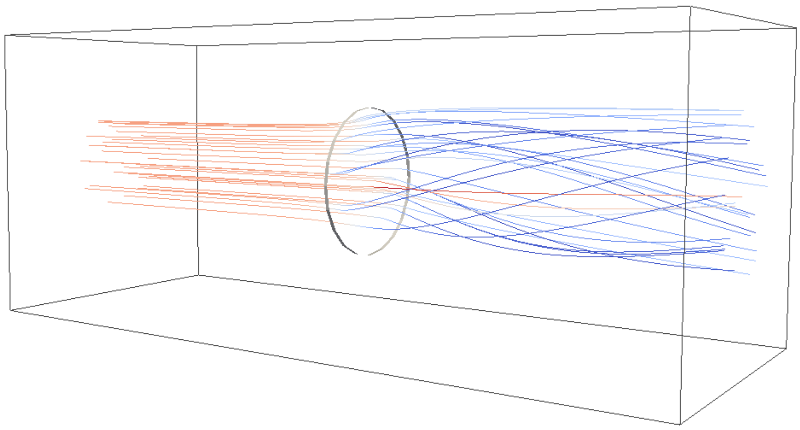
Design Optimisation of Floating Offshore Wind Turbines



First year report

Rachael Smith

rs495@exeter.ac.uk

May 2018

Lead Supervisor:	Prof Lars Johanning
Co-Supervisor:	Prof Gavin Tabor
Co-Supervisor:	Dr Ajit Pillai
Co-Supervisor:	Dr Philipp R. Thies

Table of Contents

List of Figures

1 Overview

Offshore wind offers high potential for energy production due to higher average speeds and lower turbulence than in wind over land, and floating technologies enable the installation of wind farms in water depths of up to 900m [1]. However, floating offshore wind turbines (FOWTs) present unique challenges in terms of their design due to the combined wind and wave loading and resulting platform motions to which they are subjected. This leads to increased fatigue and unsteady aerodynamic behaviour. To reduce the overall cost of energy, floating wind turbines must be designed for maximum power production in addition to minimum fatigue loads in order to maximise their operational lifetime.

In addition to maximising power output and minimising overall cost of energy in normal conditions, FOWTs must also be able to withstand extreme conditions. Storm and typhoon conditions can mean high wind speeds, large turbulence intensity, varying wave heights and rapid changes in wind direction which may result in very large responses and possible failure of the turbine, particularly in wind speeds just below the cut out speed after which the turbine changes to parked mode and therefore reduces the aerodynamic loads [2].

This project aims to develop a computational model of a floating wind turbine that accurately predicts aerodynamic performance in normal and extreme environmental conditions, with a view to optimising the design of the turbine rotor. In addition, the developed model will include a methodology for modelling morphing blades with the aim of assessing the suitability of this concept installed on a FOWT rotor. The current report presents a review of the relevant theory and literature, a description of the work currently in progress, and an outline of a research plan detailing future work to be completed.

2 Background and Literature Review

In order to accurately predict the performance of a floating wind turbine, a high fidelity model of its operating environment and resultant behaviour must be developed. In particular, this research focuses on the aerodynamic behaviour of a wind turbine installed on a floating structure, and so the behaviour of wind flow interacting with the turbine must be understood.

2.1 Challenges Specific to Floating Wind Turbines

The differences in the aerodynamics of a floating wind turbine and a bottom fixed offshore or land-based turbine can be quite significant. The floating platform is subjected to six degrees of freedom of motion: three translational modes (surge, sway and heave) and three rotational modes (pitch, roll and yaw) which result in a time varying angle of attack and therefore loading at the rotor [3]. Surge and pitch motions add a component to the flow speed at the rotor, leading to a time varying relative wind speed, and the rotor is also more likely to be misaligned with the flow due to yaw and pitch. A floating wind turbine can also experience aerodynamic effects such as dynamic inflow, where the delayed response of a rotor to a change in loading conditions leads to an overshoot in thrust, or vortex ring state, where the rotor interacts with its own turbulent wake due to surge or pitch motions and can act as a propeller.

The aerodynamics of the rotor also affect the platform motions. For example, the platform pitch motion is directly influenced by the rotor thrust, which often results in a mean pitch inclination of the platform and therefore the turbine. Therefore, the coupling between aerodynamics and hydrodynamics will need to be considered in a simulation and analysis of a FOWT system. The magnitude and aerodynamic significance of platform motions is also partly dependent on the type of floating structure used. Sebastian [4] found that the platform modes most likely to lead to unsteady loading are pitch and surge for a TLP, pitch and yaw for a spar-buoy, and pitch for a semi-submersible.

Butterfield et al. [5] identified a number of key challenges specific to FOWTs that must be considered in the design and engineering stages. Four major issues with the design of the turbine itself are the effect of turbine weight on the buoyancy tank requirements, the tower top motion due to platform motion, the use of controls to limit the turbine response to wave loading, and the effect of the tower tilt angle on loading of other components. The motions of the platform will affect the design requirements and therefore cost of the turbine, so the design must be robust enough to accommodate additional displacements. It is suggested that a flexible rotor design may be able to accommodate these extra dynamic loads more easily than rigid designs, which may also help to reduce the cost.

Multon [6] also specified challenges in the design of FOWTs, including the additional fatigue induced by the forces of inertia due to the structure being continuously subjected to movements, and the extra fluctuation in the relative wind speed due to the swaying movement of the nacelle. In addition, the natural frequencies of roll and pitch motions must not resonate with the period of the waves. Misaligned wind and waves also have a significant effect on loading. Stewart and Lackner [1] demonstrated that misaligned wind and waves can drastically increase loading in the side-side direction and reduce the effectiveness of load reduction methods.

FOWTs are on average subjected to higher loading due to the additional wave and hydrodynamic loads than their onshore equivalents. A turbine must be able to withstand normal and extreme loads in order to prevent failure of components. Ma et al. [2] used the aero-hydro-servo-elastic code FAST to study a FOWT in an extreme typhoon event. It was found that the typhoon conditions could cause a blade pitch angle error in the control system, which could result in large extreme responses of the rotor thrust and power that the control system is unable to reduce.

2.2 Analysis of Wind Turbine Aerodynamics

The most widely used method of modelling wind turbine aerodynamics is the blade element momentum (BEM) method, due its high efficiency and ability to give generally good results for steady state loading. However, the method may not be reliable when applied to floating wind turbine cases, since the nature of floating structures means they are more frequently subjected to unsteady loading. Floating wind turbines also experience more complex flow conditions that BEM theory does not take into account. Micallef and Sant [7] found that analysis using BEM was unable to show the scale of power and thrust variations due to surge motion at high tip speed ratios, and therefore suggested that BEM should be used only if the mean quantities are required. Duan and Kajiwara [8] found that BEM does not take into account the time lag between platform motion and the rotor power response while the finite volume method inherently includes this, therefore suggesting that alternative methods to BEM should be used in cases where platform motions will have a larger influence on aerodynamics such as in low wind speeds.

For this reason, the use of computational fluid dynamics (CFD) for simulation and analysis of floating wind turbine aerodynamics is becoming increasingly popular since it allows for the modelling of floating platform motions and their impact on floating wind turbine aerodynamics that cannot be captured by simpler methods.

2.2.1 CFD Modelling

CFD methods involve analysing and simulating a fluid flow by finding a solution to the governing equations of fluid dynamics, known as the Navier-Stokes equations. The flow parameters and load distribution can then be calculated. CFD methods become useful when solving particularly complex fluid flow problems, such as flow separation or wake interaction that will be present in floating wind turbine simulations.

Fluid flows are governed by the fundamental physical laws of conservation of mass, momentum and energy in a fluid flow [9]. The equations presented here are for a Newtonian fluid, which is assumed to obey Newton's law of viscosity where the viscous stresses are proportional to the rate of deformation. The laws of conservation of mass and momentum are expressed using the Navier-Stokes equations:

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \mathbf{u} = 0 \tag{2-1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla . \rho \mathbf{u}\mathbf{u} = -\frac{1}{\rho}\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{F} \tag{2-2}$$

Where $\mathbf{u}$ is the fluid velocity vector, ρ is the density, p is the fluid pressure, μ is the dynamic viscosity and $\mathbf{F}$ represents forces acting on the fluid. The energy equation only needs to be solved for problems involving heat transfer, otherwise there is no need as there is no link between the energy equation and the momentum or continuity equations. Obtaining an exact solution to the Navier-Stokes equations for a turbulent flow is near impossible due to the vast number of unknown quantities and the resulting high computational cost, so approaches such as Reynolds Averaged Navier-Stokes (RANS) or Large Eddy Simulation (LES) modelling are usually applied.

RANS modelling involves splitting each variable in the Navier-Stokes equations into two components: a time averaged and a fluctuating component. The equations are then replaced with the time-averaged equivalent, while the fluctuations are represented by a turbulence model. In the LES approach, the large turbulent structures in the flow are resolved while small scale fluctuations are represented using a sub-grid scale model. LES has been observed to produce a more accurate representation of the atmospheric boundary layer and the wake behind turbines [10], however it is significantly more computationally expensive than the RANS approach. A suggested solution is to use RANS modelling for the boundary layer flow around the blades and LES for the wake [11].

CFD methods are being applied increasingly frequently in FOWT analyses. Liu et al. developed and validated a fully coupled CFD model of a floating wind turbine, platform and mooring system, and showed that the CFD tool was able to produce detailed information on the aerodynamic loading distribution and the wake structure downstream of the turbine [12]. In another study, Make et al. showed that performance predictions using the BEM method can be significantly improved by using aerofoil data obtained from 3D CFD simulations [13].

CFD simulations of wind turbines vary in complexity, from simple actuator disk models to higher fidelity actuator line and actuator surface models, to resolving the flow around the full rotating rotor geometry. Whilst a more complex model will capture more aerodynamic effects, the computational cost increases significantly, and so the final model will involve a compromise.

2.2.2 Actuator Disk Model and Momentum Theory

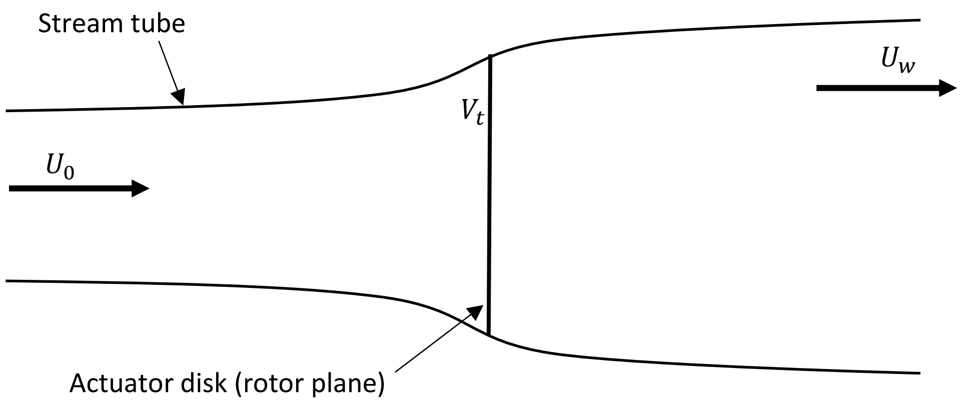


Figure 1: Stream tube and actuator disk model

Actuator disk models (ADMs) are frequently used in CFD simulations due to their relatively low complexity and computational cost, since no rotor geometry is considered and the distribution of aerodynamic forces is assumed to be axisymmetric. The simplest form of the ADM first introduced by Froude [14] as an extension of the work of Rankine [15] considers a uniform distribution of thrust over the disk area and no rotation. The model is based on simple momentum theory where the actuator disk extracts kinetic energy from the flow passing through it and therefore decreases the momentum of the flow. The axial force or thrust acting on the rotor is simply expressed as this momentum deficit:

$$T = \dot{m}(U_0 - U_w) = \rho A V_t (U_0 - U_w) \tag{2-3}$$

Where U_0 is the free stream velocity, U_w is the wake velocity, $\dot{m} = \rho A V_t$ is the mass flow through the actuator disk, ρ is the density, A is the disk area and V_t is the flow velocity at the disk. V_t is considered to be the arithmetic mean of the free stream velocity and wake velocity. The power output of the disk is then related to the thrust force:

$$P = TV_t = \frac{1}{2}\rho A V_t \left({U_0}^2 - {U_w}^2\right), \quad V_t = \frac{1}{2}(U_0 + U_w) \tag{2-4}$$

By defining the axial induction factor $a = 1 - \frac{V_t}{U_0}$, the dimensionless coefficients of thrust and power that determine the turbine performance can be expressed as:

$$C_T = \frac{2T}{\rho U_0^2 \pi R^2} = 4a(1-a), \quad C_P = \frac{2P}{\rho U_0^3 \pi R^2} = 4a(1-a)^2$$

2-5, 2-6

Since a real turbine is not uniformly distributed, the radial variation of rotor loading is analysed by splitting the rotor into a finite number of annular streamtubes, where axial and angular momentum balance for each streamtube is expressed as follows:

$$dT = 2W_z d\dot{m}, \;\; dQ = 2W_\theta r d\dot{m}$$

2-7, 2-8

Where $W_z = U_0 - U_w$ and W_θ are the induced axial and angular velocities respectively, and $d\dot{m} = \rho V_t dA = \rho\pi(U_0 + U_w) r dr$. W_θ is zero upstream of the disk, and the angular velocity is -W_θ at the disk and -2W_θ just after the disk.

A comprehensive study of actuator disk representations of wind turbines was completed by Mikkelsen [16]. This work showed that ADMs are capable of predicting the aerodynamics of coned and yawed rotors with good accuracy. However, the axisymmetric nature of an ADM means that it is not valid for high yaw angles since the rotor wake varies azimuthally. Since a floating wind turbine frequently operates in non-axial flow conditions due to the pitch and yaw motions of the platform, the validity of an ADM in these conditions will need to be investigated.

2.2.3 Blade Element Theory

The lift and drag forces that determine the thrust and torque acting on a rotor are dependent on characteristics of the blade geometry. To calculate these forces, values for the coefficients of lift and drag must be obtained, which depend on the shape of the aerofoil, the Reynolds number of the flow and the angle of attack α, where $\alpha = \phi - \beta$ (where ϕ and β are the flow angle and blade pitch angle respectively. In blade element theory, the forces acting on a blade are calculated for each radial section, known as a blade element, while assuming there is no interaction between the flow on each element. The torque and thrust acting on each rotor element are defined based on the lift and drag coefficients C_L and C_D as follows:

$$dT = \frac{1}{2} B c \rho V_{rel}^2 C_n dr, \qquad C_n = C_L cos\phi + C_D sin\phi$$

2-9

$$dQ = \frac{1}{2} B c \rho V_{rel}^2 C_t dr, \qquad C_t = C_L cos\phi + C_D sin\phi$$

2-10

Where C_n and C_t are the normal and tangential force coefficients, B is the number of rotor blades and c is the local chord length. V_{rel} is the resultant relative airflow velocity through the turbine based on the axial wind velocity and tangential velocity, where $V_{rel}^2 = (U_0 - W_z)^2 + (\Omega r + W_z)^2$. Blade element theory assumes that the radial component of the velocity is zero. The flow angle can then be defined:

$$tan\phi = \frac{U_0 - W_z}{\Omega r + W_\theta} = \frac{U_0(1-a)}{\Omega r(1+a')}$$

2-11

Where $a' = \frac{W_\theta}{\Omega r}$ is the tangential induction factor.

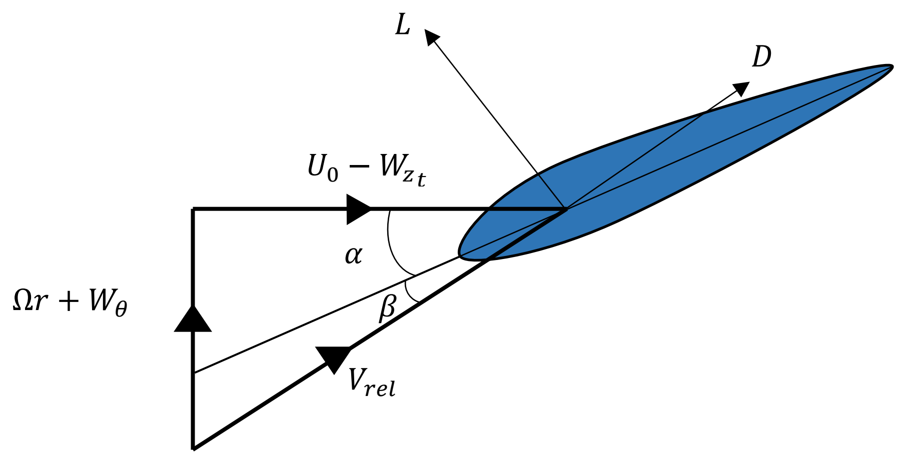


Figure 2: Velocities, lift (L) and drag (D) acting on a blade element

2.2.4 Actuator Line Model

The forces acting on a real turbine rotor are not distributed axisymmetrically due to the finite number of blades, in contrast to the assumption made by the actuator disk model. The actuator line model (ALM) considers this by representing the distribution of forces along each individual rotor blade. This concept was first introduced by Sorensen and Shen [17], who defined the forces on the blades using the blade element method with 2D aerofoil characteristics corrected for 3D effects.

In an ALM, each blade is defined by a series of points long the blade axis, where each point is the centre of an actuator element. The first blade axis is defined parallel to the x-y plane representing the ground, and the other blades are then defined using a rotation matrix. The blades rotate around the line between the points defining the nacelle and the hub, which is dependent on the turbine's yaw and tilt angle. The total force on a blade element is then defined based on the blade element theory:

$$F_i^A = \frac{dF}{dA} = \frac{1}{2} c\rho V_{rel}^2 (C_L \boldsymbol{e_L} + C_D \boldsymbol{e_D})$$

2-12

Where $\boldsymbol{e_L}$ and $\boldsymbol{e_D}$ are unit vectors in the direction of the lift and drag forces. To avoid numerical instabilities, a Gaussian smoothing function $\eta_\epsilon(r)$ is used to smoothly distribute the loading over the mesh. The projected force on a blade element then becomes:

$$f_\epsilon = \eta_\epsilon F_i^A, \quad \eta_\epsilon(r) = \frac{1}{\epsilon^3 \pi^{3/2}} \exp\left[-\left(\frac{r}{\epsilon}\right)^2\right]$$

2-13

An advantage of the ALM over the ADM is that it is able to include the root and tip vortices. As observed by Mikkelsen [16], the ALM can be used to simulate rotors at high yaw angles with greater accuracy than the ADM due to its ability to capture 3D effects from the root vortex structure.

Tossas and Leonardi presented an implementation of an ADM and an ALM [10], and found that an ALM was able to resolve the flow structures due to the presence of individual blades and asymmetric flow structures in the wake that are not captured by ADMs. It was concluded that ALMs produce better results for the near wake as they are able to capture vortical structures, however an ADM is sufficient for capturing the far wake. An ALM rotates the blades every time step and therefore applies forces to different cells each time step. The tip of the blade should not pass through more than one finite volume cell each time step, which imposes a strict limit on the time step size and therefore a significant increase in computational cost.

2.2.5 The Blade Element Momentum Method

A faster method of analysing wind turbine aerodynamics is the BEM method, which is based on a combination of momentum theory and blade element theory. From combining the equations for the axial force obtained from actuator disk theory and blade element theory, an expression for the axial induction factor a is obtained:

$$\frac{a}{1-a} = \frac{\sigma C_n}{4sin^2\phi} \tag{2-14}$$

Where the local solidity $\sigma = Bc/2\pi r$. Similarly, an expression for the tangential induction factor a' is obtained from the equations for the torque:

$$\frac{a'}{1-a'} = \frac{\sigma C_t}{4sin\phi cos\phi} \tag{2-15}$$

The axial and tangential induction factors a and a' are evaluated using an iterative procedure. Once a and a' are known, the rotor aerodynamic performance can be calculated for a given rotor geometry and flow conditions. This procedure is used in blade design processes to select the rotor geometry including the aerofoils used, chord and twist angle distributions along the blade, and the number of blades in order to optimise performance:

1. Specify initial values for the design variables
2. Resolve a and a' iteratively
    a. Guess initial values of a and a'
    b. Calculate ϕ using equation (2-11)
    c. Calculate α
    d. Calculate C_L and C_D using α
    e. Calculate C_t and C_n
    f. Calculate new values for a and a'
    g. Check for convergence, and repeat from step (2.b) until solution has converged
3. Calculate resultant rotor power and thrust
4. Repeat process as necessary for different values of the design variables

BEM theory has a number of limitations. Momentum theory becomes invalid for high values of the induction factors, when the turbine is operating at a high tip speed ratio and flow separation occurs behind the rotor due to high thrust relative to the velocity of the incoming flow. Blade element theory assumes that the forces acting on the blade are radially independent, which is untrue and can result in inaccuracies. BEM theory also does not account for certain flow effects such as losses at the blade tip, stall conditions and any 3D aerofoil data. Numerous correction models have been developed to attempt to address these issues.

Torque losses at the tip of the blade are generally accounted for using the Prandtl tip loss correction factor F, defined by Glauert [18] as follows:

$$F = \frac{2}{\pi}\arccos(e^{-\frac{B(R-r)}{2Rsin\phi}}) \tag{2-16}$$

F is then included as a factor in the equations for the axial and tangential forces on the blade element.

The equations used in BEM methods must also be modified to account for operation at high tip speed ratios, known as the turbulent wake state. In this state, turbulence in the wake increases due to more flow from outside the wake being incorporated into it. Therefore, the velocity of the flow behind the rotor decreases, but the thrust on the rotor continues to increase. BEM theory becomes invalid since it does not consider this, and momentum theory instead assumes that some of the flow in the far wake begins to propagate upstream, which is physically impossible. This is accounted for using Glauert's correction for the thrust coefficient. As high tip losses mean a high possibility of turbulent wake at the tip due to large induced velocities, the Glauert correction model includes the tip/hub loss factor and is defined by Buhl as follows [19]:

$$C_n = \frac{8}{9} + \left(4F - \frac{40}{9}\right)a + \left(\frac{50}{9} - 4F\right)a^2$$

2-17

The Glauert correction is applied to calculate a after a chosen limit, usually when a > 0.4.

Despite various corrections, the BEM method still has its limitations. However, its simplicity and speed means that it is still used as the basis for the majority of wind turbine analysis codes, and the method gives reasonably reliable results provided that the aerofoil data supplied is of sufficiently good quality [20]. BEM methods can be further improved by using aerofoil data that includes 3D flow data[13]. In order to reduce computation time, the CFD model developed in this study may be coupled with a BEM based design code such as FAST, especially if the full rotor geometry in a rotating domain is to be considered, or if flexible blades are to be modelled to represent a morphing blade concept.

2.3 Wind Turbine Optimisation

The reasoning behind creating a detailed computational model of a floating wind turbine is to gain a better understanding of the conditions in which it operates and the effect this has on turbine performance, and apply this knowledge to create an optimal design. There is a range of options for forming the optimisation process.

Optimisation problems can be solved using traditional approaches, such as mathematical gradient based methods that make use of the derivatives of the objective function to find a minimum, and have been used in several wind turbine optimisation studies [21][22]. Gradient based algorithms are quick and have the advantage of being able to handle multiple objectives, however their ability to find a global optimum is highly dependent on the initial conditions and they can only be used for differentiable functions.

Recent studies of optimisation of wind turbines have tended towards using heuristic approaches, particularly evolutionary algorithms [23]. The reason for this is that evolutionary algorithms such as genetic algorithms or particle swarm optimisation are population based, meaning they find a range of possible optimal solutions. In addition, despite being slower than gradient based algorithms on average, they are more robust due to being less sensitive to initial configurations and local optimal solutions.

Optimisation techniques applied to wind turbines often involve multiple objectives, such as maximising energy production whilst simultaneously maximising stability and minimising costs. There are a number of possible approaches to solving a multi objective optimisation problem, as outlined by Deb [24]. The simplest method is the objective weighted method, where the single objective consists of a sum of the multiple objectives, each of which are multiplied by a user defined weight factor to determine their importance. An alternative option is the ε constraint approach, where one objective is chosen as the single objective function whilst the others are instead presented as constraints.

A possible option for reducing the computation time of an optimisation process using a CFD model is to use a surrogate model. An example of this can be seen in a study by Ribeiro et al. [25], who used a genetic algorithm directly coupled with a CFD solver at the beginning of the process and then used an artificial neural network coupled with the genetic algorithm for the remaining iterations, which was shown to drastically reduce the overall solution time.

A comprehensive review of optimisation techniques applied to both onshore and offshore wind turbines has been published by Chehouri et al [23]. They recommend that wind turbine optimisation be performed with the objective of minimising the cost of energy rather than maximising aerodynamic performance, and that the use of heuristic approaches such as genetic algorithms is a promising area for investigation. Whilst the problem of wind turbine optimisation has been studied extensively, little work has been done on optimisation of floating wind turbines and their components.

The optimisation methodology applied to floating offshore wind turbines should focus on addressing the major challenges identified in their design, namely minimising the turbine weight and accommodating for the displacements and rotations resulting from the platform motions [5], while also maintaining an acceptable level of power production in order to make the design economically viable. A suitable aerodynamic analysis tool to compute the loading and the power output of the design will be needed as a reference in the optimisation process. By using a CFD method to simulate a wind turbine subjected to platform motions, the unsteady aerodynamic behaviour can be understood, which will be necessary in the optimisation process.

3 Research Progress

The objective of this research is to present an optimised design of a floating wind turbine. To achieve this, a model that can predict the aerodynamics and in particular the loading distribution on the important components with sufficient accuracy will be developed using CFD techniques.

The anticipated stages involved in developing a CFD model of a floating wind turbine are outlined as follows:

1. Develop fixed actuator disk model
   - Validate model against available data for axial flow case
   - Validate model for fixed yawed and tilted cases
   - Make modifications and improvements
2. Develop actuator line model
   - Compare results of actuator disk and actuator line models
3. Develop moving actuator model
   - Apply moving mesh method
   - Implement time varying yaw and pitch angles
   - Add platform degrees of freedom to model
4. Develop rotating turbine model
   - Investigate coupling CFD model with a BEM tool such as FAST to keep computational time within reasonable limits

3.1 Introduction to OpenFOAM

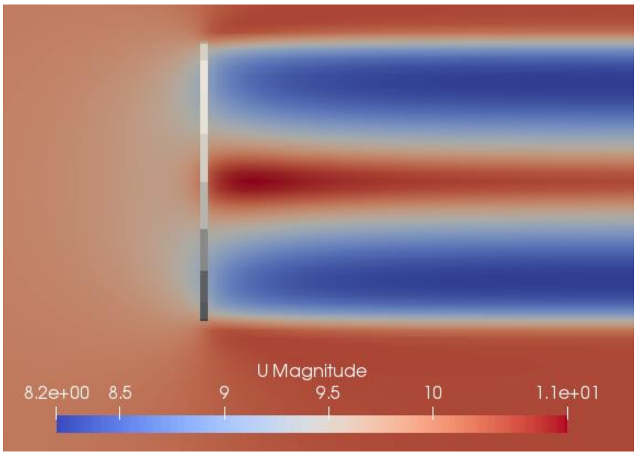
The CFD toolbox OpenFOAM will be used in this project to develop and simulate a CFD model of a floating wind turbine. OpenFOAM is an open source code written in the C++, which is designed to solve the governing equations of a fluid flow through discretisation using the finite volume method.

Due to no prior experience using the software, the beginning of this research project involved completing a training course in using OpenFOAM, followed by practising a range of tutorials in order to become familiar with the code and techniques used.

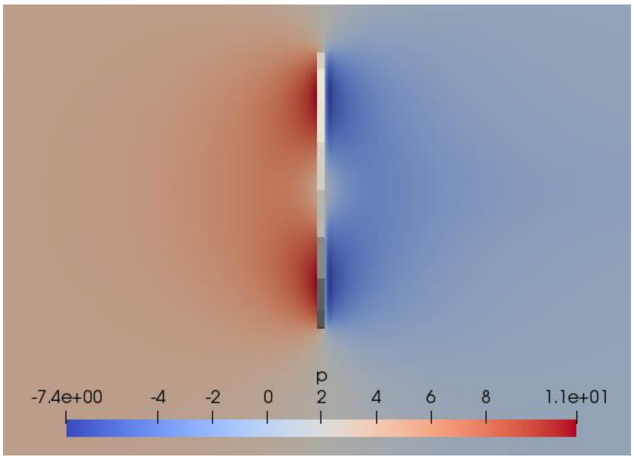
3.2 Model development

Due to its relative simplicity, it was decided that the first stage of creating a CFD model of a FOWT would be to develop an actuator disk model. The foundation for this model is an actuator disk implementation presented by Svenning [26]. Svenning's model was developed by writing a new class that defines the actuator disk properties and calculates a volume force with axial and tangential components to represent thrust and torque. The radial variation of torque and thrust over the disk area were modelled using a Goldstein optimum distribution. The simpleFoam solver is then modified to include the new class and solve the momentum equation with the calculated volume force as an additional source term.

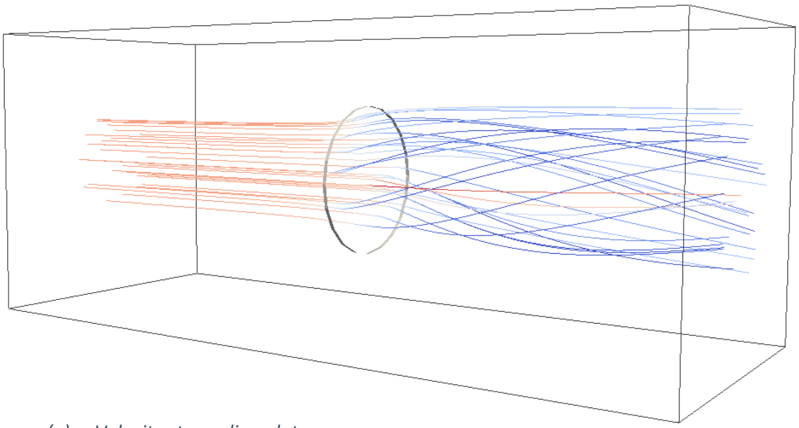
In order to simulate a floating wind turbine using an ADM, a number of modifications were made in the current model in order to include the rotor geometry and effect of misaligned flow. A new function that calculates the thrust and torque distribution based on the blade element theory was defined, which uses blade geometry characteristics and the freestream velocity of the flow as inputs. The blade chord and twist values at different radial positions and the lift and drag coefficients are obtained from lookup tables.



(c) Velocity magnitude plot



(b) Pressure plot



(a) Velocity streamline plot

Figure 3: Implementation of actuator disk model developed by Svenning [26]

The user defines the position and geometry characteristics of the actuator disk as in Svenning's model, in addition to new variables including the number of blades, blade pitch angle at the root and rotational speed. Unlike in Svenning's model where the solver loops over the entire mesh each iteration to check whether a cell is in the actuator disk region, the current implementation stores the location of the disk in a list of the position vector of each cell centre in the actuator disk region. This should significantly reduce the computation time, as it only requires the solver to loop over the mesh once at the start of the simulation. The freestream velocity is assumed to equal the inlet velocity.

The turbine will first be simulated at a range of fixed pitch and yaw angles, so an additional requirement of the model at this stage is that it is able to handle yawed and tilted conditions. This will be done using a methodology presented by Mikkelsen [16], which involves obtaining a vector of the velocity components (V_r, V_θ, V_z) calculated at the rotor and projecting these velocities onto the axial and tangential directions of the yawed or tilted rotor:

$$\boldsymbol{V_{stn}} = B\Theta^T \Phi_t \Phi_y \Theta \boldsymbol{V_{r\theta z}} \tag{3-1}$$

$$\Phi_t = \begin{bmatrix} 1 & 0 & 0 \\ 0 & cos\phi_t & sin\phi_t \\ 0 & -sin\phi_t & cos\phi_t \end{bmatrix}, \Phi_y = \begin{bmatrix} cos\phi_y & 0 & -sin\phi_y \\ 0 & 1 & 0 \\ sin\phi_y & 0 & cos\phi_y \end{bmatrix}, B = \begin{bmatrix} cos\beta & 0 & -sin\beta \\ 0 & 1 & 0 \\ sin\beta & 0 & cos\beta \end{bmatrix}, \Theta = \begin{bmatrix} cos\theta & -sin\theta & 0 \\ sin\theta & cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Where Φ_t is the transformation matrix for the tilt angle, Φ_y for the yaw angle and B for the coning/deflection angle. $\theta = \omega t$ is the azimuthal angle of the reference blade at time t where ω is the rotor angular speed. The new velocity vector resulting from this transformation is used to calculate the flow angle and relative velocity at the rotor, which are then included in the calculation of thrust and torque. At present, the coning angle of the rotor is not included.

4 Research Aims and Future Work

The current model can be improved in a number of ways. At present, the user prescribes the total torque and thrust, which is not representative of a real turbine as this means they are fixed and do not respond to changes in environmental conditions. Future work will investigate different methods of applying force and rotation to the flow that will enable the turbine to adapt to such changes, and methods of modelling the turbine hub as currently this does not apply any force on the fluid. The current model will then be used as a baseline for the development of more detailed models as outlined in the previous section.

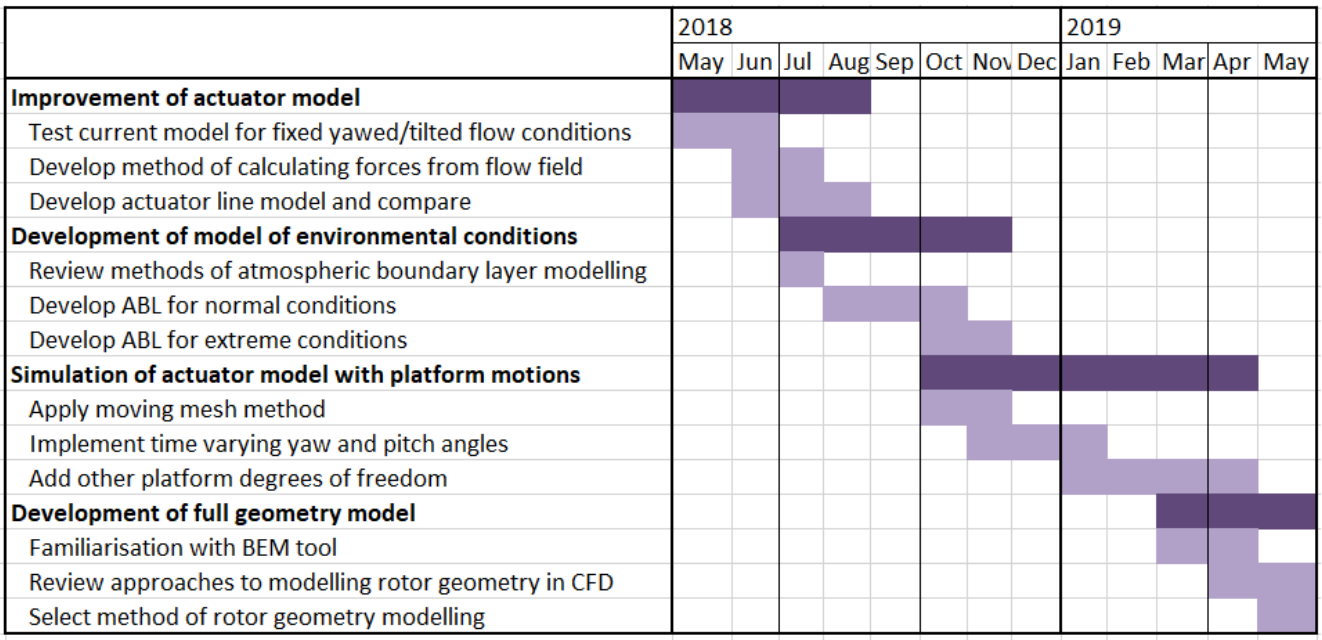
The simulations will need to be performed for different wind speeds and turbulence intensities in order to model the misaligned turbine’s behaviour in normal and extreme environmental conditions. Therefore, a realistic atmospheric boundary layer model will also need to be developed.

To further validate the CFD method used in this study, a physical model of the turbine will be created for a tank test procedure to obtain experimental data. Since offshore wind turbine rotors are typically very large, this will involve selecting an appropriate scaling approach, as well as a suitable representation of the floating structure and mooring system.

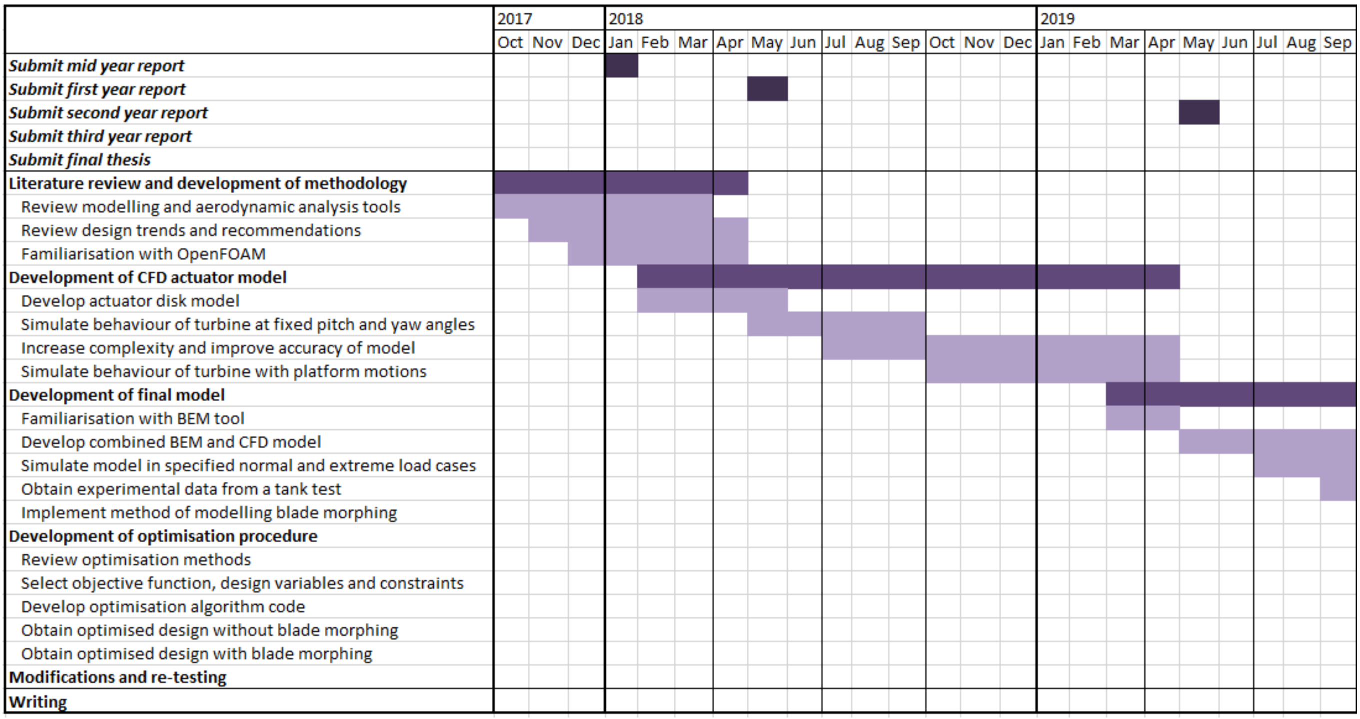
The eventual aim of this project is to produce an optimised design of a floating wind turbine. Therefore, the final model will be incorporated into an optimisation procedure so that the performance of different turbine designs can be understood.

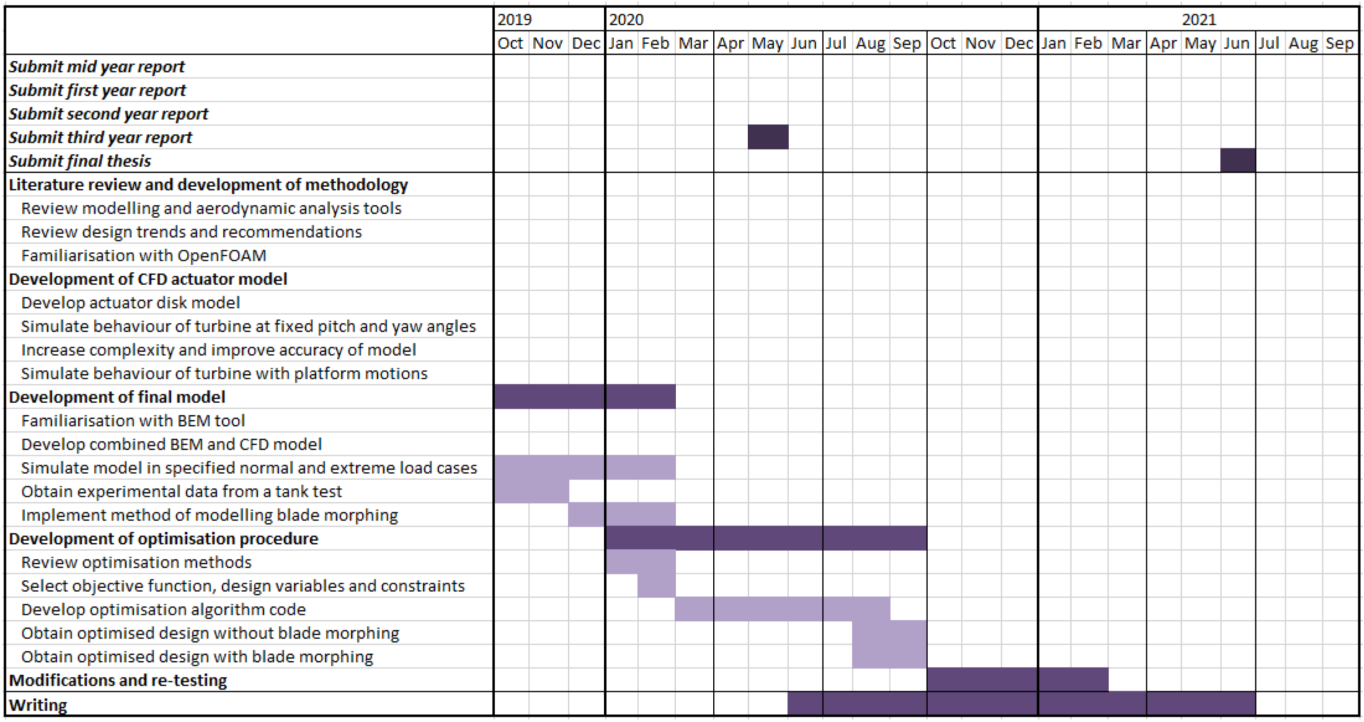
The primary objectives for the coming year are to have developed an actuator model that can be used to analyse the effects of platform motions and environmental conditions on aerodynamics, and to have selected the methodologies that will be used to develop the final model. A research plan for the coming year and a more general plan for the duration of the project are presented in the following Gantt charts.

4.1 Plan for the coming year



4.2 Full project overview





References

[1] G. Stewart and M. Lackner, "Offshore Wind Turbine Load Reduction Employing Optimal Passive Tuned Mass Damping Systems," *IEEE Trans. Control Syst. Technol.*, vol. 21, no. 4, pp. 1090–1104, 2013.

[2] Z. Ma, W. Li, N. Ren, and J. Ou, "The typhoon effect on the aerodynamic performance of a floating offshore wind turbine," *J. Ocean Eng. Sci.*, vol. 2, no. 4, pp. 279–287, 2017.

[3] J. Cruz and M. Atcheson, *Floating Offshore Wind Energy: The Next Generation of Wind Energy*. Cham, SWITZERLAND: Springer, 2016.

[4] T. Sebastian, "The Aerodynamics and Near Wake of an Offshore Floating Horizontal Axis Wind Turbine," PhD, University of Massachusetts - Amherst, 2012.

[5] S. Butterfield, W. Musial, and J. Jonkman, "Engineering Challenges for Floating Offshore Wind Turbines," presented at the Copenhagen Offshore Wind Conference, Copenhagen, Denmark, 2005.

[6] B. Multon, *Marine Renewable Energy Handbook*. ISTE Ltd and John Wiley & Sons, Inc., 2012.

[7] D. Micallef and T. Sant, "Loading effects on floating offshore horizontal axis wind turbines in surge motion," *Renew. Energy*, vol. 83, pp. 737–748, 2015.

[8] L. Duan and H. Kajiwara, "A Coupled Aero-Hydrodynamic Simulator for Offshore Floating Wind Turbines," in *OMAE2014-23828*, San Francisco, California, USA, 2014.

[9] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*, 2nd ed. Pearson Education Limited, 2007.

[10] L. A. M. Tossas and S. Leonardi, "Wind Turbine Modeling for Computational Fluid Dynamics," NREL, 2013.

[11] D. Matha, M. Schlipf, A. Cordle, R. Pereira, and J. Jonkman, "Challenges in Simulation of Aerodynamics, Hydrodynamics, and Mooring-Line Dynamics of Floating Offshore Wind Turbines," presented at the 21st Offshore and Polar Engineering Conference, Maui, Hawaii, 2011.

[12] Y. Liu, Q. Xiao, A. Incecik, C. Peyrard, and D. Wan, "Establishing a fully coupled CFD analysis tool for floating offshore wind turbines," *Renew. Energy*, vol. 112, pp. 280–301, 2017.

[13] M. Make, G. Vaz, G. Fernandes, S. Burmester, and S. Gueydon, "Analysis of Aerodynamic Performance of Floating Wind Turbines Using CFD and BEMT Methods," in *OMAE2015*, St John's, Newfoundland, Canada, 2015.

[14] R. E. Froude, "On the Part Played in Propulsion by Difference in Pressure," *Trans. Inst. Nav. Archit.*, vol. 30, pp. 390–423, 1889.

[15] W. J. M. Rankine, "On the mechanical princples of the action of propellers," *Trans. Inst. Nav. Archit.*, vol. 6, pp. 13–39, 1865.

[16] R. F. Mikkelsen, "Actuator Disc Methods Applied to Wind Turbines," Doctoral Thesis, Technical University of Denmark, 2003.

[17] J. N. Sørensen and W. Z. Shen, "Numerical modelling of wind turbine wakes," *J. Fluids Eng.*, vol. 124, pp. 393–399, 2002.

[18] H. Glauert, “Airplane Propellers,” in *Aerodynamic Theory*, vol. 4, W. F. Durand, Ed. Berlin, Heidelberg: Springer Berlin Heidelberg, 1935, pp. 169–360.

[19] M. L. Buhl, “A New Empirical Relationship between Thrust Coefficient and Induction Factor for the Turbulent Windmill State,” National Renewable Energy Laboratory, Goldon, CO, Technical NREL/TP-500-36834, 2005.

[20] T. Burton, N. Jenkins, D. Sharpe, and E. Bossanyi, *Wind Energy Handbook*, 2nd ed. John Wiley & Sons, Ltd, 2011.

[21] J. H. Sjølund and E. Lund, “Composite Thickness Optimisation of Offshore Wind Turbine Blade with Fixed Outer Geometry,” in *NSCM-30*, 2017.

[22] T. Ashuri, M. B. Zaaijer, J. R. R. A. Martins, G. J. W. van Bussel, and G. A. M. van Kuik, “Multidisciplinary design optimization of offshore wind turbines for minimum levelized cost of energy,” *Renew. Energy*, vol. 68, pp. 893–905, 2014.

[23] A. Chehouri, R. Younes, A. Ilinca, and J. Perron, “Review of performance optimization techniques applied to wind turbines,” *Appl. Energy*, vol. 142, pp. 361–388, 2015.

[24] K. Deb, *Multi-Objective Optimization using Evolutionary Algorithms*. Chichester, U.K.: John Wiley & Sons, Ltd, 2001.

[25] A. F. P. Ribeiro, A. M. Awruch, and H. M. Gomes, “An airfoil optimization technique for wind turbines,” *Appl. Math. Model.*, vol. 36, pp. 4898–4907, 2012.

[26] E. Svenning, “Implementation of an actuator disk in OpenFOAM,” Chalmers University of Technology, 2010.