Wave and Tidal Tank Flow Diffuser Design

Charlie Seviour

A thesis submitted for the degree of

Master of Mechanical Engineering and Bachelor of Renewable Energy

2011

Personal Statement

This was a new project with only a few constraints. Once the velocity profile required and geometry of the tank had been specified, I was able to get on with the project by making my own intrepretation of any implied constraints and I only needed to meet with *David Ingram* on four occasions. However this does not imply that the project ran smoothly; core texts have no detailed information on diffuser design, and potential flow programs are usually designed for testing single airfoils. Having spent the first semester examining this material (and attempting to produce my own code), it was only by persistence that the program *Flow* (Alem, 2009) and the book *Control of Flow Separation* (Chang, 1976) were found. This lead to a breakthrough; by following the references I found circa 50 papers dealing with diffusers. Amongst these was a systematic, experimental study of diffuser design (Fox and Kline, 1962). By adopting the notation, the particular design of diffuser required for this project could be validated to an extent. Potential flow theory was then used in conjunction with the experimetal results for comprehensive validation of the diffuser. Whilst I did not produce the notation or potential flow code, a thorough knowledge of the subject and a creative approach was required to customise and apply these resources. The theoretical work I have presented on the use of these resources is original.

Summary

Wave and Tidal Tank Flow Diffuser Design

Charlie Seviour

April 15, 2011

This thesis investigated the design of a flow diffuser and conditioner for the new wave and tidal tank due to be built at Edinburgh University.

Contents

Acknowledgements

I am grateful to my supervisor *David Ingram* for allowing me to approach the project in whatever way I felt most fit. I would also like to acknowledge help from *Markus Mueller*. As always the support of my friends and family has been greatly appreciated.

Glossary

β	Turning angle
θ_{eff}	Effective turning angle
a_1	Length or area of the space between vanes at the diffuser inlet.
a_2	Length or area of the space between vanes at the diffuser outlet.
L_0	Length of the diffuser outer wall
L_1	Length of the diffuser inner wall
N	Diffuser centre line arc length
R	Radius
r	Tank depth
u	Velocity parallel to the tank floor.
u_1	Velocity of the flow entering the diffuser.
u_2	Velocity of the flow existing the diffuser.
u_{max}	The maximum velocity parallel to the tank floor.
W_1	Diffuser throat width
W_2	Diffuser exit width
y	Height above the tank floor
CFD	Computational Fluid Dynamics
Re	Reynolds number

1

Introduction

1.1 Velocity Profile

A turbulent velocity profile can take 20 to 30 pipe diameters to fully develop (see figure 1.1 for an example of a fully developed profile). For a 3m deep tank it is clearly infeasible to use such an entrance distance and thus a means of accelerating the flow is necessary so that fluid velocity at the top of the tank reaches a maximum velocity quickly. A diffuser is capable of this acceleration and the topic is introduced in Section 1.2. Before the diffuser design can be considered however, it is important to establish what velocity profile is required. Many profiles exist since they are derived empirically in attempts to find the best fit to experimental results. For the purpose of this project the following equation was used:

$$u/u_{\max} = (y/r)^{8.8} \tag{1.1}$$

In 1.1 u is the velocity parallel to the tank floor at the normal distance from the wall y, r is the depth of the tank, $u_{\max}$ is the maximum velocity defined in the remit as 0.8m/s. The exponent 8.8 was selected because at the Reynolds number that the tank will be operating at this describes the best profile according to Schlichting et al. (2000). Using 1.1 the profile shown in figure 1.1 is produced.

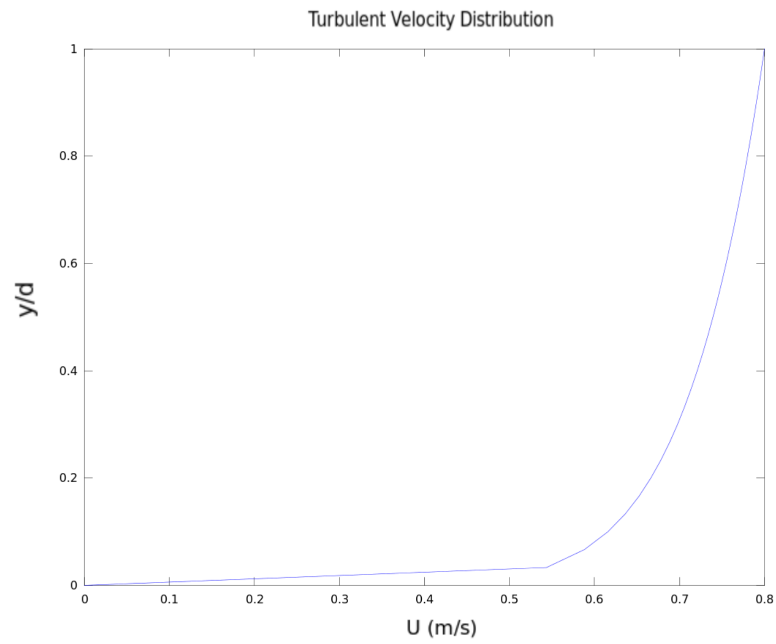

Figure 1.1: The desired velocity profile - The desired velocity profile, produced using the power-law (equation 1.1).

The inlet profile must also be defined in order to fully describe the problem. Since the water is propelled by an impeller that has not yet been specified it is not possible to define the inlet profile with certainty. Also, after the impeller a flow straightener will alter the profile. However the most likely case shall be plug flow [Do you need to explain what this is?] and this has been assumed for the design process.

An important flow characteristic is the turbulence intensity. This parameter was not specified in the remit. A maximum turbulence intensity of 10% at the inlet and exit of the design was deemed acceptable, but measures for reducing the turbulence, in the form of flow conditioners, have been investigated.

1.2 Diffuser

Diffusers are the geometric expansion at the inlet of a channel, pipe or tank. A plethora of engineering applications exists, from the air intake ducts in jet engines to wind tunnels. The simplest diffuser design is a planar, formed simply by linearly increasing the pipe cross-section. Alternatively vanes and various curved profiles can be used.

With careful design more bespoke velocity profiles can be produced after the diffuser. This is the crux of the design project–to choose a specific diffuser shape which shall deliver the desired velocity distribution.

The diffuser design in this project is intended for use in a wave and tidal tank. Schematically, as illustrated in figure 1.2, an impeller drives a current from a relatively small pipe diameter to the full height of the tank.

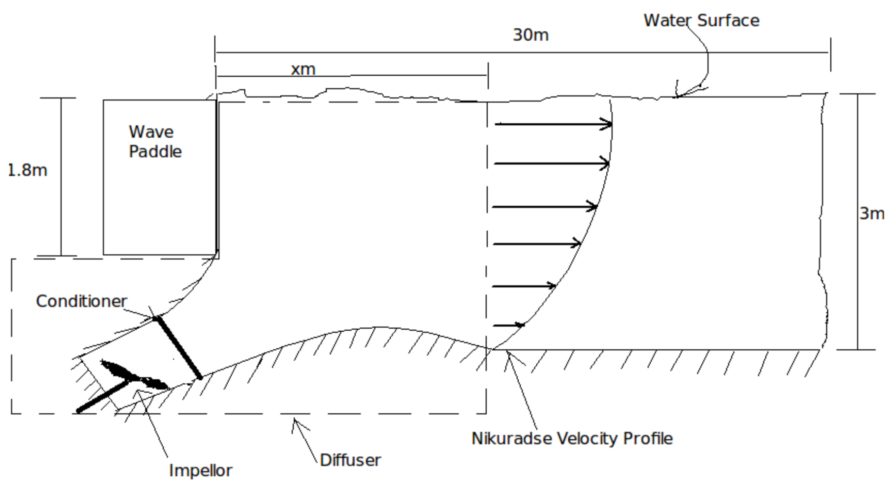

Figure 1.2: A schematic of the Diffuser - A schematic of the Diffuser in the surroundings of the tidal and wave tank.

Also in figure 1.2 four dimensions are given. These are explicit constraints of the project. As indicated a straight 1.8m section must be allowed for the wave machine. The desired velocity profile is shown at xm from the edge of the tank. The distance x must be minimised by the choice of diffuser design. In addition to these three limitations, design criteria such as cost, ease of manufacture and service lifetime are considered in deciding the merit of individual solutions.

Using the powerful tool of computational fluid dynamics, with only a desktop computer it is possible to conduct the necessary analysis for an efficient design. However since producers of CFD software can only validate a tiny fraction of the possible flow scenarios, it is still essential to confirm the results against other experimental or theoretical work. In the subsequent literary review section research by other authors is summarised.

1.3 Flow Conditioner

The flow conditioner is a simple device, by creating boundaries to the turbulent or swirling flow it is possible to obtain a clean flow. Four methods of achieving this are illustrated in figure 1.3. Since the conditioner can be easily produced for any pipe or channel geometry this has been a smaller area of focus in the design project.

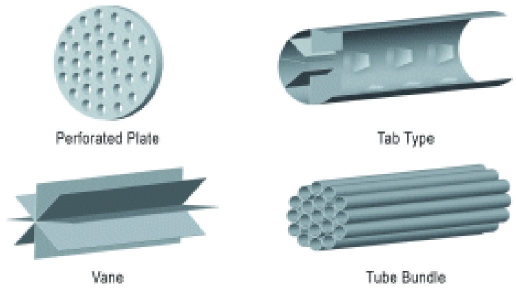

Figure 1.3: Four methods of flow conditioning - Four methods of flow conditioning

Flow conditioners are also readily available in formats which can be installed in standard pipe networks. There are two options; either to fit a standard pipe diffuser to the entrance, or to the exit of the diffuser. Although the installation is easy and the costs lower, it is possible that in placing the conditioner at the diffuser inlet my lead to unwanted turbulence.

The turbulence at the outlet of pumps is typically around 10% (Gülich, 2010). This is comparable to the maximum level of turbulence that can be present in tidal streams (**?**). Although a standard exists for the testing of wind turbines which specifies the spectra of turbulence and velocities that a turbine should be tested for, an equivalent does not exist for tidal turbines (**?**). Since the wave and tidal tank is in the conceptual design phase, the merits of various flow conditioning methods were tested and discussed but a final design will require further understanding of the clients needs.

1.4 Literature Survey

Diffusers and flow conditioners are treated only superficially in typical core texts (Douglas and Swaffield, 2005, Massey, 1975). The British Standards mention flow conditioners in the context of flow metrology but do not make any useful specifications. A one-page guide on the subject informative is included in the annex of the standard **?**. Surprisingly, there are no standards for diffusers. Discussion in greater depth can be found in articles by amateur wind tunnel builders (Bradshaw and Mehta, 2003, *Build a Wind Tunnel*, n.d., Tatman, 2006). The most enlightening text, *Control of Flow Separation* (Chang, 1976), makes the salient point that flow through a passage (internal flow) cannot be treated the same as an external flow. Due to the large ratio of wall-length to throat diameter, the growth of the turbulent boundary layer, and thus the onset of stall is different to that of external flow. A systematic study of curved and straight diffusers was conducted by **?**, to obtain the stall parameters; Reynolds number, area ratio and the turning angle. There have been attempts at finding a theoretical approach to predicting the separation point in planar diffusers by, for example, **?**. By using the momentum integral method to solve the boundary layer equations, the point at which zero stress occurs can be found, and therefore the stall point. Still no theoretical method has been developed for all possible internal flow regimes and only a few papers deal with diffuser design (approximately 50).

Potential flow theory was an approach used in this investigation and so literature in this subject was reviewed. Dated works such as *Fundamental Mechanics of Fluids* (Dekker, 2002) and *Potential Flows Computer Graphic Solutions* (Kirchhoff, 1985) proved most useful since they were written during the heyday of potential flow theory, although a common issue found in old potential flow volumes is the now antiquated programming languages such as BASIC. Examples of simple Matlab programming of potential flows can be found in guides to Matlab (Magrab et al., 2010). More complex geometries tend to be represented using the vortex panel method. Good illustrated descriptions of this method are readily available in university lecture slides (Mason, 1995, Sankar, 2008). More mathematical explanations are also in the public domain.

1.4.1 Software

An assessment of free potential flow code was made and is summarised in figure 1.4. The quality of this resource varied. The simplest code consisted of short functions for plotting the flow field around a cylinder (Isola, 2005, Nylander, n.d.). Other programs offer graphic user interfaces (GUIs) (Jayaraman, 2006). One of the most advanced codes is *Tornado* (Melin, 1991), which uses a vortex panel method. Potential flow code is typically tailored to aircraft design. *Tornado* features geometry design using a text-based input and coefficients of lift and drag plots as outputs, but since it is specific to aircraft design it does not feature a flow field output, which is required in this project. To incorporate such a feature would require finding, out of thousands of lines of code, those which calculate the flow field and modifying them – this was not feasible.

I chose to use the program *Flow*. It is compact since it uses a Matlab tool box pdetool and therefore is easy to modify.

Program	Author	Comments
Tornado	Tomas Melin	Code was designed for use in a flight simulator and has a text-based user interface. It is capable of 3D, time-dependent potential flow calculations and produces many plots of Lift and Drag but no flow visualisation.
AVL	Mark Drela and Harold Youngren	Text based program originally developed commercially but now freely avaible.
Panel Method-based 2-D Potential Flow Simulator	Divahar Jayaraman	A nicely finished code with graphical interface. It only allows geometries from the supplied database [Check that this means what you intended] or user defined objects. There is no facility for multiple objects and since there are 1000s of lines of code, mostly associated with the GUI, it would be hard to reverse-engineer to permit this.
CMARC	AeroLogic	Full version costs $3125. Highly developed code capable of inviscid and viscous simulations. Demo does not allow display of the results.
Potential Flow	Maxim Vedenyov	A very simple program which allows airfoils to be designed by clicking on a flow field. The position of the clicks are joined using a spline function. Requires skilled mouse positioning.
Flow	Peter van Alem	A very succinct code which consists only of 6 files, each about 2 pages in length. Coded using Dutch variable names with sparse commenting in English.

Figure 1.4: Potential flow-The various open source potential flow codes tabulated

2

Theory

The theory required for this project revolves around computational fluid dynamics. The need for CFD is the consequence of being unable to solve the Navier Stokes equations analytically, except for a few simple cases. CFD has became available to students with the emergence of powerful personal computers and free or inexpensive software. By dividing a flow domain into separate cells and solving the Navier Stokes and continuity equations discretely for each cell, it is possible to obtain an approximated solution. If the divisions are small enough, an exact solution may be obtained. For this, cells of the Kolmogorov microscale are required, this results in huge numbers of mesh points and the need for a super computer to do the processing. An alternative is to use a turbulence model. This carries the risk that the model may be inaccurate. Turbulence models are designed using only a small number of experimental results and cannot be accurate for every flow scenario. Therefore in using CFD it is standard practise to validate and verify a simulation.

The approach for this was to run the simulation in question a second time using potential flow code, instead of the viscous solver, to obtain a second set of results for comparison. Potential flow theory employs a different set of equations from CFD and has been used for diffuser design in the past by **?**.

2.1 Potential Flow

Potential flow theory works by assigning an energy level or potential, ϕ, to a flow. The magnitude is arbitrary and merely scales the problem. In a given flow scenario a source of potential is required to drive the flow through the system. This source could be represented by the free stream velocity in airfoil testing, or specific to this project, the flow driven by an impeller.

Two conditions are then applied. One, that the fluid is incompressible, where u and v are the velocity components in the horizontal and vertical directions:

$$u + v = 0 \tag{2.1}$$

and two irrotationality:

$$\nabla^2 \phi \tag{2.2}$$

Having applied these conditions it is then possible to solve the latter equation which is known as the Laplacian equation when it is equated to zero or in the case that the right hand side so equal to some function; the Poisson equation. With a function which satisfies these equations the velocity can be found at any point. It is the rate of change from one point to another that dictates the velocity which expressed mathematically, is the following derivative:

$$u = \frac{\partial \phi}{\partial x} \tag{2.3}$$

$$v = \frac{\partial \phi}{\partial y} \tag{2.4}$$

A great range of potential flow functions are published but for complicated shapes a composite of functions is required. One common means of compiling functions is the panel method, which uses a combination of sources and vortices along surface contours. These can be arranged in a system of equations to create an influence matrix ,which determines the required strength of individual sources and vortices. An interesting property of potential flow is that not only can it be used for modelling conventional fluid flow, but also for acoustics, water waves or electroosmotic flow.

Another method of solving a potential flow problem is to find a solution on some domain D such that on the boundary of D, ϕ is equal to some given function. Stating the problem in this format it is known as Dirichlet problem. Using a finite element method the Poisson equation can subsequently be solved computationally. This method takes an electrical engineering approach. This is possible because the flow of electricity is dicatated by the same Laplacian equation. To illustrate figure shows the path taken by an electrical current around a cylinder which is identical to flow pattern an inviscid fluid takes.

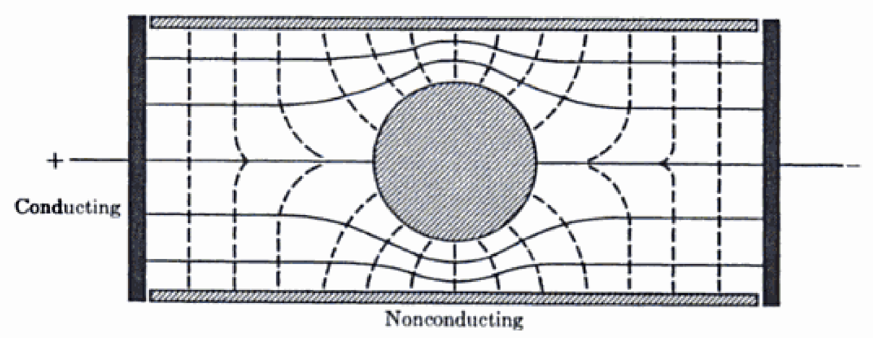

Figure 2.1: A electrical analogy - The governing equation for electrical and fluid flow is the same (**?**)

By solving an electrical flow problem a more sophisticated and concise code is produced, which also works for the other types of flow regimes. The latter coding is present in the program *Flow* (Alem, 2009).

Potential flow problem can be solved graphically. The fluid always follows streamlines, which can therefore also be considered as physical boundaries where suitable. Perpendicular to equi-potential lines are streamlines. This implies that a desired flow geometry can be formed by prescribing streamlines to follow a particular route. The simplest way of doing this is to draw a flow net. The flow net is constructed by sketching the channel that is intended to be used and then, by hand, adding streamlines that follow the correct course. The streamlines are then bisected at 90 degrees by equi-potential lines. This results in varied spacing of the lines, which corresponds to changes in velocity. The method is intuitive, although probably less accurate than the computational approaches described previously since the construction of the net relies on a good eye for judging the correct angles for the equi-potential lines to take. An example flow net is sketched in figure 2.2. The example depicts the flow over a back step. A diffuser could be this shape. Shortly after the step, the flow regime returns to a plug flow profile, caused by the parallel channel walls. This is a shortcoming of potential theory which is contemplated now and in examining the results from this project. In most real flow situations initial conditions determine the characteristics of flow a long way downstream. This is exemplified in the swirling flow, caused by pipe bends, which can be detected up to 100 diameters from the bend (?). Indeed the motivation for this investigation stems from the large entrance length required to develop a particular velocity profile.

Figure 2.2: A flow net - Produced by sketching 90 degree bisecting lines corresponding to the geometry in question. In this simple example flow over a back step has been drawn.

It is also important to remember an assumption made with potential flow theory; it deals in Dry Water, Dry meaning inviscid and irrotational. However, in the case of a flow diffuser, neither assumption may be true. Furthermore, separation cannot be predicted using this theory. If separated flow were to occur in a design it would be necessary to know where this happens and if required correspondingly change the geometry of the potential flow problem by placing a physical boundary along the affected region. This leads to the conclusion that separation analysis complements potential flow well, as will be discussed in the next section (section 2.1.1).

2.1.1 Flow Separation

Before considering what is the best diffuser geometry, it is necessary to understand separation – the most common and best understood example being airfoils when experiencing a stall. Airfoils stall at high angles of attack. For efficient designs such as the naca 0012 this is at an angle of around 15 degrees for an Re of a million. The fluid turns an angle of twice that as it follows [Should this be flows?] from the leading edge of the wing to the tail.

In this project there were two reasons for avoiding stall. First, if the water no longer follows along the contours of the channel, then the subsequent curvature would not influence the current as planned. Second, if the flow is unseparated at the maximum flow rate, then this will also be the case when the tank is operating at lower velocities. In a design which incorporates separation, the velocity profile would indirectly be a function of flow rate, since this would determine the separation point. Because it is probable that at certain velocities stall would cause an undesired velocity profile,this design has been not been chosen.

Flow separation was also a principal factor when choosing the angle of the impeller. The greatest velocity component across the tank occurs parallel to the tank floor. [Check that the previous sentence means what you intend] Since the impeller must be positioned below the wave paddle, it would be impossible to direct the maximum velocity to the top of the tank without a large change in the flow direction, and as a result risk separation and turbulent eddies. Equally, with the impeller directed upwards, at the tank entrance a large correction in the flow direction would be needed. [Check that the previous sentence means what you intend] As a compromise, an angle of 45 degrees was selected.

The other factor determining the stall point is the Reynolds number that a wing is operating at. At the lowest Re, separation does not occur and a "creeping" flow exists. At higher Re, separation occurs at lower angles of attack. The explanation for this phenomenon is that Re is a ratio of the inertia and viscous forces. A larger inertia flow for a given viscosity (higher Re) will clearly be less inclined to change course. In this project [Check that project means what you intend] Re can be considered a constant since only the fastest operating conditions are being designed for. As a result there remains a single option for preventing stall, which is to minimise the rate of change of gradient that the flow is subjected to.

In principle the design of the diffuser walls emulates airfoil design. This analogy cannot be taken further because the growth of the turbulent boundary layer is different in internal flow (Chang, 1976), otherwise the diffuser could be designed using the vast published data on airfoils. Instead the work on separation in diffusers by Fox and Kline (1962) has been referred to. Whilst the shape of my diffuser must contain a corner for the wave maker, I hypothesise that it can still be treated in the same manner as the diffusers examined by Fox and Kline (1962).

3

Methodology

This chapter deals with the design approach used in the project. An analytical starting point was the work conducted by Fox and Kline (1962), who systematically studied straight and curved diffusers. The resource was useful for determining the separation point in diffusers. The bespoke design required for this project could not be attained from here however. Knowledge of the stall point is only sufficient for a small part of the overall design. The main part was to determine the right shape of diffuser to obtain a suitable velocity profile, there is no other similar work. Nonetheless potential flow software released to the public domain has been great use in assessing the resultant flow from a particular geometry. The design was an iterative process which integrated three stages of development as shown in figure 3.1.

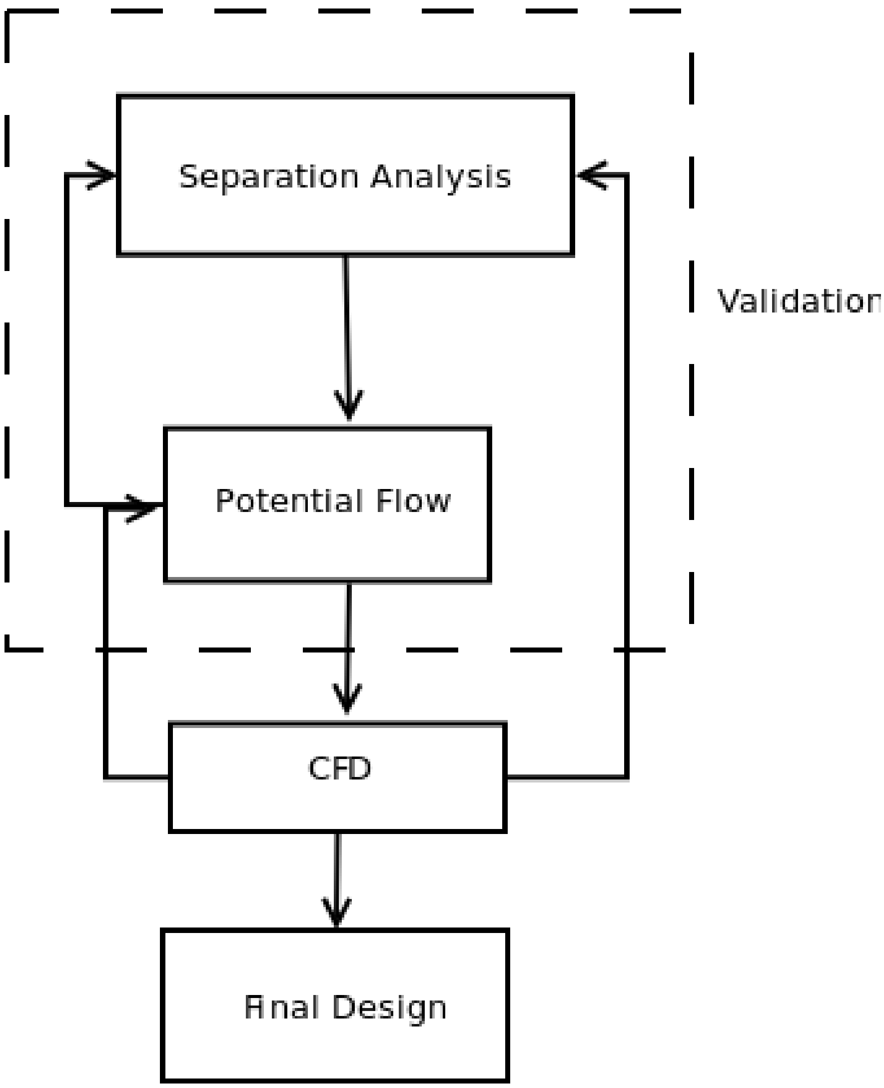

Figure 3.1: The design process diagram - The design process diagram

3.1 Validation

Due to minimal literature on diffusers the bulk of the project focused on methods of validating the CFD aspect of the design process. A common and reliable method is to conduct an experimental investigation of the flow regime in question and comparing key parameters with CFD results. This is relatively expensive. A cheaper alternative is to use a simplified study of the problem for validation. A pertinent approach in this case is the use of potential flow theory.

3.2 Mitigation of Flow Separation by Consultation of Available Data

Initially the possibility of using just a rectangular section was examined. Immediately it became apparent that such a geometry is insufficient to produce the desired velocity profile. Also the flow over the sharp edges created unsteady vortices. The shape that does minimise stall is a circle, as shown in figure 3.2. Having decided the position and angle of the inlet (points B and C), a circle of certain radius that can be selected connects points A and D. The intersection of (point D) the tank floor and the circle R is also a varible. At some point after the mid point of the circle R a spline can be used to smooth the transition to the tank floor.

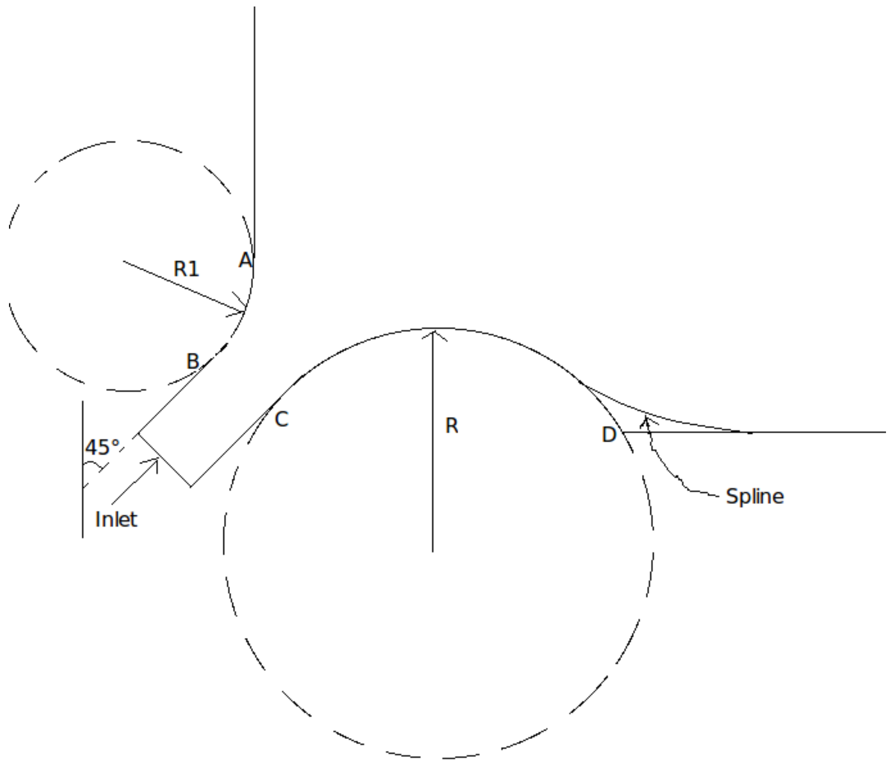

Figure 3.2: The design of a diffuser - The design of a diffuser simplified to two circles connecting the tank walls and the inlet. This circular cross-section was chosen to prevent stall.

A comparison to the separation points found by Fox and Kline (1962) was made to establish whether a diffuser design would stall or not. A diagram was produced using the original notation and applying it to a typical diffuser geometry in this project (see figure 3.3). As indicated in figure 3.3 there is an area of unavoidable separation in the corner. As such the length of the outer wall L_0 has been approximated as a line that follows the general curvature of the flow. A mean distance between the outer and the inner wall (L_1) is described by the line of length N. To identify whether separation shall occur, two key parameters have to be calculated; the turning angle β and the effective turning angle θ_{eff}. β can be measured as the angle between N and the horizontal at the throat or from the equation:

$$\beta = \frac{360N}{2\pi R} \tag{3.1}$$

θ_{eff} is found in the relationship:

$$2\theta_{\text{eff}} = \arctan\left(\frac{W_2 - W_1}{N}\right) \tag{3.2}$$

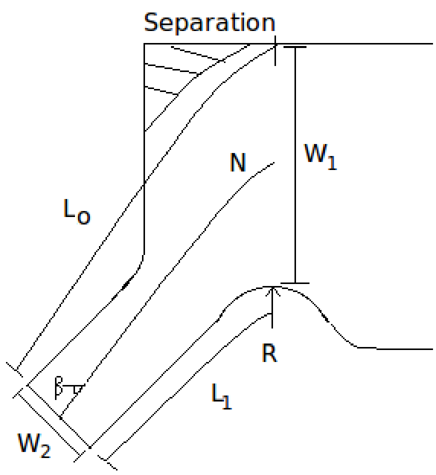

Figure 3.3: Separation - Separation

Looking up these parameters in figure 3.4 it can be determined whether stall is initiated.

Figure 3.4: Stall lines - The onset of stall for various angles according to Fox and Kline (1962)

3.2.1 Potential Flow

To use the program *Flow* it is necessary to first create a mesh. This can be achieved using the pdetool in matlab. The desired geometry is made from a composite of shapes–either adding or subtracting from a rectangle. The construction a of diffuser is shown in figure 3.5.

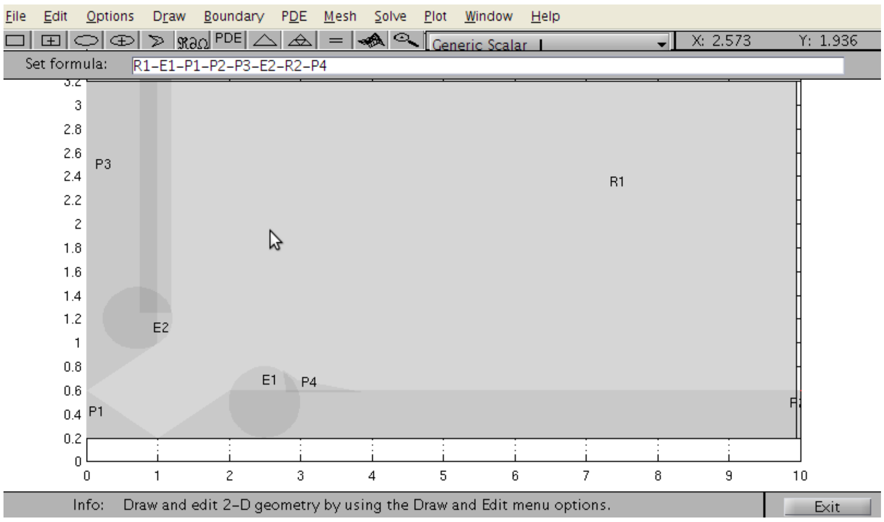

Figure 3.5: Construction of a diffuser mesh - Construction of a diffuser mesh using pdetool, which is later used in the program *Flow*

3.3 Diffuser Designs Investigated

Two criteria were examined; the shape of the diffuser walls and the use of vanes. It was hoped that the desired velocity profile could be achieved using only the former. The advantage would be that a less complex and cheaper design would be possible as a result. However it was found in this simple configuration that a jet was always produced and the required diffusion was not possible. The use of vanes allowed a precise stipulation of the velocity at the exit of the vanes. A further advantage is that the vanes have a secondary effect of removing swirl from the flow and hence reducing the flow conditioning requirements.

To begin with just a 45 degree diffuser exit was modeled to find what the required vane exit velocities should be. In this model the desired velocity profile was used as the input. An important point arose from this; at the point where the diffuser meets the tank floor an area of recirculation is present. There are two ways of dealing with this. One method is to increase the velocity component parallel to the tank floor. The other is to create a protrusion in the tank which fills the area of recirculation. The latter is more reliable and was thus used in the subsequent designs.

The tank is a closed system and as such the water has to travel from one side, then back again under the ground. This means that the water entering the diffuser must at some point have turned 180 degrees. For the initial design this was ignored, but once a successful methodology had been developed, the logical next step was to incorporate the bend into the design. The benefits are a potentially more compact design and the fact that the natural velocity distribution after a bend conincides favorably with the required turbulent velocity profile (and thus fewer vanes may be required).

3.4 Design of Vanes

Continuity was the govering equation used in the design of the vanes. Given a inlet velocity u_1, inlet area or in the case of 2 dimensional design a length a_1, the outlet velocity u_2 the required area (or length) at the outlet (a_2) can be calculated:

$$a_2 = \frac{a_1 u_1}{u_2} \tag{3.3}$$

As cost is a concern a varible which was varied was the number of vanes in a design: Fewer vanes allow less redirection of the flow but reduce the material and manufacturing burden.

3.5 Computation Fluid Dynamics

The CFD study was conducted in Starccm+. The advantage of using such a commercial software is that it well recognised and has a more friendly user interface than the open source alternatives such as open foam. The simulation was kept as simple and efficient as possible. Modelling was conducted in 2 dimensions, which provides sufficent detail for preliminary design, whilst being fast to compute. The entire tank domain was modelled using approximately 50 000 cells but the mesh was locally refined in the diffuser area where detailed information on the flow is important.

To produce the mesh an internal program included in the Starccm+ package was used. The program is capable of automatic generation of a grid from a model produced from a computer aided design (CAD) interface. As a result the vast range of geometries tested in this project where quickly produced.

The trimmer meshing model was used. This was selection was based on obtaining an optinum skewness as doing so reduces the numercial error. Skewness describes how far from 90 degrees the angles in a tetrahedral are. For a square tank it is possible to have exactly square cells. Where the more complex shape of the diffuser is modelled a perfect skewness is not possible. It would be possible to, on an idividual basis, to determine the best meshing model for each simulation. Such further optimisation was not justifiable however as convergence was typically reached after only 1000 iterations, with the residuals reaching machine zero.

To confirm that the resolution was high enough to obtain a grid indepent solution a grid convergence study was conducted. A grid convergence study requires that a vital property is compared from simulations of different grid size. A comparison of the velocity profile was made between the simulations. For the refinment studies a halving of the average cell size was used for the successively finer meshes.

Another tool which was exploited was the macro function. Once a standard set a inputs where decided apon it was possible the record these to a java file, which was executed when the same commands where desired. The macros used in the project are included on the CD accompanying the report.

The key continua parameters were the $k - \epsilon$ and the steady model. The selection of the turbulence model primarily determines the coefficients used in the simulation for calculating the turbulent viscosity and energy. The best model is the one in which these constants have been derived from experiments which are similar to the simulation in question. The $k - \epsilon$ model is well established and based on the empirical results of the same ilk as the diffuser in this project.

Computationally the steady model is greatly preferable. Determining whether a flow is likely to be unsteady numerically is possible by first using an unsteady model and recording the time dependent change in the flow field. If this change is small then it can be assumed to be quasi-steady. Because of the time restraints of the project this is not feasible. Alternatively an indication of unsteady flow can be obtain by watching the simulation as it is running. Common unsteady flow behaviour such as vortex shedding can be identified in a vector scene, since Starccm+ uses a psuedo timestep in this mode. Also it is unlikely that convergence shall be obtain if an unsteady flow is modeled as steady. Whilst to model all of the diffuser designs in the unsteady mode is not viable, the final design was modeled using the unsteady option to check that the flow was indeed steady.

Starccm+ was also used for post-processing of the results. As well as being able to view vector scence of the flow scenario it was also possible to create user field function in order to display customised plots. An example being the use a the field function to plot the theoretical velocity profile and the actual velocity profile on a graph of non-dimensionalised velocity and position.

4

Results

In this chapter are the CFD results from the design work completed in this project. These results are presented in conjunction with the product of the associated validation and verification. To place the final design in the context of the other attempts, colour figures depicting the flow in a vaneless design are presented and described in the next subsection (see figures 4.1 and 4.2 in section 4.1). Details of the design, such as separation and unsteady flow behaviour, are considered in depth in the subsequent chapter "Discussion" (chapter 5).

4.1 Vaneless Design

In figure 4.1 it can be seen that a jet is produced as the flow detaches from the inner diffuser wall. It was found from the experimental work by Fox and Kline (1962) that the jet should be at an angle of 40 degrees from the horizontal (the working can be found in the appendix). As expected, the potential flow describes a different scenario; at the point where separation should occur, the flow clings to the the inner diffuser wall. However up to that point the velocity profile is similar.

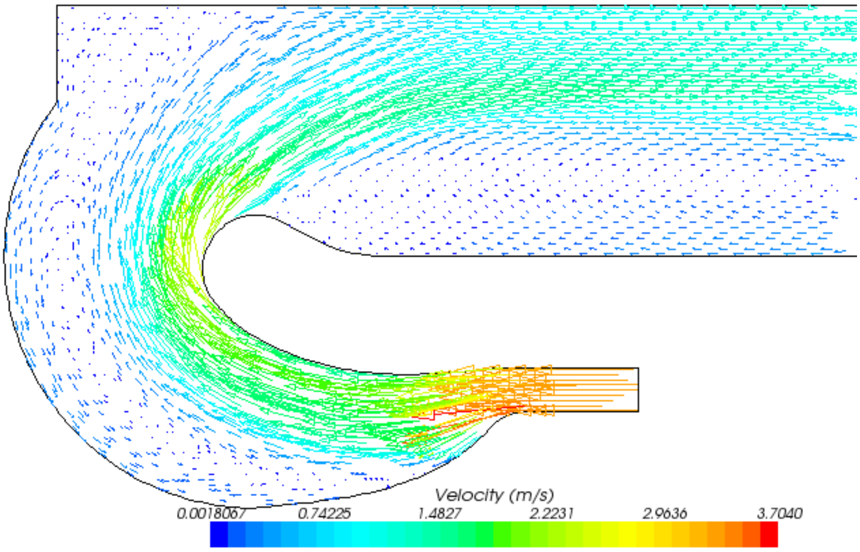

Figure 4.1: CFD vector scene - The flow field produce by a vaneless diffuser

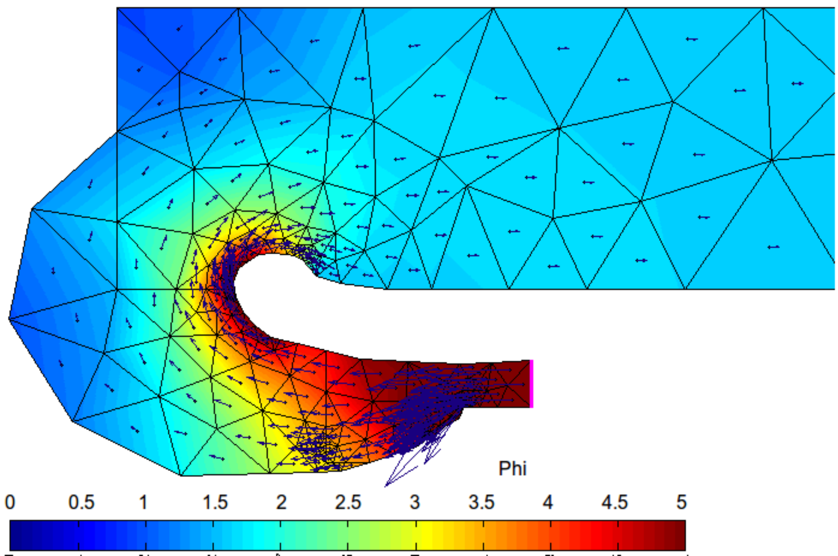

Figure 4.2: The potential flow result - Using potential flow code it was possible to obtain a solution for validation of the CFD

4.2 Vaned Design

4.2.1 Velocity Profile

In comparing the potential flow scenario in figure 4.3 to the CFD equivalent (figure 4.4) it can be seen that there tends to be a higher velocity on the inside of the guide vanes in both models. Also in agreement is the reduction in velocity in the wave making zone. Recirculation is not described by potential flow, as with the separation experienced in the vaneless design, because separation is not accounted for in potential flow theory.

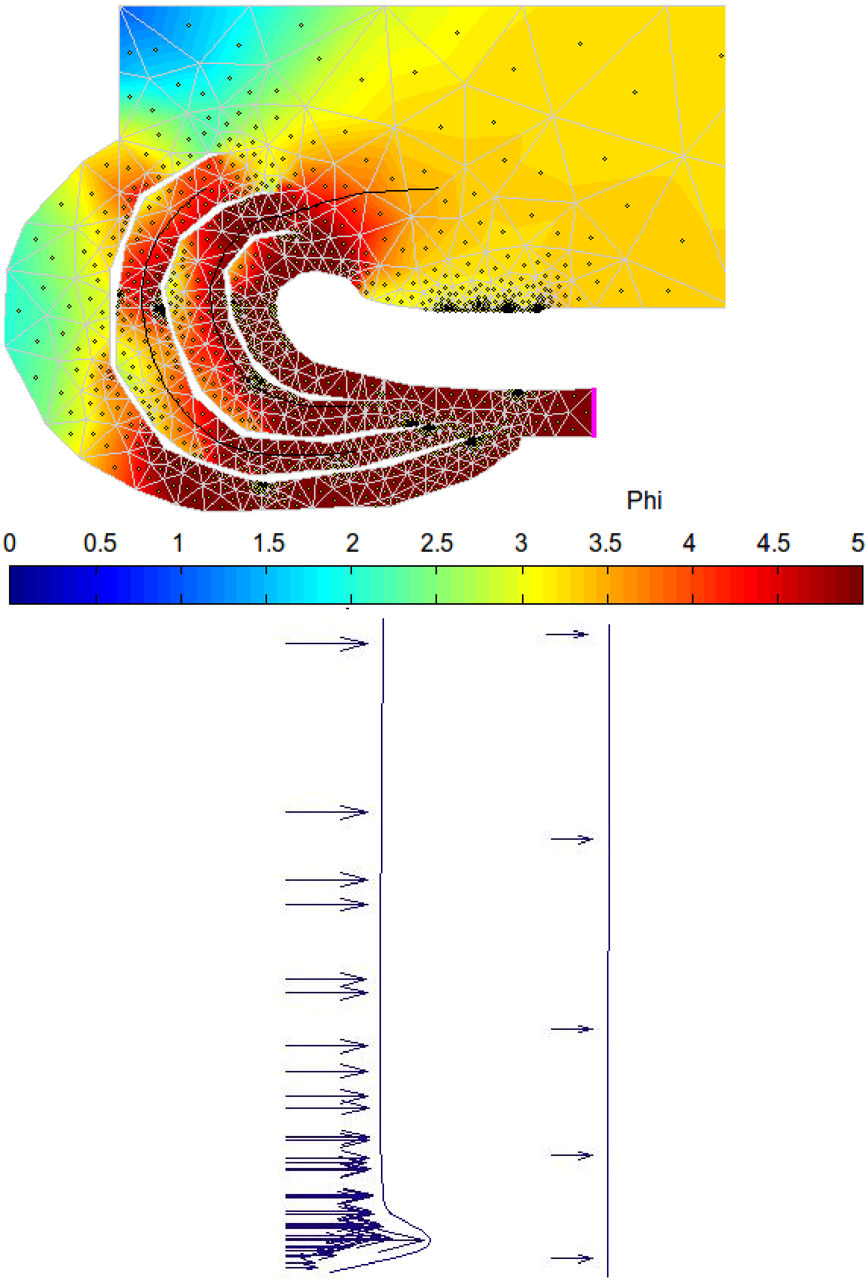

Figure 4.3: A vector scene of the output of final design, modelled using potential flow - This is the result of modelling the final design using the modified potential flow program *Flow* by Alem (2009). The two black lines running between the vanes are the streamlines.

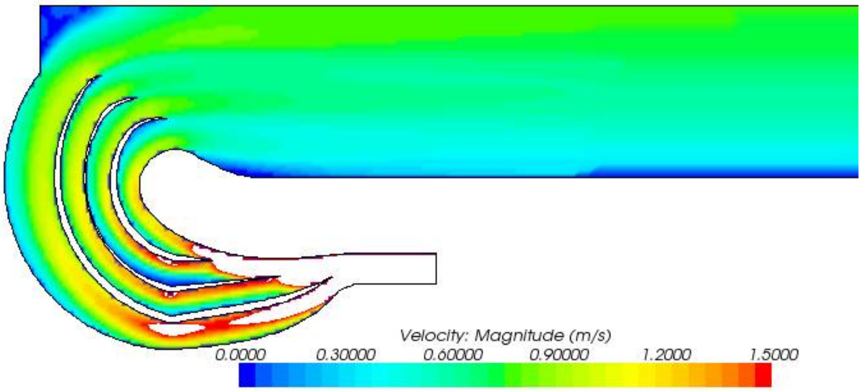

Figure 4.4: A scalar scene of the final design simulation - The scale has been clipped to 1.5m/s (velocity in exceedance is shown as white regions) in order to make the the colours correspond to the equivalent image produced from the potential flow code.

At the tank entrance the fluid is diffused (causing a reduction in velocity) in both cases but different profiles are produced. The potential flow solution is driven by the geometry of the problem and not intrinsic properties of the working fluid, such as viscosity. The acceleration seen within the vane structure was a consequence the corner geometry. As can be illustrated by coding the simple potential flow equation for flow around a 270 degrees corner the flow accelerates to infinity on the tip of the corner. The high velocity seen next to the vanes was produced in line with the same principles. At the diffuser exit there are no longer any edges capable of inducing a higher fluid velocity. Here the situation is equivalent to a collection of sources, one source between each vane. The velocity from a source decays logarithmically with distance so that at further lengths from the tank inlet a constant change in potential is approached and thus plug flow is prevalent. This is shown by the two velocity profiles produced beneath the vector scene in figure 4.3. In this example the vanes are staggered towards the wave machine. As a consequence the first profile describes a faster lower region as the flow from vane outlet closest to the wave machine has already reached a constant speed, whereas the outlet next to the tank floor has not. Once the flow has reached the location of the other velocity profile plot, plug flow is produced as indicated.

By contrast, in CFD the formation of a turbulent profile can be seen after 5m (see figures 4.5 and 4.6). In this case the upstream flow conditions have a long-lasting effect on the velocity profile further downstream. This can be seen in the slow decay of a jetting effect caused by the diffuser vanes (figures 4.7 and 4.8).

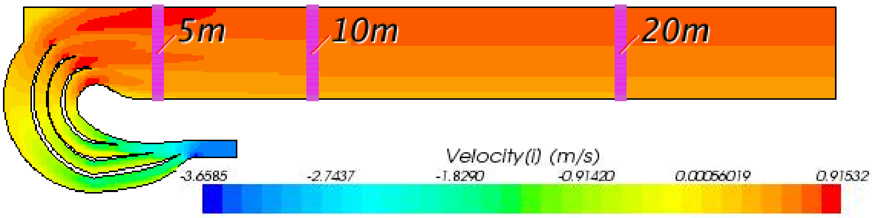

Figure 4.5: The velocity profiles - The location of the velocity profile probelines at 5m, 10m and 20m. In this image the velocity of the flow parallel to the tank floor is also shown.

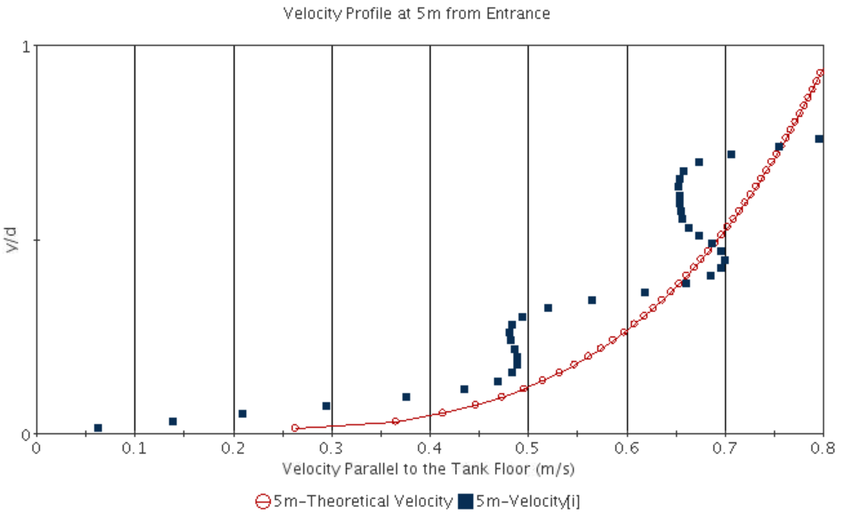

Figure 4.6: Profile after 5m - The velocity profile after 5m shows that the general trend of the flow matches that of the theoretical, but there are significant deviations caused by jets from the diffuser.

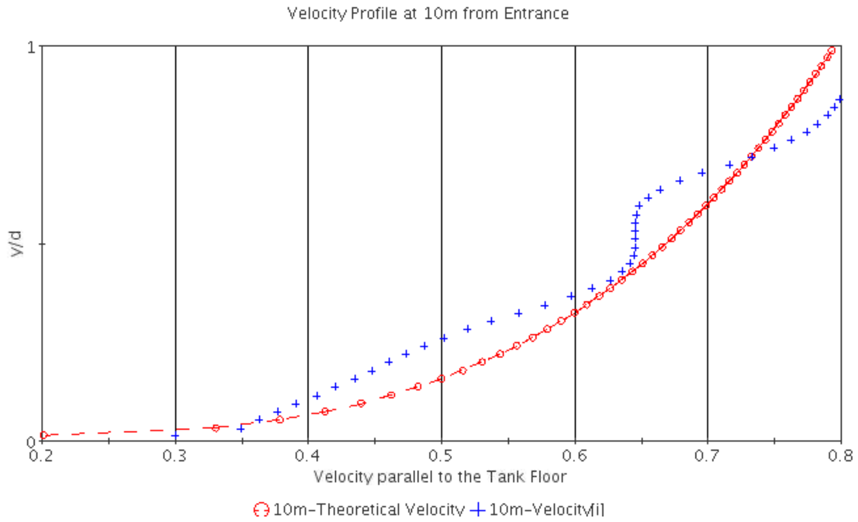

Figure 4.7: Profile after 10m - After the flow has travelled 10m the velocity fluxes from the diffuser have been smoothed.

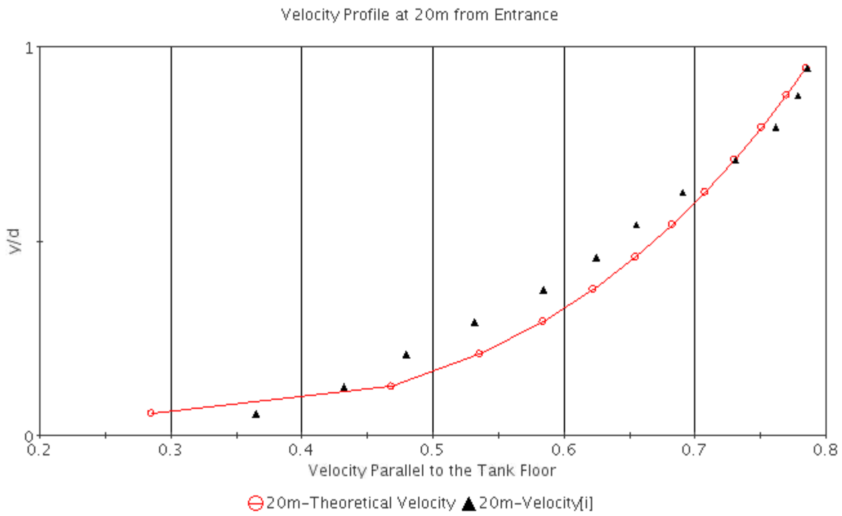

Figure 4.8: Profile after 20m - A smooth profile which matches the theoretical profile.

4.2.2 Grid Convergence

Three grids were used for a grid convergence study, using the Octave code by Ulerich (2009), the results are presented in figure 4.9. An order of convergence of 0.85 was found and a safety factor of 1.25 was used. By using the Grid Convergence Index (GCI) method of presenting the results, as recommended by Celik et al. (2008), it can be seen that a grid converged solution was obtained.

Grid	Base Spacing (m)	Mean Velocity at 5m from Entrance	GCI (%)
1	0.5	0.597	-
2	1	0.588	2.35
3	2	0.583	1.32

Figure 4.9: Grid convergence studyA coarse, medium and fine grid where used to determine that grid convergence was achieved.

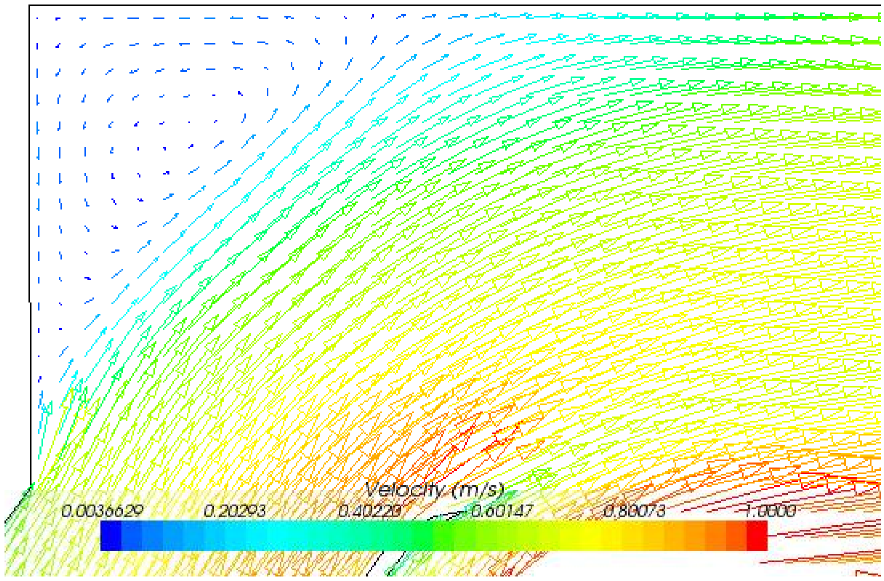

Figure 4.10: Separation adjasent to the wave maker - The recirculation in this zone is likely to be of interest to the wave paddle designers

5

Discussion

This chapter discusses the results obtained in the project and examines their realism. An appraisal of the design is made and reasoning that lead to its conception. A few practical considerations are given with respect to the construction of the diffuser and flow conditioner.

5.1 Validation of Results

Good agreement between the CFD and potential flow models was achieved in the immediate vicinity of the diffuser. In potential flow theory once the water departs the diffuser region plug flow is quickly developed. This is a inevitable consequence of using this theory. In the general tank zone the flow boundaries become parallel. After a very short distance this results in a constant gradient in potential and thus no variation in the velocity across the tank cross-section.

The risk of the eventual development of plug flow was already highlighted in the theoretical section (section 2.1). As was mentioned, on creating a flow net for a rectangular section, initial velocity fluxes are smoothed quickly. Alternatively this can be explained more mathematically; since the flow is driven by a source, without the presence of other influences the velocity must decay logarithmically towards a constant velocity. By either reasoning it is clear that some alternative or improvement is required.

It is likely that the long range behaviour of potential flow has not been dwelled upon in the literature as the subject is predominately focused on the near flow field and the far flow field is simply considered as an infinite expanse of constant velocity. The reason for typically only focusing on the near field velocity is due to the fact that the technique is most commonly used in aerospace engineering. In this research area the performance of wings is important and an assessment of lift characteristics, for example, can be made using only the flow field around an airfoil cross-section.

Reasoning for whether the CFD results beyond the diffuser exit are presumably correct can be made from a variety of perspectives. Since the validation of the diffuser itself has been possible by a number of independent approaches, one could therefore make the assumption that the rest of the simulation is also valid. After the diffuser the problem is just a case of water completing the last section of the bend and then continuing on a straight course. Since this is not very complex it can be hypothesised that the flow is likely to be correctly computed in this area. The constituent parts are well documented. Flow around a bend is known to be fastest on the outside of the bend as is the case in the final design. Further along the channel a fully turbulent profile is expected. Ignoring a jetting effect after the diffuser, a turbulent velocity profile is already present so the rest of the domain only serves to show how the profile smooths out. A counter argument is that by inputting wrong parameters in CFD it is possible produce incorrect result viz, it is possible to use to coarse a mesh which causes too much numerical dampening. A mistake with the setting could have been made resulting inaccurate results downstream of the diffuser.

Because not all parts of the simulation could be validated the cautious recommendation would be that experimental validation is required. For such big budget project it would be prudent to produce a scaled down prototype of the wave and tidal tank anyway. Like this any unforeseen issues can be revealed at a scaled down cost.

5.2 Design Appraisal

This is partly the reason why it was necessary invent new design as published in this thesis. The other influence responsible for the novel ex-cogitation demonstrated in this case, is that a circular tidal or wave tanks are uncommon. As a result of the round shape the usual closed system recirculation of the water required rethinking. An alternative routing of the water was considered–by running piping under the tank the water return flows on 90 degree plane to that of a conventional water tunnel. The resulting lack of a free surface necessitates a means of converting the velocity profile developed in the conduit into the correct profile for channel flow.

5.3 Practical Design Considerations

6

Further Work

This was the first project to investigate diffuser designs for the wave and tidal tank; the time available as a thesis project was limited and more development work is needed. Under these circumstances, the CFD has been kept as simple as possible without compromising the quality of the results. This was achieved using grid convergence studies and validation against theoretical work. Going forward this principle could be applied to more complex models, where additional complexity could be introduced in a number of ways; the current model could be extended to the 3rd dimension - it is not expected that this would alter the results significantly but it would be prudent to confirm this; instead of using a rigid lid model for the water surface, an alternative such as the volume of fluid (VOF) method could be used. A more sophisticated model could then incorporate the action of the wave machine and determine what flow is produced by the interaction between the the diffuser and wave machine and whether it is acceptable.

This thesis relied heavily on potential flow theory for validation of the CFD results. However the potential flow does not agree well with the CFD. If the progression towards the final design includes a prototype design, as is likely, the prototype should be used for the validation of the CFD results so that further computational optimisation of the design can be done with confidence.

References

Alem, P. v. (2009). Flow. Acessed 03.02.11. iii, 11, 26

Bradshaw, P. and Mehta, R. (2003). Wind tunnel design. Accessed 27.01.11.
URL: *http://www-htgl.stanford.edu/bradshaw/tunnel/wadiffuser.html* 5

Build a Wind Tunnel (n.d.). 5

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H. and Raad, P. E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in cfd applications, *Journal of Fluids Engineering* **130**(7): 078001.
URL: *http://link.aip.org/link/?JFG/130/078001/1* 30

Chang, P. (1976). *Control of flow separation: energy conservation, operational efficiency, and safety*, Series in thermal and fluids engineering, Hemisphere Pub. Co. iii, 5, 13

Dekker, M. (2002). *Fundamental Mechanics of Fluids*, Mechanical engineering, Marcel Dekker. 5

Douglas, J. and Swaffield, J. (2005). *Fluid mechanics*, Pearson/Prentice Hall. 4

Fox, R. W. and Kline, S. J. (1962). Flow regimes in curved subsonic diffusers, *Journal of Basic Engineering* pp. 303–315. iii, 13, 15, 17, 18, 23

Gülich, J. (2010). *Centrifugal Pumps*, Springer.
URL: *http://books.google.co.uk/books?id=9xAKk7Zi5RQC* 4

Isola, D. (2005). Joukowski airfoil transformation. Acessed 30.01.11.
URL: *http://www.mathworks.com/matlabcentral/fileexchange/8870-joukowski-airfoil-transformation* 5

Jayaraman, D. (2006). Panel method based 2-d potential flow simulator. Acessed 30.01.11.
URL: *http://www.mathworks.com/matlabcentral/fileexchange/12790* 5

Kirchhoff, R. (1985). *Potential flows: computer graphic solutions*, Mechanical engineering, M. Dekker. 5

Magrab, E., Balachandran, B., Azarm, S., Duncan, J. and Herold, K. (2010). *An engineer's guide to MATLAB: with applications from mechanical, aerospace, electrical, civil, and biological systems engineering*, Prentice Hall. 5

Mason, W. H. (1995). Incompressible potential flow using panel methods. Accessed 29.01.11.
URL: *http://www.aoe.vt.edu/ mason/Mason*$_f$*/C* 5

Massey, B. (1975). *Mechanics of fluids*, Van Nostrand Reinhold Co. 4

Melin, T. (1991). Accessed 30.01.11.
URL: *www.redhammer.se/tornado/thesis.pdf* 5

Nylander, P. (n.d.). Potential flow. Accessed 30.01.11.
URL: *bugman123.com/GANNAA/* 5

Sankar, D. L. N. (2008). Panel method. Acessed 28.01.11.
URL: *soliton.ae.gatech.edu/people/lsankar/AE2020/Panel.Methods.ppt* 5

Schlichting, H., Gersten, K. and Gersten, K. (2000). *Boundary-layer theory*, Physic and astronomy, Springer.
URL: *http://books.google.co.uk/books?id=8YugVtom1y4C* 1

Tatman, N. (2006). Wind tunnel design and operation. Accessed 29.01.11.
URL: *www.radford.edu/ chem-web/Physics/images/nathan-tatman-thesis.pdf* 5

Ulerich, R. (2009). richardson.m. Accessed 120111.
URL: *http://www.mathworks.com/matlabcentral/fileexchange/24388* 30