University of Exeter coursework header sheet

ECM3102 Individual Project (Meng) (A, TRM1+2 2017/8)

071505

1032717

650002170

Coursework: Final Report

Submission Deadline: Wed 9th May 2018 12:00

Personal tutor: Dr Yang Liu

Marker name: GT

Word count: 11911

First marker's comments	Indicative mark
Second marker's comments	Second mark
Moderator's comments	Agreed mark

Final Report

The Effect of Swansea Bay Tidal Lagoon on Hydrodynamic Flow Patterns

Matt Postles

2018

3rd Year Individual Project

I certify that all material in this thesis that is not my own work has been identified and that no material has been included for which a degree has previously been conferred on me.

Signed..mnpostles..

College of Engineering, Mathematics, and Physical Sciences
University of Exeter

Final Report

ECM3102

Title: The Effect of Swansea Bay Tidal Lagoon on Hydrodynamic Flow Patterns

Date of submission: Wednesday, 09 May 2018

Student Name: Matt Postles
Programme: MEng Mechanical Engineering
Student number: 650002170
Candidate number: 071505

Supervisor: Professor Gavin Tabor

Abstract

This primary objective of this project is to investigate the potential impacts that Swansea Bay Tidal Lagoon could have on the hydrodynamics of the region, including the effect on sediment transfer processes, water quality and changes to navigation and shipping routes. Tidal energy in recent years has become a more widely considered and attractive option to reduce the world's dependency on unsustainable methods of energy generation such as fossil fuels, and with this, new research must be undertaken to fully understand its impact.

As Swansea Bay tidal lagoon, if completed, would be the first of its kind, there is no existing evidence on how it would affect the environment, and so detailed modelling and analysis must be undertaken to ensure that the useful benefits of the project are not overshadowed by significant environmental damage. The main deliverables of this project were to identify how the combination of the current in the bay and flow exiting the turbines would affect the shear stress on the sea bed, specifically due to changing current speeds and tides, and how this could lead to processes such as the erosion and deposition of sediment.

Analysis was completed by use of the computational fluid dynamics (CFD) code ANSYS Fluent. The conclusions from the simulations ran were used to infer what potential impact this could have on the Swansea Bay area, which was compared to previously completed assessments by various companies for the planning application of the lagoon.

From the collected results, it was found that high velocity current flow and low tides led to the greatest shear stress on the sea bed, with current flow speeds affecting stress much more significantly than the tides. This would likely lead to erosion of the surface and transport of the material away from the turbine housing, although due to the rarity of such high velocities, this is unlikely to be significant enough to be a cause for concern. However, it should be considered in the construction process and appropriate precautions and preparation undertaken to reduce its impact if necessary.

Keywords: ANSYS, Hydrodynamics, Swansea Bay Tidal Lagoon

Table of contents

1. Introduction and Background

The purpose of this project is to investigate the potential impact that could be caused by the introduction of the proposed Swansea Bay Tidal Lagoon (**Figure 1.1**). The proposed lagoon is formed of a large sea wall in a loop shape which is connected to the coast. Sluice gates are shut to create a height difference between the sea and the lagoon and vice versa, allowing flow through the turbine when opened and causing the generation of electricity [1]. Marine locations such as Swansea Bay are important ecological sites and are host to a wide variety of organisms and organic matter. Therefore, it is important to clarify the extent to which the tidal lagoon will affect the ecosystem of the area. Furthermore, a tidal lagoon would have several major impacts on other industries; one such example is the Swansea docks, which have an important contribution to the local economy. A disruption of flows in the bay may have a resulting effect on shipping routes in the area, which could impact trade and have economic implications.

Figure 1.1: The location of Swansea Bay Tidal Lagoon [2]

This investigation will be completed with the use of ANSYS Fluent, which is a computational fluid dynamics (CFD) software package, primarily used for small-scale applications such as cars, airfoils and mechanical parts. Therefore, analysis will be focused on the relatively small-scale area of the turbine housing and one singular turbine exit, rather than modelling the entire lagoon and looking at large scale shifts in flow patterns. The primary objectives are to model the interaction between current flow past the turbine housing, and how this is influenced by the expulsion of water from the turbine during ebbing stages. The height of the tide and the current velocity will be varied in order to identify maximum points of stress.

It is well-known that in the UK and globally, the current reliance on non-sustainable sources of energy, fundamentally the use of fossil fuels such as crude oil and natural gas, contributes greatly to the rise of carbon emissions in the atmosphere. Therefore, it is vital to investigate alternative solutions to generate electricity. There are various targets set that the UK must achieve; for one, the European Commission demand that 20% of total energy consumed must be generated using sustainable methods by 2020, with this increasing to 27% by 2030 [3]. In 2016, only 8.9% of total energy consumption used renewable energy. Of the renewable energy sources currently used, solar photovoltaics and wind energy dominate the industry, which account for approximately 33% and 45% respectively, with marine energy, including wind and tidal generation, contributing only 0.04% of total generation, equating to a mere 13 MW [4].

The UK is naturally suited to tidal power due to the vast amount of coastline and the presence of many bays and estuaries. A report by The Royal Society indicates that all 20% of the renewable energy target could be generated by tidal power alone, with the combination of tidal barrages, which use the same technology as Swansea Bay, as well as the use of tidal turbines situated underwater [5]. Swansea Bay is currently only a proposition, and if completed, would be the first tidal lagoon in the world. Tidal lagoons are based on the same theory behind tidal barrages, which use the tides to activate turbines installed in the barrage, converting the kinetic energy of the currents into usable power. Unlike other hydropower methods such as dams, barrages can incorporate flow from two directions, and therefore generate a greater amount of energy; as there are two high tides and low tides every 24 hours in the bay, this gives the opportunity for the lagoon to generate electricity 4 times a day. This is because there will be a period of generation for both times the high tide comes in, after which the sluice gates will shut to create a height difference between each side of the lagoon, and when the gates are released, energy will be generated [1].

Swansea Bay in particular has been highlighted as a suitable location - it is connected to the Severn Estuary, which has previously been considered for tidal barrages as it has a wide tidal range [6]. Furthermore, it presents significant advantages over other proposed tidal projects, such as the Severn barrage, primarily the cost – Swansea has been predicted to cost £1.3 billion [2], compared to the Severn's estimated cost of £20 billion [7]. If completed, the lagoon will have a direct positive impact on the UK's energy problem, contributing enough power for 155,000 homes over 120 years [2]. Despite the promising outlook of the lagoon, it is important to carefully examine the potential impacts that it could have on the hydrological processes in the area; even if it has the capacity to produce vast amounts of energy and change the way energy is generated in the UK, if this comes at a cost to the environment, then the benefits may not be justified.

2. Literature Review

This review analyses the previous work that has been completed in this field and identifies the opportunities for this project to focus on, as well as showing how previous work could influence its scope. Covered in this section are several aspects of the area of study such as: the hydrodynamics of tidal lagoons, the use of CFD programs, particularly ANSYS, to investigate other tidal energy structures; the use of turbulence models for marine applications; previous work completed regarding the environmental impacts of Swansea Bay; analysis of sediment transport and bed shear stress, which is a primary concern that may occur due to the lagoon; and the interaction of jets with crossflow for the purpose of validating the model.

Several studies have previously been carried out to investigate the hydrodynamic effects of tidal lagoons in general, as well as of Swansea Bay. An assessment by Cornett *et al.* [8] presents the impacts of the introduction of a tidal lagoon in the Bay of Fundy, Canada. This study was performed using the TELEMAC modelling system using finite element analysis in order to calculate the velocities and directions of the currents, in addition to the effect on the maximum tidal range due to the introduction of several proposed tidal lagoons in the area. The conclusions drawn in this article indicate that the greatest area of impact is directly next to the lagoon and the changes become a lot more insignificant further away from the lagoon. This highlights that the focus on analysing Swansea Bay should be in the immediate area surrounding the lagoon, ensuring that these locations are analysed in a correct manner, rather than trying to incorporate too much area into the model and losing detail. This also suggests that hydrodynamic impacts due to the lagoons will likely be localised and only affect the bay, rather than spreading to other areas and having a wider effect.

A similar study by Ferrarin and Umgiesser [9] simulates the hydrodynamics of a coastal lagoon in Sardinia, Italy, by investigating the effects of natural forces exerted on the water by the tides and waves. This was completed with the use of SHYFEM, a 2D finite element modelling software. This paper describes how the results of the initial simulations can be used to further develop other models, such as water quality, which can then be used to observe the effect on the ecology of the area. This paper has influenced the scope of the study on Swansea Bay, as the project could include similar ideas by using the results to incorporate analysis of the effect on ecosystems within the area.

Kadiri *et al.* [10] evaluate the damage to water quality due to tidal energy generation methods, with specific interest in the Severn Estuary, which feeds into the Bristol Channel and subsequently flows past Swansea Bay. The 2-D DIVAST model was used in this study to predict the changes in tide levels due to the placement of various lagoons and barrages in the area, and this is used to analyse the changes in current of the area. This is researched further, to include the impact on sediment transport, salinity and nutrients in the estuary, which may have significant geological effects.

Of these previous investigations, there is a variety of examples which use Delft3D in order to analyse the lagoons. One such example, by Bielecka and Kazmierski [11], uses Delft3D-FLOW to analyse Vistula Lagoon on the Baltic Sea. The correlation between velocities of wind and salinity has been investigated, as well as the variation of water level throughout the course of the year. This paper will help shape the investigation on Swansea Bay by emphasising certain considerations when undertaking hydrodynamic modelling, mainly the use of varying cell sizes; there are certain points where larger changes in velocity are expected, which in this paper occur in close proximity to the strait connecting the lagoon to the Baltic Sea, and therefore, smaller cell sizes are used in these locations to more accurately model the velocities. A similar technique can be implemented in this project to generate precise results at important regions, such as near the wall, so that the effect of the turbines can be closely monitored.

In addition to studies performed regarding tidal lagoons, research has been undertaken focusing on other tidal energy generation methods. In particular, some of these studies focus on the effects of oceanic processes on a singular turbine; for instance, Tatum *et al.* [12] used ANSYS to model a tidal stream turbine (TST), subjecting it to various conditions of waves and current velocities to identify its ability to withstand these conditions. The methodology used to perform this study is particularly relevant; unlike many previous studies which exclude surface effects [13-15], which in some cases is due to the lack of computational resources, this study employs the Volume of Fluid (VOF) method, which is important for use in solving interactions between two fluids, such as water and air, and is therefore applicable for studies involving coastal processes. This method highlights the potential importance of using the VOF method in this project and will be explored in later sections to identify whether use of this model will be applicable, taking computational demands into account. As many previous studies discount surface effects, it is clear that useful results can still be achieved without the more complex VOF model. An additional facet of CFD which this paper explores is the importance of meshing. The model employed a fine tetrahedral-based mesh and emphasises the use of inflation layers in various parts of the model to ensure a high resolution. This technique could be considered for aspects of the model such as the sea bed to ensure that the boundary layer is resolved in full detail.

Further studies have explored the hydrological impacts of similar tidal structures with direct application to the Severn Estuary. Ahmadian *et al.* [16] use the 2D hydrodynamic transport model DIVAST to identify how such a system could impact tidal currents and other parameters such as sediment transport. The study reports that while little impact is made on water depths, current flows around the tidal turbine array are increased and currents passing through the turbines themselves were reduced. Finally, sediment concentrations similarly decreased in line with the turbines, while increasing around the turbines. These results are useful to provide a comparison with general, large-scale data; however, this study does not investigate changes on a closer scale and observe the direct impacts of the water around the turbine, which presents an avenue for further research.

Other research on TSTs focus more exclusively on the impacts on sediment, recognising that in the area of operation, friction will be created on the bed of the region [17]. Furthermore, the need to consider the tidal ranges of the location has been identified, as this will greatly affect the morphodynamics of the location; this is especially applicable to regions such as Swansea Bay, which have large tidal ranges and therefore exacerbate these issues. The study concludes that for the example of the Bristol Channel, the difference of whether tidal currents in the region have ‘symmetry’ or ‘asymmetry’ can affect the decrease of the bed level by up to 20%. Therefore, this highlights the necessity to include tidal levels in future studies to fully recognise the impacts that they can have on not only immediate scour, but also the levels of deposition throughout the entire region. Scouring is the process by which flow patterns cause the removal of sediment from the sea floor in such a way that it can dig out holes and lead to the collapse of marine structures [18]. This could prove important for Swansea Bay, as if the motion of the fluid is severe enough to cause the vast removal of sediment, the wall of the lagoon could be at risk.

The flow in this situation is definitely turbulent and therefore the selection of the correct turbulence model is an important consideration. There are many studies [19-21] that focus on the comparison between various turbulence models to identify which is most suitable for the application of jets in crossflow, including standard k-ε (and its variations such as realisable k-ε, and RNG k-ε), k-ω and SST k-ω. From these studies, it is commonly concluded that k-ε can most accurately reflect experimental and published data, while SST k-ω could also produce acceptable results. Furthermore, RNG k-ε and SST k-ω have often been deemed unsuitable for use as they deviate too much from expectation.

In addition to a wide range of academic research on this topic, there is considerable corporate literature published by the company responsible for Swansea Bay, Tidal Lagoon Power (TLP). An initial scoping report for the proposed project was first submitted in 2012 [22] and describes the initial research regarding coastal processes and sedimentation. The document primarily covers existing research in the area, such as the general action of the waves transporting sediment in a north-easterly direction. Initial research using a basic 2D Delft3D model predicted that while there would be some influence on sediment patterns directly next to the lagoon, these would not extend to the rest of the Severn Estuary; this conclusion is echoed by a further study by ABP Marine Environmental Research (ABPmer), which was contracted by TLP [23]. Also outlined in this document is the scope of the environmental impact assessment (EIA), including the importance of understanding the potential for sediment transport and what effect this could have on marine species and their habitats.

A review [24] of the environmental statement was issued in 2014, completed by White Consultants. This report cited other completed work by companies such as KPAL [25] and heavily criticised the previous research, including the work of ABPmer, stating that there is a lack of detail, and recommended further research to establish the true results. A later 2014 environmental statement [26] predicted the possibility for major changes to the morphology of the area, including erosion, scour and deposition. The study theorises that initially, rapid changes will occur to the hydrological processes of the area, but following this unsettled period, there will be a slower progression to a “morphological equilibrium”. Of particular importance therefore is the initial change to the environment immediately after installation. It is highlighted that the primary interest is next to the turbine housing structure, which will lead to erosion of the sea bed. Furthermore, it is emphasised that the most extreme results will occur during spring tides (when low tides are extremely low and high tides are extremely high), as this is when there will be the greatest levels of deposition and erosion, although it is unclear if maximums would occur at extremely high or low tides. These results have influenced the scope of this project, by emphasising the need for the project to focus on the immediate area around the turbine structure, and also by analysing the full range of tidal levels to identify what creates the largest impact. Finally, it is highlighted that a significant area to assess is the region of influence of the jet flow out of the turbine during ebb flow, which is when water flows through the turbines from the lagoon to the bay [27]. This has influenced the analysis to focus on purely ebb flow, rather than looking at both the ebbing and flooding stages.

An important aspect of this project is to identify the immediate impact of tidal lagoons on shear stress exerted on the sea bed and the resulting deposition of sediment, and how this could affect the geology of the area. General hydrodynamic studies, such as that completed by Collins *et al.* [28] describe the current patterns as complex, partly due to the large tidal range of the area, which leads to fluctuating levels of sediment. However, the overall condition of the bay has been described to be in 'equilibrium'. It has been predicted by several studies that the placement of renewable energy generation methods in the Severn estuary will impact bed shear stress and therefore sediment deposition; it is important to note that studies by Wolf *et al.*, Willis *et al.*, and Kirby and Retière [29-31] conclude that, as opposed to what might be expected, the placement of such tidal extractors could in fact reduce bed shear stress and scour, and allow for greater deposition to occur. However, these studies look at smaller-scale turbine systems, which may have different effects to larger-scale projects such as Swansea Bay. Other studies focus on the general effect of flows on bed shear stress; Thompson *et al.* [32] conclude that an increase in velocity results in an increase in shear stress, and the relationship between the two is approximately linear. Furthermore, work by Gonzalez-Santamaria *et al.* [33] which studies the hydrological effects of wave farms identifies that low tides lead to more significant damage to the sea bed through increased shear stress than water at high tide.

More applicable to this project is the direct research on bed shear stress. A wide variety of existing research explores this area using experimental methods, mainly the prediction of stresses using depth-averaged velocity measurements using flumes in hydraulics laboratories [34-36]. However, in more recent years it has also become common to explore this field using computational methods, partly as validation of the experimental methods, but also partly as research in its own right [37]. A 2008 study analysing hydrological changes to the bed of the Danube river [38] confirms the suitability of using 3D CFD for the purpose of morphological investigation, demonstrating its accuracy to previously found results. Further studies include work by Ma *et al.* [39] which specifically use the program Fluent to show the accuracy of computational results by comparing them to findings using a scale model, proving that numerical models are a valid method of generating data.

There has been extensive research in the area of the interaction of jets with crossflow, and this information can be directly related to the interaction of fluid flowing through turbines and interacting with the current flowing past. Therefore, this is an area of research which can be used to validate the actions of the fluids interacting. Mahesh, a prominent academic in this field, has written a number of publications of the actions of crossflow, ranging from a comprehensive amalgamation of current knowledge on compressible and incompressible jets [40], to more specific applications of jets analysed using computational methods [41]. The latter study in particular analyses the effects of varying velocity ratio (the ratio between the jet velocity and the crossflow velocity) and velocity profile on flow trajectories; it is demonstrated that a higher velocity ratio causes the jet to disrupt the crossflow more than a jet with a lower velocity ratio. This builds on work completed by Kelso *et al.* [42] which employs experimental methods to quantify the behaviour of round jets in crossflow. This study focuses more on the features which can be observed in such interactions, including wall vortices, which form in close proximity with the wall flush to the exit of the jet and upright vortices (or wake vortices) which are a result of vortex shedding from the jet combining with the crossflow. More applicable to this work are results from rectangular jets in crossflow such a by Pathak *et al.* [43]. **Figure 2.1** displays the results of their simulations, in which the combination of the jet with the crossflow and the formation of various vortices can clearly be seen. This can later be used to validate simulations.

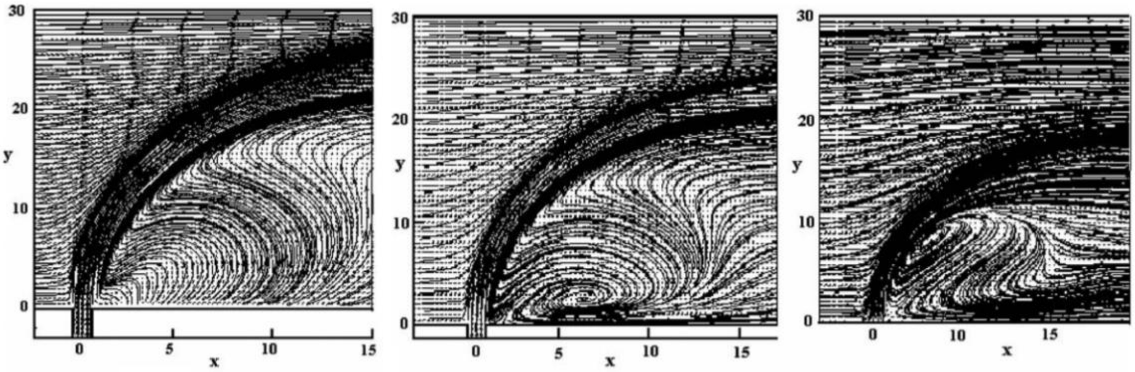


Figure 2.1: Velocity streamlines of the jet discharge interacting with the crossflow for different jet widths

While extensive previous research has been completed in the field of small-scale tidal energy generation projects using CFD programs such as ANSYS, these focus on TSTs or other tidal barrage schemes, but are very rarely, if ever, directly related to tidal lagoons such as that proposed at Swansea Bay. Furthermore, the majority of analysis for tidal lagoons involves large hydrodynamic models, but there is little emphasis on investigating aspects of the lagoons on a smaller scale. This presents a clear opportunity for research for this project.

3. Methodology and Theory

The research method used in this project involves the program ANSYS Fluent, which is used for modelling small-scale applications. Fluent is a computational fluid dynamics (CFD) program, which utilises numerical methods to solve various equations, namely the Navier-Stokes equations, which describe the motion of fluids [44]. While there are various methods of solving computational problems, including the finite difference (FD), finite element (FE) and finite volume (FV) approaches, FD and FE only have very limited applications in the field, and therefore FV is commonly used for most situations. Fluent incorporates the FV method into solving the Navier-Stokes equations, which is a method of converting partial differential equations into solvable equations in a process known as discretisation [45].

Using a generated mesh, the area of interest is divided into smaller regions, known as cells, which gives an individual control volume for each region. These control volumes can be analysed for fluid flowing in and out of each cell, which is achieved by integrating the Navier-Stokes equations over each individual volume, hence ‘Finite Volume’ [46]. Using a CFD code involves three main steps: definition of the inputs of the problem, generating the solution, and processing the results into a useable format. There are a wide range of variables in each of these processes that can affect the outcome, and these will be explored in further detail in this section.

3.1. Governing Equations

The fundamental equations solved in the use of CFD are the Navier-Stokes equations. In incompressible form using Cartesian coordinates, the Navier-Stokes equations can be written as follows for x, y and z components:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\rho g_x \quad (3.1)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+\rho g_y \quad (3.2)$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\rho g_z \quad (3.3)$$

Where ρ is the density of the fluid, μ is the dynamic viscosity, and u, v and w are velocities in the x, y and z directions respectively. The conservation of mass equation can also be written as:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (3.4)$$

In addition to the fundamental equations that are necessary for solving fluid flow, there are many more which are important to the process of computational fluid dynamics. For instance, it is necessary to define boundary conditions for regions of the model such as the intersection of fluid with air on the top face of the model, the intersection with the bed of the estuary on the bed of the model, and the walls of the turbine housing. These regions are all defined as a 'wall' by ANSYS Fluent, and therefore employs the no-slip condition. This condition means that at the boundaries, the velocity of the fluid is zero [47].

Finally, an important aspect of this project is identifying the shear stress on the bed due to the action of the turbines. For a no-slip condition, although the tangential velocity has been defined as zero, Fluent approximates the shear stress on the wall using flow properties immediately next to the boundary [48]. For laminar flow, this is defined as:

$$\tau_w=\mu\frac{\partial u}{\partial y} \quad (3.5)$$

In the case of turbulent flow however, the calculation of shear stress at the wall is more complex and depends on the turbulence model chosen.

3.2. Turbulence Models

Selecting a turbulence model is vital for ensuring that the results generated by the program are accurate. There are many considerations in choosing a model and making sure that it is the most applicable choice to the problem; for instance, the application required and therefore the physics of the fluid, and the computational cost of the simulation and whether there are sufficient resources to complete the task.

Firstly, the Reynolds number for the flow should be calculated to ensure that the flow is turbulent. As this model is approximated by using open channel flow, in which the flow is bounded on three sides with a free surface at the top, the equation for calculating the Reynolds for open channel flow can be used [49]:

$$Re=\frac{vR_H}{\upsilon} \quad (3.6)$$

Where v is the speed of the flow, R_H is the hydraulic radius of the channel, determined by dividing the cross-sectional area of the channel by its wetter perimeter, and υ is the kinematic viscosity of the fluid.

The minimum Reynolds number will occur at the lower velocity possible, 0.1 m/s, and the smallest channel size, which is at a tidal height of 14m. The channel is therefore modelled as a 14m high by 50m wide channel, and therefore the hydraulic radius is:

$$R_H = \frac{14 \times 50}{50 + 2 \times 14} = 8.97$$

Therefore, the Reynolds number for the flow is:

$$Re = \frac{0.1 \times 8.97}{1.373 \times 10^{-6}} = 653{,}314$$

Using data for the kinematic viscosity of water at 10°C [50], assuming a density of 1027 kg/m^3. It is generally accepted that if flow has a Reynolds number greater than 4000, the flow will be considered turbulent; therefore, the flow in this simulation is clearly turbulent, and all other simulations which use greater tidal heights or flow velocities will also be turbulent.

3.2.1. K-Epsilon

All variations of the k-epsilon (k-ε) models share the same fundamental transport equations in k (the turbulent kinetic energy) and ε (the rate of dissipation of turbulence energy), using a two-equation model. However, they vary in their definition and calculation of various constants in the equations, such as the Prandtl numbers [48], which are parameters related to the motion of eddies in fluid flow [51]. The standard version of the model is a widely used, industry-standard model which is reliable and has a wide range of applications in the field of CFD. While the standard model is generally good, it has several limitations; there are certain applications which it is not appropriate for, such as problems involving large adverse pressure gradients. Furthermore, k-ε has difficulty with the calculation of fluid properties at the wall of the model, and therefore relies on the use of wall functions instead [52].

The realisable version of the model improves on this in a number of important ways, including its ability to accurately predict solutions involving large adverse pressure gradients, which as previously mentioned is a flaw of the standard model [48]. It is also able to more accurately solve model aspects such as flows adjacent to planes and models involving round jets; however, the realisable model was primarily developed to deal with strong rotary motion and so it not particularly relevant for this model. Furthermore, it is more well-suited to rotating frames of reference, which is not applicable here. The other variation of the model considered is the renormalization group model (RNG) model [48]. In this version, various aspects of the standard model are enhanced, such as the modelling of swirl and the calculation of the Prandtl numbers. However, it has been reported that the model can wrongly estimate turbulence levels, and for that reason has been removed from use in some programs [53].

From analysing previous work in Section 2, it is clear that the literature favours using k-epsilon (k-ε) as this is most directly applicable to work in coastal and tidal applications. This has also been validated by experimental data. However, other types of turbulence model including Shear Stress Transport (SST) and K-Omega (k-ω) have been considered; their use has been dismissed as they are more commonly used for simulations involving turbomachinery, aeronautics, and strong rotational flows [53], which is not applicable for this project. Furthermore, k-ω often takes longer to converge than k-ε, which may be unsuitable for this model due to the lack of computational power.

3.2.2. Computational Requirements

In addition to the capabilities of the models themselves, it is important to consider what impact the different choices have on the computational cost of the simulation, specifically the CPU time and the memory required. Electing to use a model with significantly greater computational cost will result in a dramatic increase in time needed to collect the required results and will also need the use of computers with far more processing power. The k-ε group of models generally require less computational effort than k-ω, of which the standard model is the least intensive and the realisable model requires the most processing ability. The greatest computational demand comes from SST k-ω [52].

3.2.3. Chosen Model

After analysing the various turbulence models available, the standard k-ε has been chosen, as this is a generally accepted model for a variety of applications and has been validated by experimental data for similar tidal applications and has a relatively small computational cost. Despite this, there are several limitations of the k-ε model, and it is important to be aware of what they are. Previously mentioned were the issues involving large pressure gradients and wall functions, but another important disadvantage to note is that k-ε is known to have problems correctly calculating the spreading rate of jets, which is important in this application [54]. For a full analysis, a range of models should be tested against each other and their weaknesses and strengths evaluated for the geometry in question, but for this project, only k-ε will be used, as this has been predicted to perform well under the circumstances.

3.3. Meshing

Meshing is one of the most important aspects of the CFD process, as it can dramatically alter the results of the simulation and affect the accuracy of the results. Included in the process is the necessity for adequate refinement of the mesh, so that its resolution is high enough for the application, and quality checking, to ensure that awkward shaped cells, which would reduce the accuracy of the results, are removed.

3.3.1. Generation and Refinement

It is vital to perform a mesh convergence study in order to identify the optimum mesh size to run the simulations at. The more refined the mesh is (and therefore the smaller the cells in the mesh are), the more accurate the result will be; however, this also means that the simulations will take longer to run and require more computational expense. A mesh convergence study involves analysing the output of the mesh with respect to the size of the mesh to identify how the results of the simulation change, and at what point the simulation can be deemed accurate enough. There is a balance between the size of the mesh and the accuracy of the answer that must be considered; if the mesh is too big, the output will be inaccurate, but if the mesh is refined too far unnecessarily, processing power will be wasted, and the simulation will run for longer than needed. This study will identify the point at which the results no longer change significantly, and therefore can be used as the mesh size to produce the required level of accuracy.

There are a variety of methods for performing a mesh convergence study; one method, which is generally the most common, is to choose a parameter which is of significance to the study, such as one which is required as a result. It can be observed how this output changes with respect to the number of cells in the mesh. At the point at which this value no longer varies significantly, it can be determined that using that number of cells is the optimal solution.

Additionally, mesh convergence can be performed by observing the changes to a range of data points when the mesh size is changed. For instance, the values across a certain line or plane through the model domain can be plotted on a graph and overlaid to identify how the profiles resemble each other as the resolution of the mesh is increased. To identify the optimum mesh resolution for this model, both methods were employed. The velocity profiles across a central line through the model were extracted from the simulation and plotted on the same graph to demonstrate how they converge to a final solution; this is described in more detail in **4.2.2**.

3.3.2. Quality Checking

In addition to properly sizing the cells forming the mesh, there are several checks that should be performed to analyse the quality of the cells themselves, mostly relating to their shape. This further impacts on the quality of the results, as well as the computational power required, and the time to achieve convergence. The first parameter that must be considered is the shape of the cells used to generate the mesh; this can be tetrahedral, hexahedral or polyhedral among other. Hexahedral meshes are more applicable to simple models, whereas tetrahedral meshes are more commonly used for complex models. Following this, the element quality can be inspected by analysing various factors, such as the element quality, aspect ratio, skewness, smoothness and orthogonal quality.

Element quality is a representation of the shape of the cell, which rates each individual element from 0 to 1, where 1 is a higher quality element, producing a perfectly accurate shape and where 0 is completely misshapen. Element quality is calculated as follows [48]:

$$Element\ Quality = C\left[\frac{Volume}{\sqrt{[\sum\ (Edge\ Length)^2\ ]^3}}\right] \tag{3.7}$$

Where C is a constant depending on the shape of the element. The quality should be maximised to ensure a well-structured mesh.

The aspect ratio of an element is the ratio of the longest dimension to the shortest dimension, where the dimensions are from the centre of a face to the central point of the cell, and from a corner node to the central point of the cell. Therefore, the ratio will be equal to 1 for a perfect triangle or square. A ratio as close to 1 as possible is desirable [48].

$$Aspect\ Ratio = \frac{Longest\ Dimension}{Shortest\ Dimension} = \frac{A}{B} \tag{3.8}$$

Skewness indicates how close a cell is to being perfect, where 1 indicates a degenerate cell, and 0 demonstrates a perfectly equilateral cell. Skewness can be calculated using **Equation 3.9** [48]:

$$Skewness = \frac{Optimal\ Cell\ Size - Cell\ Size}{Optimal\ Cell\ Size} \tag{3.9}$$

It is recommended that the maximum skewness of any cell in the mesh is kept below 0.95, but this should clearly be minimised to be as close to 0 as possible [48].

Finally, the orthogonal quality is a very important evaluator of the quality of the mesh. It is a calculation involving vectors between the central point of the cell and various centroids of adjacent cells. The optimal value for a cell is 1, and an unacceptable value is 0.

3.3.3. Error in CFD

Whilst CFD is a powerful method of calculating properties of fluids and has commonly been validated, it is important to be aware of the limitations of the software, which can fall into a number of categories such as numerical errors, modelling errors and convergence errors.

Modelling errors refer to the disparity between the model created in ANSYS and the actual physical properties of the flow. There is much room for error in this, as it is impossible to model properties of fluids such as turbulence with complete accuracy, and therefore approximations have to be made. Furthermore, manual inputs such as the boundary conditions and the description of the geometry can lead to uncertainty if they are not properly defined [55]. This highlights the importance of ensuring that the correct turbulence model is selected and checking that the geometry of the problem is accurately represented. Further errors can also be introduced due to the way CFD codes operate by approximating the exact equations with algebraic equivalents, in a process known as discretisation. The discretisation error is therefore the difference between the exact result of the original equations and the solution of the approximated equation [56]. An increase in the resolution of the mesh can reduce this error.

Finally, convergence errors will arise; these are the difference between the solution of the discretised equations and the solution generated by the iterative process. The residuals (which represents the error in the solution) will reduce gradually with time, but it is impossible to fully reduce these values to zero, and therefore there will always be some discrepancy between the true answer and the calculated answer. These errors can be reduced by allowing the simulation to run for greater amounts of time, but this further increases the need for greater processing power [56].

3.4. The Volume of Fluid Method

Volume of fluid is a method for modelling how two fluids interact with each other, and how the interface between the two fluids change. This is a method for modelling multiphase flow, as it models two phases simultaneously [48]. It can be used for applications where there is mixing of such fluids, for instance bubbles in a fluid and water flowing through a dam. In order to do this, a parameter describing the volume fraction must be incorporated into the equations, which ranges between 0 and 1, where 0 implies that cell is comprised only of one of the fluids (for instance water), and 1 indicates that the cell is filled with the other fluid. Therefore, if the value lies between 0 and 1, this describes the fraction of the cell that is filled with each fluid [57].

Although using this method has several advantages, such as a more accurate result and its ability to capture the true nature of the physics of the flow, this also creates several disadvantages; it comes with a significantly greater computational cost and can cause great difficulty when used in three dimensions [57]. There is potential for its use in this project as there will be mixing of fluids, particularly on the top face of the model where water and air interact. However, in this application it may not be required, as the area of interest is at the bottom of the model and may be situated far enough away from the free surface on the top of the model for this not to affect the results. Its need can be assessed by calculating the Froude numbers for the flow.

3.4.1. Calculation of Froude Numbers

The Froude number is a parameter which describes the ratio between the inertial fluids of the fluid and the forces that gravity exerts on the fluid. Physically, it evaluates the likely distortion of the free surface due to the flow conditions. The Froude number is defined by **Equation 3.10** [58]:

$$Fr = \frac{Inertial\ Forces}{Gravitational\ Forces} = \frac{u}{\sqrt{gl}} \tag{3.10}$$

Where u is the velocity of the fluid, g is the acceleration due to gravity and l is a characteristic length. In this case, the characteristic length is often defined as a hydraulic mean depth (**Equation 3.11**):

$$l = \frac{A}{T} \tag{3.11}$$

Where A is the cross-sectional area of the flow and T is the width of the free surface in the direction of the flow. A Froude number of less than 1 implies that the flow is subcritical, and the gravitational forces dominate. This means that the free surface is calm and stable. However, if the Froude number is greater than 1, then this demonstrates that the flow is supercritical, and the inertial forces have greater influence on the flow, and therefore the flow is less stable, and this can lead to the formation of bubbles via aeration [59].

In the case of the initial model used for mesh convergence and the model used for investigation for current velocity, the dimensions of the channel used to model the open estuary are 50m by 30m, leading to a cross-sectional area of 1500 m^2 and the width of the free surface is 50m; this leads to a hydraulic mean depth of 30. The greatest flow velocity used in the analysis of current speed is 1.2 m/s, and therefore the Froude number can be calculated using **Equation 3.10**:

$$Fr = \frac{1.2}{\sqrt{9.81 \times 30}} = 0.0699$$

Therefore, as the Froude number is well below the critical value of 1, this proves that the gravitational forces have a greater effect on the flow than the inertial forces, and so the free surface will not be affected. Furthermore, the maximum Froude number associated with the analysis of changing tidal height is 0.0427 from a height of 14m, which is also well below the critical value.

The distortion of the water surface due to the turbine is not great enough to cause sufficient mixing of the air and the water, and therefore it is not necessary to use a multiphase calculation. Therefore, a single-phase calculation will be employed in this project. In place of modelling the free surface with the volume of fluid method, a no slip boundary condition will be applied to the top face as described in **Section 3.1**.

4. Experimental Work

4.1. Modelling

All geometry creation was completed using DesignModeler, which is one of the built-in packages in ANSYS for modelling. While there is a wide variety of other options for modelling, meshing and post-processing in CFD, the standard modules have been used as they connect easily to one another and allow for continuous use without importing or exporting of files. The design of the model was based on artists' impressions of the lagoon turbine housing (**Figures 4.1** [60] **and 4.2** [61]). Rather than model the full housing and waste computational resources, purely the fluid domain was modelled; this includes the large section representing the body of water immediately outside of the housing, and the small rectangular section which represents the output from the turbine. If the housing had been of a more complex shape, for instance a sloping dam, it would have been necessary to incorporate this as it would have likely affected the outcome, however as the housing is merely a sloping wall, this will not impact the flow patterns greatly. Firstly, a 2D model was created for use in the 2D mesh convergence, with a 3D model created later for full analysis; **Figures 4.3** and **4.4** show engineering drawings of this model.

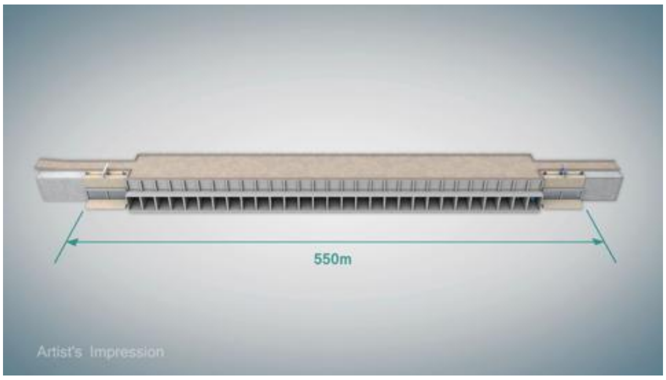


Figure 4.1: Artist's impression of the front view of the turbine housing

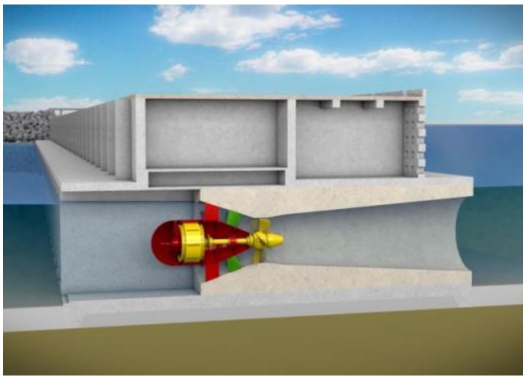


Figure 4.2: Artist's impression of a cross-section of the turbine housing

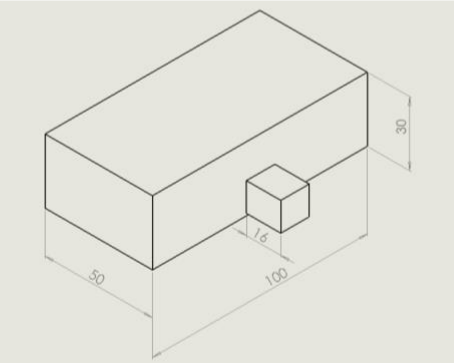


Figure 4.3: Isometric view of the engineering drawing for the 3D model

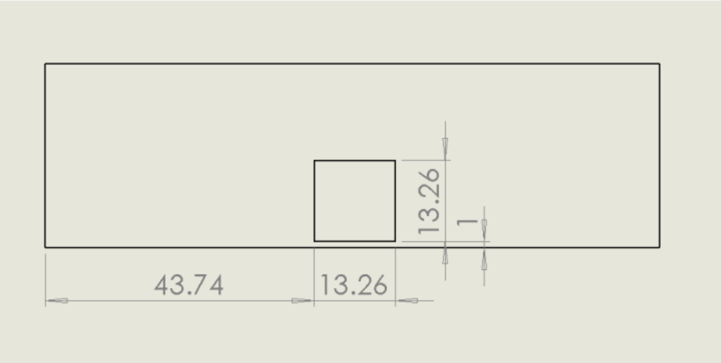


Figure 4.4: Side view of the engineering drawing for the 3D model

The dimensions used for the model were derived from detailed design drawings completed by Atkins in 2014 [62]. **Figure 4.5** displays the completed 3D model in DesignModeler; the red arrows were added to indicate the current flow past the turbine housing, and the orange arrows to represent the flow out of the turbine.

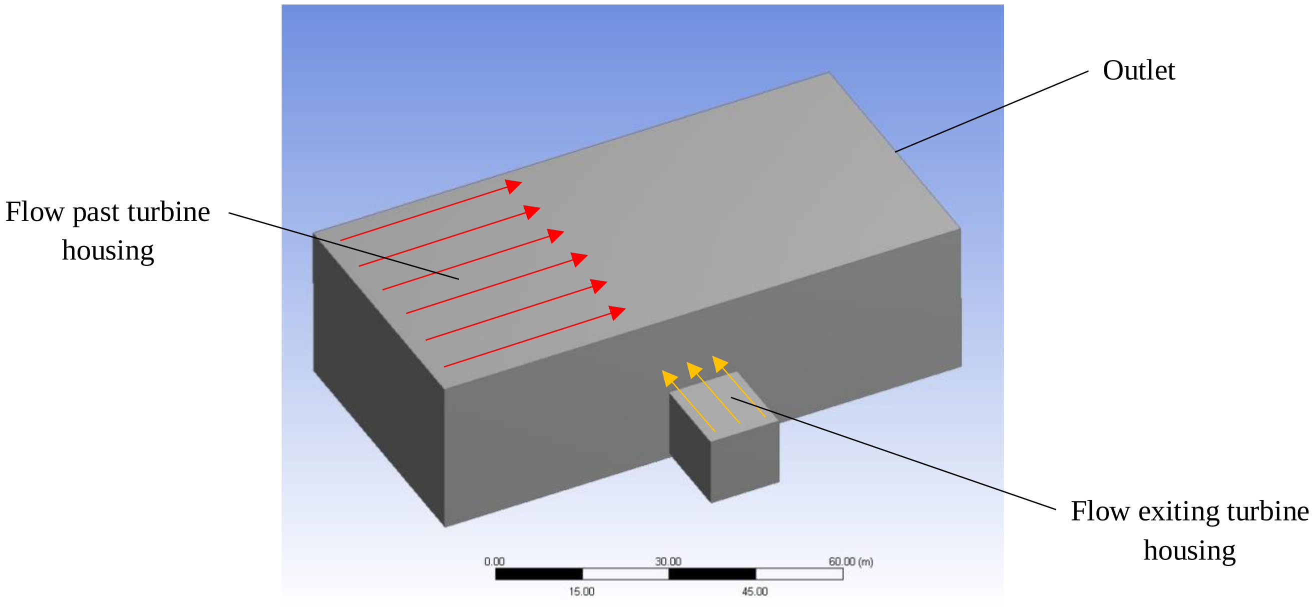


Figure 4.5: The 3D model created in ANSYS representing the flow past the turbine housing

In order for the simulation to operate as expected, there are a number of parameters which must be defined and selected as necessary, such as the chosen turbulence model, named selections and initial conditions. For the turbulence model, the standard k-ε model was selected.

In the meshing process, certain areas of the model must be identified as 'named selections' in order to define important aspects of the body, such as the inlets, outlets and walls. For instance, one end of the channel was defined as a velocity inlet, along with the exit from the turbine housing, and the other end of the channel was labelled a pressure outlet. Furthermore, the outside edges of the model were labelled as walls; importantly, the top face of the model includes a 'no slip' condition, as opposed to modelling it as a free surface and using the VOF method. Finally, the bottom of the model (the 'sea bed') was defined as a separate selection, so that it could be used later in the results section, such as for defining contour plots on that face.

To change the materials of the simulation, the Fluent material database was used - water (liquid) was selected. Using the options for cell zone conditions, the fluid for the whole model was changed to water.

For the mesh convergences, the initial velocity conditions were set to arbitrary values; 0.5 m/s for the current flow past the turbine housing, and 1.2 m/s out of the turbine. For the 3D simulations, these were adjusted to different values to investigate their effects on the results.

4.2. Meshing

4.2.1. Mesh Generation

Mesh generation was performed in Mesher, which is a module built in to ANSYS. To achieve the required mesh resolution, a sizing function was inserted across the whole body, which allows the size of the cells across the model to be adjusted.

4.2.2. Mesh Convergence

To identify the optimum mesh resolution for this model, a mesh convergence study was performed as described in **Section 3.3.1.** Firstly, a mesh convergence study for the 2D model as demonstrated in **Figure 4.6** was used to gain an estimate of what mesh would be needed. The velocity profiles across a central line through the model were extracted from the simulation (by creating an xy plot of the velocity magnitude against the position throughout the model, and then exporting this information into the form of a text file, from which the raw sampled data can be used to create the plots) and plotted on the same graph to demonstrate how they converge to a final solution. The figure demonstrates how the velocity magnitude across the line varies for maximum cell sizes of the mesh ranging from 10m to 1m.

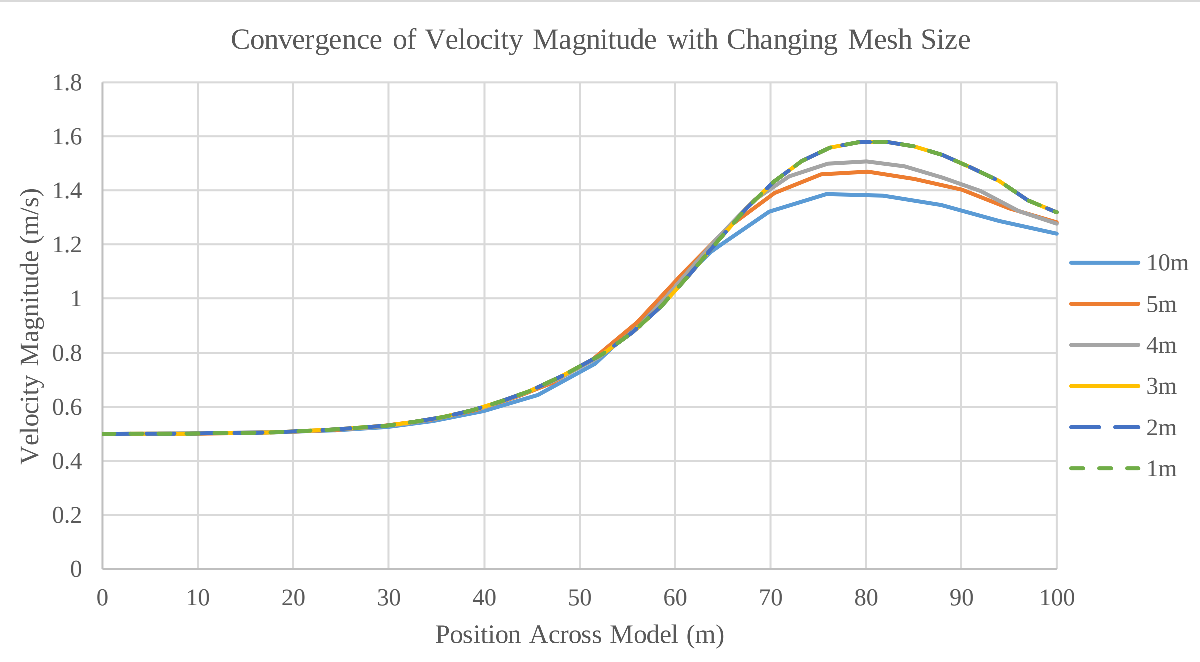


Figure 4.6 – Mesh convergence analysis for the 2D model with maximum cell sizes ranging from 10m to 1m

It can clearly be seen that the profiles for 3m, 2m and 1m all lie directly on top of each other, demonstrating that the solution has converged. This suggests that a 3m mesh could be used for this model to achieve the required level of accuracy; however, the 3D model contains significantly more cells, and therefore the resolution will likely need to be increased. As a result, it was necessary to complete further mesh convergence studies to ensure that the correct cell size was selected. In the same way as with the 2D model, mesh convergence for the 3D model involved plotting the velocity profile across a specified line throughout the model and observing how it changes for varying mesh sizes. This was completed for three different lines across the model, all on the centre line throughout the model;

one just above the sea bed, one halfway up the channel and one just below the top face. The velocity profile for the line in close proximity to the sea bed is shown in **Figure 4.7:**

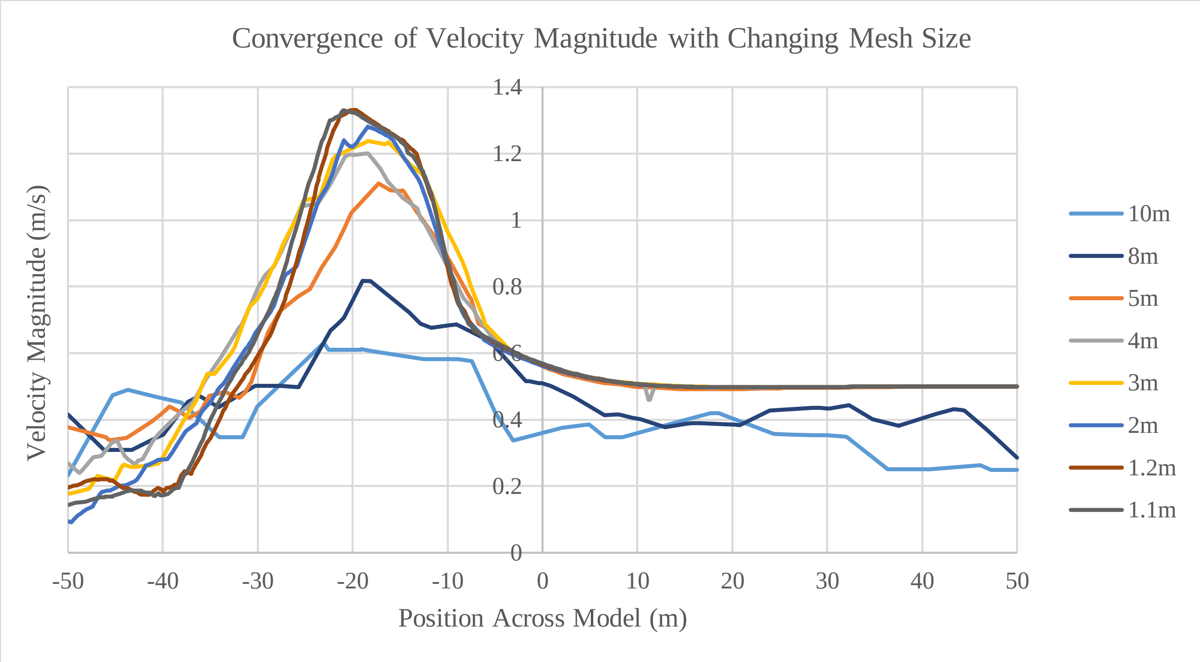


Figure 4.7: Mesh convergence analysis for the 3D model with maximum cell sizes ranging from 10m to 1.1m

It is clear from this graph that as the cell size decreases, the velocity profiles of the flow begin to resemble each other more and more. The difference between the profiles becomes smaller and smaller until 1.2m and 1.1m which resemble each other very closely. Therefore, it can be concluded that 1.2m is an acceptable mesh size to use, as using a more refined mesh will hardly change the outcome. This is validated by both other lines, which display similar characteristics from which it can be shown that 1.2m is the optimum size.

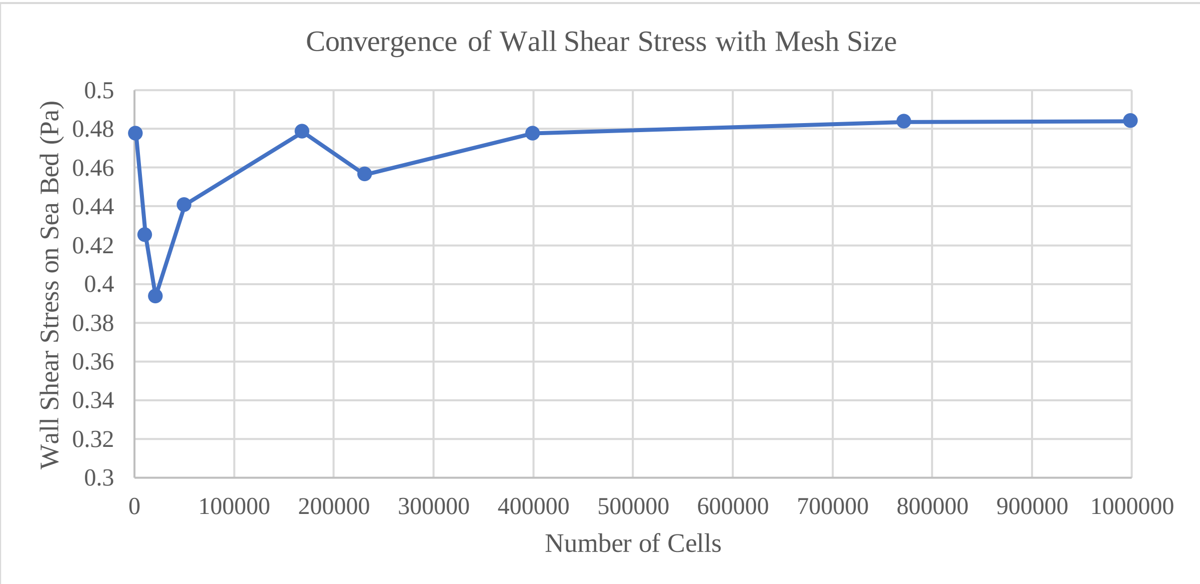


Figure 4.8: Mesh convergence analysis of the 3D model using the changing wall shear stress on the sea bed, with maximum cell sizes of 10m to 1.1m, equating to 1500 and 1 million cells respectively

Finally, it is important to observe the change to the parameter of interest with a varying mesh size. The average wall shear stress on the sea bed was plotted against the number of cells to further check that the chosen mesh would be suitable (**Figure 4.8**). From this, it is evident that at 1.2m (approximately 770,000 cells), the result is close to the stress at 1.1m (approximately 1 million cells). As the result no longer changes after this point, 1.2m can be selected to decrease the computational requirements.

4.2.3. Mesh Quality Analysis

As described in **Section 3.3.2**, it is important to properly check the quality of the mesh to ensure the collected results are as accurate as possible. The parameters considered in this check were the orthogonal quality, aspect ratio, element quality and skewness. This was completed by using the mesh metrics option built in to Mesher. For the best results, the orthogonal quality should be maximised, the aspect ratio should be minimised, the element quality should be as close to 1 as possible and the skewness should be kept below 0.95. The mesh for 1.2m was also compared to 1.1m, to identify how this changes the quality of the mesh. **Table 4.1** displays the findings from the check.

Table 4.1: The mesh quality metrics for the 1.1m mesh and the 1.2m mesh

Mesh Size	Minimum Orthogonal Quality	Maximum Aspect Ratio	Minimum Element Quality	Maximum Skewness
1.1m Mesh	0.162	8.934	0.237	0.838
1.2m Mesh	0.165	10.912	0.205	0.836

Contrary to expectation, some of the parameters for the 1.2m mesh improve upon that of the 1.1m mesh, including a greater orthogonal quality and a lower skewness. While some of the values seem extremely inaccurate, these only apply to a handful of cells, and the vast majority are significantly more well-formed and acceptable for use. The average parameters of the 1.2m mesh are shown in **Table 4.2.**

Table 4.2: Average values of the mesh quality parameters for the 1.2m mesh

Average Orthogonal Quality	Average Aspect Ratio	Average Element Quality	Average Skewness
0.78275	1.8232	0.84266	0.2159

4.2.4. Inflation Layers and Refinement

To accurately model the flow near the boundary layers, inflation layers can be used in the meshing process. Due to the no-slip condition at the wall, the fluid will have a velocity of 0, and therefore the velocity will decrease non-linearly to zero as it nears the wall. It is important to model this situation correctly to ensure that the result is accurate. Another technique to increase the accuracy of the mesh is edge sizing, which has been incorporated into the model to further advance the resolution. This has been used at the intersection between the exit from the turbine housing into the main channel flow of the sea, increasing the number of divisions between the two bodies.

The use of inflation layers was investigated, involving the calculation of the first layer height according to the relevant y+ values, and, if used, would have allowed for a greater increase in the accuracy of the modelling of the sea bed floor. However, after their implementation, the mesh metrics were analysed, and it was clear that they negatively impacted the quality of the mesh; for instance, the minimum element quality was lowered to 0.00144, the maximum aspect ratio was increased to 1260 and the minimum orthogonal quality was increased to 0.0203. This is due to the element size of the model being set to 1.2m, but the layers of the inflation layers being considerably smaller than this, with the first layer being 1.62mm. This results in extremely flat tetrahedral cells and causes poor quality metrics. This could be rectified by using local sizing in this area and dramatically decreasing the size of the cells to match the width of the layers; however, the amount of computational processing power that would be needed in the calculation of the solution with such a vastly increased number of cells would be very great. Therefore, this method has been deemed too computationally intensive for this project, and inflation layers will not be incorporated into the boundary. The edge sizing however does not significantly impact the mesh metrics of the model, and in some cases increases the quality, and therefore this will be incorporated into the model. The edge sizing can be seen in **Figure 4.9:**

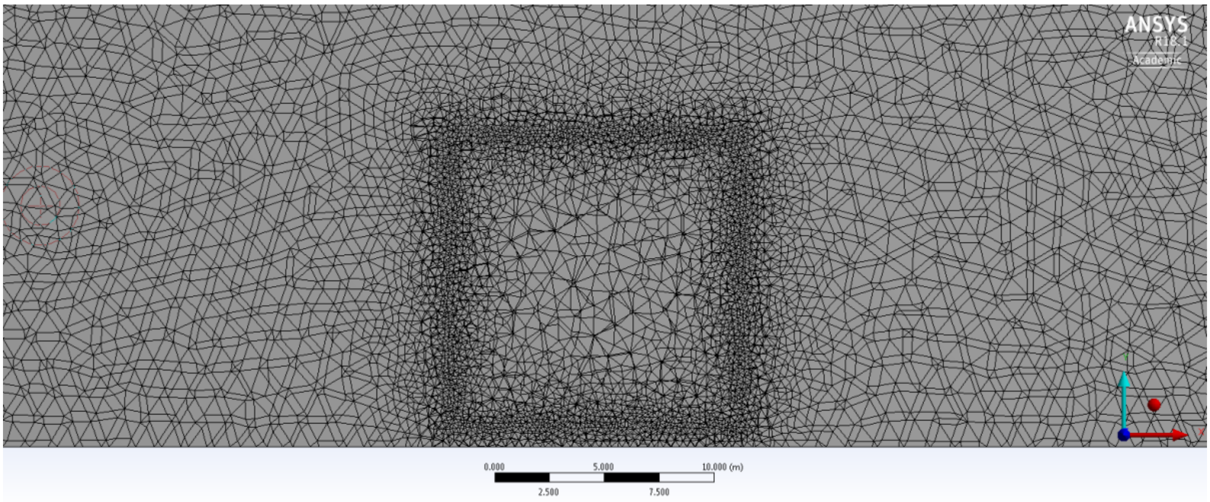


Figure 4.9: Cross-section of the 3D mesh displaying the increased refinement around the turbine housing intersection with the channel

4.3. Running the Solver

The final step before running the simulation was to set the initialisation options. There are two initialisation methods that can be chosen when running a simulation; hybrid and standard. Hybrid initialisation is when initial estimates of the values are unknown and Fluent is required to create an initial estimate using the boundary conditions, whereas standard initialisation uses an initial estimate provided by the user. Hybrid initialisation will be used in these simulations, as this allows for a better approximation of the whole area and computes more information, rather than relying on an initial guess from only one of the inlets.

For this simulation, the residuals were turned off and the simulation was allowed to run with no convergence criteria. This was because the residual values at which the solution has fully converged is unknown and turning off the residuals allows the solution to run indefinitely, and so the point of convergence can be identified. After initialising, the number of iterations for this solution was chosen. For all simulations, this was set to 10,000 to ensure that the convergence was fully captured, but they were often stopped manually at approximately 5,000 iterations, at which point they had usually converged.

4.4. Post-Processing

For obtaining numerical results, such as average and maximum wall shear stresses, this was computed using the 'surface integrals' option, with area-weighted average and facet average for the average results, and facet maximum and vertex maximum for the maximum results. These are simply different calculation methods based on faces (facet), nodes (vertex) and the area of the plane (area-weighted). The contours, streamlines and other plots will be developed using ANSYS' built-in processing package, CFD Post. 20 simulations were completed, each of which took two hours, to obtain a full range of data from varying velocity speeds and changing tidal heights.

5. Presentation of Experimental Results

5.1. The Effect of Current Velocity

The simulations developed an iterative solution to each case, and the simulation was stopped when it was deemed convergence had been achieved, which is most clear by observing when the residuals become constant. **Figure 5.1** demonstrates the residuals for one case, and it is clear that at 3500 iterations, all residual values have become constant.

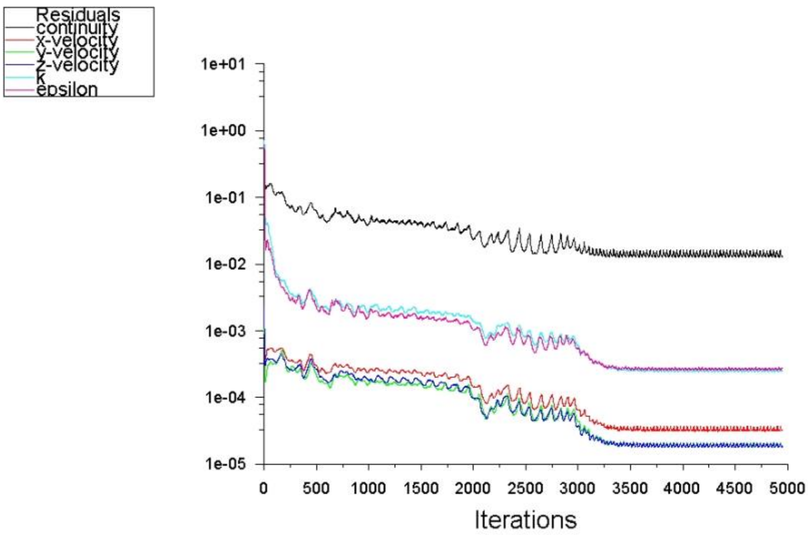


Figure 5.1: Residuals plot against iterations for the case of current flow at 0.9 m/s

Firstly, the impacts of current velocity flowing past the turbine housing and its interaction with discharge from the turbines was investigated. This was achieved by varying the velocity of the current from 0.1 m/s to 1.2 m/s, assuming a constant turbine output of 1 m/s. The current flow speeds have been derived from a hydrodynamic model of the bay completed by Intertek [63], which indicates that the flow speeds range widely, peaking at approximately 1.2 m/s. From operating these simulations, the effect on the wall shear stress, including maximum points and averages across the whole plane, were identified using contours and graphs. **Figure 5.2** displays a graph analysing the effect of increasing current flow velocity on the maximum shear stress on the sea bed. This has been calculated for two types of maximums in CFD; facet maximum and vertex maximum. **Figure 5.3** displays the relationship between the average wall shear stress across the whole sea bed and the current flow velocity.

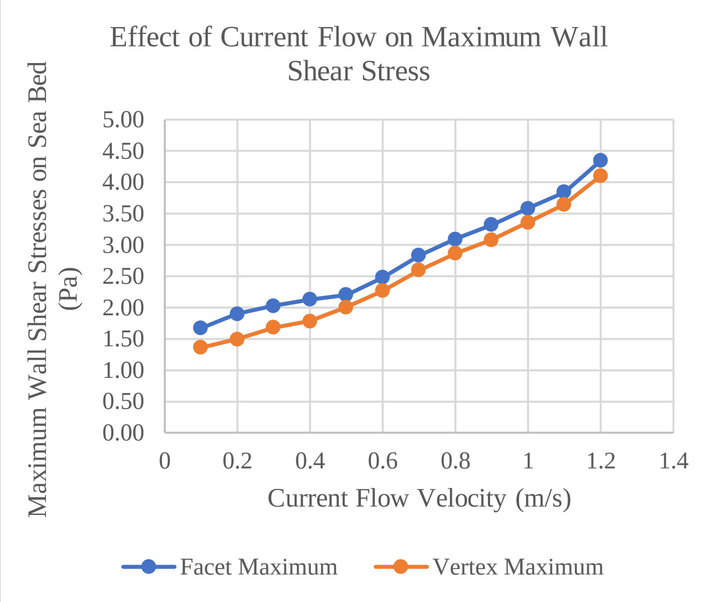


Figure 5.2: Facet and vertex maximum wall shear stresses plotted against flow speed past the housing

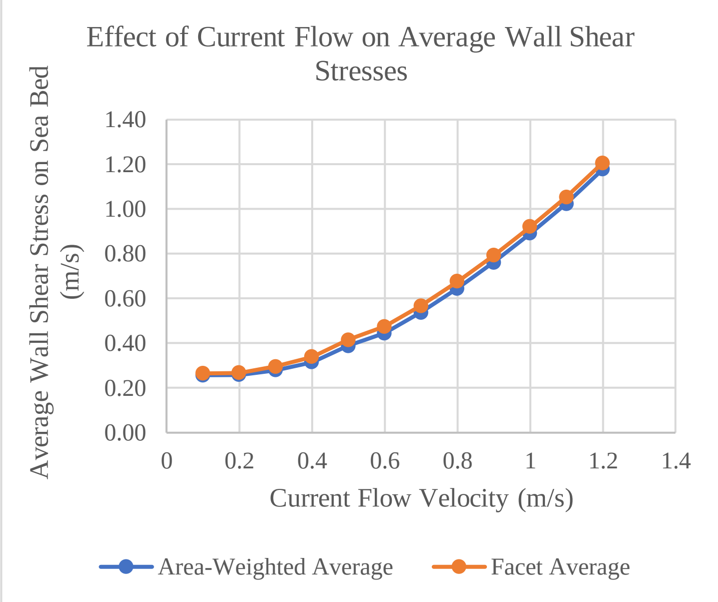


Figure 5.3: Area-weighted and facet average wall shear stresses plotted against flow speed past the housing

From these simulations, wall shear stress contours were produced for the sea bed (**Figures 5.4** and **5.5**).

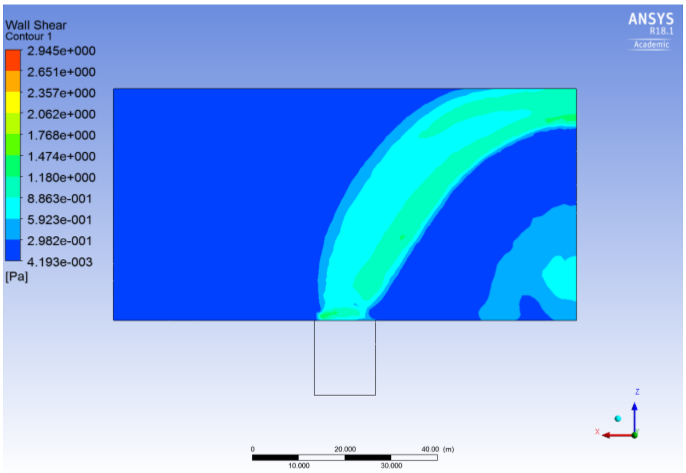


Figure 5.4: Wall Shear Stress contour on the sea bed at a current velocity of 0.2 m/s

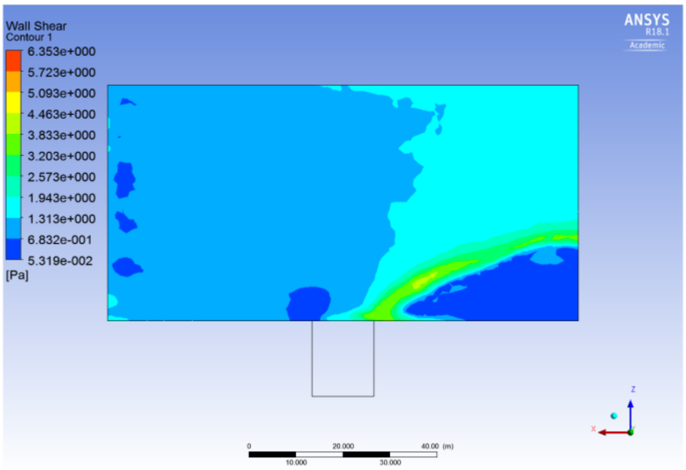


Figure 5.5: Wall Shear Stress contour on the sea bed at a current velocity of 1.2 m/s

Additionally, plots of the turbulent kinetic energy were created to identify areas of high energy (**Figures 5.6** and **5.7**). These were calculated on a horizontal plane across the model which intersects the centre of the turbine housing to ensure that the full effects of the turbine jet were realised.

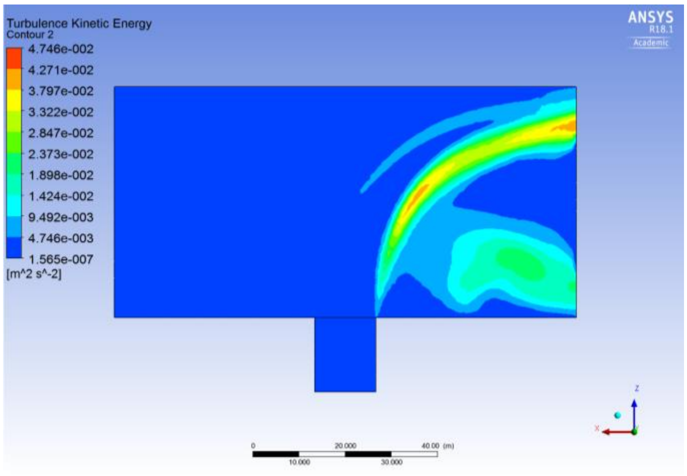


Figure 5.6: Turbulent Kinetic Energy contour on a plane in line with the centre of the turbine exit at a current of 0.2 m/s

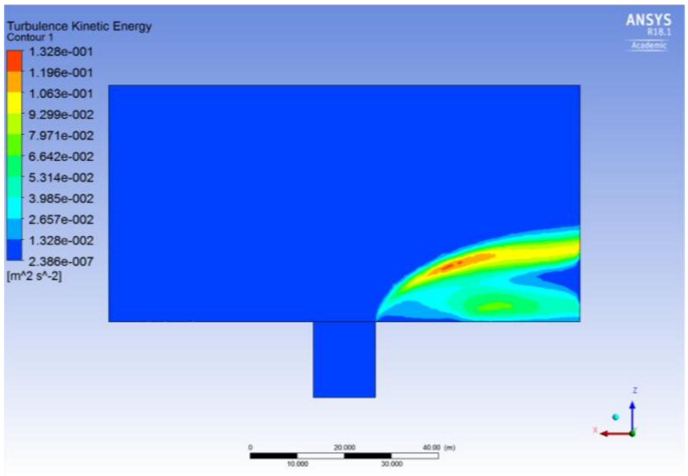


Figure 5.7: Turbulent Kinetic Energy contour on a plane in line with the centre of the turbine exit at a current of 1.2 m/s

Figure 5.8 demonstrates the velocity vectors on a cross-section through the model, and clearly displays the interaction of the flow leaving the turbine housing with the current flowing past it. The turbine flow is evidently influenced by the current and is diverted to flow in the same direction as it, and this leaves a region to the right of the exit where the flow becomes more unpredictable. There are clear formations of vortices in this region, giving rise to the high turbulent kinetic energy.

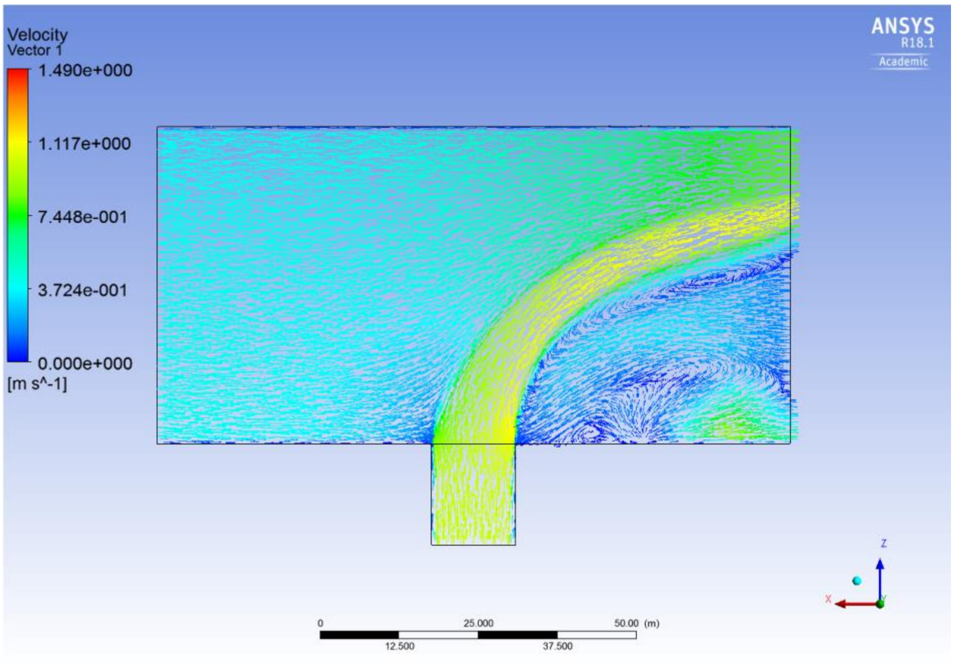
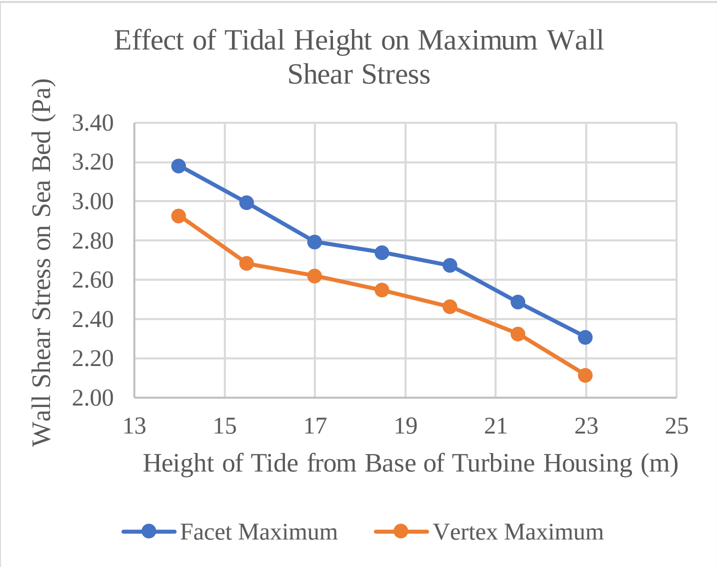


Figure 5.8: Velocity vectors on a plane in line with the centre of the turbine at a current speed of 0.4 m/s, demonstrating the formation of vortices due to the interaction of the jets

5.2. The Effect of Tidal Height

The effect of the varying tidal height on the hydrodynamics of the area was also examined. The height ranged from low tide, which has been estimated at 14m above the base of the turbine housing, up to a high tide of 23m. These estimations are based on detailed design drawings by Atkins [62]. During this, the current flow speed was kept constant at 0.5 m/s and the flow from the turbine at 1 m/s. The effect of this on the wall shear stress on the sea bed, the turbulent kinetic energy and the turbulent dissipation rate has been recorded. **Figure 5.9** displays the graph of how the changing tides affect the wall stress on the sea bed; from this, it is clear that the maximum stress occurs when the sea is at low tide. However, the average wall shear stress (**Figure 5.10**) does not appear to have a correlation with the tidal height, with the shear stress initially decreasing with the rising tide until 17m, beyond which the shear stress increases again, peaking at 20m, before finally decreasing again. This is contrary to expectation, as it would be assumed that it would simply decrease in the same way that the maximum stress decreases.



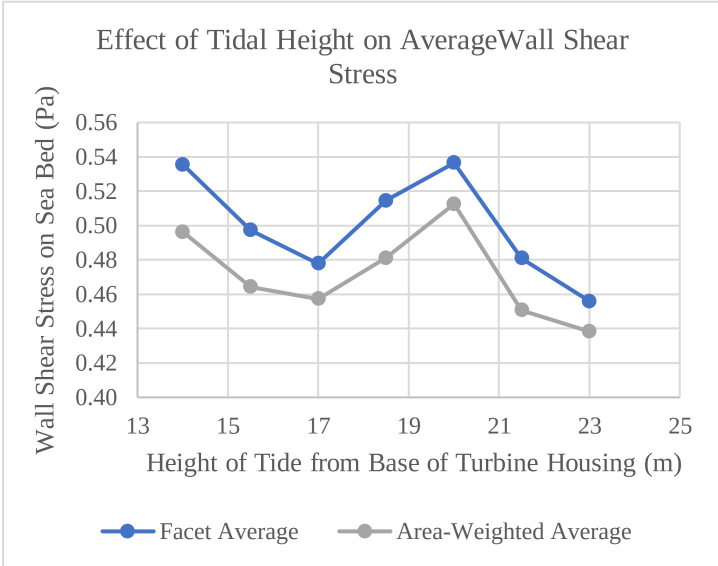
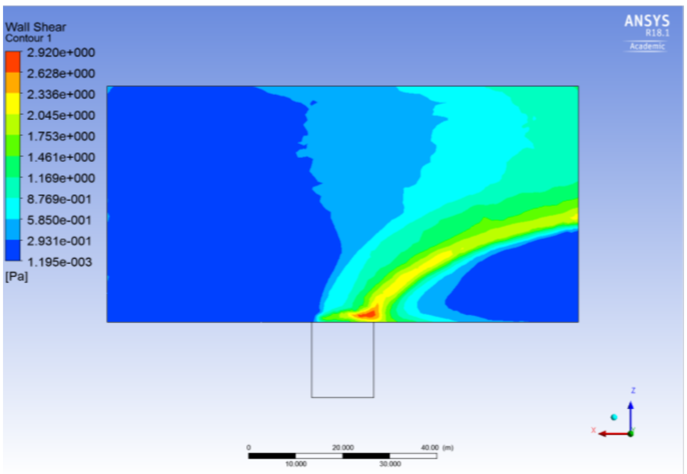


Figure 5.9: Facet and Vertex maximum wall shear stresses plotted against the tidal height

Figure 5.10: Facet and Area-Weighted average wall shear stresses plotted against the tidal height

Figures 5.11 and **5.12** demonstrate the contrast between the wall shear stress produced on the sea bed at low tide (14m) and at high tide (23m).



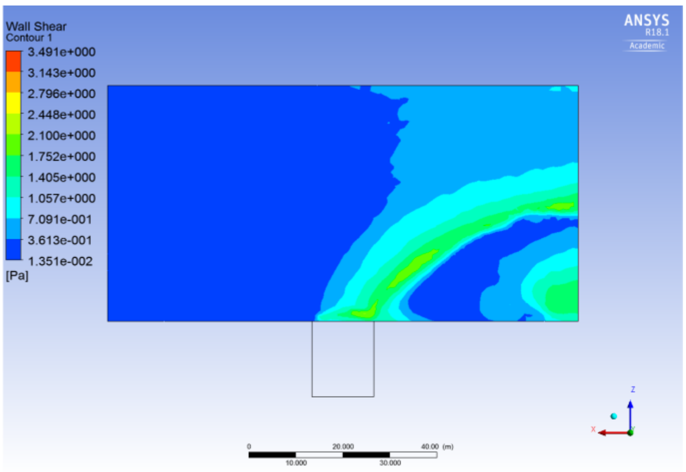


Figure 5.11: Wall Shear Stress contour on the sea bed at a tidal height of 14m

Figure 5.12: Wall Shear Stress contour on the sea bed at a tidal height of 23m

6. Discussion and Conclusions

6.1. *Findings*

From the results, it is evident that the maximum wall shear stress on the sea bed increases linearly due to increasing current flow past the turbine; the maximum wall shear stress at 0.1 m/s is 1.67 Pa, increasing to 4.35 Pa at 1.2 m/s. This is as expected – other such studies, including work by Thompson *et al.* [32] for the most part demonstrate that shear stress and velocity are linearly related, although at the higher range of velocities, the shear stress begins to plateau – this may suggest that if the studies were continued at higher velocities for Swansea Bay, higher velocities would lead to less and less variation in stress. The evidence of increased swirling motion at higher velocities is noticeable in the turbulent kinetic energy plots (**Figures 5.6** and **5.7**); at 0.2 m/s, the kinetic energy has a maximum value of 0.0427 m^2/s^2, compared with 0.133 m^2/s^2 at 1.2 m/s, an increase of over 200%. However, contrary to expectation, the average wall shear stress on the sea bed increases exponentially with the current velocity, where it would be assumed to be linear. Therefore, this implies that at lower velocities, the only regions impacted are the paths directly underneath the jet from the turbine housing, which is a small, contained area, but as the current flow increases, the combination of the two streams combine and exacerbate each other's effects, leading to a greater area of increased shear stress.

Unlike the current speed, an increase in tidal height causes a linear decrease in maximum wall shear stress. It can therefore be concluded that at low tide, the turbine jet has a greater effect on the water and causes it to move more chaotically and give rise to a number of eddies. This gives localised points of high shear stress, of which there are fewer as the tide rises to its maximum. The average wall shear stress appears to have little correlation with the height of the tides. From inspecting the contours, it can be seen that the area the turbine jet has an impact on does not change greatly across different tidal heights. Therefore, the speed of the current has the main effect on average stress across the whole surface, as this defines how the two currents combine, and the severity of the vortices formed. As the current and turbine velocities were kept constant throughout the variation in tide, this meant that the primary area of impact was kept common throughout all the simulations, which could explain the unchanging average wall shear stress. Analysis by Gonzalez-Santamaria *et al.* [33] confirms the conclusions that lower tides lead to greater shear stress; the study establishes that areas of shallow water led to the maximum shear stresses in the area, with deeper waters creating a less significant impact on the bed.

Comparing **Figures 5.2** and **5.9**, it is obvious that the current flow past the turbine housing is of much more significance in terms of stress than the tidal height; from one extreme of velocity to another, the maximum wall shear stress ranges from 1.67 Pa to 4.35 Pa, a difference of 2.68 Pa, whereas the effects of the tide amount to merely an increase of 0.87 Pa from low tide to high tide. Therefore, it is clear that when preparing for environmental impacts, the rarer, more powerful high currents should be investigated, rather than focusing on the relatively consistent tidal pattern.

Validation is an important part of CFD simulations as without reference to existing results or theoretical calculations, it is very difficult to tell whether the results of a simulation are accurate. These simulations have proved difficult to validate as there are no simple equations that can be used to compare the solution with, and there is little research in this area to compare results with. However, there is significant research in the field of jets in crossflow, which was covered in Section 2, which is applicable to this situation. This included the visual identification of flow properties such as vortices; the expected noticeable vortices were wake vortices and wall vortices. The wake vortices can clearly be seen in **Figures 6.1, 6.2 and 6.3**, where there is obvious formation of several vortices downstream from the turbine housing exit. Furthermore, **Figure 6.4** clearly displays a wall vortex just above the top of the turbine exit on the right-hand side of the model. This can be compared to the plots (**Figure 2.1**) and results discussed in **Section 2**; the results of the simulations accurately reflect the published results, and so it can be inferred that the simulations have operated as expected and the outputs will be accurate.

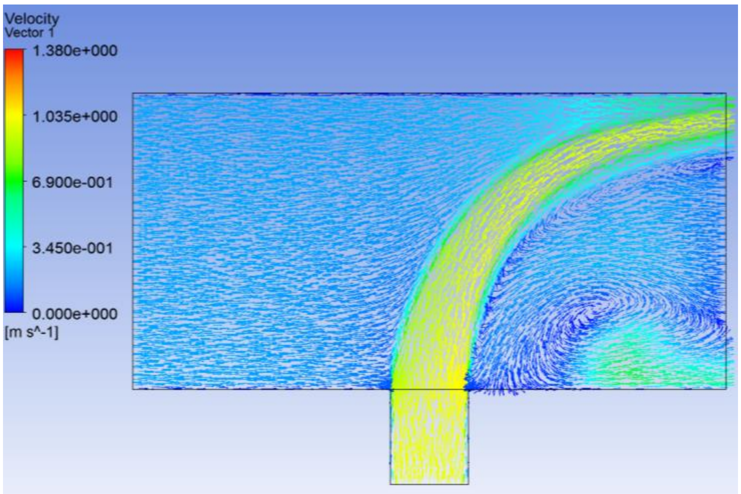


Figure 6.1: Velocity vectors of the turbine flow interacting with the current flowing at 0.2 m/s

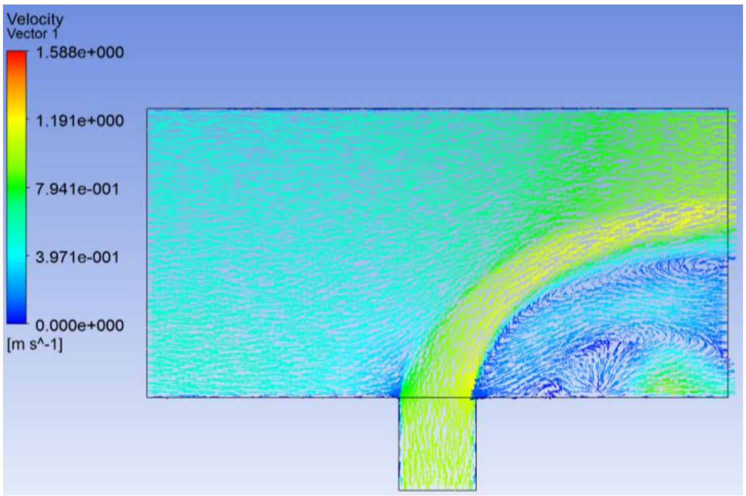


Figure 6.2: Velocity vectors of the turbine flow interacting with the current flowing at 0.5 m/s

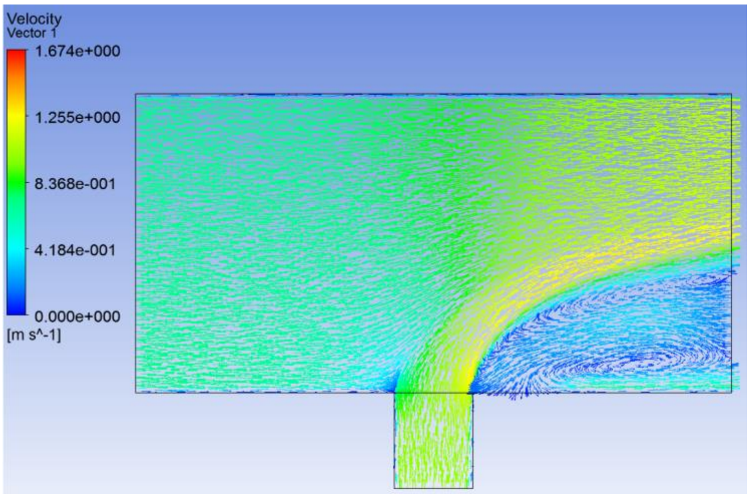


Figure 6.3: Velocity vectors of the turbine flow interacting with the current flowing at 0.7 m/s

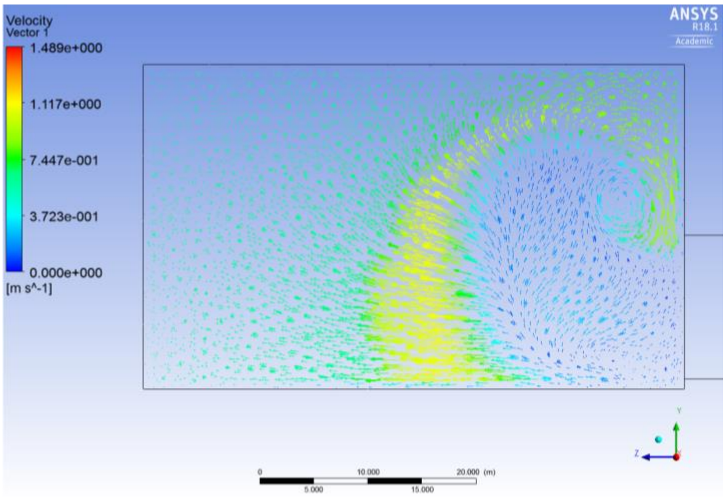


Figure 6.4: Velocity vectors of the turbine flow at a cross section through the model at 0.4 m/s

6.2. Significance of Results

The increased velocity of flow and generated shear stress on the sea bed will have a number of further impacts on the hydrology of the region, most notably on sediment transport. It is well-known that an increase in the shear stress in the downstream direction leads to erosion, and therefore it can be inferred that downstream from the turbine housing exits will be the primary region liable to erosion. This reflects inferences contained in ABPmer's report [23], which state that due to greater velocities in the Swansea Bay area, local scouring and erosion will likely occur, and could also cause the suspension, and therefore transport, of sediment. It is clear from the shear stress contours that the region most heavily impacted follows the path of the jet exiting the turbine housing, and therefore this will cause sections of erosion within this path and cause the sediment to be transported downstream with the current flow. This confirms predictions by TLP's own EIA [26], which highlights that the 'jetting' area is an area of interest and states that in the region in close proximity to the housing, erosion will occur, and there will be increased flow velocities and therefore sediment suspension.

As previously identified, scouring is an important consideration for marine structures as it can lead to their instability and collapse through the removal of sediment. This is clearly relevant in this context, as there is significant formation of vortices which could lead to scouring and the structural collapse of the turbine housing if left unprotected. It is important to be aware of this during construction to allow for any necessary changes to avoid structural damage.

Furthermore, the evidence of increased erosion on the sea bed and dispersion of sediment could lead to the transfer of sediment from the sea floor into the lagoon. Although this process would likely be reversed when the turbine operates in the opposite direction, it is still a consideration that should be investigated further, as it could allow sediment to build up inside the lagoon and affect the operation of the turbines. Erosion has a number of further impacts related to the hydrology of the region, such as the impact on water quality, ecology and navigation patterns. The removal of sediment from the sea bed could cause destruction of habitats for various organisms, and the release of various nutrients contained in this matter, which means certain species of plants may no longer be able to grow there. Additionally, the increased viscosity and velocity of the flow could lead to the dispersion of pollutants and contaminants and carry these to other areas in the estuary. Finally, variations in sediment levels in the Swansea Bay area could impact shipping routes; for instance, if sediment is deposited in certain places significantly enough to build upon the bed of the estuary, this may make existing shipping routes unsuitable for use, which has further impacts in the form of reduced space for boats to pass each other.

Although a number of potential impacts due to the effects of the turbine housing on the sea bed have been presented, Swansea Bay is known to be a region which possesses great variation in sediment patterns with ability to maintain an equilibrium despite changes [28], so it is unlikely that this would have a significant impact on the hydrology of the area and require further protective measures to be enforced. However, it is vital to be aware of these potential impacts so that if, in the future, previously unexpected results begin to occur, there are plans in place to reduce the risk of hazard.

6.3. Limitations of Model and Opportunity for Further Research

Although the model has been successfully used to apply flow theory to the tidal lagoon, there are a number of shortcomings which could be improved on in future work. Firstly, due to time limitations, it was only possible to model the tidal side of the lagoon and observe the effects of the jet operation during ebb flow; however, it would be of further interest to investigate the effects on the lagoon side too, to identify what immediate impact the turbines have on the bed stress without the combination of the current flow past the housing.

Additionally, in this model the current flow was only modelled as a flow in one direction, but the flow should also be considered acting in the other direction, and with the addition of waves to fully realise the flow past the housing. In addition to this, the sea bed was modelled as a smooth surface for ease, as the import of bathymetry data was attempted but unsuccessful. This presents an avenue for further work, as the varying height of the sea floor will further impact the shear stress. Finally, a limitation of this analysis is that while general inferences can be drawn based on the stress contours and flow patterns produced, without collected data and larger scale modelling, the necessary parameters are not able to be found in great enough detail, and therefore numerical evaluations cannot be given.

6.4. Conclusions

Swansea Bay Tidal Lagoon is an extremely interesting proposition, which would have the potential to dramatically change the way energy is generated in the UK and be at the forefront of tidal energy generation across the world. Despite its potential, it is important to clarify the extent to which it could affect the hydrology and ecology of the area.

This study has demonstrated that both the current velocity and the changing tides will have a noticeable impact on the wall shear stress applied to the sea bed. In the case of current velocity, an increase leads to a linearly increasing maximum wall shear stress but an exponentially increasing average wall shear stress, and in the case of tidal height, an increase leads to a linearly decreasing maximum wall shear stress yet has no correlation with the average wall shear stress. Finally, it has been identified that the current velocity past the housing affects the stress on the bed more significantly than the tides do, and therefore when considering the possible impacts of the lagoon on the surrounding ecology, the focus should be on extremes of current flow, which have the capacity to affect the hydrological processes far more.

7. Project Management, Consideration of Sustainability and Health and Safety

7.1. Health and Safety

As the project is not based in laboratories or workshops, this decreases the amount of risks possible. However, there are still considerations due to the amount of desk work being undertaken, which have been analysed in the risk assessment (**Table 7.1**).

7.2. Procedures for Project Management

In order to plan the project and identify how long each individual process would take and in what order they need to be completed, a Gantt chart was used (**Figure 7.1**). An initial chart was created in September 2017 to outline the tasks that needed to be undertaken that term, and as part of the preliminary report, a plan of the tasks for the next term was completed. This was later revised to manage the project in more detail and breakdown the tasks into small sections to identify whether the progress of the project was on schedule or not. Furthermore, the length of each task was slightly overestimated to allow for any unforeseen delays in the project and allow for their resolution without impacting the overall project plan. The Gantt chart breaks down the process into each individual section of the project including the preliminary report, main report and interview. Milestones and deliverables for the project are also shown, including the first project meeting, expected dates of completion of each section, and the submission deadlines.

A further procedure for management used which is vital to the success of the project is the logbook, in which all notes, meeting minutes, calculations and observations were recorded so that they could be reflected upon at a later date. The consolidation of all useful material into a single place allowed for quick access to important information, and meant no time was wasted searching through notes to find the required details. Furthermore, supervisor meetings took place every week, so that problems could be discussed, and solutions developed quickly, ensuring that progress was aligning with the Gantt chart. Finally, weekly tasks were set so that the progress made during the week could be evaluated and tasks set could be adjusted accordingly.

7.3. Sustainability

There is no physical impact on sustainability as part of this project, as there is no use of machines or materials during experimental work, and the project does not focus on the construction aspects of the lagoon, which would inherently impact the environment. Despite this, the nature of the project means that sustainability is at the forefront of analysis, as this project assesses the feasibility of a renewable energy generation method, and therefore aims to decrease emissions and the carbon footprint of the UK.

Table 7.1: Risk assessment identifying the potential hazards of the project.

Risk	Effect	Cause	Likelihood	Severity	Importance	Action to Minimise Risk
Lack of computers available	Unable to use the programs and run simulations	Other user occupying all other computers	1/10	7/10	7%	Use computers at quieter times or install on personal laptop
Insufficient computational power	Inability to run simulations at correct resolution	Computers do not have enough processing power to run program in high enough detail	5/10	6/10	30%	Find computers with greater computational power or reduce demand of task
Loss of work	Waste time re-writing previously completed work	Not correctly saving, unexpected computer errors	6/10	8/10	48%	Ensure work is backed up daily to a hard drive and digitise physical notes
Poor time management	Not running the required amount of simulations and not completing work before the deadline	Not planning deliverables correctly, not following the Gantt chart	3/10	4/10	12%	Revise Gantt chart if problems are encountered and allow room for error
Discomfort and injuries from working at a desk	Repetitive strain injuries (RSI), eyestrain, muscular pains, headaches and poor posture	Not following computer guidelines correctly, not taking regular breaks	4/10	3/10	12%	Ensure that equipment used is ergonomic, ensure that lighting conditions are suitable for looking at a screen, and take regular breaks to reduce the time spent looking at a screen

	Task Name	Duration	Start	Finish	Predecessors
1	**Preliminary Report**	**54 days**	**Mon 02/10/17**	**Fri 15/12/17**	
2	Prepare ideas for project	6 days	Mon 02/10/17	Sun 08/10/17	
3	Initial project meeting	0 days	Mon 09/10/17	Mon 09/10/17	2
4	Plan tasks for report	4 days	Mon 09/10/17	Thu 12/10/17	3
5	Research history of Swansea Bay	6 days	Fri 13/10/17	Fri 20/10/17	
6	Research background theory of tidal lagoons	11 days	Wed 18/10/17	Wed 01/11/17	
7	Read previous literature	11 days	Thu 02/11/17	Thu 16/11/17	5,6
8	**Preliminary Work**	**15 days**	**Mon 06/11/17**	**Sun 26/11/17**	
9	Follow tutorials	6 days	Mon 06/11/17	Sun 12/11/17	
10	Model structure	6 days	Mon 13/11/17	Sun 19/11/17	9
11	Run simulations	6 days	Mon 20/11/17	Sun 26/11/17	10
12	Write report	16 days	Mon 20/11/17	Mon 11/12/17	
13	Preliminary Report Deadline	0 days	Fri 15/12/17	Fri 15/12/17	
14	**Full Report**	**111 days**	**Mon 18/12/17**	**Tue 22/05/18**	
15	Plan deliverables for project	5 days	Mon 18/12/17	Fri 22/12/17	
16	Review report feedback	3 days	Mon 15/01/18	Wed 17/01/18	
17	**Literature Review**	**8 days**	**Mon 08/01/18**	**Wed 17/01/18**	
18	Research Topics	7 days	Mon 08/01/18	Tue 16/01/18	
19	Write section	5 days	Thu 11/01/18	Wed 17/01/18	
20	Finish literature review	0 days	Wed 17/01/18	Wed 17/01/18	
21	**Methodology and Theory**	**11 days**	**Mon 22/01/18**	**Mon 05/02/18**	17
22	Research theory	6 days	Mon 22/01/18	Mon 29/01/18	
23	Write up methodology	6 days	Mon 29/01/18	Mon 05/02/18	
24	Finish methodology section	0 days	Mon 05/02/18	Mon 05/02/18	
25	**Experimental Work**	**22 days**	**Tue 06/02/18**	**Wed 07/03/18**	
26	Model structure	5 days	Tue 06/02/18	Sun 11/02/18	
27	Mesh convergence	5 days	Mon 12/02/18	Fri 16/02/18	26
28	Analyse mesh quality	2 days	Mon 19/02/18	Tue 20/02/18	26,27
29	Run simulations	11 days	Wed 21/02/18	Wed 07/03/18	28
30	Finish experiments	0 days	Wed 07/03/18	Wed 07/03/18	
31	**Results**	**5 days**	**Thu 08/03/18**	**Thu 15/03/18**	
32	Collect numerical values	3 days	Thu 08/03/18	Mon 12/03/18	29
33	Post-process images	5 days	Thu 08/03/18	Wed 14/03/18	29
34	Finish results section	0 days	Thu 15/03/18	Thu 15/03/18	
35	**Discussion and Conclusions**	**10 days**	**Thu 15/03/18**	**Wed 28/03/18**	
36	Analyse data	6 days	Thu 15/03/18	Thu 22/03/18	32,33
37	Draw conclusion on hydrological impact	4 days	Fri 23/03/18	Wed 28/03/18	36
38	Finish conclusions section	0 days	Wed 28/03/18	Wed 28/03/18	
39	**Report Writing**	**62 days**	**Mon 12/02/18**	**Wed 09/05/18**	
40	Write up main report	51 days	Mon 12/02/18	Mon 23/04/18	
41	Format report	5 days	Tue 24/04/18	Sun 29/04/18	40
42	Complete draft	0 days	Mon 30/04/18	Mon 30/04/18	41
43	Read through report	6 days	Mon 30/04/18	Sun 06/05/18	42
44	Submission	2 days	Mon 07/05/18	Tue 08/05/18	43
45	Final report deadline	0 days	Wed 09/05/18	Wed 09/05/18	44
46	**Interview**	**6 days**	**Mon 14/05/18**	**Tue 22/05/18**	
47	Prepare for interview	6 days	Mon 14/05/18	Sun 20/05/18	
48	Interview	0 days	Tue 22/05/18	Tue 22/05/18	

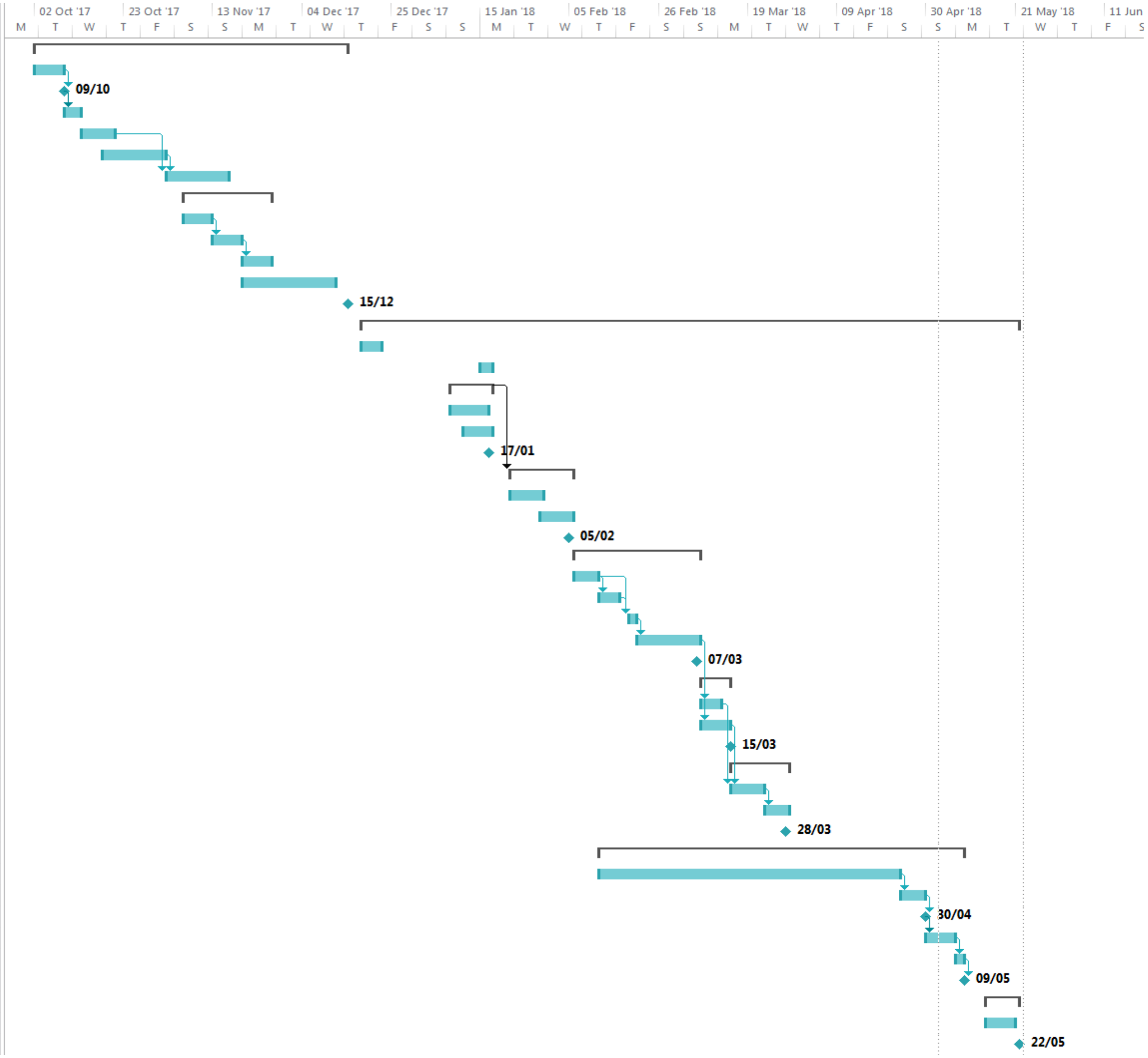


Figure 7.1: The full Gantt chart for the project from October 2017 to May 2018

References

[1] G. Aggidis and S. Waters, “A World First: Swansea Bay Tidal lagoon in review,” *Renewable and Sustainable Energy Reviews,* vol. 56, pp. 916-921, April 2016.
[2] Tidal Lagoon Power. (n.d.). *An iconic, world-first infrastructure project in South West Wales* [Online]. Available: http://www.tidallagoonpower.com/projects/swansea-bay/ [Accessed 14th December 2017].
[3] European Commission. (n.d.). *Renewable energy* [Online]. Available: https://ec.europa.eu/energy/en/topics/renewable-energy [Accessed 13th December 2017].
[4] Department for Business, Energy and Industrial Strategy, *“Digest of United Kingdom Energy Statistics 2017”* London: UK Government, 2017.
[5] BBC. (2013). *UK tidal power has huge potential, say scientists* [Online]. Available: http://www.bbc.co.uk/news/science-environment-20983645 [Accessed 13th December 2017].
[6] J. Xia, R.A. Falconer and B. Lin, “Hydrodynamic impact of a tidal barrage in the Severn Estuary, UK,” *Renewable Energy,* vol. 35, issue 7, pp. 1455-1468, July 2010.
[7] J. Brammer, R.A. Falconer, C. Ellis and R. Ahmadian, “Physical and numerical modelling of the Severn Barrage,” *Science China Technological Sciences,* vol. 57, issue 8, pp. 1471-1481, August 2014.
[8] A. Cornett, J. Cousineau, I. Nistor, “Assessment of hydrodynamic impacts from tidal power lagoons in the Bay of Fundy,” *International Journal of Marine Energy,* vol. 1, pp. 33-54, April 2013.
[9] C. Ferrarin and G. Umgiesser, “Hydrodynamic modeling of a coastal lagoon: The Cabras lagoon in Sardinia, Italy,” *Ecological Modelling,* vol. 188, issues 2-4, pp. 340-357, November 2005.
[10] M. Kadiri, R. Ahmadian, B. Bockelmann-Evans, R.A. Falconer and D. Kay, “An assessment of the impacts of a tidal renewable energy scheme on the eutrophication potential of the Severn Estuary, UK,” *Computers & Geoscience,* vol. 71, pp. 3-10, October 2014.
[11] M. Bielecka and J. Kazmierski, “A 3D mathematical model of Vistula Lagoon hydrodynamics – general assumptions and results of preliminary calculations,” in: *Proceedings of the 7th International Specialised Conference on Diffuse Pollution and Basin Managements and 36th Scientific Meeting of the Estuarine and Coastal Sciences Association (ECSA),* University College, Dublin (2003).
[12] S. Tatum, M. Allmark, C. Frost, D. O’Doherty. A. Mason-Jones and T. O’Doherty, “CFD modelling of a tidal stream turbine subjected to profiled flow and surface gravity waves,” *International Journal of Marine Energy,* vol. 15, pp. 156-174, September 2016.
[13] A. Mason-Jones, D.M. O’Doherty, C.E. Morris, T. O’Doherty, C.B. Byrne, P.W. Prickett, R.I. Grosvenor, I. Owen, S. Tedds, R.J. Poole, “Non-dimensional scaling of tidal stream turbines,” *Energy,* vol. 44, issue 1, pp. 820-829, August 2012.
[14] C. Morris, *“Influence of Solidity on the Performance, Swirl Characteristics, Wake Recovery and Blade Deflection of a Horizontal Axis Tidal Turbine,”* PhD Thesis, School of Engineering, Cardiff University, Cardiff, 2014.
[15] A. Mason-Jones, *“Performance assessment of a Horizontal Axis Tidal Turbine in a high velocity shear environment,”* PhD Thesis, School of Engineering, Cardiff University, Cardiff, 2009.
[16] R. Ahmadian, R. Falconer, B. Bockelmann-Evans, “Far-field modelling of the hydro-environmental impact of tidal stream turbines,” *Renewable Energy,* vol. 38, issue 1, pp. 107-116, February 2012.
[17] S.P. Neill, E.J. Litt, S.J. Crouch, A.G. Davies, “The impact of tidal stream turbines on large-scale sediment dynamics,” *Renewable Energy,* vol. 34, issue 12, pp. 2803-2812, December 2009.
[18] B.M. Sumer, R.J.S. Whitehouse and A. Tørum, “Scour around coastal structures: a summary of recent research,” *Coastal Engineering,* vol. 44, issue 2, pp. 153-190, December 2001.
[19] F. Davoudzadeh, D. Forliti, A. Le and H. Vu, “Numerical Simulation of Confined Multiple Transverse Jets,” in: *42nd AIAA Fluid Dynamics Conference and Exhibit,* New Orleans, 25-28 June, 2012.
[20] A. Karvinen and H. Ahlstedt, “Comparison of Turbulence Models in Case of Jet in Crossflow Using Commercial CFD Code,” in: *ERCOFTAC International Symposium on Engineering Turbulence Modelling and Measurements,* Sardinia, 23-25 May, 2005, pp. 399-408.
[21] B. Guo, T.A.G. Langrish and D.F. Fletcher, “An assessment of turbulence models applied to the simulation of a two-dimensional submerged jet,” *Applied Mathematical Modelling,* vol. 25, issue 8, pp. 635-653, August 2001.
[22] Tidal Lagoon Power, *Proposed Tidal Lagoon Development in Swansea Bay, South Wales: Environmental Impact Assessment Scoping Report.* Cheltenham, 2012.
[23] ABPmer, *Swansea Bay Tidal Lagoon: Feasibility Modelling,* Report R.1956TN. Southampton, 2012.
[24] White Consultants, *Review of Environmental Statement: Seascape, Landscape and Visual.* 2014.
[25] KPAL, *Tidal Lagoon Swansea Bay: Sediment Sources and Depositional Processes in Swansea Bay,* Report 16033. Solihull, 2014.
[26] Tidal Lagoon Power, *Environmental Statement: Chapter 6. Coastal Processes, Sediment Transport and Contamination.* Cheltenham, 2014.
[27] S. Petley and G. Aggidis, “Swansea Bay tidal lagoon annual energy estimation,” *Ocean Engineering,* vol. 111, pp. 348-357, January 2016.
[28] M. Collins, G. Ferentinos and F.T. Banner, “The hydrodynamics and sedimentology of a high (tidal and wave) energy embayment (Swansea Bay, Northern Bristol Channel),” *Estuarine and Coastal Marine Science*, vol. 8, issue 1, pp. 49-50, January 1979.
[29] J. Wolf, I.A. Walkington, J. Holt and R. Burrows, “Environmental impacts of tidal power schemes,” *Proceedings of the Institution of Civil Engineers - Maritime Engineering,* vol. 162, issue 4, pp. 165-177, December 2009.
[30] M. Willis, I. Masters, S, Thomas, R. Gallie, J. Loman, A. Cook, R. Ahmadian, R. Falconer, B. Lin, G. Gao, M. Cross, N. Croft, A. Williams, M. Muhasilovic, I. Horsfall, R. Fidler, C. Wooldridge, I. Fryett, P. Evans, T. O’Doherty, D. O’Doherty and A. Mason-Jones, “Tidal turbine deployment in the Bristol Channel: a case study,” *Proceedings of the Institution of Civil Engineers - Energy,* vol. 163, issue 3, pp. 93-105, August 2010.

[31] R. Kirby and C. Retière, “Comparing environmental effects of Rance and Severn barrages,” *Proceedings of the Institution of Civil Engineers - Maritime Engineering*, vol. 162, issue 1, pp. 11-26, March 2009.
[32] C. Thompson, C. Amos, M. Angelaki, T. Jones and C. Binks, "An evaluation of bed shear stress under turbid flows," *Journal of Geophysical Research*, vol. 111, 2006.
[33] R. Gonzalez-Santamaria, Q. Zou and S. Pan, "Modelling of the impact of a wave farm on nearshore sediment transport," *Coastal Engineering Proceedings*, vol. 1, no. 33, 2012.
[34] P.M. Biron, C. Robson, M.F. Lapointe and S.J. Gaskin, “Comparing different methods of bed shear stress estimates in simple and complex flow fields,” *Earth Surface Processes and Landforms*, vol. 29, issue 11, pp. 1403-1415, October 2004.
[35] P.R. Wilcock, “Estimating Local Bed Shear Stress from Velocity Observations,” *Water Resources Research*, vol. 32, issue 11, pp. 3361-3366, November 1996.
[36] N.D. Pope, J. Widdows and M.D. Brinsley, “Estimation of bed shear stress using the turbulent kinetic energy approach - A comparison of annular flume and field data,” *Continental Shelf Research*, vol. 26, issue 8, pp. 959-970, June 2006.
[37] G.T. Török, S Baranya and N. Rüther, “3D CFD Modeling of Local Scouring, Bed Armoring and Sediment Deposition,” *Water*, vol. 9, issue 1, January 2017.
[38] T. Fischer-Antze, N.R.B. Olsen and D. Gutknecht, “Three-dimensional CFD modeling of morphological bed changes in the Danube River,” *Water Resources Research*, vol. 44, issue 9, September 2008.
[39] L. Ma, P.J. Ashworth, J.L. Best, L. Elliott, D.B. Ingham and L.J. Whitcombe, “Computational fluid dynamics and the physical modelling of an upland urban river,” *Geomorphology*, vol. 44, issues 3-4, pp. 375-391, May 2002.
[40] K. Mahesh, “The Interaction of Jets with Crossflow,” *Annual Review of Fluid Mechanics*, vol. 45, pp. 379-407, January 2013.
[41] S. Muppidi and K. Mahesh, “Study of trajectories of jets in crossflow using direct numerical simulations,” *Journal of Fluid Mechanics*, vol. 530, pp. 81-100, May 2005.
[42] R.M. Kelso, T.T. Lim and A.E. Perry, “An experimental study of round jets in cross-flow,” *Journal of Fluid Mechanics*, vol. 306, pp. 111-144, January 1996.
[43] M. Pathak, A. Dewan and A.K. Dass, “Computational prediction of a slightly heated turbulent rectangular jet discharged into a narrow channel crossflow using two different turbulence models,” *International Journal of Heat and Mass Transfer*, vol. 49, issues 21-22, pp. 3914-3928, October 2006.
[44] A. Sayma, *Computational Fluid Dynamics*. Ventus Publishing, 2009.
[45] R. Eymard, T. Gallouët and Raphaèle Herbin, “Finite volume methods,” *Handbook of Numerical Analysis*, vol. 7, pp. 713-1018, 2000.
[46] R.J. Leveque, *Finite-Volume Methods for Hyperbolic Problems.* Cambridge: Cambridge University Press, 2002.
[47] S. Richardson, “On the no-slip boundary condition,” *Journal of Fluid Mechanics*, vol. 59, issue 4, pp. 707-719, March 2006.
[48] ANSYS, Inc., “ANSYS Fluent 18.1 User’s Guide,” 2017.
[49] H. Chanson, *The Hydraulics of Open Channel Flow: An Introduction*, 2nd ed. Oxford: Elsevier Butterworth-Heinemann, 2004.
[50] Engineering ToolBox. (2005). *Sea Water Properties* [Online]. Available: https://www.engineeringtoolbox.com/sea-water-properties-d_840.html [Accessed 23rd April 2018].
[51] R.L. Simpson, D.G. Whitten, R.J. Moffat, “An experimental study of the turbulent Prandtl number of air with injection and suction,” *International Journal of Heat and Mass Transfer*, vol. 13, issue 1, pp. 125-143, January 1970.
[52] ANSYS, Inc., “Modeling Turbulent Flows: Introductory FLUENT Training,” December 2006.
[53] S. Wasserman. (2016). *Choosing the Right Turbulence Model for Your CFD Simulation* [Online]. Available: https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/13743/Choosing-the-Right-Turbulence-Model-for-Your-CFD-Simulation.aspx [Accessed 25th April 2018].
[54] S.B. Pope, “An explanation of the turbulent round-jet/plane-jet anomaly,” *AIAA Journal*, vol. 16, issue 3, pp. 279-281, March 1978.
[55] F. Juretic, *“Error Analysis in Finite Volume CFD,”* PhD Thesis, Department of Mechanical Engineering, Imperial College London, London, 2004.
[56] J.H. Ferziger and M. Perić, “Further Discussion of Numerical errors in CFD,” *Numerical Methods in Fluids*, vol. 23, issue 12, pp. 1263-1274, December 1996.
[57] V.R. Gopala and B.G.M. van Wachem, “Volume of fluid methods for immiscible-fluid and free-surface flows,” *Chemical Engineering Journal*, vol. 141, issues 1-3, pp. 204-221, July 2008.
[58] F.M. White, *Fluid Mechanics*, 7th ed. New York: McGraw-Hill, 2011.
[59] S. Longo, “Turbulence and free surface flow at relevant Froude number,” in” *6th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering*, Prague, 9-11 September, 2008, pp. 115-118.
[60] Wales Online. (2016). *The Swansea Bay tidal lagoon: What is it? How big will it be? When will we know if it’ll be built?* [Online]. Available: https://www.walesonline.co.uk/business/business-news/swansea-bay-tidal-lagoon-what-11999320 [Accessed 2nd May 2018].
[61] Maritime Journal. (2013). *Plans Unveiled For Swansea Bay Tidal Lagoon* [Online]. Available: http://www.maritimejournal.com/news101/marine-civils/port,-harbour-and-marine-construction/plans-unveiled-for-swansea-bay-tidal-lagoon [Accessed 2nd May 2018].
[62] Atkins. (2014). *Turbine House Typical Section* [Online]. Available: https://infrastructure.planninginspectorate.gov.uk/wp-content/ipc/uploads/projects/EN010049/EN010049-001913-Tidal%20Lagoon%20(Swansea%20Bay)%20Plc%2029.pdf [Accessed 22nd April 2018].
[63] Intertek, *Swansea Bay Tidal Lagoon Water Quality: Hydrodynamic Model Update and Validation Report.* Cardiff, 2013.