Development of Dispersion Model of a Two-stroke Engine Outboard Plume

STACY BUGEJA AND MOHAMMAD G RASUL*
School of Engineering and Technology, Central Queensland University
Rockhampton, Queensland 4702
AUSTRALIA
*Corresponding author email: m.rasul@cqu.edu.au

Abstract: - Many studies have shown that the conventional two-stroke engine produces more emissions than most other types of marine engines; this is due to the incorporation of the total-loss lubrication system into the design of the engine. As a result of stricter regulations recent developments in the outboard have seen the two-stroke engine go from the convention oil/petrol mix to direct fuel injection. Direct fuel injection has the potential to significantly reduce two-stroke engine emissions by 75%-95%. Whilst this is a significant development there is still a number of conventional two-stroke engines operating, with the typical two-stroke engine having a life span of between 10-20 years; consequently the environment still experiencing the effects of unburned residual and partially burnt oil being emitted by the engine exhaust. The aim of this study was to investigate the properties of dispersion of a two stroke engine. This was achieved through the development of a computational fluid dynamics (CFD) model that would simulate the flow downstream of a propeller and be validated by experimental data obtained from literature. Data was taken at four major points downstream of the propeller hub on the z-axis, from those locations several sub points of data were recorded tangentially to the y-axis. The simulated data was validated against experimental data sourced from literature. The findings yielded a large variation between simulated model data and the five sets of data that were taken downstream of the propeller.

Key-Words: - Dispersion model, CFD Fluent, Two-stroke engine, pollution, marine life.

1 Introduction

Many studies have shown that the conventional two-stroke engine produces more emissions than most other types of marine engines due to the system depositing a significant quantity of the oil-fuel mixture during the exhaust cycle of the combustion process [1,2]. The design of the conventional outboard is orientated in a position that directs the exhaust fumes away from the passengers, venting these emissions through the propeller hub below the surface of the water. Consequently the exhaust by-products are captured in the underwater line plume created by the forward by the forward motion of the outboard and spread via the propeller turbulence. The propeller turbulence dilutes the products of combustion emitted into the water column allowing the emissions to be further mixed throughout the environment; subsequently natural water currents and turbulence will further increase the spreading. The exhaust fumes produced during the combustion process are a combination of gases and condensable emissions; part of these by-products will be evaporated into the atmosphere, the rest will remain in the water column be further diluted.

Studies have shown that water contaminated by the toxicity of two-stroke engine emissions can persist for up to 14 days [3]. Comparative studies have indicated that the amount of hydrocarbons and nitrogen oxides produced by a single two stroke powered personal watercraft (Jet Ski) are the same as a 1998 model car that travels 160 000km [4]. Based on t he findings produced by a number of studies the USEPA introduced legislation in 2006 [5] that required a 75% reduction in two-stroke outboard nitrogen oxide emissions requiring a change in the design of the conventional two-stroke outboard. It should be noted that Australia does not have any regulations or standards that regulate marine outboard engine emissions. Figure 1 illustrates the dispersing action of the outboard propeller. The objective of this study was to investigate the dispersion of two-stroke engine emissions in seawater and the further impacts that are passed onto marine life. This was done through developing a CFD model using ANSYS Fluent. The model was then validated against experimental data taken from Loberto's PhD thesis [6]. Finally, the study identified the potential environmental impacts of two-stroke engine emissions in sea water.

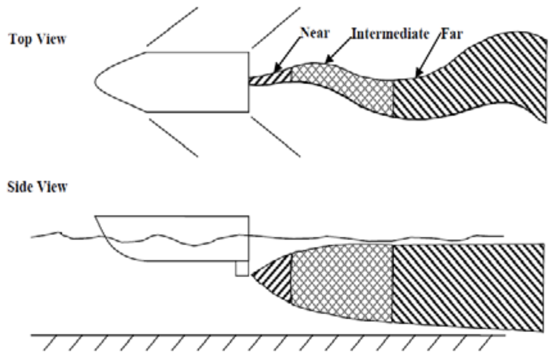

Figure 1: Diagram of the outboard propeller wake plume [6]

2 Model Development

Dispersion modelling is a type of computational model that utilises mathematical simulations and is performed to simulate exactly how pollutants are spread throughout the environment [7, 8]. When developing a dispersion model based on combustion emissions it is important to understand the following factors need to be taken into consideration; properties of the oil, oil transport process and mass or volume of the particle cloud.

It was identified within the user guide that there are essentially four main steps that should be considered when developing a model in Fluent; these are a) defining the goals of the model, b) determining the correct computation model, c) determining the physical model and d) determining the solution procedure (Fluent Inc., 2006). Modelling dispersion requires the turbulent flow downstream of the propeller to be modelled in order to understand how particles are dispersed throughout. Currently the Reynolds-Averaged Navier Stokes (RANS) set of equations is the more common approach to solving turbulent flow. Within fluid dynamics, Navier stokes are essentially considered the governing equations which define the conservation of conservation of mass, momentum and energy. Turbulent model was considered for this model development. When modelling turbulence it is classically characterised by the k-ε model. The k-ε model is one of the more common turbulence model approaches used for a solution to a viscous flow. The k-epsilon model is a two equation model that utilises the transport equations of turbulent dissipation (epsilon) and kinetic energy (k). There are three main k-ε models used in Fluent; the standard, RNG and realisable model. Each of these models have differences within the transport equations and/or use different values for the constant in the equations [9]. According to the Fluent User manual the three main differences in the equations are turbulent viscosity calculation method, ε equation for generation and destruction and differing turbulent Prandtl numbers for k and ε diffusion [9]. The standard k-ε model was used in this study. When the standard k-ε model was derived, it was assumed that the flow in the model would be fully turbulent negating the molecular viscosity. The equations for the standard k-ε model were based on the basic transport equations of turbulent kinetic energy and dissipation rate; with the kinetic energy equation being derived from the exact equation and the epsilon transport equation being derived from physical reasoning [9]. The transport equations are as follows [10]:

Kinetic Energy, K:

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

Dissipation rate equation, ε:

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

Turbulent viscosity:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where, G_k is generation of turbulent kinetic energy due to velocity gradients, G_b is the generation of turbulent kinetic energy due to buoyancy, Y_M is the contribution of fluctuating dilation in compressible turbulence to overall dissipation rate, C_{1e} = constant = 1.44, C_{2e} = constant = 1.92, C_μ = constant = 0.09, σ_k = turbulent Prandtl number for k = 1.0, σ_e = turbulent Prandtl number for ε = 1.3, S_k = user-defined source term and S_e is the user-defined source term.

The values for the constants mentioned above were determined from a number of experiments conducted with air and water for shear turbulent flows. ANSYS utilises these values as the defaults as they are capable of fitting for a w ide range of different flow types [6].

2.1 Model Geometry and Grid Information

The first step involved in developing a solution to the model requires that geometry is modelled and a grid is generated. The geometry was drafted in AutoCAD 2016 and imported into the ANSYS workbench to begin the simulation process. The mesh for this project was created using the ANSYS meshing program. To generate the model AutoCAD must be opened to build the geometry of the system. The CAD sketch for the project was modelled off of the propeller utilised from the experimental data provided. The CAD sketch includes the propeller blades and the hub o f the propeller; the diameter was 20mm from tip to tip. The tank was generated within the ANSYS design modeller stage, with the dimensions of 0.2m wide and 0.15m high as per the flume dimensions in the experimental. In order for the model to be compatible with the version of Fluent being utilised for the simulation, the sketch must be exported to an IGES file. This file format will allow the ANSYS workbench to open the geometry file in order to begin the mesh process. Figures 2 and 3 shows the propeller geometry and mesh respectively.

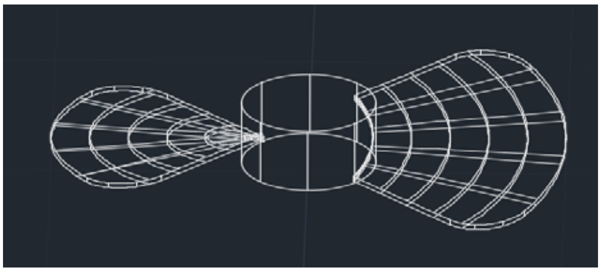

Figure 2: Propeller geometry

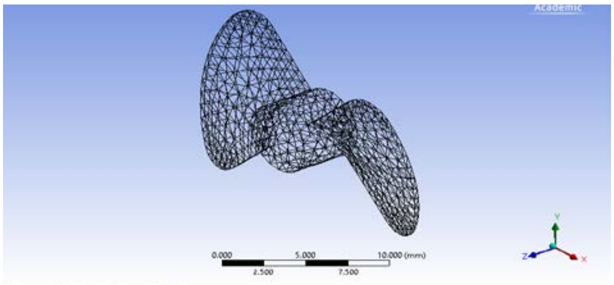

Figure 3: Propeller mesh

2.2 Solving the Problem

In the solution methods tab, the pressure-velocity coupling scheme was set to SIMPLE. The gradient was set as 'least squares cell based' and the pressure was set as s tandard. The momentum, turbulent kinetic energy and turbulent dissipation rate was set to second order upwind. As the model is running through the simulation a plot of residuals is present within the graphics window. The plot of results produced during the simulation was viewed in the side window. If it appears as if the simulation is proceeding in a satisfactory manner, a more detailed amount of iterations can be specified. After a detailed simulation is complete the data was saved, examined, validated and refined. There were a number of trials conducted for this investigation in order to determine the best possible model.

3 Results and Discussion

The propeller for this model was developed in AutoCAD and was potentially used for all modelling scenarios. The propeller went through rigorous amounts of trial and error in order to develop a model that would not cause too much skewness within the mesh and would potentially not cause divergence. It was identified that in order to be able to predict how far downstream pollutants may go, the velocity profile of the propeller needed to be modelled.

3.1 Experimental Set-up and Data

The experimental data was gathered from Loberto [6]. The jet for the scale-model boat propeller, with a tip-diameter of 20mm, was powered by a variable speed electric motor with a flexible cable transmission. The propeller was fixed in place by a frame where the axis is held in place with the direction of flow. Dye was introduced at several vertical positions downstream of the propeller and measured. The measurements at the vertical points ranged from x/D =2 to x/D=50, in this instance x is the distance downstream of the propeller and D is the propeller diameter. These measurements were recorded with a 2D Dantec forward scattering Laser Doppler Anemometer. The test was conducted in a closed loop flume that had dimensions of: 12m x 0.4m x 0.2m; with the water level held at a height of 150mm. The speed of the water used within the flume was 0.04m/s. Two jet speeds were used within this study, however only one is analysed in this study. A schematic diagram of the experimental set-up is shown in Figure 4.

3.2 Simulation Results

The SRF model was used for this study to validate against the experimental data. For this model the cylindrical housing was removed and replaced with a square tank similar to that utilised within the experimental conditions. The mesh around the model in this instance was refined to allow for a more accurate result to be obtained. The model was subsequently imported into Fluent and the boundary conditions were set. Table 1 shows the boundary and solver conditions that were allocated and Table 2 depicts solution details. Figures 5, 6 and 7 shows the turbulence contour, velocity contour and velocity vector respectively.

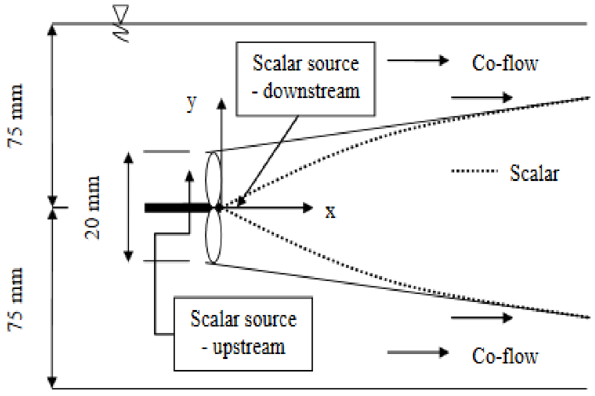

Figure 4: Schematic diagram of experimental set-up [6]

Table 1: Boundary and solver conditions

Boundary	**Condition**
Walls	Default. Set as liquid water and stationary
Velocity inlet	Velocity: 0.04m/s Turbulence: Intensity – 5% Hydraulic diameter – 0.1486m
Outlet	Default
Propeller	Set as liquid water. Velocity: 3000 rpm in the z-axis
Solver	**Conditions**
Type	Pressure-based
Velocity formula	Absolute
Time	Steady

Table 2: Solution details

Turbulence model	Standard k-ε
Wall treatment	Standard wall functions
Discretisation scheme for convective fluxes and turbulence parameters	Second order upwind
Pressure	Standard
Solution method	SIMPLE

From the velocity contours, it can be seen that the flow of the propeller had reached approximately 0.2m downstream from the start point. It can also be seen that the velocity moving directly behind the blades is very small and almost nil to begin with.

Figure 8 below better illustrates exactly how the flow profile is generated initially, it can be seen that the flow is moving as r eadily backwards as it is forwards, with the greatest amount of velocity being seen adjacent of the rotating blades. What is interesting though, is that the largest contour behind the blades appears to have spread right back to the inlet which would have mixed back in with the incoming velocity from the inlet. The velocity vector diagram also demonstrates the significant amount of vectors clustering around the blades due to revolution of the propeller during the simulation.

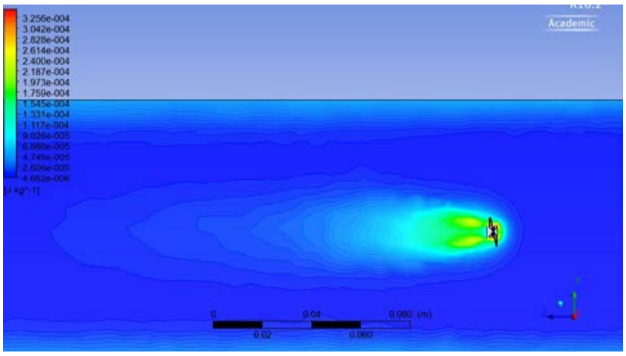

Figure 5: Turbulence contour

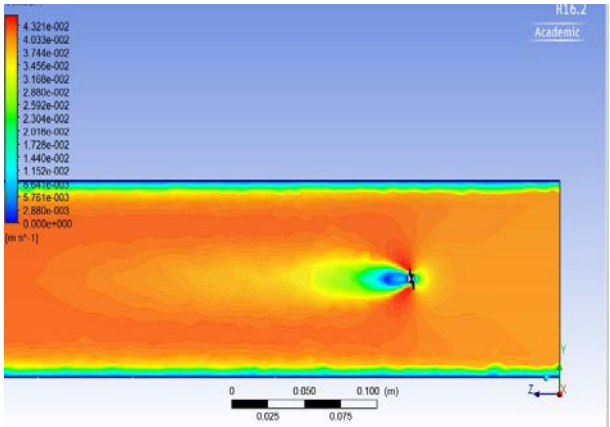

Figure 6: Velocity contour

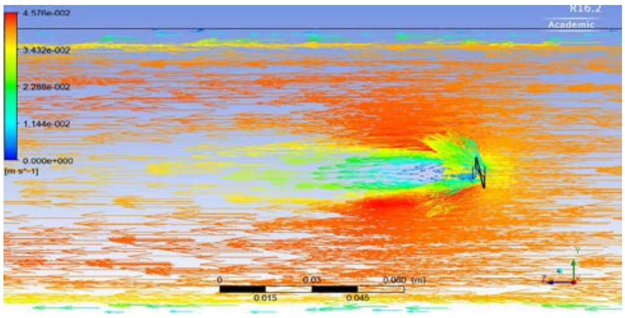

Figure 7: Velocity vector

It does not seem to have been quite captured in the data, but the water whilst having quite a small velocity is continuously mixing below the hub directly downstream of the propeller. This potentially appears to be demonstrated better in the stream line and is what causes the turbulence seen in Figure 5. It can be seen from Figure 5 that the greatest amount of turbulence is experienced directly downstream of the propeller, this is assumed to be where the greatest amount of mixing would occur within the water and potentially pollutants before it is pushed downstream.

Whilst pressure was not a variable that was set within the boundary conditions of this model; there appeared to be a greater amount of pressure built up behind the propeller and a negative pressure in the front of the propeller. The negative pressure experienced was of 1.227×10^3Pa, indicates that there is a lower pressure in front of the geometry then in the rest of the tank, which was consistent throughout the simulation process.

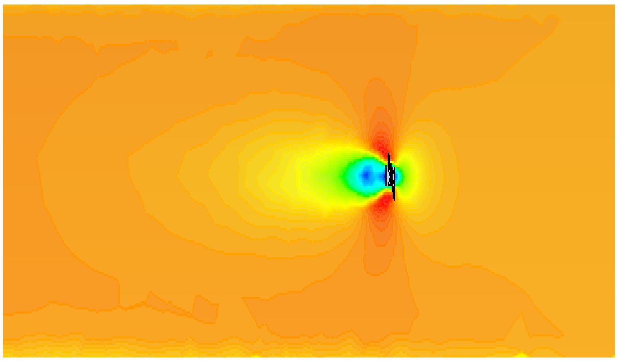

Figure 8: Velocity profile

3.3 Model Validation

The validation process for the model was carried out using the data from Loberto [6] to determine whether the model is of a good fit, or there are significant inconsistencies present. The data from literature was conducted using a 2D Dantec forward scattering Laser Doppler Anemometer, with a sample propeller 20mm in diameter. The test was performed in a flume, with data measurements recorded at a number of longitudinal points downstream of the propeller i.e. x/D. The points x/D refer to the ratio of the distance downstream of the propeller by the propeller diameter, so at 100mm downstream of the propeller the point x/D will be equal to 5. Figure 9 compares the velocity profiles of model data with experimental data for x/D = 20, while Figure 10 s hows the velocity profile for simulated data range (i.e. x/D = 2,5, 5, 10 and 20). The Figure 9 demonstrates that there is a very large variation between model and experimental data, though the trend of ther profile is similar. The velocity profiles of simulated data are consistent as shown in Figure 10. From the results it is clear that the maximum velocity that was achieved by the model in the simulation was 0.044 m /s which is quite small and proves that there is inaccuracies in the model and potentially the geometry.

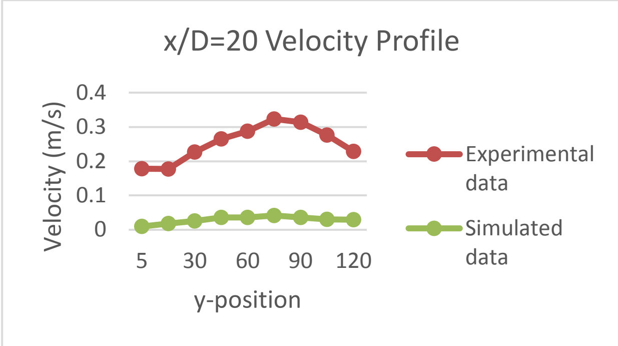

Figure 9: Velocity profile at x/D = 20

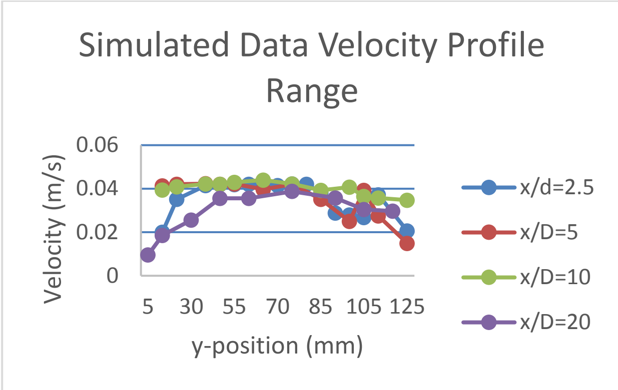

Figure 10: Velocity profile for simulated data range

Based on the validation and the data provided by this model alone for the given parameters, there is a significant degree of error; thus drawing the conclusion that the model is not a accurate representation to be used as a future model for dispersion. To be applied to a dispersion model further development into a more accurate model is necessary. There are known limitations to the MRF model, it is suggested that if a model of higher complexity is developed with the correct geometry and boundary conditions, far more accurate results could be achieved. A sliding mesh model would have been a better choice for this investigation. The sliding mesh model is able to far more realistically produce data, as it has the capacity to simulate the interaction between the rotating domain and the fluid domain through the use of interfaces. There were a number of issues that arose within the came up during the modelling phase, with regards to the geometry and developing a potential scalar function to simulate the dispersion of the model. It is assumed that the issues faced within geometry generation could potentially be the direct result of the errors of accuracy demonstrated within the model. It is essential that the development of the interfaces and splitting the geometry be done correctly otherwise a result similar to that found in the initial MRF model could result.

It is known that the emissions from the exhaust are initially mixed and spread due to the turbulence created by the propeller before those pollutants are spread further into the flow field by the downstream velocity flow. The particle track was able to demonstrate how the particles would spend a period of time being mixing with in the intiial flow field before being projected further downstream as more pollutant particles were released by the propeller. The particles were spread to potentially the entirety of the downstream flow field after a significant amount of time, which indicated that even with a flow field that demonstrated such a small downstream velocity the particles were able to be spread quite far and wide. It is assumed that if the propeller was able to produce a greater downstream velocity speed, then the particles may have been able to be dispersed far enough to reach both the top and bottom of the flume. Indicating that a 20mm diameter propeller has the capacity to disperse a pollutant flow of 150mm high and a minimum of twice that downstream of the hub. If a scale model propeller with a 20mm diameter and known limitations has the capacity to produce a reasonable sized particle flow; then it is estimated that a larger propeller would potentially demostrate a greater pollutant particle dispersion given that it would have a far more substantial flow field, coupled with the capacity to produce a greater amount of tubulence.

4 Conclusions and Recommendations

The primary purpose of this project was to investigate the properties of dispersion of a two stroke engine and the further impacts it way have on marine life. Data was taken at four major points downstream of the propeller hub, several sub points were taken tangentially from the propeller. The simulated data was plotted and validated against experimental data sourced from literature. The findings yielded a very large variation between simulated data and the five sets of data that were taken downstream of the propeller (for example, see Figure 9 for x/D = 20). A number of parameters were altered in order to generate a flow field with a higher downstream velocity, however these attempts were not successful in yielding a better result. It is clear however that the co-flow has a significant effect on the downstream velocity results, as when the velocity and speed was altered the downstream flow increased. However based on the data produced for validation, it is clear that there is an error present somewhere within the model or the geometry; as a result it has been deemed that more work is required to refine the model before it could be utilised to simulate the downstream pollutants from a two stroke outboard.

References:

[1] Gaca, H et al. 2014, *Total-loss Lubrication Systems, Overview*. Springer, Heidelberg, Berlin.

[2] Holmes, N & Morasakg, L 2006, 'A Review of Dispersion Modelling and Its Application to the Dispersion of Particles: An overview of Different Dispersion Models Available', *Atmospheric Environment*, Vol 40, no. 30, pp. 5902-5928.

[3] Juttner, F et al. 1995. 'Emissions of Two and Four Stroke Outboard Engines – II: Impact of Water Quality', *Water Resources,* Vol 29, no. 8, pp. 1983-1987

[4] Martin, L. C. 1999. *Caught in the wake: The Environmental and Human Health Impacts of Personal Watercraft,* Izaak Walton League of America.

[5] U.S. Environmental Protection Agency 1996, *Control of Air Pollution Emission Standards for New Nonroad Spark-ignition Marine Engines,* viewed 24 August 2015, http://www.epa.gov/otaq/regs/nonroad/marine/marnfria.pdf

[6] Loberto, A 2007, *An Experimental Study of the Mixing Performance of Boat Propellers,* viewed 6 August 2015, http://eprints.qut.edu.au/16619/1/Anthony_Loberto_Thesis.pdf

[7] Egerton, J. O, Rasul, M. G and Brown, R. J 2002, 'Outboard Engine Emissions: Modelling and Simulation of Underwater Propeller Velocity Profile Using the CFD Code FLUENT', *Proceedings 16th Australasian Fluid Mechanics Conference,* University of Queensland, pp. 777-781, Viewed 5/8/2015. http://eprints.qut.edu.au/11823/1/11823.pdf

[8] Subhas, S et al. 2012, 'CFD Analysis of a Propeller Flow and Cavitation', *International Journal of Computer Applications*, Vol 55, no 16, pp 26-33.

[9] Fluent Inc. 2006, *FLUENT 6.3 User's Guide*, viewed 1 May 2016, https://www.sharcnet.ca/Software/Fluent6/pdf/ug/flug.pdf

[10] Autodesk Inc. 2009, *Specifying turbulence options,* viewed 8 October 2015, http://download.autodesk.com/us/algor/userguides/mergedProjects/setting_up_the_analysis/fluid_flow/Analysis_Parameters/Steady_or_Unsteady_Fluid_Flow_%28Turbulence_Options%29.htm