward to reach the lowest position at t/T = 0.5 and moves back to maximum position at the end of the revolution (Figure 4.12d). From Figure 4.12, it can also be seen that separation is initiated on the leading edge of the suction surface (LEV) when the foil vibrates in its equilibrium position. The separation starts from the leading edge and flows downstream to behind the trailing edge. The separation is a result of the fluctuating pressure distribution over the foil surface as explained earlier in the foil standard model in Section 7.5. The pressure distribution for 6 m/s velocity model which corresponds to the wake profile in Figure 4.12, is shown in Figure 4.13.

Figure 4.13 shows that the pressure distribution exhibits fluctuations with the highest pressure peak existing at x/c=0.49 when t/T=0.286, this corresponds to the wake profile in Figure 4.12b. The LEV is detected at t/T = 1 when the foil is at its maximum position.
Chapter 4 Fluid Structure Interaction of a Foil with Steady Incoming Flow

Figure 4.13: Pressure distribution in the case of high velocity response, corresponds to Figure 4.12

The fluid flow regime in this at this time similar to stall phenomenon. The vortex is shed due to the high pressure gradient in the direction of the flow which is indicated by the high amplitude pressure peak. Thus the fluid induced vibration is likely to experience lock-in phenomenon as depicted by the high displacement shown in Figure 4.12b. The LEV flows towards the trailing edge as indicated by the pressure peaks relocating to trailing edge. For the later stages of the vibrating revolution, the pressure fluctuation is found to be very low amplitude which means a smaller vortex occurs in the fluid flow regime. A strong vibration is also recorded from a point displacement as shown in Figure 4.14.

Figure 4.14 shows the time history of displacement at the point $x/c = 0.49$ where the high pressure occurs when $t/T = 0.286$ shown in Figure 4.11. The vibration is dominated by the y direction motion and small displacement occurs in x. The maximum displacement at the steady state is approximately 0.2 m and considered high when compared to the standard operation model which is only 0.014 m (Figure 4.7). The angular motion cannot be detected in this plot however it can be captured from the vibration image in Figure 4.9 and the moment coefficient as depicted in Figure 4.16. A moment coefficient expresses a pitching moment produced by production of forces time distances in respect to a quarter chord length. Therefore a moment coefficient can be an indicator of a torsional vibration or of angular displacement.

The vortex shedding and forces from the case of high velocity and the standard model will be compared to the low velocity model. The low velocity case at 2 m/s will be discussed next. The original, reference and phase averaged signals are shown in Figure 4.15.
Figure 4.14: Displacement vs time step at x/c = 0.49 for original foil response in the high velocity model (6 m/s). pointDisplacement (magnitude) corresponds to magnitude displacement, pointDisplacement (0) equals to x direction displacement, pointDisplacement (1) equals to y direction

Figure 4.15a shows that all velocity signals are found to be very regular with very low amplitude. The frequency domain plot is also quite regular as shown in Figure 4.15b. The dominant frequency is at 1.85 Hz. None of the frequencies coincide with either the heave or pitch foil natural frequency. Thus it is likely that the lock-in condition will not happen and resonance will not occur in the model. The foil experiences no vibration (foil displacement is zero) during operation and no vortices are found. The force coefficients for 2 m/s, 4 m/s and 6 m/s are shown in Figure 4.16 -4.18.

The lift coefficient for the velocity variation case is shown in Figure 4.16. From that figure it can be seen that the lift coefficient at 6 m/s has the highest amplitude with the average amplitude approximately the same as the velocity in the standard model (4 m/s). The lift coefficient for 2 m/s model has a higher average value than the other two models. This model also has a small amplitude thus the lift coefficient less fluctuates and is almost constant. Although the mean lift force is larger, it is constant hence vibration does not occurs and a vortex is not generated. The drag and moment coefficient are plotted in Figure 4.17 and 4.18 respectively.

The drag coefficient graph for the velocity variation case is plotted in Figure 4.17. Similar to the lift coefficient for the 6 m/s model, the drag coefficient has the highest value. However the drag force frequency at 6 m/s is smaller than that at 4 m/s whereas in the 2 m/s model the drag coefficient is constant. In contrast to the lift coefficient, the drag coefficient average for the 2 m/s model is the lowest. It is also found that the drag
Figure 4.15: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the fluid velocity of 2 m/s model

coefficient frequency magnitude is twice the lift coefficient frequency. This only happens in the high velocity case since in this case a stronger vortex is shed which appears to affect the lift and the drag coefficient. At the low velocity model the drag and lift frequency cannot be detected. This also happens for the moment coefficient shown in Figure 4.18.

The moment coefficient for the velocity variation case is shown in Figure 4.18. In contrast to the lift and drag coefficient, the average moment coefficient for all the three velocity variations coincide with almost at the same value. However the high velocity model still exhibits highest magnitude. There is also evidence for the appearance of resonance in the pitch motion as plotted in Figure 4.11b. Another condition which also affects the response behavior is the initial angle of attack variation which will be discussed in the next section.
4.1.3 Initial Angle of Attack Variation

The initial angle of attack is set in this model in order to identify how its change will affect the behavior of the foil response and vortex shedding. This parameter is varied according to Table 2.4. The reference, original and phase averaged velocity at an angle of $8^\circ$ are shown in Figure 4.19a and the vortex shedding frequency from the PSD method is shown in Figure 4.19b.

Figure 4.19a shows that the original frequency is quite chaotic which results from the chaos of the vortex generation. From Figure 4.19b, it is clearly seen that there are three
main frequencies existing in the case of 8° initial angle of attack model, these are 0.468 Hz, 1.094 Hz, and 2.188 Hz. The frequency of 1.094 Hz is close to the heave natural frequency which likely causes a resonance on the foil. The resonance intensifies the foil vibration as shown in Figure 4.20.

Figure 4.20 shows one oscillation of a strong foil vibration time history for the case of 8° initial angle of attack. The fluid flow regime in this model is very chaotic (Figure 4.19a) with a strong vortex generation (Figure 4.20a-e). At the lowest position of vibration shown in Figure 4.20a, the vortex detaches from the trailing edge on the suction surface. The detachment can also be detected from the pressure distribution plot in Figure 4.21. At $t/T = 0.04$, the pressure signal is flat which demonstrates a small and uniform pressure over the surface. This constant pressure indicates that fluid flow regime is laminar with no separation or vortex generation occurring. The pressure distribution line when $t/T = 0.04$ also shows that the pressure decreases on the suction surface close to the trailing edge. At the location where high gradient pressure exists, the bottom-most layer is rolled by the above layers to form vorticity as seen in Figure 4.20a. From this point, the foil moves to the position in Figure 4.20b.

The foil starts to move upward to a position at $t/T = 0.22$ as shown in Figure 4.20b. No vortices are shed from either upper or lower surface as demonstrated from the pressure distribution line at $t/T = 0.22$ in Figure 4.21. The contour lines are flat but show high pressure at the leading edge which includes stagnation pressure. From this position onward, the foil moves upwards until reaches a maximum point as shown in Figure 4.20c. Figure 4.20c displays that the leading edge vortex beginning to form which is also indicated in the pressure distribution contour plot. When $t/T = 0.35$, the pressure peak is found at the midpoint position $x/c=0.297$. This peak corresponds to vorticity generation as shown in Figure 4.20c. The high pressure gradient at that point induces reverse fluid velocity which cause vortex formation. The vortex formation causes strong
vibrations (resonance) which is proven from the foil displacement during operation as shown in Figure 4.22. From the uppermost position the foil moves downward, sheds the vorticity downstream as shown in Figure 4.20d and reaches its original position (lowest position) as shown in Figure 4.20e. The pressure distribution at the last two positions are flat. This indicates that the pressure is constant and the flow remains attached at the front of the foil. When $t/T = 1$ separation is found at the back of foil which is also known the decreasing pressure in Figure 4.21. The high amplitude vibration can also be configured from the displacement at $x/c = 0.297$ as illustrated in Figure 4.22.

The displacement plot in Figure 4.22 displays a resonance behavior in the response at 8° initial angle of attack. The high amplitude vibration is shown. The maximum amplitude reaches 0.2 m at time step of 63. This is a response from the fluctuating pressure as described previously. It is also evidence for the lock-in condition in heave motion as
the vortex shedding frequency coincides with natural heave frequency shown in Figure 4.19b.

The fluctuating pressure distribution affects the forces which will be discussed later along with the effect of the 4° initial angle of attack in Figure 4.25, 4.26, and 4.27. The other initial angle of attack to be discussed is at 4°. At 4° initial angle of attack, the reference, original and phased averaged velocity are plotted in Figure 4.23a and the vortex shedding frequency from the PSD method is shown in Figure 4.23b.

In Figure 4.23a the original and reference velocities are shown to fluctuate irregularly with smaller magnitude and amplitude than in the case of 8° initial angle of attack. This means the vibration response when the initial angle is 4° is weaker than in the 8° model as shown in the amplitude in the frequency domain plot Figure 4.23b. However it is also a concern that the one of the main frequencies approach the heave natural frequency. There are three main frequencies as shown in Figure 4.23b, these have magnitudes of
Chapter 4 Fluid Structure Interaction of a Foil with Steady Incoming Flow

Figure 4.21: Pressure distribution in the case for 8° initial angle of attack, at $t/T$: a. 0.04, b. 0.22, c. 0.35, d. 0.57, e. 1

Figure 4.22: Displacement in y direction vs time step at $x/c = 0.297$ for 8° initial angle of attack

1.015 Hz, 2.65 Hz, and 3.672 Hz respectively. The 1.015 Hz frequency coincides with the heave natural frequency. This situation is likely to amplify the response and cause the foil to vibrate strongly, however the magnitude of vibration is not as high as the 8° angle of attack. The strong vibration can be recognized through the displacement of point $x/c=0.297$ as shown in Figure 4.24. The other two frequency peaks lie far from each of the natural frequencies and are not expected to damage the system.

In the initial angle of attack variation cases, all model results have three frequency peaks and the maximum peak in each model is written in Table 4.1. The maximum dominant frequency decreases with the increasing of initial angle of attack. This result agrees with
Figure 4.23: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the 4° initial angle of attack model

Jung and Park’s prediction (2005). For the model with 4° and 8° angle of attack, one of the main frequencies approaches the heave natural frequency with the higher power present at the 8° angle. This shows that raising the initial angle of attack has a potential risk to synchronize the vortex frequency with the natural frequency and thus produce high amplitude vibrations as listed in Table 4.1. The time history displacement for 4° angle of attack is shown in Figure 4.24.

Table 4.1: Dominant frequency and vibration amplitude in initial angle of attack variation

<table>
<thead>
<tr>
<th>Initial angle of attack</th>
<th>8°</th>
<th>4°</th>
<th>0°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. frequency (Hz)</td>
<td>2.188</td>
<td>3.672</td>
<td>4.76</td>
</tr>
<tr>
<td>Max. Displacement (m)</td>
<td>0.2</td>
<td>0.09</td>
<td>0.014</td>
</tr>
</tbody>
</table>
Figure 4.24: Displacement in y direction vs time step at x/c = 0.297 for 4° initial angle of attack

Figure 4.24 shows that the displacement of the 4° initial angle of attack model has maximum peak to peak amplitude of approximately 0.09 m occurring at time step 88. The displacement is more irregular than in the case of the high initial angle of attack which indicates the vibration in the 4° angle of attack model has a higher frequency. However the 4° angle of attack model has less power and less displacement (smaller vibration amplitude) than the case of 8° angle of attack as discussed previously. The fluid induced vibration is strongly influenced by the force coefficients which is illustrated by Figures 4.25, 4.26, 4.27.

Figure 4.25: Lift coefficient in the case of initial angle of attack

Figure 4.25 shows time history for lift coefficient in the initial angle of attack variation models. The highest lift coefficient magnitude, the highest average magnitude, and the
largest amplitude occurs in the $8^\circ$ initial angle of attack model. This supports the result of highest amplitude vibration (displacement) existing in the $8^\circ$ angle of attack simulation. The drag and moment coefficients are shown in Figure 4.26 and 4.27 respectively.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{drag_coefficient.png}
\caption{Drag coefficient in the case of initial angle of attack.}
\end{figure}

Figure 4.26 shows the drag coefficient results for the initial angle of attack variation models. The drag coefficient fluctuations are found in the results for all models. Similarly to the lift coefficient plot, the $8^\circ$ angle of attack model drag coefficient has the highest amplitude, highest magnitude and highest average value. From the number of peaks passing fixed point, one can tell that the drag coefficient frequency is decreasing with increasing angle of attack. This result is verified by Chandravanshi et al. (2010) and it is shown that the higher amplitude of vibration which occurs in the $8^\circ$ angle of attack model induces drag coefficient frequency decrease. The high amplitude vibration which is produced by high amplitude fluid forces should be avoided because it can expose the structure to higher risk of dynamic failure. Thus the conditions which can cause high forces during operation should be avoided, these include higher angles of attack. For moment force analysis, it refers to time domain moment coefficient plot in Figure 4.27.

In a similarly manner to lift and drag coefficient, the moment also fluctuates with the highest amplitude, magnitude and average value in $8^\circ$ angle of attack model. The magnitude and frequency of the moment force is less than the drag and lift forces which support the result of no existing resonance. Therefore there is synchronization in frequency in pitch direction for all the models. Another important criterion for a foil response is the number of degrees of freedom for which the foil vibrates. This variation model allows a foil to move in three different ways with one, two or three degree of motion when in contact with the fluid as described below.
Chapter 4 Fluid Structure Interaction of a Foil with Steady Incoming Flow

4.1.4 Number of Response Motions

The number of response motions denotes the number of degree of freedom involved in the motion of the foil. Number of response motions is investigated for four different models. For one degree of freedom, only pitch motion is permitted since this motion is the main mode of the motion (rotation) during the vertical axis turbine operation. Two degrees of freedom is analyzed for two cases of pitch heave (standard model) and pitch surge, neglecting the heave surge motion. The last type of motion combination is three response motions in 2D analysis, this is pitch, heave and surge and will be discussed separately in this section.

The phase averaged, reference and original velocity signals for the case of pitch heave and surge model and the PSD method of dominant frequency analysis are depicted in Figure 4.28.

Figure 4.28a shows that the reference and original velocity signals oscillates regularly. The vibration frequency is detected from the reference velocity signal as well as from the original signal. Consequently there is only one distinct peak in the transformed signal which is associated to the dominant vortex shedding frequency. The dominant frequency of the three degree of response is at 0.782 Hz which is close to the heave natural frequency. This condition is similar to the simulation with 6 m/s incoming velocity, 8° and 4° initial angles of attack. In contrast with these models, aside from having one distinct peak, the power of the dominant frequency in the three degree response model is not high thus the vibration response is weaker. The weak vibration is detected from the maximum displacement which is only found to 0.0003 m for this case. The weak vibration of the three degree of freedom model can also be viewed in the force coefficient plots which are illustrated in Figure 4.33, 4.34, and 4.35. These will be discussed later. The fluid force acting on the foil incurs a foil vibration when the foil interacts with the
Figure 4.28: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch heave surge model

A stronger fluid-induced vibration on a foil indicates higher forces operating on the foil. The other motion responses will be analyzed in the following section.

The two degree of motion response model is developed for two cases, these are pitch surge and pitch heave models. The pitch heave model is the standard model which has been presented in Section 4.1.1. For the pitch surge model, the phase averaged, original and reference velocity signals are given in Figure 4.27a and the frequency domain results of vortex shedding from the PSD method is shown in Figure 4.29b.

From Figure 4.29a, it is shown that all velocity signals in the pitch surge model decay and at 15 seconds the reference velocity is almost static with a very small velocity amplitude existing. This also indicates that the foil vibrations are damped and the foil becomes almost static in fifteen seconds (the displacement is found zero). Figure 4.29b shows the
frequency domain obtained from the PSD method. The dominant frequency is found to be 2.5 Hz which is far from either the heave or the pitch natural frequency. The decaying vibration is also recordable in the frequency domain by the decreasing amplitude trend as the vortex shedding frequency increase. The fact that the dominant frequency is far from the structural natural frequency shows that a resonance effect does not exist in the system during operation.

The next response investigated is pitch motion. Three velocities of pitch motion and the vortex shedding frequency from the PSD method are shown in Figure 4.30a and 4.30b respectively. Figure 4.30a shows irregular oscillations in the reference and original signal. The vortex shedding velocity signal is obtained by subtracting the reference from original velocity using the Phase Averaged method. The vortex shedding frequency is transformed into vortex shedding frequency domain result using the PSD method as

Figure 4.29: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch surge model
shown in Figure 4.30b. Similar to the three degree of freedom model, the dominant frequency is found at 0.55 Hz (Figure 4.30b). Although the dominant frequency is not close to either the heave or the pitch natural frequencies, the magnitude of the dominant frequency is quite high with respect to other two and three degrees of response. The high power contained in a fluid provide strong forces acting on a foil and it can be detected in the force coefficients as shown in Figure 4.33, 4.34, and 4.35. The strong force from the fluid exerts a strong vibration on the foil structure as shown in the vibration sequence and pressure distribution plots in Figure 4.31 and 4.32 respectively.

Figure 4.30: a. Phase averaged velocity signal, b. frequency domain of phase averaged signal from the pitch model

Figure 4.31 shows the fluid flow regime in a revolving motion. This has correlation to the pressure distribution plot shown in Figure 4.32. From Figure 4.31, it is clearly shown that the vortex is shed during a cycle. The cycle starts with the foil in the maximum position, shown in Figure 4.31a. Vortex shedding starts to form at the middle of the foil
on the suction surface. This vortex generation is induced by a high pressure peak in the direction of the fluid flow when $t/T = 0.03$. This is at approximately the same location of the vortex generation. After the location of vortex generation in the streamwise on the suction surface, the pressure increases and reaches higher than the lower surface pressure. At the trailing edge region the suction surface pressure decreases and reaches a suction condition again as shown by a peak at $x/c = 0.94$ as shown in Figure 4.32. The trailing edge region peak indicates the vortex is shed from the trailing edge region. The middle high pressure induces forces to a foil and move downward to minimum position of the oscillation as shown in Figure 4.31b.

Because of the pressure distribution at the midpoint on the suction surface, the foil moves downwards with a vortex continuing to shed and flow downstream as shown in Figure 4.31b. In this condition, a vortex is shed from both surfaces which can be ascertained from the pressure distribution in Figure 4.32. On the lower surface the pressure distribution profile is not flat and small peaks are found at $x/c = 0.394$ and $0.756$ where separation is detected. At the leading edge on the suction surface two high peaks exist at $x/c$ equal to $0.104$ and $0.266$. These induces small vortices shown in Figure 4.29b. The high pressure gradient in that location contributes to fluid flow stagnation near the surface. The fluid in the upper layer which has higher velocity rolls the fluid particle near the surface of the foil and produces reverse velocity fluid flow. The fluid flows back against the flow at the layer near the surface which creates a vortex in the fluid flow regime. The vortex shedding causes vortex induced vibration in the foil which
can be observed in the force coefficient plot in Figures 4.33 - 4.35. From that location, the foil moves up to its maximum position as shown in Figure 4.31c.

Figure 4.31c shows that when the foil is at $t/T =1$, it moves to the upper-most position again, the same as the original position shown in Figure 4.31a. At this location the pressure distribution and vortex shedding condition are also similar to the beginning of the cycle. Vortex shedding is found on the suction surface only and a constant pressure distribution is observed on all the surfaces except at the midpoint and at trailing edge as shown in Figure 4.32.

In the pitch response model as shown in Figure 4.31, the response is dominated by torsional vibration with some small heave vibrations occur. The heave maximum displacement of 0.001 m is detected during the operation.

In Figure 4.33, the time history lift coefficient for a number of motion responses variation is presented. The time history for four models is shown for fifteen seconds simulation time. Drag and moment coefficients are depicted in Figure 4.34 and 4.35 respectively.

From Figure 4.33 it is shown that the lift coefficient of pitch response is quite strong compared to the other three modes. This is likely due to the high power content of the vortex shedding signal as depicted in Figure 4.30b. The pitch heave surge lift coefficient is oscillated regularly with the lowest frequency propagation. This indicates that the vibration in heave direction of the pitch heave surge response has regular oscillations and the lowest frequency. In contrast to the pitch heave surge response, the second degree of freedom responses have higher frequency of oscillation. However the pitch surge motion response has a smaller lift coefficient than pitch heave response. The pitch surge motion lift coefficient also decays gradually and is almost constant at 15 second. The pitch response foil model yields a chaotic lift coefficient which is a result of a vigorous vibration during its operation. This result proves that pitch is the dominant motion
response. The pitch response model is sensitive to excitation during the vertical axis tidal turbine operation and produces a strong vibration even without any resonance condition or being in synchronization with the natural frequency. The sole effect of pitch motion can reduced by a heave or surge motion. However surge motion is found to be better for damping the pitch motion effect than heave. Surge motion reduces the amplitude and vibration frequency. Interestingly the damping effect is not further reduced when the three motion response is applied at the same time on the foil as shown in pitch heave surge result in Figure 4.33. At the three motion response condition, the lift coefficient amplitude is higher than that of the two degree of freedom response model. However the lift coefficient frequency is reduced in the three motion response condition. The drag and moment coefficients are discussed with reference to Figure 4.34 and 4.35.
Figure 4.34 shows the drag coefficient for a number of motion response combination. Similar to the lift coefficient, the drag coefficient for pitch motion response is chaotic and displays the highest amplitude appearance. The chaotic profile does not exist in the other motion response variation. The other three motion responses show a regularly oscillating drag coefficient. The pitch surge response foil, drag coefficient decays and reaches a constant magnitude at 0.05 after fifteen seconds simulation time. The pitch heave foil has the highest amplitude and frequency compared to the other three foil motion. In contrast to the lift coefficient, the combined three motions response together reduces the drag coefficient by decreasing significantly the amplitude and lowering the frequency. The last coefficient to be described is the moment coefficient as shown in Figure 4.35.

![Graph showing moment coefficient](image)

Figure 4.35: Moment coefficient in number of response mode variation

As predicted from the lift and drag coefficients, the pitch response moment coefficient is chaotic while the other three response modes exhibit regular oscillations. The pitch surge response moment coefficient also decays. The pitch heave surge motion response has the lowest frequency although amplitude is similar magnitude to the pitch heave model. From the analysis of the three of force coefficients, it can be concluded that the pitch heave surge response model provides an advantage for lift force by yielding higher amplitude forces with lower frequency. The same effect is obtained from the pitch heave response model for drag and moment coefficients. Generally overall forces can be reduced by applying a pitch surge motion on a foil. The next foil mechanical property which will be described is related to the elasticity of the foil which is discussed in the next section.
4.1.5 Stiffness Constant Variation

The structural stiffness coefficient is a parameter which expresses the structural elasticity or how much a structure is distorted when a force is applied to it. This parameter influences the vibration or the turbine response when it interacts with a fluid. In this analysis only the heave frequency is altered because in the standard model the heave stiffness coefficient is greater than the pitch coefficient so that the heave stiffness is more crucial. The governed equation for the heave stiffness coefficient is from Young Modulus theory as shown in Equation (4.1).

\[ k = \frac{EL}{A} \]

(4.1)

In this work the stiffness constant variation is adopted from Wang et al. (2013a) who studied the effect of Young modulus variation in the application of wind turbine. Wang developed a numerical model to investigate the effect of glass fibre reinforced polymer composite stiffness on its Young Modulus for a wind energy device application. His model showed that Young Modulus was strongly influenced by the fibre orientation in material manufacturing process. When the stiffness constant of a material is altered, the natural frequency of the turbine will also change. The various stiffness frequencies simulated in the model is presented in Table 2.4. The structures natural frequency is derived from the material stiffness as discussed in Section 4.4.4. Altering the material stiffness constant changes the structure’s natural frequency therefore the various heave stiffness constant in this case will alter the foil heave and pitch natural frequency. This analysis is further described in Subsection 6.5.2. The modified natural frequency corresponding to each stiffness constant is listed on Table 4.2.

A higher heave stiffness constant provides a larger foil natural frequency and increases the natural frequency value in the frequency domain plot. For a stiffer material, the foil displacement will be less for a given load application. According to Equation (2.25), for a given time period high stiffness foil will vibrates with higher frequency than a low stiffness one. In the high stiffness model, forces coefficients contain of a high frequency component as shown in Figure 4.40, 4.41, and 4.42. For a high stiffness coefficient (1500 N/m), three velocity signals and the dominant frequency from the PSD method are presented in Figure 4.36a and 4.36b respectively.

<table>
<thead>
<tr>
<th>Heave stiffness constant (N/m)</th>
<th>Heave natural frequency (Hz)</th>
<th>Pitch natural frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1500</td>
<td>1.0364</td>
<td>1.6436</td>
</tr>
<tr>
<td>1000</td>
<td>0.9425</td>
<td>1.4758</td>
</tr>
<tr>
<td>800</td>
<td>0.8767</td>
<td>1.4191</td>
</tr>
</tbody>
</table>

Table 4.2: Natural frequency of each model in stiffness variation
Figure 4.36a shows the reference, original and phase averaged velocity signal for the high stiffness model. All velocities display regular oscillation. The original velocity also has a high amplitude signal. The vortex shedding power of the dominant frequency is obtained by subtracting of the original signal from the phase averaged signal as shown in Figure 4.36b. There are two main frequencies at 2.578 Hz and 1.72 Hz which coincide with the pitch natural frequency. This situation is potentially harmful to the structure due to the existence of the resonance phenomenon. In the resonance phenomenon, the vortex shedding is enhanced which strengthen the vortex induce vibration. The strong vibration can danger the structure. The amplifying effect can be distinguished from the high frequency component in the fluid force coefficient plots presented in Figure 4.40, 4.41, and 4.42. The resonance can also be seen from the vortex shedding behavior in the fluid flow regime illustrated in Figure 4.37. Although the strong vibration occurs in the pitch direction, there is no vibration detected in the heave direction. The heave displacement of this model is found to be zero.

Figure 4.37 shows one revolution for the high stiffness foil response model. The motion starts at the equilibrium position shown in Figure 4.37a. The wake begins to form at the leading edge, flows downstream and finally detaches from the trailing edge of the foil at the upper surface. The vorticity observed during the separation process creates a higher fluid induced vibration on the foil and causes the foil to move upward to the uppermost position depicted in Figure 4.37b. Because the frequency of the vortex shedding is close to the pitch natural frequency, the pitch response is dominant and responsible for increasing angle of attack when the foil vibrates. As the foil moves upward in Figure 4.37b the angle of attack is also raised and produces a stronger wake at the leading edge. From this position the foil moves downward but the angle of attack remains constant, producing more wake and causing separation at the front of the foil which flows downstream as shown in Figure 4.37c. At this stage, the separation and vortex shedding is strong and causes a decrease in the lift force and forces the foil back to equilibrium position. The angle of attack is reduced as shown in Figure 4.37d. The wake and vorticity is also reduced similar to the condition presented in Figure 4.37a.

The fluid flow regime shown in Figure 4.37 is closely related to the pressure distribution over the foil surface as depicted in Figure 4.38. At the beginning and at the end of the vibration cycle, when $t/T = 0.04$ and $1$, the foil is at equilibrium position. In this position the flat pressure distributions are identical with a small peak found at quarter chord length from front. The shed vortex detaches from the trailing edge and the fluid flow regime on the foil surface at this point is regular as shown in Figure 4.37a and 4.37d. At $t/T = 0.4$ the pressure distribution fluctuates more with the highest pressure found at $x/c = 0.149$ and $x/c = 0.23$. At these locations the fluid flow direction reverse and an initial LEV is generated. When $t/T = 0.6$ the vortex grows which is represented by the large sloping peak in the pressure distribution contour plot. The vortex detaches from trailing edge at $t/T = 1$. The high stiffness and the moderate stiffness models in
Figure 4.36: a. Reference, original and phase averaged velocity signal for higher stiffness, b. frequency domain for higher stiffness

the standard model will be compared to the low stiffness model. This will be explained in the next section.

Figure 93a and 93b depict three velocity signals and the dominant frequency for the low stiffness material turbine respectively.

From Figure 4.39a, it is found that the reference and original velocity in the low stiffness model have smaller amplitude and oscillates less frequently than the high stiffness model. The frequency domain signal is shown in Figure 4.39b. The dominant frequency occurs at 0.312 Hz which is lower than the standard and high stiffness material models and is also far from either the material heave or pitch natural frequency. The weak vibration is found with a maximum displacement of 0.009 m in this model. This condition will
yield a safer foil performance with resonant vibration unlikely to exist. The vibration is also related to the fluid forces as shown in Figures 4.40 - 4.42. The lift coefficient is plotted in Figure 4.40. It can be seen that lift force for all of the three material models have identical mean value. The high stiffness material has the highest amplitude and frequency. In the low stiffness model, the lift coefficient is
Figure 4.39: Reference, original and phase averaged velocity signal for lower stiffness material, b. frequency domain for lower stiffness.

almost constant with small fluctuations appear in the contour plot. The drag coefficient is presented in Figure 4.41.

The drag coefficient for the high stiffness model has the highest amplitude and frequency. The high frequency component also appears and the mean value of the three models are similar. The drag coefficient for the low stiffness model has constant value at a steady state.

The moment coefficient plot is depicted in Figure 4.42. Similarly to the lift and drag coefficient, the moment coefficient for the high stiffness material model exhibit a high frequency component. The mean value of the low stiffness model is lower than the other stiffness models with the value remaining almost constant. The mean value for the standard and the high stiffness model are similar. The other material property
which affects a foil response is the damping constant which will be explained in the next section.

4.1.6 Damping Constant

Similar to the material stiffness, the damping is a structural property of a material and strongly affects a tidal turbine response. Damping is a property that acts to resist material vibrations. Higher material damping reduces a turbine blade vibration. In this analysis, the effect of the damping constant variation on the turbine blade response is investigated with only two condition applied. The conditions are underdamped state and over damped state. The damping constants are listed in Table 2.4. The standard model describes the case of an under damped material model and was detailed in Section
7.5. In this section the discussion is focused on the overdamped material model. The velocity signal for the overdamped foil model is given in Figure 4.43a whereas the vortex shedding frequency is given in Figure 4.43b.

Figure 4.43a shows that the amplitude and frequency velocity for the overdamped condition is lower than that of the underdamped condition (Figure 4.5a). The original velocity shows a damping effect which reduces the signal amplitude at each cycle. The dominant frequency (Figure 4.43b) is found to be 0.935 Hz coinciding with the heave natural frequency. This condition should be highlighted because resonance effect can be involved and a strong vibration is created. The heave displacement which has maximum value of 0.013 m is found to be constant and is not damped during the simulation. This is due to the heave motion response on the foil is less dominated than the pitch motion thus the heave vibration is not being decayed. The vortex shedding can also be analyzed by the pressure distribution which is plotted in Figure 4.44.

From Figure 4.44, at t/T = 0.06, a small leading edge vortex is initiated as demonstrated by the dynamic overshoot at x/c = 0.243. Following this the vortex grows and flows downstream as shown at t/T = 0.28 and continues until at t/T = 1. The vortex flow can be detected from the pressure magnitude decrease at each selected times on the upper surface. From the pressure distribution profile, it can be recognized that a weak vortex is shed from the leading edge and detaches from the back of the foil. Hence the fluid induced vibration experienced by the foil is also weak. This condition can also be distinguished from the force acting on the foil in Figures 4.45, 4.46, and 4.47.

Figure 4.45 shows the lift coefficient plot for the underdamped (standard model) and overdamped models. The overdamped model has a lower lift coefficient amplitude and frequency. The lift coefficient for overdamped model decays whereas the underdamped model oscillates regularly during the operation. The drag and moment coefficients which
4.2 Modified Trailing Edge Foil Response

Modified trailing edge foils including blunt, sharp and rounded foil are developed and simulated using the standard model properties listed in Table 2.3. The results are compared to the standard model to identify the influence of modifying the trailing edge.
Chapter 4 Fluid Structure Interaction of a Foil with Steady Incoming Flow

Figure 4.44: Pressure distribution for high stiffness response, at t/T: 0.04, 0.4, 0.6, 1

Figure 4.45: Lift coefficient for damping variation

shape to a foil response. The parameters which have been detailed when investigating the original foil response are also applied. The reference velocity signal is sampled at the leading edge to take a purely periodic vibration. This signal is then averaged in phase using the Hilbert transform and subtracted to the original signal. The original signal is sampled at 0.1c behind the trailing edge. The sampled points for the original and the reference signal are at similar locations as shown in Figure 3.25. The phase averaged and the PSD methods are used to obtain vortex induced frequency. Force coefficients are also extracted using ParaView to determine the fluid forces.
4.2.1 Blunt Foil

The blunt foil is constructed by truncating the trailing edge of an original foil by 15% (Ramjee et al., 1986). The reference, original, and phase averaged velocity signals for a blunt foil are shown in Figure 4.48a and the vortex shedding frequency plot is presented in Figure 4.48b.

The reference velocity is almost constant which represents the small displacement experienced by the foil. The displacement is shown in Figure 4.49. The original velocity signal measured in the near wake of the foil is very chaotic. It is assumed that the vortex shedding fluctuates irregularly and the signal comprises of many fundamental frequencies as shown in Figure 4.49b. Figure 4.49b shows that there are no distinct dominant
frequency in the fluid flow regime for the blunt foil response. It indicates that the vortex shedding is more chaotic and disordered than the case of original trailing edge foil. Although the vortex is irregularly shed, it does not generate a strong vibration response on the foil itself. This is because the vortex shedding frequency does not coincide with the natural frequencies and also due to the small power contained in the high frequency vortex shedding signal. The foil experiences weak vibrations as can be deduced from the displacement at quarter length of the chord in y direction in Figure 4.49.

During the first 70 time steps, the simulation is in a transient condition and the displacement is found to oscillate with high amplitude. The vibration transitions to steady state with small amplitude displacement. The small displacement reflects the small vibration experienced by the foil. The vortex is shed on the truncated surface at the back of the foil in a similar manner to the case of the static foil (Figure 3.23). Two source of
vorticity are found swirling on the truncated surface. The vortices is shed alternatively generating Von Karman vortex street as seen in the vortex shedding history in Figure 4.50.

Figure 4.49: Displacement in y direction on the blunt foil response model

Figure 4.50 shows the history of alternating vortex generation at the back of a blunt foil surface. At the back of the blunt foil, a vortex core occurs at the bottom part of the truncated surface as seen in Figure 4.50a. The vorticity increase (Figure 4.50b) and detaches from the surface toward the near wake (Figure 4.50c). At the same time at the top of the truncated surface, a vortex core begins to form (Figure 4.50c) and grows larger (Figure 4.50d and 4.50e). Finally it is released into the stream flow along with the generation of an additional vortex at the bottom core (Figure 4.50d and 4.50e, 4.50f). The process repeats and the top and bottom vortices are generated alternatively.

Figure 4.51 shows the pressure distribution on a blunt foil at $t/T = 0.08$ and 0.28 which corresponds to the fluid flow regime in Figure 4.50b and 4.50d respectively. Both pressure distributions are almost identical with a small discrepancy at $x/c = 1$. It can be concluded that in the blunt foil response model the pressure distributions are same at all times. The small difference occurs on the truncated surface due to alternating vorticity described earlier as shown in Figure 4.50. The pressure similarity indicates the same forces acting at all times thus very small vibration exists during the simulation. The forces which are shown in Figure 4.58 - 4.60 will be discussed later. These results will be confirmed by other trailing edge shape foil responses which will be analyzed in the next section.
4.2.2 Rounded Foil

The rounded foil model is developed with the same simulation properties as normal and blunt foil. The foil is truncated and rounded at a distance of 0.15c from the trailing edge of the original foil. All the signals observed for the vortex shedding frequency using the Phase Averaged method and the PSD are depicted in Figures 4.52a and 4.52b respectively.

Figure 4.52b shows that the rounded foil vibrates at 1.328 Hz which is quite close to the pitch natural frequency. The power at this vibration frequency is also quite high, reaches $2.828 \times 10^4$ W. The high power response combined with the proximity of the dominant frequency to natural frequency, generates a strong vibration on the foil as shown in the vibration revolution in Figure 4.53. The vibration at the dominant frequency and pitch natural frequency are in synchronicity. This creates a high power intense vibration.
The pitch natural frequency enhances the foil pitch response causing angle of attack to increase significantly. This induces separation on the upper surface of the foil. The separation and strong vortex shedding are observed in the fluid flow regime and this forces the foil to vibrate violently. The separation can be clearly seen at all stages of the foil vibration in Figure 4.53 while the vortex is generated when foil returns from its maximum position (Figure 4.53b) to its equilibrium position (Figure 4.53c). This is caused by pressure gradient over the surface as shown in Figure 4.54.

Figure 4.54 shows the pressure distribution over the foil surface in relation to the fluid flow regimes shown in Figure 4.53. At the beginning and the end of the cycle oscillation at $t/T = 0.04$ and $1$, the pressure gradient is almost zero which is indicated by the flat line with no pressure peak. Separation with slight vortex shedding from trailing edge is found in the fluid flow regime over the foil surface as illustrated in Figure 4.53a and 4.53e. At the maximum position ($t/T=0.022$), a high pressure is observed over the foil surface with the maximum pressure coefficient reaching $-5$. LEV is created at this point as seen in Figure 4.53b with no separation found in the overall fluid flow regime. From this position, the foil moves back to its equilibrium position as shown in Figure 4.53c.

At this stage when $t/T= 0.035$, the pressure distribution contour is found to fluctuates and a large pressure peak is identified at $x/c = 0.389$. The LEV grows and flows downward as shown in Figure 4.52c. The LEV increases in size and flows toward the trailing edge as evidenced by the receding peak at $x/c = 0.609$ when $t/T =0.57$. The pressure peak which indicates a high pressure gradient, creates reverse flow which causes the vortex shedding. The location of the peak indicates where the vortex shedding starts. The shifted peak demonstrates the flowing vortex shedding from the leading edge to the
4.2.3 Sharp Foil

Similarly to the rounded foil, the sharp foil is cut off at 0.15c from trailing edge and is inclined at 45° from the horizontal on the upper and lower surfaces. The signals for the phase averaged method, original and reference velocities are shown in Figure 4.55a. It can be seen from this figure that the velocity amplitude is high although the fluctuation is regular. The vortex shedding frequency which is determined from the PSD method for the original signal subtracted from the phase averaged signal, exists at 1.406 Hz, 2.89 Hz, 5.7 Hz, and 8.594 Hz as illustrated in Figure 4.55b. The first main vortex
shaking frequency coincides with the pitch natural frequency and is synchronized with
the vibration response. The vibration is driven by fluid forces acting on the foil as shown
in Figures 4.58 - 4.60. The vibration can also be identified from the images of the fluid
flow regime shown in Figure 4.56.

Figure 4.56 shows an image of fluid flow regime history on a sharp foil for one vibration
cycle at three selected times. The flow regimes correspond to the pressure distributions
which are depicted in Figure 4.57. A sharp foil is shown at its minimum position at
t/T = 0.06 in Figure 4.52a. At that the time foil does not experience pitch motion (the
foil is in the horizontal position), however strong vortex shedding starts to form at the
leading edge (LEV) which is caused by the resonance phenomenon. The LEV is also
demonstrated by the peak pressure gradient at the associated time in Figure 4.57. This
vortex induces vibration on the foil and causes the foil to move to the maximum point
as illustrated in Figure 4.56b.
At maximum point the LEV flows downward to the trailing edge and is released into the freestream flow as shown in Figure 4.56b. The detached vortex can also be detected from the shifted pressure peaks toward trailing edge shown in Figure 4.57 at the corresponding time. At this position the angle of attack is high, creating a chaotic fluid flow regime which reduces the lift force (associated with to dynamic stall). A lower lift force brings the foil back to its minimum position as depicted in Figure 4.56c. In this state, the fluid flow regime is identical to Figure 4.56a and both pressure distribution profiles are also similar.

The force coefficients for four different trailing edge shapes are detailed in Figures 4.58 - 4.60. The lift forces for the original and the three modified trailing edge are shown in Figure 4.58.

From Figure 4.58, the blunt foil has the smallest amplitude and the sharp and rounded foils have the highest amplitude. The blunt foil also has the highest mean lift coefficient. The lift force drives the turbine to rotate and produce power thus it is likely that the blunt foil will yield the most power for a vertical axis tidal turbine. The small amplitude indicates that the blunt foil experiences weak fluid dynamic loading hence it has the weakest vibration response. The blunt foil is expected to improve a vertical axis tidal turbine efficiency when utilized for a blade construction. The actual incorporation of a modified trailing edge blade for a vertical axis tidal turbine will be discussed in Section 7.7. The benefit is low drag forces as shown in Figure 4.59. A lower drag force lowers resistance to turbine rotation. The sharp and rounded foil drag coefficient exhibit high amplitude fluctuations. This fluctuating drag forces increases the dynamic loading caused by the lift force.

Similarly to the lift and drag coefficient, the moment coefficient amplitude for the blunt foil is also the smallest whereas the sharp and rounded foils have very high amplitudes as

![Figure 4.54: Pressure distribution for the rounded foil model at t/T: 0.04, 0.22, 0.35, 0.57, 1](image-url)
Figure 4.55: a. Reference, original and phase averaged velocity signal for sharp foil model, b. frequency response for the sharp foil model

shown in Figure 4.60. This implies that the pitch response for the blunt foil is weaker in comparison to other foils. The pitch response for the sharp and round foils are found to be very high as discussed earlier and as shown in Figure 4.56 and 4.53 respectively. The modified trailing edge foil response result is an early prediction for the modified vertical axis tidal turbine blade response. An assessment is required to ensure the improvement gained in the usage of each modified blade for a vertical axis tidal turbine. This will be detailed in the next section.

4.2.4 An Asymmetric Foil for Singing Reduction

The asymmetric foil model is developed to reduce a singing effect from fluid induced vibration as discussed in Section 4.2.4. The asymmetric foil fluid regime is shown in
Figure 4.56: Fluid flow regime at sharp foil model, at $t/T$: a. 0.06, b. 0.28, c. 1

Figure 4.57: Pressure distribution for the sharp foil model at $t/T$: 0.04, 0.22, 0.35, 0.57, 1

Figure 4.61. From that figure it is seen that the vortex generation is observed during the operation. In 2D model the vibration reduction effect can be captured by the model. However this cannot be justified that the singing effect is not being reduced either. The reduction occurs when the frequency of vibration lies away from the blade’s natural frequencies.

Another modified trailing edge profile is introduced to reduce the vibration effect. The profile is assymetric foil which allows the foil’s trailing edge to be truncated in deeper
in the lower side. This typical truncation is proven to reduce the vibration and its effect such as singing as investigated by Fischer (2008). Singing itself is recognized as a strong tone which has a range of frequency between 100-100 hz and emitted from a vibrating structure such as a blade (blade singing) or a propeller (singing propeller) (Ross, 2013). Fischer (2008) in his paper defined a "singing" as a tone generated by the interaction between a Karman vortex shedding mechanism from trailing edge of the blade and a
blade natural frequency. He also emphasized that the coincidence of vortex shedding frequency and the blade natural frequency induced a “singing”. He suggested a singing reduction by introducing an asymmetric blade as also studied by Blake (1984, 2017). They demonstrated that a foil with 45° bevel angle did not sing.

In this work an asymmetric foil is modeled to reduce the blade singing. The asymmetric foil is adopted from Zobeiri et al. (2012). The blunt foil is truncated with 30° bevel angle on the lower side of trailing edge as shown in Figure 4.62(a). The asymmetric foil mesh in a rectangular grid is shown in Figure 4.62(b). The asymmetric foil attempts to reduce the vortex shedding in fluid regime, minimize the fluid-induced vibration and eventually reduce the blade singing.

### 4.3 Summary

The original foil model is developed using the standard simulation properties listed in Table 2.3. The simulation properties is varied as shown in Table 2.4. The vortex shedding frequencies for all the standard and varied simulation property models using the original foil are presented in Table 4.3.

The vortex shedding frequency tends to be synchronized with the natural frequency when both frequency values are close. In this case a resonance occurs. In a resonant condition, the vortex shedding is amplified and the vortex induced vibration on the foil is enhanced which can be signified by a large displacement as shown in Table 4.3. In
Figure 4.61: Fluid flow regime on an asymmetric foil at $t/T$: a. 0.2, b. 0.4, c. 0.6, d. 0.8, e. 0.99

Figure 4.62: Asymmetric Foil: a. Model, b. Mesh
the 6 m/s inlet velocity model, 8° initial angle of attack model, and 1500 N/m stiffness constant model, one of the main vortex shedding frequency is found to be close to the pitch natural frequency. In these models the vibrations are stronger and signified by larger displacement than the other models which do not have a frequency close to the blade natural frequency. In the 4° initial angle of attack model, the pitch surge heave response model and 300 Ns/m damping constant model, one of the main vortex shedding frequency coincides with the heave natural frequency. Similarly to the pitch natural frequency case, these models have a large amplitude vibration as indicated by a greater displacement in each model. However in the heave natural frequency models, the vortex induced vibrations are weaker which are indicated by smaller displacements compared to the pitch natural frequency case. This is due to the smaller vortex shedding frequency of the models which coincide with the heave natural frequency. To control the foil vibration, the foil should operate in a 4 m/s fluid velocity or lower. Additionally it should be oriented by zero initial angle of attack, and be allowed to have three degree of freedom response.

Table 4.3: Maximum Heave Displacement and vortex shedding frequencies for all models with the associated natural frequency at each foil geometry. (P): a resonance to pitch natural frequency, (H): a resonance to heave natural frequency

<table>
<thead>
<tr>
<th>Variation</th>
<th>Model</th>
<th>Heave (m)</th>
<th>Main frequency (Hz)</th>
<th>$f_n$ (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Heave</td>
<td>Pitch</td>
</tr>
<tr>
<td>Original</td>
<td>Standard</td>
<td>0.014</td>
<td>0.31</td>
<td>2.34</td>
</tr>
<tr>
<td>Velocity</td>
<td>6 m/s</td>
<td>0.2</td>
<td>1.328(P)</td>
<td>1.476</td>
</tr>
<tr>
<td></td>
<td>2 m/s</td>
<td>0</td>
<td>1.85</td>
<td>0.9425</td>
</tr>
<tr>
<td>Initial angle</td>
<td>8°</td>
<td>0.2</td>
<td>0.468</td>
<td>3.672</td>
</tr>
<tr>
<td></td>
<td>4°</td>
<td>0.09</td>
<td>0.015(1)</td>
<td>2.65</td>
</tr>
<tr>
<td>Orientation</td>
<td>Pitch surge</td>
<td>0.0003</td>
<td>0.782</td>
<td>0.8767</td>
</tr>
<tr>
<td></td>
<td>heave</td>
<td></td>
<td>2.55</td>
<td>1.4191</td>
</tr>
<tr>
<td></td>
<td>Pitch surge</td>
<td>0</td>
<td>0.001</td>
<td></td>
</tr>
<tr>
<td></td>
<td>pitch</td>
<td></td>
<td>0.55</td>
<td></td>
</tr>
<tr>
<td>Damping constant</td>
<td>Overdamped</td>
<td>0.013</td>
<td>0.935(H)</td>
<td></td>
</tr>
<tr>
<td>Stiffness constant</td>
<td>1500 N/m</td>
<td>0</td>
<td>1.72(P)</td>
<td>1.0364</td>
</tr>
<tr>
<td></td>
<td>300 N/m</td>
<td>0.009</td>
<td>0.312</td>
<td>0.8767</td>
</tr>
</tbody>
</table>

The mechanical properties of a foil such as the stiffness constant and the damping constant also impact the vortex generation and vortex induced vibration. The less stiff material can control the vortex shedding and produce less vortex induced vibration on the foil. In contrast with the stiffness model, the heave damping property does not affect the vortex induced vibration much. The pitch damping constant should be also further analyzed in a later study to identify the effect of both properties on the foil vibration response. From the simulation properties variation, the standard model gives a good result with no resonance occurrence and less vortex induced vibration. This model is
employed for the modified foil response model and the vertical axis tidal turbine response model using the original and modified blades.

The response of the modified foils are investigated using the standard model properties shown in Table 2.3 and the vortex shedding frequencies as listed in Table 4.4.

Table 4.4: Maximum displacement and vortex shedding frequency for the original and modified foils with the associated natural frequency. (P): a resonance to pitch natural frequency

<table>
<thead>
<tr>
<th>Foil</th>
<th>Max. Displacement (m)</th>
<th>Main Frequency (Hz)</th>
<th>$f_n$ (Hz)</th>
<th>Heave</th>
<th>Pitch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.014</td>
<td>0.31</td>
<td>2.34</td>
<td>4.76</td>
<td>0.9425</td>
</tr>
<tr>
<td>Blunt</td>
<td>0.01</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Rounded</td>
<td>0.16</td>
<td>1.328(P)</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Sharp</td>
<td>0.13</td>
<td>1.406(P)</td>
<td>2.89</td>
<td>8.594</td>
<td>-</td>
</tr>
</tbody>
</table>

From Table 4.4, it can be seen that the blunt foil has the lowest displacement which indicates the occurrence of the weakest vibration. This shows that the blunt trailing edge profile controls the vortex shedding more effectively than the other modified foils. In the blunt foil model no dominant vortex shedding frequency is observed. The absence of a predominant vortex shedding frequency in the blunt foil is likely due to the location of the vortex shedding. In the blunt foil the vortices are shed from the flat back of the trailing edge rather than from the upper or lower surface such as occurs in the other trailing edge shapes. In contrast with the blunt foil, the rounded and sharp trailing edge profile cause strong vortex shedding and shift the vortex shedding frequency close to the pitch natural frequency. However the heave displacement of the rounded and sharp foils are similar to the original foil model.

Force coefficients are also affected by the trailing edge modification. The blunt foil lift coefficient has the highest mean value with the smallest amplitude compared to the other trailing edge profiles. The main benefit from the blunt foil is that it reduces the drag force magnitude more than the other trailing edge shapes. However, the blunt foil has the highest moment coefficient which excites the pitch response motion.
Chapter 5

Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

The development of a vertical axis tidal turbine blade response model is detailed in this chapter. The model utilizes design parameters which have been observed and selected from their variations as explained in Chapter 4. The design parameters used in the original and modified vertical axis tidal turbine blade response models are the parameters in the standard model as presented in Table 5.1.

Table 5.1: Design parameters used in the original and modified blade response model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial velocity (m/s)</td>
<td>4</td>
</tr>
<tr>
<td>Initial angle of attack (°)</td>
<td>0</td>
</tr>
<tr>
<td>Orientation</td>
<td>Pitch, heave</td>
</tr>
<tr>
<td>Stiffness (N/m): heave, pitch</td>
<td>1000, 200</td>
</tr>
<tr>
<td>damping (N.s/m)</td>
<td>2</td>
</tr>
</tbody>
</table>

The blade response model is developed in the dynamic mesh as implemented in the oscillating foil and foil response models. The mesh which has been validated and discussed in Section 3.7 will be applied to the original and modified blade response models. However, the boundary condition of the original blade model needs to be further verified before it is set to examine the influence of trailing edge shapes on fluid structure interaction of a vertical axis tidal turbine blade. The boundary condition and the CFD is verified using Almohammadi et al.’s model (2015). Almohammadi et al. developed a rotating three bladed vertical axis tidal turbine model using Fluent and analysed the effect on trailing edge profiles to the turbine performance. Their rotating turbine model developed in a global frame of reference. In the blade response model, it is assumed to model one of the three Almohammadi et al.’s turbine blades using the rectangular grid which is discussed in Subsection 3.2.1.2 and the CFD properties as explained in Section 2.3. The single vertical axis turbine blade has a fixed position with no response motion which adopts
the Almohammadi et al.’s turbine model. The turbine rotational motion is replaced by time varying velocity magnitude and angle of attack entering the CFD domain using the Periodic Inflow Equivalence Method (Subsection 2.2.1). The result is compared to Almohammadi et al.’s result for the process prior to further use for the modified vertical axis tidal turbine blade.

## 5.1 Vertical Axis Tidal Turbine Response Representative

The single original vertical axis tidal turbine blade response model is verified using Almohammadi et al.’s result (2015). In this stage the blade is static, adopting the reference model which does not allow the three blades of a vertical axis tidal turbine to have individual response motions except for the rotating motion. In the static response model the turbine rotation is represented by the Periodic Inflow Equivalence Method as described in Subsection 2.2.1. This method assumes that the observer is sitting on the observed turbine blade while it is rotating and experiencing the steady tidal current passing the blade in a time varying angle of attack and velocity magnitude. The turbine design parameters of the reference model are applied in the static response turbine model as listed in Table 5.2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Static response model</th>
<th>Almohammadi et al.’s model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model</td>
<td>a single VATT blade</td>
<td>Three bladed VATT</td>
</tr>
<tr>
<td>Blade</td>
<td>NACA 0012</td>
<td>NACA 0015</td>
</tr>
<tr>
<td>Chord length (m)</td>
<td>0.745</td>
<td>0.4</td>
</tr>
<tr>
<td>Turbine radius (m)</td>
<td>2</td>
<td>1.25</td>
</tr>
<tr>
<td>$Re$ (based on the resultant velocity)</td>
<td>$7.796 \times 10^5$</td>
<td>$7.796 \times 10^5$</td>
</tr>
<tr>
<td>TSR</td>
<td>2.5</td>
<td>2.5</td>
</tr>
<tr>
<td>Free stream velocity (m/s)</td>
<td>5.324</td>
<td>10</td>
</tr>
<tr>
<td>Angular velocity (rad/s)</td>
<td>6.655</td>
<td>20</td>
</tr>
<tr>
<td>Range of angle of attack</td>
<td>-23.54° to 23.54°</td>
<td>-23.54° to 23.54°</td>
</tr>
</tbody>
</table>

From Table 5.2, some differences between the reference and static blade response models can be stated. Almohammadi et al.’s model has a rotational frame of reference which represents the rotating vertical axis tidal turbine with all the attached three blades. The selected blade thickness, blade chord length and turbine radius which influences
the angular velocity are also different. The angular velocity of the reference model is calculated using Equation (2.3). This affects to the free stream velocity and angular velocity of the static response model because of the different selected chord lengths and maintaining geometric similarity of both models. The free stream velocity and angular velocity of the static response model are calculated using Equations (2.7) and (2.3) respectively. The free stream and angular velocity of both models are shown in Table 5.2.

The static blade response model is developed using the mesh which is applied in the oscillating foil and foil response models. There are some dissimilarities found in the mesh resolution between the Almohammadi et al. and the static blade response models. The mesh properties are shown in Table 5.3.

Table 5.3: CFD parameters of static blade response model and Almohammadi et al.’s model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Blade response model</th>
<th>Almohammadi et al.’s model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid type in boundary layer</td>
<td>structured</td>
<td>structured</td>
</tr>
<tr>
<td>Grid type in rotor region</td>
<td>unstructured</td>
<td>unstructured</td>
</tr>
<tr>
<td>Boundary layer thickness (m)</td>
<td>0.7</td>
<td>0.752</td>
</tr>
<tr>
<td>Number of layers in the boundary layer</td>
<td>7</td>
<td>7 (approx.)</td>
</tr>
<tr>
<td>Number of cells</td>
<td>12,432</td>
<td>23,693 (approx.)</td>
</tr>
<tr>
<td>Domain area per one blade</td>
<td>27.28</td>
<td>12.822</td>
</tr>
</tbody>
</table>

In the static response model, the blade does not show the blade response motion. This can be obtained by removing the spring and damper system. As such the blade becomes static with 0° pitch angle when the azimuth angle is 0° (refering to Figure 2.9(a)). This condition is adopted from Almohammadi et al.’s model. The turbine rotation in the static blade model is performed by the Periodic Inflow Equivalence method which provides the time varying angle of attack and incoming velocity magnitude. The method is applied in the boundary condition of velocity inlet of blade response model’s domain.

In the stage, the lift and drag coefficients of the original blade in the static response model is compared to the drag and lift coefficients from Almohammadi et al.’s model as seen in Figure 5.1. The ’s result gives an insight if the the static response model is representative and favorable to perform the rotating vertical axis tidal turbine manner. The drag coefficients of the rotating various vertical axis tidal turbine blade profiles from Almohammadi et al.’s model are shown in Figure 5.1(a). The original blade drag
Chapter 5 Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

coefficient of the static response model and Almohammadi et al.’s model which is taken from Figure 5.1(a) are confronted in Figure 5.1(b).

![Figure 5.1](image-url)

Figure 5.1: a. $C_d$ from Almohammadi et al. 2015. Only sharp trailing edge data is considered in b. $C_d$ from Almohammadi et al.’s model (in orange, taken from Figure 5.1(a)) and the static response model (blue)

The design parameters of both models are shown in Table 5.2. The reference drag and lift coefficient is replicated using the online software provided (Rohatgi, 2017). Drag coefficient of the rotating turbine in Figure 5.1(b) is calculated from Figure 5.1(a) using the blade angular velocity which returns to lower drag coefficient magnitude.

From Figure 5.1(b) it can be observed that the drag coefficient from the static response model result represents the reference model’s drag coefficient. In general, the result shows similar drag coefficient contour. The angle of attack range is identical which supports the justification of the static response model. However disimilarities are found from both results. The non identical results of the static model and reference model
occur due to the different frame of reference applied in both models. In the global frame of reference which allows the turbine to rotate, the fluid behaviour in the reference model is more chaotic. The irregularity is caused by the faster angle of attack change which is indicated by higher angular velocity (Table 5.2). This creates higher level of unsteadiness in which the boundary layer evolves very quickly. The rapid boundary layer evolution drives the larger change of drag forces working on the turbine blade and affects the change of $C_d$ magnitude which is indicated by higher slope in the drag coefficient line. The different blade thickness also contributes to the drag coefficient discrepancy of both models. The maximum drag coefficient magnitude for the reference model is found higher. In the negative angle of attack the $C_d$ magnitude is found higher than the positive angle and creates bigger hysteresis. The trend is also experienced by the lift coefficient of the reference and static response model which are shown in Figure 5.2.

![Figure 5.2](image-url)

Figure 5.2: a. $C_l$ from Almohammadi et al.. Only sharp T.E data is considered in validation b. $C_l$ from the response model
In Figure 5.2(a), lift coefficient of the reference model using various blade profiles are shown. Lift coefficient of the original blade in the reference model (refers to sharp T.E) is replicated in Figure 5.2(b) and plotted along with the lift coefficient of the static reference model. As occurs in the drag coefficient result, the lift coefficient of the static response model agrees qualitatively to the lift coefficient contour of the reference model. Hysteresis is also found in the lift coefficient magnitude. However disimilarities are found due to the level of fluid unsteadiness and different blade section used as detailed in drag coefficient in the previous paragraph. The lift coefficient magnitude of the reference model is higher than in the static response model due to larger blade thickness. This result is consistent with the static angle of attack lift coefficient trend of symmetric blades as shown in Figure 5.3.

![Figure 5.3: Lift coefficient contour of various different symmetric blade thickness (Sheldahl and Klimas, 1981)](image)

From his experiment’s result which is shown in Figure 5.3, Sheldahl and Klimas (1981) found that a blade’s thickness affected to lift forces acting on a static foil. Larger thickness blade gains better lift forces hence experiences higher lift coefficient.

In Figure 5.2(b), at -7° angle of attack, the lift coefficient of reference model drops drastically in which does not exist in the static response model. This different condition happens due to the smaller blade thickness used in the static response model. At -7° angle and beyond, the lift coefficient magnitude of the static response model decreases until it reaches the minimum angle of attack. This prediction is consistent with the lift coefficient magnitude from Sheldahl and Klimas’s experiment. From Figure 5.3 it is
observed that NACA 0015 foil has higher lift coefficient magnitude than NACA 0012. The lift coefficient of NACA 0015 foil increases beyond the angle of attack in which the maximum lift coefficient of NACA 0012 foil occurs. This is reflected from the decreasing lift coefficient magnitude beyond \(-7^o\) angle of attack in the reference model.

From the comparison between the static response and reference model’s drag and lift coefficients, it can be justified that the boundary condition and the mesh of the response model is qualitatively represented. Therefore it will be further applied in the response model using original and modified foils. Although the response model is verified, it is interesting to identify the TSR and \(Re\) variations implemented in the model. This helps to predict the blade response behaviour resolved by the model in various condition. The TSR and \(Re\) variations which is modeled in this section are depicted in Table 5.4.

Table 5.4: TSR and \(Re\) variation for the static response model validation

<table>
<thead>
<tr>
<th>Variation</th>
<th>model 1</th>
<th>model 2</th>
<th>model 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>TSR</td>
<td>2.5</td>
<td>5.1</td>
<td>5.1</td>
</tr>
<tr>
<td>(Re)</td>
<td>(3.07 \times 10^6)</td>
<td>(3.07 \times 10^6)</td>
<td>(7.796 \times 10^5)</td>
</tr>
</tbody>
</table>

5.1.1 TSR variation

TSR is a significant parameter in the Periodic Inflow Equivalence method used in the response model. The method develops the range of angle of attack which is experienced by a vertical axis tidal turbine blade hence determines the possibility of a stall occurrence. The static response model is applied to identify the influence of TSR variation on the vertical axis tidal turbine performance. Two models of different \(Re\) with the same TSR are developed as depicted in Table 5.4. The TSR variation is covered by model 1 and model 2. The drag and lift coefficients of the static response model 1 and model 2 are depicted in Figures 5.4 and 5.5 respectively.

Figure 5.4 shows the drag coefficient of the static response model with TSR variation. The different TSR produces different range of angle of attack which is experienced by the blade. From Figure 5.4 it is shown that both models have similar contour. Hysteresis occurs in both models. It is found that higher TSR returns to lower maximum drag coefficient magnitude due to lower angle of attack at which the maximum drag coefficient magnitude occurs. In model 1 the maximum angle is \(20^o\) which generates stall in the fluid flow and cause higher maximum drag coefficient magnitude. In model 2 the upward and downward motions are not overlapped which provides lower drag coefficient in downward motion for all angles of attack. The non overlapping trend also occurs in lift coefficient of both models as shown in Figures 5.5 corresponds to Table 5.4.

Figure 5.5 shows the lift coefficient of the static response model 1 and model 2 of Table 5.4. From that figures it is shown that both models have similar pattern. Hysteresis
is found in both models. Model 2 has lower maximum lift coefficient magnitude due to lower range of angle of attack. The larger range of angle of attack occurs in model 1 produces larger lift coefficient magnitude increase which creates higher maximum lift coefficient magnitude. Although model 2 in which the turbine design parameters are selected in this work has lower lift coefficient magnitude, the drag coefficient is lower. It also has lower angle of attack which is required to prevent the stall condition. However the stall condition can also created by the $Re$ which will be varied and discussed in Subsection 5.1.2.
5.1.2 \( Re \) variation

\( Re \) is varied in the static response model as shown in Table 5.4. Model 2 and model 3 cover the \( Re \) variation with the same TSR of 5.1. The drag and lift coefficients plots from both models versus angle of attack are depicted in Figures 5.6 and 5.7 respectively.

![Figure 5.6: Drag coefficient from models with \( Re \) variation and TSR = 5.1](image)

From Figure 5.6 it can be observed that both models have quite identical drag coefficient shape with the slightly different maximum drag coefficient magnitude. The two models also have the same range of angle attack as a result of the same TSR. Similar to drag coefficient, lift coefficient of model 3 and 2 are found identical as shown in Figure 5.7.

![Figure 5.7: Lift coefficient from models with \( Re \) variation and TSR = 5.1](image)

Figures 5.7 shows that the maximum lift coefficient of model 3 is slightly lower due to lower \( Re \). The result of drag and lift coefficients conclude that \( Re \) variation gives a small influence to the vertical axis turbine performance.
The representative single blade response model requires sufficient time to reach a steady state simulation to obtain an accurate result. All models are set to capture 15 seconds real time motion which is considered to be sufficient to provide a steady state simulation. All velocities and force coefficients will be simulated for 15 seconds. As explained in Subsection 4.1.1, the reference velocity is investigated at the leading edge and trailing edge point as shown in Figure 4.2. In the leading edge point a periodic vibration signal which is not affected by any fluctuating components is obtained. At this location, vortex shedding is not yet formed thus the fluid flow is likely to have no vorticity and exhibits purely periodic response.

The Phase Averaged method is applied to calculate the vortex shedding frequency for each case. Further the Power Spectrum Density method is implemented to obtain the vortex shedding frequency domain results from time domain results. Both methods are applied after the transient condition is finished. This happens after three seconds from the simulation starts. The velocity signal is analysed using the Phase Averaged and PSD method to find the vortex shedding frequency. The reference velocity is obtained at the nearest cell to the leading edge and the original signal is taken from the velocity signal at a point 0.1c from the trailing edge. Figure 4.2 shows the location of the original and reference points probe (left and right pink spot respectively) taken from an original foil. The modified foil parameters are also sampled at the same location to extract the signals of the original and reference velocities. OpenFOAM models are constructed in 3D space model thus the result is generated with three components in the x, y, and z directions. The velocities obtained from in the Phase Averaged method is the resultant velocity which is calculated from the square root of the three velocity components. The procedure is repeated for both signals.

5.1.3 Lift to Drag Ratio

Lift to drag ratio is defined mathematically as the ratio of lift coefficient over drag coefficient working on a turbine blade. It is an important aspect for a vertical axis turbine as reflects the amount of torque which drives the turbine rotation. The higher the lift to drag ratio of a turbine contributes to higher power the turbine gains from its rotation. The lift to drag ratio of model 2 vertical axis tidal turbine as shown in Table 5.4 is depicted in Figure 5.8.

From Figure 5.8 it is shown that the high torque as a result of zero drag coefficient occurs at angle close to 6.2° in upward motion, 1.8° and -4.5° in downward motion. The angle of 6.2° upward motion happens when a vertical axis tidal turbine blade rotates at the first quadrant section of a turbine revolution as shown in Figure 5.9. From that figure, it can also be observed that the angle of 1.8° and -4.5° downward motion occur at second and fourth quadrant respectively. The angle of attack ranges from -11.3° to
11.3° in which the vortex is unlikely to develop. Therefore the fluid flow is laminar and generates smaller drag coefficient which produces higher lift to drag ratio.

5.2 Response of a Vertical Axis Tidal Turbine Using Original Blade

The response of a vertical axis tidal turbine using the original and modified NACA 0012 blades are investigated numerically in this section. The three blades modification includes blunt, sharp and rounded profiles. The turbine response model is developed using the CFD and blade properties as listed in Table 2.3. The turbine design properties correspond to data in Table 3.7. The model also analyses the turbine rotation by the Periodic Inflow Equivalence Method which have been explained in Subsection 2.2.1. This method converts the turbine resultant velocity into the inlet fluid velocity entering the vertical axis tidal turbine blade model domain. The turbine blade model performs only the response motion and neglecting the turbine rotation as in the turbine model in Figure 2.9. The response is a result of the tide and blade interaction.

The four different blade geometries will be analysed separately in four different models. All models employ the Periodic Inflow Equivalence method. This assigns an unsteady incoming flow passing through the domain inlet which represents the turbine’s rotation. The method creates an unsteady incoming flow which models the turbine rotation as a time varying inlet velocity and angle of attack calculated using Equation (2.4) as discussed in Subsection 5.2.1. Time step resolution is also investigated in this section.
considering the importance of the time step in the simulation. The time step independence will be explained in Subsection 5.2.3. The investigation includes time step independence in vertical axis tidal turbine case using original blade.

### 5.2.1 Unsteady Angle of Attack and Velocity of the Incoming Flow

The incoming flow experienced by a vertical axis turbine model is unsteady. In this turbine model the unsteady flow comes from a time dependent changing angle of attack and inlet velocity magnitude. The angle of attack which is calculated using Equation (2.4) from Gosselin et al. (2013) is found to be in the range of \(11.307^\circ\) to \(-11.307^\circ\). The angle of attack for one turbine revolution is illustrated in Figure 5.9.

![Figure 5.9: Angle of attack during one turbine rotation](image)

The turbine blades experienced the maximum and minimum angle of attack of \(11.31^\circ\) and \(-11.31^\circ\) at an azimuth angle of \(102.41^\circ\) and \(259.06^\circ\) regardless blade geometry and blade trailing edge shape. The time dependent angle variation determines the time varying velocity resultant as illustrated in Figure 2.9. The fluid resultant velocities act on the blade over one revolution are shown in Figure 5.10 and it is also independent to the blade geometry and trailing edge shape.

The resultant velocity over one blade revolution reaches the highest magnitude at 4.0016 m/s when the turbine azimuth angle is at \(0^\circ\) and at \(360^\circ\). In this position, the angle of attack between the incoming velocity and the blade tangential velocity is zero. Both velocity vectors are in the same direction and create the highest velocity magnitude during a revolution. The minimum velocity occurs at an azimuth angle of \(180^\circ\). In this position the azimuth angle is \(180^\circ\) in which the incoming velocity direction is opposite to the tangential velocity. The velocities moderate each other and creates the lowest resultant velocity magnitude. The magnitude of the angle of attack and the resultant velocity are independent of the blade geometry and the blade trailing edge shape. The
shape of the trailing edge generates different fluid flow behavior on the trailing edge region. This influences the overall pressure, velocity and forces acting on the blade. The pressure, velocity and forces from one trailing edge shapes to another are different. The response result of original foil will be detailed in Subsection 5.2.2 and the modified trailing edge results will be discussed in Section 5.3.

### 5.2.2 Response Result

The analysis starts with sampling two velocities to find the periodic reference velocity and the original velocity. The points where the velocities are monitored are at the leading edge and at 0.1 m behind the trailing edge of each blade. The reference signal is further converted to the phase averaged signal is using The Hilbert Transform. The reference signal is subtracted from the phase averaged signal to find the vortex shedding velocity. Finally the vortex shedding frequency is obtained by applying the PSD method to the vortex shedding velocity to convert from the time domain to the frequency domain. The analysis process is undertaken using the calculation routine as discussed in 4.1.1.

Original NACA 0012 is utilized in for a vertical axis turbine blade model. The periodic inflow equivalence velocity is implemented in this model. The reference, original and phase averaged velocities are depicted in Figure 5.11(a) and the vortex shedding frequency is shown in Figure 5.11(b).

Figure 5.11a shows that the velocities are repeated for approximately three and a quarter revolutions during twelve seconds simulation time. The revolutions reflect the number of rotations experienced by the turbine. The exact turbine revolution can be calculated using the TSR, solidity and chord length listed in Table 2.2. The turbine revolution is obtained by first calculating the turbine radius and the angular velocity of turbine using Equation (2.5) and Equation (2.3) respectively. The turbine radius and turbine
angular velocity are found to be 1.96 m and 1.71 rad/s (97.91°/s) respectively. From this calculation, over twelve seconds the turbine rotation is found to be 3.268 cycles. The number of repeating velocity plot shown in Figure 5.11a is consistent with number of the turbine rotations from the calculation. The $\text{Re}$ based on the maximum resultant velocity is $3.07 \times 10^6$.

A vertical axis tidal turbine using the original blade exhibits four main vortex shedding frequencies. These are 0.234 Hz, 0.55 Hz, 0.78 Hz, and 1.095 Hz as shown in Figure 5.11b. The 1.095 Hz frequency is close to the heave natural frequency. This condition is affected by the time varying incoming fluid velocity and angle of attack as shown in Figure 5.9 and 5.10. At the location where the positive angle of attack acts on the blade during the first half cycle of turbine rotation, the vorticies is stronger behind the trailing
edge and influences the fluid induced vibration on the blade. The vortex shedding in the fluid flow regime can be seen in Figure 5.12. The vortex shedding can be also detected from the pressure distribution contour shown in Figures 5.13 to 5.16.

Figure 5.12: Fluid flow regime during a rotation of a vertical axis tidal turbine using original blade model. From $0^\circ$, shown every $30^\circ$ azimuth angle clockwise.

U magnitude in m/s

Figure 5.12 shows images from an original blade vertical axis tidal turbine fluid flow regime during one revolution shown every $30^\circ$ azimuth angle, starting from $0^\circ$ clockwise. The fluid flow regime is strongly influenced by the pressure distribution which is plotted in Figures 5.13 to 5.16. At $0^\circ$ angle, the vortex is fully detached from the surface as indicated by a flat surface pressure in Figure 5.13. A small suction pressure is detected at the leading edge, demonstrated by a vortex creation at that location. At an angle of $30^\circ$, formed LEV is moving further along the upper surface which can be seen in the
pressure contour plot in Figure 5.13. A small peak is found in the plot at $x/c = 0.25$ caused by a vortex generation pressure. The LEV grows and flows downward at $60^\circ$ azimuth angle marked by higher and receding peak at $x/c = 0.46$. The fluid begins to detach and a smaller vortex is attached to the blade surface as shown by a smaller peak at $90^\circ$. The peak is located at $x/c = 0.65$ which signifies the moving vortices.

The pressure distribution in the second quadrant which covers the azimuth angle from $90^\circ$ to $180^\circ$ is shown in Figure 5.14. At $120^\circ$ azimuth angle, the vortex is detached from almost the entire upper surface to the free stream and only some attachment is left near the trailing edge. The pressure distribution profile is also shown a constant line at the upper and lower surfaces which indicates that the pressure is almost constant over the entire blade surface. The vortex detachment continues at the azimuth angle of $150^\circ$. The separation flow is found on the back of the blade. This is indicated by a small peak at $x/c = 0.81$. The vortex is fully detached at $180^\circ$ azimuth angle as reflected by a smooth fluid flow regime shown in Figure 5.12.

![Figure 5.13: Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: $0^\circ$, $30^\circ$, $60^\circ$, $90^\circ$](image)

The fluid flow regime is consistent with the angle of attack result shown in Figure 5.9 and the velocity result in Figure 5.10. In the first quadrant, the velocity decreases, however its magnitude is still high. The high velocity magnitude combined with the increasing angle of attack generates a LEV on the upper surface. The vorticity weakens in the second quadrant due to decreasing velocity and decreasing angle of attack. The fluid flow regime becomes laminar with no vortex detected at $180^\circ$ as a result of the low velocity and zero angle of attack.

The pressure distribution contour in the third quadrant of turbine revolution from $180^\circ$ to $270^\circ$ azimuth angle is shown in Figure 5.15. The pressure distribution during this stage are flat with no pressure peak observed. This indicates that no vortex is generated on either surfaces. Although the velocity magnitude increases from the previous condition,
the angle of attack acting on the blade decreases. This weakens the effect of the high velocity and prevents the vortex from becoming stronger.

The last quadrant of the revolution cycle is observed at azimuth angles from $270^\circ$ to $360^\circ$. The pressure distribution associated with the fluid flow regime is shown in Figure 5.16. At $300^\circ$ azimuth angle, a suction peak is found at $x/c = 0.15$ on the upper surface. This peak indicates vortex initiation at that location as can also be seen in the fluid flow regime image in Figure 5.12. The vortex flows downstream until the blade reaches $330^\circ$ azimuth angle. The moving vortices are also captured in the pressure distribution plot indicated by peak moving away from the leading edge. The vortices generated on this section are weaker than in the first quadrant section as shown in the pressure distribution magnitude and the fluid flow regime plot. This is due to the lower angle of attack experienced by the blade. The vortex detaches from the upper surface at $360^\circ$.
Chapter 5 Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

Figure 5.16: Pressure distribution for vertical axis tidal turbine model with original blade at azimuth angles of: 270°, 300°, 330°, 360°

The vortex induces vibrations on the blade which can be observed from the blade displacement are shown in Figure 5.17. Figure 5.17 shows the time history of the blade displacement in x (pointDisplacement (0)) and y (pointDisplacement (1)) directions. The displacement of the foil when the blade is at 0° azimuth angle is taken as a point of vibration reference. At this point the blade is assumed to be in equilibrium position, although this is not the initial non-force condition. The time axis is presented in unit of the time step. The unit conversion from the time step to seconds at one turbine rotation (azimuth angle) and angles of attack experienced by the blade at that specific azimuth angles can be found in Table A1 in appendix.

At 0° azimuth angle, the vortex is fully detached from the surface. The blade surface has no vortex induced vibration acting on it and is at the equilibrium position at this instance. At 30°, an initial LEV begins to form which produces vibrations on the blade, marked by some fluctuations between 0° and 30° angle. The upper surface experiences lower pressure which produces more suction pressure on the blade and cause it move to the upper position. From that position, the vortex flows towards the trailing edge and creates further vibrations until it reaches 90°. At this point the pressure on the lower surface decreases and reduces the lift force which causes the blade to move back to its equilibrium position.

In the second quadrant from azimuth angle of 90° to 180°, the distribution of the pressure magnitude decreases at all locations over the foil surface. The decreasing pressure reduces the total lift acting on the foil thus the foil moves downwards during this second quadrant of the turbine revolution. In the third quadrant of revolution, the pressure increases over the foil, thus creating a higher force that moves the blade to its upper position again.
In the last quarter of revolution, it can be seen from the distribution pressure contour that total pressure on the blade becomes higher as illustrated by the larger total area inside the upper and lower surface pressure contour. The larger total pressure contributes to a higher force acting on the blade which tends to move the blade to the upper position until it reaches 330° angle. After that position the pressure becomes lower causing the blade to return to its mid position. The x direction displacement is very low compared to the y direction displacement in which a sign of a weak vibration in x direction acting on the blade. The maximum displacement reaches approximately 0.11 m. This exists at azimuth angle between 68° and 108°.

### 5.2.3 Time step Independence Study

Time step independence study is employed on the vertical axis tidal turbine using original blade. The time step study affects time step resolution employed in the simulation. Time step resolution is a parameter to identify the length of time period for running an iteration in a simulation. The resolution is strongly influenced by cells size, fluid velocity and selected Courant Number.

The purpose of the time step independence study is to identify the influence of time step to the vertical axis tidal turbine response model. In all models time step is generated automatically by determining a Courant Number. The appropriate Courant Number in
the CFD models are less than 1. In this time step investigation, some Courant Numbers are selected and implemented in the simulation. A smaller Courant Number leads to a smaller time step. The displacement at a quarter chord length graph of those results are compared and observed. The displacement of quarter chord length at Courant Number equals to 0.9, 0.7, 0.6, 0.5, and 0.4 are shown in Figure 5.18.

![Displacement Graphs](image)

Figure 5.18: Displacement of quarter chord length point at Courant Numbers of: a. 0.9, b. 0.7, c. 0.6, d. 0.5, e. 0.4

Figure 5.18 shows that there are slightly different displacements occurred at azimuth angle between 150° to 210° and between 240° to 300° (refering to Figure 5.17). At Courant Number equals to 0.9, 0.7, 0.6 and 0.5, the displacement difference at those two ranges are quite obvious. This indicates the simulations are still affected by those Courant Numbers. At Courant Number of 0.4, the displacement at the two ranges are similar to the simulation with Courant Number of 0.5. From that result it can be seen that the time step independence appears at Courant Number equals to 0.5.
5.3 Response of A Vertical Axis Tidal Turbine Using Modified Blade

Modified blades are employed for the vertical axis tidal turbine blade design to avoid the possibility of locked-in frequency occurrence. The employed modified blades include blunt, rounded and sharp trailing edge shapes which will be detailed in the next sections.

5.3.1 Blunt Blade

A vertical axis tidal turbine using a blunt blade is analyzed in this section. The model is constructed with the same mesh topology as that one utilized for the blunt foil response model. The phase averaged method for finding the vortex velocity and the PSD method for converting the vortex velocity to a frequency domain are also used in this model. The reference velocity is taken from the leading edge point and the original velocity is sampled at a point of 0.1 m distance from the trailing edge. The original, reference and phase averaged velocities are illustrated in Figure 5.19a while the vortex shedding frequency from the PSD method is shown in Figure 5.19b.

The incoming velocity is the same as in the original blade model. Therefore, similarity to the original blade vertical axis tidal turbine model, the velocity plot exhibits of 3.2 repetition which reflects the numbers of turbine revolution. The velocity profile repeats and is quite regular and less chaotic for each revolution. The time varying resultant incoming velocity and angle of attack for one rotation are plotted in Figure 5.9 and 5.10 respectively. Figure 5.19b shows that the main vortex shedding frequencies on the blunt blade are 0.313 Hz and 0.85 Hz. The 0.85 Hz frequency coincides with the heave natural frequency which causes a resonance effect on the blade. The vortex is amplified and a strong vortex induced vibration will happen. The vibration can be observed from the displacement at a quarter chord length point shown in Figure 5.22. The vortex shedding in the fluid flow regime is detailed and correlated with the pressure distribution in Figure 5.21. Figure 5.21 shows pressure distribution in four selected azimuth angles since the vortex does not change much during the turbine rotation. As predicted in the blunt foil response model, vortex generation is also not observed in the blunt blade vertical axis tidal turbine response fluid flow regime as shown in Figure 5.20. The vortex induced vibration is also predicted to be weaker than original blade vertical axis tidal turbine model.

In the first quarter of turbine revolution, vortex shedding frequency is initiated at 0° azimuth angle at the middle of the blade. The LEV initiation is also detected from the pressure distribution plot in Figure 5.21. In the pressure plot, two small peaks are found at \(x/c = 0.33\) and 0.55 where the LEV initiation is observed. The blade is at the midpoint of motion response as shown in Figure 5.22. From this position, the LEV
moves towards the trailing edge as seen in fluid flow regime image in Figure 5.20. The moving LEV can also be predicted by the receding peaks from the pressure distribution plot at 90° azimuth angle shown in Figure 5.21. The first flowing vortex is occurs at x/c = 0.36 and the second vortices has likely been detached from the surface.

The LEV is generated and flows from 0° to 90° azimuth angle. This is created by dynamic loading which induces vibration signified by fluctuating signals in the displacement plot. At 90°, both surfaces exhibit negative pressure which is found to be lower than at the previous position. At 180° azimuth angle, the LEV is fully detached from the blade upper surface which is indicated by the absence of peak pressure. The pressure magnitude also decreases which generates a lower total force acting on the blade. The blade moves to lower position. From this position, the blade rotates toward 270° azimuth angle with no vortex generation as presented in the pressure contour plot. The vortex is fully detached and the pressure increases. As a result, all surfaces experience positive pressure. The
positive pressure causes the blade to move upward. The wake is laminar and the vortex is weaker compared to the original blade. The laminar fluid flow regime is due to the trailing edge flat profile at which two sources of fluid pulse recirculation is created and shed alternatively as happened in the foil response model. The recirculation flows downstream and prevents the regime to form a reverse fluid flow.

From figure 5.22, it can be seen that the blunt vertical axis tidal turbine experiences a strong vibration response in the heave direction when it interacts with the fluid. The peak to peak amplitude of vibration reaches 0.3 m acting in the range of 51.2° to 203° azimuth angle. Compared to the original blade vertical axis tidal turbine, the blunt blade creates a higher vibration response amplitude in heave but with lower frequency.
The high vibration amplitude proves the prediction of vortex vibration which has almost the same frequency as the heave natural frequency as shown in Figure 5.19b.
5.3.2 Rounded Blade

The rounded blade vertical axis tidal turbine response model is developed using the same domain as the vertical axis tidal turbine using blunt blade. The phase averaged and PSD methods are also used to calculate the vortex velocity and shedding frequency. The reference velocity is measured at the leading edge and the original velocity is taken from a point at 0.1 m behind the trailing edge. The velocity of the reference, original and phase averaged signals are shown in Figure 5.23a and the result of vortex shedding frequency using the PSD method is illustrated in Figure 5.23b. The incoming fluid flow properties are the same as those used in the blunt and original vertical axis tidal turbine blade models as listed in Table 2.2. Since the foil is truncated by the same amount as the blunt blade, the $Re$ is the same as a value of $2.45 \times 10^6$.

(a)

Figure 5.23: a. Reference, original and phase averaged velocity signals for a vertical axis tidal turbine with a rounded blade model, b. frequency response for a blunt vertical axis tidal turbine model
The velocity signals in Figure 5.23 are showed for 3.2 cycles of the turbine revolution. The original signals are quite chaotic with a high amplitude found in each revolution. The high amplitude indicates strong vibrations acting on the blade. The vibration is also demonstrated by the frequency domain plot shown in Figure 5.22b. The main vortex shedding frequencies are at 0.313 Hz and 1.1 Hz which are near to the heave natural frequency. The vibration in the heave direction is strong as a consequence of synchronization with the heave natural frequency which is marked by a high displacement as shown in Figure 5.25. The heave lock-in vibration is caused by vortex shedding as shown in Figure 5.24. The vortex shedding is predicted from the pressure distribution over the surface as plotted in Figure 5.25.

Figure 5.24: Fluid flow regime for one cycle of a vertical axis tidal turbine model using a rounded blade
The rounded blade vertical axis turbine displacement in Figure 5.26 is similar to the blunt blade pattern as shown in Figure 5.22. However the fluid flow velocity in rounded bladed is found much higher. At $0^\circ$ azimuth angle, a vortex is formed on the upper surface leading edge at $x/c = 0.25$ and 0.6, indicated by pressure peaks in the contour plot. At the trailing edge of the blade upper surface, the pressure is higher than on the lower surface and force fluctuation along the chordwise of the blade. The fluctuating forces creates fluid induced vibration on the blade which can be detected from the blade displacement shown in Figure 5.26. As the blade rotates to $90^\circ$ azimuth angle, the vortex shedding increases and flows to the trailing edge as shown in the receding peak to $x/c = 0.75$. The vortex shedding is also stronger which is signified by larger peak than at $0^\circ$ azimuth angle. At this state, both blade surfaces have very low pressure compared to the pressure at $0^\circ$. The lower pressure lowers the lift forces and moves the blade in $y$ direction to 0.05 m downwards from equilibrium position as indicated in the displacement plot.

In the second quadrant of turbine revolution, the vortex detaches from the trailing edge and flows to the freestream. This is caused by the higher pressure over the blade surface. However the pressure magnitude at both the upper and lower blade surface is reduced and this creates a lower lift force which causes the blade to rotate to its minimum position. Small vibrations exist when the blade reaches approximately $180^\circ$ as a result of vortex detachment.

From $180^\circ$ to $270^\circ$ azimuth angle, the flow is laminar. This is distinguished by a fully detached vortex from the trailing edge region. At $270^\circ$ the pressure on the entire lower and upper surfaces are positive which indicates that both surfaces becomes pressure surfaces. The higher pressure creates higher lift force and moves the blade to higher position. The laminar fluid flow regime yields to no fluctuation in the displacement. The blade moves from its minimum to its maximum position between $210^\circ$ and $360^\circ$ azimuth angle without any fluctuating motion as shown in Figure 5.26.

The rounded blade has a similar displacement pattern to the blunt blade vertical axis tidal turbine although with a higher amplitude. The vibration displacement for the rounded blade is found to be 0.45 m. This is likely caused by a higher power spectrum and synchronized vortex shedding frequency. Although the pitch motion is not dominant, the vortex is generated by the strong heave vibration as stated by Khalid et al. (2014) in Figure 2.16. He showed that the effect of plunging motion on vortex shedding is equal to pitch motion. He also derived an equal angle formula as written in Equation (2.16) for both motions when those motions have the same configuration fluid flow regime. Thus the rounded blade gives rise to a more chaotic fluid flow regime than the blunt blade vertical axis tidal turbine, however both geometries have the same pattern of vibration.
Figure 5.25: Pressure distribution for vertical axis tidal turbine model with a rounded blade at azimuth angle of: 0°, 90°, 180°, 270°

Figure 5.26: Time history displacement for the blunt blade response with selected azimuth angle positions sampled at a position of one quarter chord length

5.3.3 Sharp Blade

The sharp blade is used for the vertical axis tidal turbine and the model is developed using the same domain as in the sharp foil response. The Phase Averaged and PSD methods are used to find the vortex shedding velocity and frequency. The reference, original and phase averaged velocity signals are shown in Figure 5.27a and the vortex shedding frequency is shown in Figure 5.27b.
The sharp blade vertical axis tidal turbine velocity signals are much more chaotic than the other geometry. This reflects a chaotic fluid flow regime. The vortex shedding frequency plot shows that the dominant frequency at 0.31 Hz which lies far from the heave or pitch natural frequencies. Some small peaks are also found at higher frequencies. These small peaks indicate that the vortex is shed at some frequencies and produces a distracted fluid flow regime as illustrated in Figure 5.28. This fluid flow regime is also supported by the pressure distribution over the blade as depicted in Figure 5.29. The chaotic fluid flow causes the blade to vibrate and produce a fluctuating response as shown in Figure 5.30. Although there are no dominant frequencies which are close to the blade natural frequency, the power spectrum for the dominant frequency is high. This power also contributes to the frequent the vibration.
The strong LEV starts to form at $0^\circ$ azimuth angle determined from three peaks at the upper front part of the blade shown in Figure 5.29. The upper surface of the blade is subjected to a small suction pressure which induces the LEV. However this condition does not occur over the entire upper surface. The pressure fluctuates as the mid part of the upper surface is shown to experience a positive pressure. The detached vortex is also detected at this point. This is characterized by a larger peak at the trailing edge. From $0^\circ$ angle, the LEV continues to flow downstream as shown by the pressure peaks moving toward the trailing edge at $90^\circ$ azimuth angle. A vortex also forms at the lower surface of the blade as signified by a pressure peak approximately at $x/c = 0.3$. The pressure on the lower surface is very irregular signified by a disordered pressure signal on the entire lower surface. The irregularity identifies the chaotic fluid flow regime which happens...
on the lower blade. From 90° azimuth angle onward the vortices starts to detach from both surfaces until 180° azimuth angle. While the vortices detach from trailing edge, the LEV also starts to form on both surface at the front part of the blade. Overall the fluid flow regime is laminar with low pressure on both surfaces. The blade rotates to 270° azimuth angle where both surfaces pressure become positive. The LEV creation continues with the vortices moves downstream toward the trailing edge. In general the wide scattering of LEV on both surfaces indicates that the vortices propagates with a wide range of shedding frequencies. This is shown by different peaks at the frequency domain plot. From 270° angle, the vortices become stronger as the blades rotates back to 0°.

Figure 5.29: Pressure distribution for vertical axis tidal turbine model with a sharp blade at azimuth angle of: 0°, 90°, 180°, 270°

The vortices generated during a turbine rotation induce vibrations on the blade which can be characterized by the turbine displacement. The displacement measured at a quarter chord length position on the sharp blade vertical axis tidal turbine is shown in Figure 5.30. This figure shows that sharp blade vibrations is fluctuated more than the blunt or rounded blade vertical axis turbine. The fluctuating vibration occurs due to the vortex generation which happens during the turbine revolution. However the rounded and the blunt blade turbine vibration has higher amplitude than the sharp or the blunt blade due to the proximity of those two blades dominant frequencies to the blades natural frequency. The maximum displacement is found to be occurs when the turbine revolves between 46° and 66°.

The lift, drag and moment force of normal and all modified trailing edge shapes are shown in Figures 5.31, 5.32, and 5.33 respectively. The lift coefficient for the original, blunt, rounded and sharp blade profile are shown in Figure 5.31. For fifteen seconds simulation time with an angular velocity of 1.71 rad/s, the blades rotate by approximately 4.08 revolutions. The blade modifications increase the lift force by approximately 3.5 times
from the original blade lift coefficient value. This confirms that the modification to the blade trailing edge gives a benefit to a vertical axis tidal turbine by improving its lift performance. However for sharp blade vertical axis tidal turbine, the lift force increase is accompanied by highly fluctuating condition. The sharp blade has the highest amplitude and vibrates more vigorously. The lift coefficient plot for the blunt and the rounded blades confirm that the heave vibration observed for the blunt blade is less than the sharp or original blade. This is shown in Figures 5.22 and 5.26. The rounded and blunt blade lift coefficients have similar trend as also shown in the displacement graph.
Chapter 5 Fluid Structure Interaction of a Vertical Axis Tidal Turbine Blade

The fluctuations demonstrate the effect of vortex shedding during the turbine rotation which contribute to vortex induced vibration. Starting from $0^\circ$ angle the lift coefficient fluctuation happens as a result of LEV initiation. The fluctuation continues to $90^\circ$ while the vortex moves downstream towards trailing edge. During the second quarter of revolution the lift force reaches its minimum with less fluctuation. The vortex is detaches from trailing edge and the fluid flow regime is laminar. The blades rotates to $270^\circ$ and continues to $360^\circ$ with vortex disturbances found in the fluid flow regime which induce fluctuations for this part of the cycle.

![Drag coefficient for a vertical axis tidal turbine with modified blade profiles](image)

Figure 5.32: Drag coefficient for a vertical axis tidal turbine with modified blade profiles

The lift coefficient for the sharp blade profile has higher amplitude fluctuations comparing to the original, blunt and the rounded blade. The fluctuations exist for almost the entire turbine revolution. The lift coefficient results agree with the result from the vortex shedding in the fluid flow regime which is shown the fluctuating signals presence in the first and fourth quarter of the turbine revolution. The higher amplitude and frequency experienced by the sharp blade likely cause increased fluid dynamic loading which is more harmful for the turbine structure. The high dynamic loading observed causes detrimental effects such as fatigue failure. This is more damaging than the high lift force obtained in the original, blunt and rounded blade vertical axis tidal turbine. However an experiment should be conducted to confirm this result and assess the intensity of the dynamic loading and the damage it can cause to the vertical axis tidal turbine.

Figure 5.32 shows the drag coefficient for the original, blunt, rounded and sharp blade vertical axis tidal turbine. Similarly to the lift force result, the drag force for the modified foil is higher with higher frequency fluctuations compared to the original blade vertical axis tidal turbine. The fluctuations observed between $0^\circ$ to $90^\circ$ azimuth angle are due to vortex formation. The vortex formation increase the viscous effect in the fluid flow.
regime. This increases the drag force. This situation agrees with the fluid flow regime for all the modified trailing edge profiles. From 90° to 270°, the drag coefficient is reduced. This indicates the lack of vortex generation and a laminar fluid flow regime. From 270° onward, the drag force increases slightly and is more fluctuated due to the occurrence of vortex shedding. However within this period, the blunt blade turbine drag force has the lowest frequency fluctuations which implies that the truncated shape can reduce drag force fluctuations. The sharp blade drag force is also found to fluctuate strongly which indicates that the vibration response in surge direction is quite strong and chaotic. The surge vibration is represented by the displacement in x direction (pointDisplacement (0)) for all the blades shown in Figures 5.17, 5.22, 5.26, and 5.30. From the drag force analysis, the sharp blade is not recommended to be implemented in vertical axis tidal turbine blades since the vibration response is enhanced. The increasing drag force should be taken into account when designing a vertical axis tidal turbine because the efficiency and performance of the tidal turbine can also be reduced. The last force coefficient to be analyzed is moment coefficient which is shown in Figure 5.33.

Figure 5.33 shows the moment coefficient for the original and modified blade vertical axis tidal turbines. Similarly to the lift and drag coefficient, the moment coefficients for the modified blades are higher than the original blade turbine. The frequency of the moment coefficient fluctuations for all blades are similar although the sharp blade has a higher amplitude. This result shows that the pitch response (torsional vibration response) which is driven mostly by the moment force is similar for all blades. This also suggests that the pitch response is not dominant when compared to the heave vibration response which was presented in the frequency domain plot for all the blades on Figures 5.11b, 5.19b, 5.23b and 5.27b.
The result for moment coefficient confirms the result of lift and drag force observation which shows that the sharp blade has a highly fluctuating response and cannot reduce a vortex induced vibration. Thus the sharp profile is not suitable for a vertical axis tidal turbine application. The blunt blade is more favourable to be used than a rounded blade for vertical axis tidal turbine application. The disadvantage of rounded blade turbine comes from the stronger vibration which is represented by higher heave displacement.

5.4 Summary

The original and modified NACA 0012 are utilized for vertical axis tidal turbine blades employing the equivalent incoming fluid velocity method. The response is investigated by identifying the vortex shedding frequencies and further analysis to determine if any of the frequencies induce the locked-in condition. The vortex shedding frequency results are listed in Table 5.5.

The modified trailing edge blades control the vortex shedding frequency of a vertical axis tidal turbine and reduce the number of frequencies peaks. The first frequency peak for the turbine modified blades are identical thus the frequency is independent of the trailing edge profiles. Although the number of main frequencies are reduced, one of the values is closer to the heave natural frequency for blunt and rounded blades. This is a concern to be considered in the vertical axis tidal turbine design as a resonance effect is potential to be created. However the resonance effect can be minimized by choosing a less stiff material or installing the turbine in a medium ocean with 0.65 m/s or less tidal velocity as predicted in the foil response model.

Table 5.5: Vortex shedding frequency for a vertical axis tidal turbine with modified blade geometries with the associated natural frequency. (H): lock-in by heave natural frequency

<table>
<thead>
<tr>
<th>Model</th>
<th>Main Frequency (Hz)</th>
<th>$f_n$ (Hz)</th>
<th>$f_h$ (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.234 0.55 0.78 1.095</td>
<td>0.9425</td>
<td>1.4758</td>
</tr>
<tr>
<td>Blunt</td>
<td>0.313 0.85(H) - -</td>
<td>- -</td>
<td>- -</td>
</tr>
<tr>
<td>Rounded</td>
<td>0.3136 1.1(H) - -</td>
<td>- -</td>
<td>- -</td>
</tr>
<tr>
<td>Sharp</td>
<td>0.31 - - -</td>
<td>- -</td>
<td>- -</td>
</tr>
</tbody>
</table>

Modifying the blade can also magnify the force coefficients for a vertical axis tidal turbine. The lift coefficient increases significantly by up to 350%. However for the rounded and sharp cases, the increasing lift coefficient gives rise to higher frequencies. For the drag coefficient, again, the rounded and sharp trailing edge foils produce higher value with highly fluctuating signals. The blunt vertical axis tidal turbine experiences a higher drag force than the original foil but with less fluctuations. Similarly to the lift and drag coefficient, the moment coefficient trend shows that the rounded and sharp profiles
have the highest amplitude although all trailing edge shapes have similar mean moment coefficient value of approximately zero. The force coefficient is strongly related to the vertical turbine vibration response which can be identified from the point displacement. The maximum displacement for four types of vertical axis tidal turbines are listed in Table 5.6. The maximum displacement is sampled at a quarter chord length point from leading edge.

Table 5.6: Total displacement for four vertical turbine blades profiles

<table>
<thead>
<tr>
<th>Maximum displacement (m)</th>
<th>Original</th>
<th>Blunt</th>
<th>Rounded</th>
<th>Sharp</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.111</td>
<td>0.416</td>
<td>0.422</td>
<td>0.573</td>
</tr>
</tbody>
</table>

From Table 5.6, it can be seen that the largest displacement occurs in the vertical axis tidal turbine with sharp blades. Although the blunt and rounded blades give larger displacement than the original blade, the displacement fluctuates less. This indicates that the blunt and rounded vertical axis turbine have less frequent vibration but with higher amplitude. The high amplitude vibration of the blunt and rounded blades is caused by the vortex shedding frequency occurring simultaneously with the heave natural frequency. This effect can be reduced by choosing an appropriate material for the turbine blade.

The lock-in condition is determined from the vibration main frequencies created by the interaction with the tides. This fluid loads is detected from the pressure distribution which affects the fluid flow regime over one turbine revolution. The pressure distribution on inner and outer surface of the blade which influences the vortex generation in the fluid regime, is plotted over one turbine rotation in twelve selected azimuth angles. The obtained lock-in condition is then verified by the blade displacement which is recorded during one turbine revolution.

For a vertical axis tidal turbine with the original blade, it is found that the high amplitude vibration response occurs at the second quadrant of the turbine revolution. However the more fluctuated vibration which is likely to generate a resonance or lock-in condition occurs in the first quadrant which is induced by the strong vortex shedding found in the chaotic fluid regime in that section. Unlike a vertical axis tidal turbine with the original blade, a vertical axis tidal turbine with a truncated blade does not experience lock-in condition as identified in the blade displacement. The truncated blade movement plot does not detect high fluctuated displacement.

A vertical axis tidal turbine with a rounded blade experiences a lock-in frequency which is likely to occur during second and third quadrant. Small fluctuated displacements also found in the first quadrant but apparently it does not produce a resonance. For a vertical axis tidal turbine with a sharp blade, a resonance is found in all quadrants although the sharp blade displacement is less than in the rounded blade.
Chapter 6

Conclusions and Recommendation for Future Work

6.1 Conclusion

The dynamic structural response of a three bladed NACA 0012 vertical axis tidal turbine is investigated numerically in this research. The response which analyses the interaction of a vertical axis tidal turbine blade and the tide is modeled by a vibration system of up to three degrees of freedom (pitch, surge, and heave) with a spring damper component modeling the time domain response representing the effect of the supporting structure. A 2D model of a single foil which represents a turbine blade in rotational operation is developed in a structured rectangular grid domain using k-ω SST turbulence model and is simulated using OpenFOAM 2.2. The blade is constructed using SolidWorks 2014 and imported into the OpenFOAM 2.2 domain. The domain mesh is refined using the snappyHexMesh utility.

A method to control the vibration of a vertical axis tidal turbine blade is proposed in this research by introducing trailing edge modifications on the blade. These modifications are assigned at a section with 15% of chord length distance from the blade’s trailing edge. The modifications include foil truncation, sharp and rounded profiles. The model has the $Re$ of $3.07 \times 10^6$ based on a 0.75 m chord length for the original blade and the $Re$ of $2.54 \times 10^6$ for a 0.64 m chord length for the modified trailing edge blades. The turbine solidity and tip speed ratio is 0.1829 and 5.1 respectively. The tidal velocity is assumed to be uniform in the spanwise direction and has a constant magnitude of 0.65 m/s (Tillinger, 2011). The blade’s vibration is a response caused by the interaction between the unsteady incoming flow and the blade. The unsteady inflow is due to the time dependent angle of attack and incoming flow velocity magnitude which represent the
turbine rotation. The incoming flow velocity is a resultant velocity which is calculated from the tidal and the turbine angular velocity. All vibrations are considered as first mode vibrations. The blade is designed to be manufactured from a polymer material which is constructed from sixty percent of e-glass and forty percent of epoxy by volume.

Preliminary works are conducted prior to the investigations of the vertical axis tidal turbine blade response models. The preliminary work is required to ensure the simulation domain including mesh topology and selected boundary conditions yield accurate results in a reasonable execution time. The domain is constructed in a structured rectangular grid and is simulated using k-ω SST turbulence model. The preliminary work applies a steady incoming fluid velocity and analyse for fifteen seconds simulation time. The work consists of developing nine static angles of attack models for the original and modified foils using the pimpleFoam solver. The lift and drag coefficients for the original foil model are validated using Abbott and Von Doenhoff’s (1959) and Eleni et al.’s (2012) results. The purpose of the validation is to ascertain that the CFD design parameters resolve the fluid flow condition accurately. The analysis of the static angle of attack models is followed by the oscillating foil investigation using a dynamic mesh configuration and the pimpleDyMFoam solver. The oscillating foil is validated using Frederich et al.’s (2009) numerical results.

Initially two domain topologies (the rectangular and c-structured grids) are designed to handle the fluid flow condition as a result of the fluid blade interaction. The c-structured grid is designed to have more control for adjusting the domain mesh thus it has more number of cells than the rectangular grid. This strongly affects to dynamic foil model’s execution time. The rectangular grid is further selected in the oscillating foil model and the response model for both using original and modified foils. The rectangular grid is also employed to model a vertical axis tidal turbine blade response by using Periodic Inflow Equivalence method to replace the turbine’s rotation.

Preliminary works in static and oscillating foils models are in agreement with the result from the reference data. This means that the mesh and topology grid can resolve the fluid flow condition around the foil. For the original foil model, a maximum lift coefficient discrepancy is found to be 8.5% from Abbott and Von Doenhoff’s experimental data and 19.35% from Eleni et al.’s simulation results. The drag coefficient displays a much higher discrepancy due to the mesh resolution which is not able to fully resolve the viscous effect in the fluid flow regime close to the foil’s surface. It is suggested that the mesh should be modified and that the boundary layer should also be refined. The future modification and the refinement are expected to create the y+ value in the viscous sub layer region (y+<1) in order to resolve the viscous effect. The domain is further adopted to the static angle of attack models using modified foils and found that the blunt foil has the highest lift coefficient.
The domain is employed to the dynamic foil models using the original and the three modified foils. However, the refined mesh is not easily incorporated into the dynamic mesh model because the execution time is very sensitive and is significantly affected. A refinement in the dynamic mesh case takes a longer execution time. Thus, a second validation is required to ensure that the selected mesh resolution without any further refinement are suitable for the dynamic mesh used in the foil and blade response model. The dynamic mesh validation is performed for the oscillating foil model which is adopted from Frederich et al.’s model (2009). The lift coefficient is recorded at angles below 16°; the results concur with the lift coefficient from Frederich et al.’s model. The agreement with the reference data confirms the robustness of the domain to resolve turbulence and fluid flow equations for a dynamic model. The mesh is further utilized in a vertical axis tidal turbine blade response model.

A foil response model is developed to identify the effect of each various parameter which influences the actual condition of a vertical tidal turbine operation. Five predominant parameters are selected and these are the tidal velocity magnitude, blade initial angle of attack, response orientation, stiffness constant and damping constant. The Phase Averaged Method and the PSD method are applied to the velocity magnitude of two points at leading and trailing edges. The results examine the main frequencies of the foil response vibration and predict the existence of lock-in condition in the response. All parameters are varied according to data shown in Table 2.3. It is observed that the response becomes stronger with the increase of tidal velocity magnitude, non-zero initial angle of attack, lone pitch response, the increase of spring constant and damping constant. From the foil response’s result, the turbine operation for a vertical axis tidal turbine blade’s response is selected as shown in Table 5.1. This further becomes standard operational parameters implemented in the vertical axis tidal turbine blade response model using the original and modified blades. This blade response model assigns the Periodic Inflow Equivalence method to model the turbine rotation. Applying this method, the inlet velocity entering the domain is the resultant velocity obtained from the freestream velocity and the tangential velocity.

A vertical axis tidal turbine blade response is developed in the dynamic mesh model domain using the original and modified blades. The original blade model’s lift and drag coefficients are qualitatively verified using Almohammadi et al.’s vertical axis tidal turbine model (2015). Almohammadi et al. developed a rotating three bladed vertical axis tidal turbine model. They used rotational frame of reference in their model. Except for the frame of reference, the Re, TSR and blade thickness of their model are also different from the blade response model. Therefore, identical results are not possible to achieve and the verification process is taken qualitatively. By introducing the Re and TSR variations in the verification process, it can be identified if the blade response’s result has the reference trends of lift and drag coefficients. The variations of the Re and TSR are shown in Table 5.4. From the results, it is observed that the lift and
drag coefficients contour of the original blade model are qualitatively satisfied with Almohammadi et al.'s model. The $Re$ variation affects a little to the lift and drag coefficients profile. This stage is also to confirm that the designed response model is robust to be used further in the modified trailing edge blade response models. The modified trailing edge profiles applied to a vertical axis tidal turbine blade are blunt, sharp and rounded. From the response model results, it can be observed that the blunt trailing edge is recommended to be applied in a vertical axis tidal turbine blade to reduce turbine’s vibration. This is contrast with the sharp blade which produce higher frequency vibration and thus should be avoided.

From the result it can be seen that one of key findings in this research is the prediction of fluid behaviour at all positions on the turbine rotation in all models as depicted in Figures 5.12, 5.20, 5.24 and 5.28. The result is of importance for industry as an information to design, to install and to operate the turbine efficiently. The data can be used to design a controlling system to point the blade in a certain direction to avoid turbulence when it reaches a position with a strong interaction. Other key findings which are also beneficial for the renewable energy technology are the prediction of lock-in condition and the vibration history as a result of the turbine blade response. The lock-in condition result is significant for a blade designer to achieve a turbine design with lower fatigue failure risk. A turbine with less fatigue failure increases the turbine's lifetime.

The successful process in modeling the vertical axis tidal turbine blade response in this research lies on the application of the Periodic Inflow Equivalence Method. The methodology has successfully adapted a rotating vertical axis tidal turbine motion to the time varying tidal inflow velocity magnitude and angle of attack. As a result, a single blade can be modeled solely in which creates a less complex CFD domain mesh. The number of cells composed in the model with the same final mesh resolution are also reduced. This will help to run the model with a less computational cost and execution times. The better performance turbine with less vibration and noise can be modeled more reliably.

### 6.2 Future Work

Future work is suggested to improve the current vertical axis blade response’s result. From the CFD perspective, the selected domain might be improved by refining the mesh or by constructing a different topology to obtain a boundary layer which will produce $y+$ value in the viscous sublayers region. The requirement of $y+ > 1$ is needed to solve the viscous effect at the innermost cell layer. With such boundary layers, the model is expected to produce more accurate force coefficients, especially the drag coefficient result. An independent mesh analysis should also be conducted after the refinement.
process to achieve a compromise aspect between the number of cells and the execution time.

From the fluid dynamics perspective, the inlet velocity should model and reflect an actual tidal wave velocity variation. This will affect the unsteadiness incoming fluid velocity entering the CFD domain. In this project the unsteadiness of the incoming velocity comes from the time varying angle of attack and the resultant velocity magnitude variation which is calculated from a constant tidal velocity and the turbine's angular velocity. In this work, the periodic inflow equivalence method is used to model the incoming tidal current. The approach approximates the actual vertical axis tidal turbine operation. However by applying a tidal wave model, the incoming velocity entering the domain will be more reliable and closer to actual tidal conditions. Thus the model is expected to give more accurate results.

The unsteady incoming flow which is employed in this work is not the only turbulence source experienced by a turbine blade. Other source of turbulence experienced by a rotating vertical axis tidal turbine blade also comes from the near wake of the front blade. During the turbine rotation, the near wake from the front blade affects the fluid flow regime experienced by the next blade. This turbulence effect is also necessary to be modeled in the incoming fluid flow entering the CFD domain.

In the turbine design point of view, the use of asymmetric blades can be further considered to be utilized in a vertical axis tidal turbine. An asymmetric blade provides a higher lift force than a symmetric blade although it also tends to create a higher drag force. The drag force can be controlled by modifying the blade trailing edge shape or assuring that the asymmetric vertical axis turbine operates at high tips speed ratio. In this case the range of the blade’s angle of attack is below a stall angle. Another blade geometry is also proposed to improve the blunt vertical axis turbine performance as suggested by Zobeiri et al. (2012). This profile is an improvement of a symmetric blade by constructing an asymmetric trailing edge shape on the blade. The asymmetric trailing edge profile on a symmetric blade should reduce the vortex shedding by moderating the vorticity generation.

There are not so many experimental data existing to validate the foil and the vertical axis tidal turbine blade response. The vertical axis turbine blade response experiment can be a promising stage in this area to gain better knowledge and to improve the response model.
Appendix A

OpenFOAM codes

A.1 U for Vertical Axis Turbine Blade Response

/*---------------------------------------*- C++ -*--------------------------------------*
---

/F ield — OpenFOAM: The Open Source CFD Toolbox — —
/O peration — Version: 2.2.2 — —
/A nd — Web: www.OpenFOAM.org — —
/M anipulation — — —————————————————————————*/ Foam-

File version 2.0; format ascii; class volVectorField; location "0"; object U; // * * * *
* * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *

#include "include/initialConditions"

dimensions [0 1 -1 0 0 0 0];

internalField uniform $flowVelocity;

boundaryField inlet type groovyBC; value uniform (4.0016 0 0);//resultant velocity of incoming fluid and tangential velocity, calculated in dropbox/postProcessing/azimuth.xlsx file //variables "u=0.656;tanVel=3.3456;angVel=1.709;beta=pow(pow(u,2)
+pow(tanVel,2)+2*u*tanVel*cos(angVel*time()),0.5);azimuth=angVel*time();
w=atan(pow((5.1*sin (angVel*time())),-1)+ pow(tan(angVel*time())),-1))+(angVel*time())-
p/2;alfax=mag(beta*cos(atan(pow((5.1*sin(angVel*time())),-1)+ pow(tan(angVel*time())),-1))+(angVel*time())-pi/2);alfay=beta*sin(atan(pow((5.1*sin (angVel*time())),-1))+(angVel*time())-pi/2);angle=azimuth

variables
"u = 0.656; tanVel = 3.3456; angVel = 1.709; beta = pow(pow(u, 2) + pow(tanVel, 2) +
2 * u * tanVel * cos(angVel * time()), 0.5); alfax = mag(beta * cos(atan(pow((5.1 *
sin(angVel * time())), -1)) + pow(tan(angVel * time()), -1)) + (angVel * time()) -
pi/2)); alfay = beta * sin(atan(pow((5.1 * sin(angVel * time())), -1) + pow(tan(angVel * time()), -1) + (angVel * time()) - pi/2); azimuth = angVel * time(); angle = azimuth valueExpression
"vector (alfax, alfa, 0)"; // modulo in openFoam is much different from original definition

outlet type inletOutlet; inletValue uniform (0 0 0); value $internalField;

wing type movingWallVelocity; value uniform (0 0 0);

#include "include/frontBackTopBottomPatches"

// ************************************************************************* //

A.2 pointDisplacement for Vertical Axis Tidal Turbine Response

/*———————————————————*- C++ -*———————————————————* — ========= —
— —
/ F ield — OpenFOAM: The Open Source CFD Toolbox — —
/ O peration — Version: 2.2.2 — —
/ A nd — Web: www.OpenFOAM.org — —
/ M anipulation — — —————————————————————————*/ FoamFile version 2.0; format ascii; class pointVectorField; location "0.01"; object pointDisplacement; /* ****************************************** */

dimensions [0 1 0 0 0 0];

internalField uniform (0 0 0);

boundaryField

wing

type sixDoFRigidBodyDisplacement;

mass 16.73;

centreOfMass (0.3355 0 0.1025);

momentOfInertia (0.0564 0.8089 0.7543);

orientation

( 1 0 0 0 1 0 0 0 1 );
velocity (0 0 0);
acceleration (0 0 0);
angularMomentum (0 0 0); torque (0 0 0); rhoName rhoInf; rhoInf 1025; g (0 -9.81 0);
report on; constraints maxIterations 500;

fixedLine1 sixDoFRigidBodyMotionConstraint fixedLine; tolerance 1e-9; relaxationFactor 0.7; fixedLineCoeffs refPoint (0.18625 0 0.1025); direction (0 1 0);

fixedAxis1 sixDoFRigidBodyMotionConstraint fixedAxis; tolerance 1e-06; relaxationFactor 0.7; fixedAxisCoeffs axis (0 0 1); restraints verticalSpring sixDoFRigidBodyMotionRestraint linearSpring;

linearSpringCoeffs anchor (0.18625 -0.1 0.1025); refAttachmentPt (0.18625 0 0.1025); stiffness 1000; damping 2; restLength -0.1; axialSpring sixDoFRigidBodyMotionRestraint sphericalAngularSpring; sphericalAngularSpringCoeffs axis (0 0 1); stiffness 200; damping 0.2; referenceOrientation $orientation; value uniform (0 0 0);

front type empty;
back type empty;

"**" type fixedValue; value uniform (0 0 0);

// *************************************************************************/
References


REFERENCES


REFERENCES


REFERENCES


REFERENCES


Tabassian, R. (06-12-2013). Torsional vibration analysis of shafts based on adomian decomposition method.


REFERENCES


