17th Australasian Fluid Mechanics Conference
Auckland, New Zealand
5-9 December 2010

CFD Analysis of a High Speed Paddlewheel

W.C. Bowen and K.V. Alexander

Department of Mechanical Engineering
University of Canterbury, Christchurch, New Zealand

Abstract

The primary goal of this CFD study is to model the forces generated by a high speed paddlewheel used for propulsion of a water craft. The simulation involves modelling transient analyses in Ansys CFX v12.1 of solid body water impact, which includes solving the Volume of Fluid (VOF) equation for two fluids dispersing rapidly throughout a flow domain. 2D and 3D paddlewheel models are compared. A transient 2-D water piercing, vertical flat plate slamming case is also used for model verification. Both the flat plate slamming case and the paddlewheel cases are compared to experimental results.

Introduction

The goal of the study is to verify that CFD can accurately model a lifting paddlewheel (LPW) system in order to optimize the overall wheel shape in the future. The lifting paddlewheel is a high speed wheel with blades shaped such that a momentum exchange with the water propels a vehicle above and across the water surface, i.e. water skimming. Earlier studies by one of the authors (KA) included measuring propulsive forces generated by one wheel immersed in a long flow rating tank with a car which travels the length of the tank. [1] A range of tests were performed for a variety of wheel shapes, immersion depths, rotational speeds, and forward speeds in order to gain an understanding of the performance capabilities of a wheel, in particular the thrust and lift. Various 4-wheeled remote controlled 1/10th scale models built by both authors demonstrated the concept. [2]

Method

The "standard wheel", experimentally tested in Alexander's thesis, [1] is modelled using ANSYS CFX v12.1 in the present work. The model consists of six blades (solid walls) located on a rotating domain within a static domain. The static domain includes the boundary inlet/exit conditions. The water free surface location is initialized at the inlet and solved throughout the domain via the VOF (volume fraction) solver. The interface between the two domains is modelled as a transient rotor/stator with General Grid Interface (GGI) mesh connection. 2D and 3D simulations are compared, though due to computational expense 3D modelling was limited. The simulation is unsteady, with the time step size selected so that for each time step the flow passes through roughly one new computational element. This is crucial to accurately solve for the free surface location within the domains. The computational grid was created using CFX Mesh (a tetrahedra dominant method). Note the refined zones around the boundary interface and blade walls which are both regions where the free surface is subject to rapid change. Each cell is outlined in green; the refined zones show up as a dense green in figure (1). The flow enters the domain from the right at constant velocity (Vo) for all cases of 2.36 m/s with a water surface height 30mm above the bottom of the wheel. Note- the wheel refers to the bladed wheel of which only the blades are modelled, not to the domain boundary. Wheel rpm (n) is different for each case rotating clockwise at constant speed matching the flow conditions used in the experimental tests. By varying the rotational speed multiple flow cases are simulated according to the velocity ratio (Vr) which is the ratio of forward speed to wheel tip speed. Vr is the equivalent to screw propeller slip, an indicator of propulsive efficiency. The outlet on the left is of constant velocity, with zero gradient of all other flow parameters (ex. VOF). The top is also an opening of constant velocity, zero gradient. The bottom boundary is a slip wall. The wheel is 242mm in diameter (D), 72mm in width (s), and blade chord (c) is 25mm. The shear stress transport turbulence model is used for both domains and fluids, with second order, high resolution options selected.

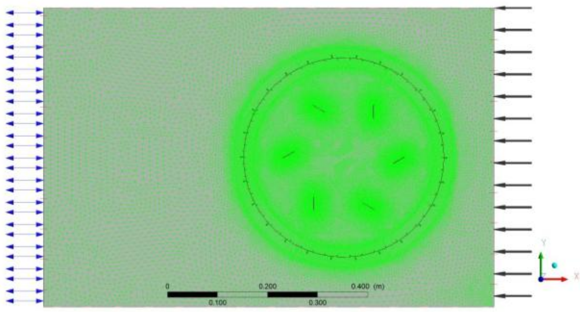

Figure 1 LPW Mesh

2D Results

The most challenging aspect of the CFD problem has been modelling the transient free surface. The free surface is not just a big water splash, but splash, interacting with splash created by the previous blades, at each blade impact. Contours of volume fraction for the LPW case are not mesh independent, but rather model smaller and smaller water droplets dispersed throughout the air. Due to the free surface changing, propulsive forces also fluctuate with each blade impact. The resulting flow can be seen below with the contours of volume fraction plotted (i.e. fraction of water and air in each cell), which define the free surface of water. The result has been compared to stroboscopic studies of a similar wheel in a testing tank. Visually the flow is similar. The flow seen in figure (2) is a 2-D representation of the actual flow seen in figure (3).

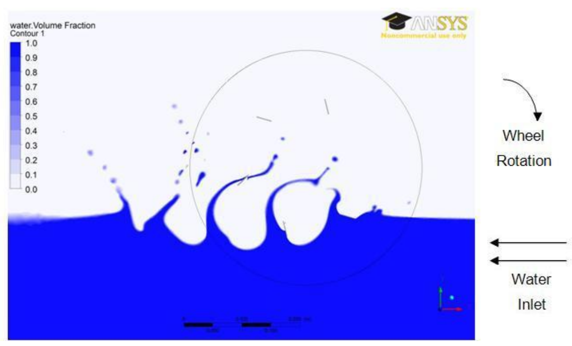

Figure 2 LPW 2D Case (n=5rps)

Figure 3 LPW Stroboscopic Studies [1]

Propulsive forces generated by the LPW are almost entirely due to the momentum exchange between the blade and water which happens on the bottom surface of the blade. Very little force is generated by viscous stresses as they can be monitored easily in the simulation. For the tank test case, forces are measured on the wheel via a force balance. For the CFD case, forces are monitored for each individual blade (a wall boundary) and summed to give a time dependent force as shown in figure (4). Convergence criterion for each time step was set to 1e-4 RMS.

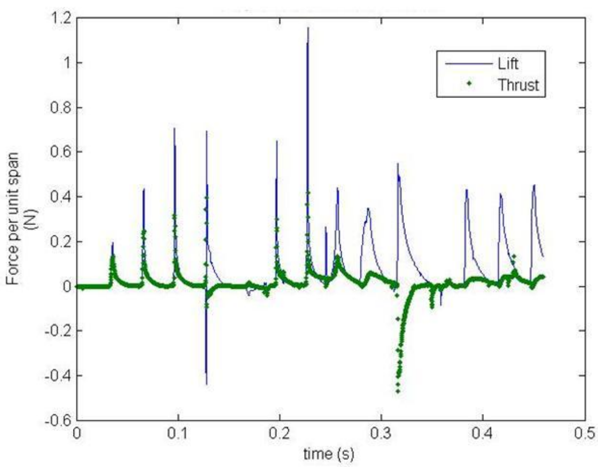

Figure 4 CFD Propulsive Forces (n=5rps)

For the time duration modelled, the simulation does not approach any steady state force, or repeating force pattern. Modelling a greater time duration was not within a feasible limit of solver time (see discussion). In order to approximate the average force acting on the wheel for comparison to the experimental values, a time interval must be selected over which to average the force of impact. Averaging the solution over a particular time (i.e. a series of blade impacts) comes with an uncertainty on the order of 50%, due to there not being one steady solution of force over the flow time modelled. Example- choosing a time interval after .3 seconds in figure (4) yields a ~50% difference with force averaged from a time interval after .1 seconds (both of which appear to have achieved a steady state when observing the volume fraction contours). Theoretically a well refined case, if allowed to run for a time much greater than that showed in figure (4) would result in a "steady" repeating force pattern. As an approximation, the averaged results of lift force for each case have the "uncertainty" applied and plotted against the experimental results shown in figure (5).

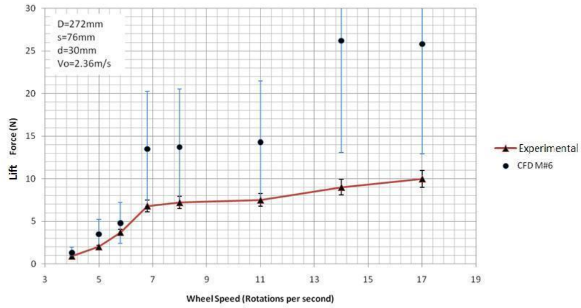

Figure 5 CFD - Experimental Comparison

Below a rotational wheel speed of about 6.5rps the force comparison error is within the 50% uncertainty. For wheel speeds above 6.5rps the error it is not within the uncertainty. The uncertainty in the test balance used in the experimental measurements is the greater of 0.5N or 10%.

When velocity ratio (Vr) is too low most of the power is used to maintain the depression in the water surface and little thrust is generated. This state is termed cavity intrusion- a state of LPW operation in which the descending blade meets the air cavity created by the previous blade. This is similar to when a wheel "digs a hole" once a car gets stuck in snow or sand. The flow result of cavity intrusion is essentially a wave being pushed forward by each blade as seen in figure (6) on the right. This causes increased splash (termed "bow-splash" in the 1983 thesis [1]) which is very difficult to model in CFD due to the rapidly changing free surface of water. For the immersion depth modelled, at velocity ratios less than 0.5 (n > 6.5rps for the given Vo = 2.36 m/s) cavity intrusion occurs as illustrated in the right in figure (6). LPWs are designed to operate at velocity ratios greater than 0.5 and in this zone (seen in the left in figure (6)) the model is within the CFD uncertainty.

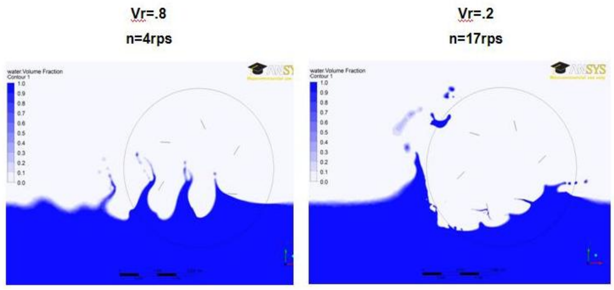

Figure 6 Cavity Intrusion

3D Results

As seen in figure (4) the 2D simulation does not converge to a steady state within the flow time modelled. A 3D model was run as a comparison to the various 2D simulations in an attempt to obtain a more accurate solution. However, due to the computational resources needed to model the 3D transient water splash, only one case (i.e. one velocity ratio) has been modelled. The case modelled is for n=5rps. Note- in figure (7) the iso-surface of VOF=.5 is plotted (i.e. the water free surface or surface of constant VOF=50% water, 50% air) with contours of pressure plotted on that surface. The flow in figure (7) is coming in from the left and the wheel is rotating to the left. There is also a symmetry plane in the middle of the blade (low z value fig (7)), and only half of the wheel is modelled about this plane. The picture is taken at an angle to visualize the 3D distribution.

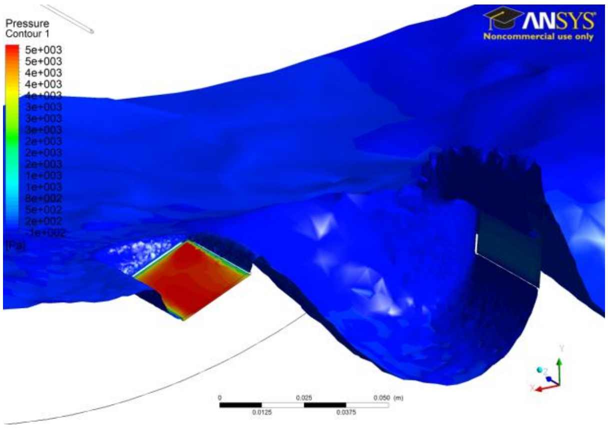

Figure 7 3D Pressure Contour Result on Free Surface

In figure (8) below the 2D and 3D models are compared for the n=5rps case. It appears as if the 3D model might be a better approximation for the experimental values, though no conclusions regarding this can be made as more cases have not been modelled.

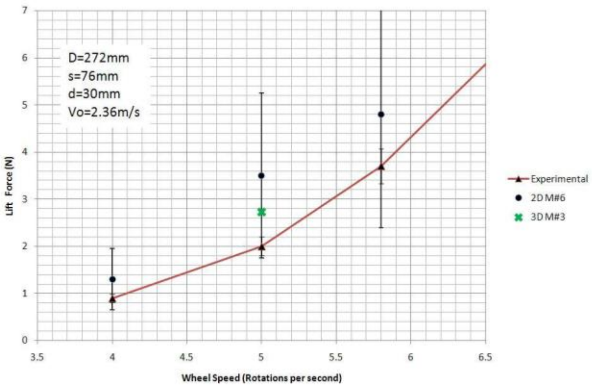

Figure 8 3D Case Comparison

Model Verification

For both the 3D and 2D models, simulations were run of increasing mesh density to determine whether or not the flow was mesh independent in a mesh independence study (MIS). However, the complexity of the LPW case makes getting a converging or mesh independent result difficult. To simplify the system a 2D flat plate was modelled in ANSYS CFX making purely vertical impact. The model is similar (same turbulence, VOF, boundary settings, etc.) to the LPW CFD model though with just one blade, and a flow of water approaching from the bottom of the domain which is initially just air (i.e. one blade is falling on the surface of the water). The purpose of this was to verify that water impact can be modelled accurately using the software, and to determine what mesh density is necessary to do so. The same time step convergence criterion is implemented as was for the LPW case, 1e-4 RMS.

Shown in figure (9) is the MIS result for a simple 2-D flat plate slamming case with purely vertical velocity (Vv) of 7 m/s. An earlier study compared a similar flat plate case using Fluent, to various experimental results with the empirically derived formula from Chuang [3] having the closest comparison. The empirical formula is for maximum impact pressure on a flat plate, not the time averaged values (i.e. peak values shown in fig (4)).

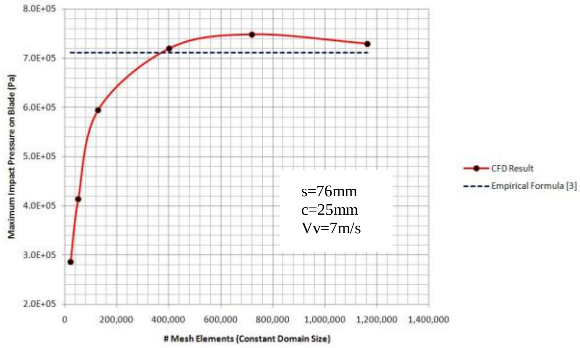

Figure 9 Max Impact Force for Flat Plate Impact

As an initial result of the study, it appears that very high mesh density (approx. 400,000 elements for the simple 2D flat plate case) is necessary to solve the force monitors accurately. The reason the mesh must be so dense is to solve the VOF equation at the rapidly changing free surface during water splash. Figure (10) illustrates the difference in free surface contours between simulations of different mesh densities which also affects the forces monitored on the blade surface.

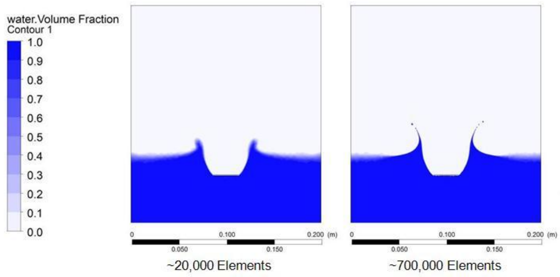

Figure 10 VOF Contour Variation with Grid Density

Discussion

Knowing the results of the mesh independence study, in order to get a closer approximation of the LPW case it would be necessary to refine the LPW grid to a density equivalent to the mesh independent grid for flat plate slamming, which shows >400,000 elements are necessary for the flat plate case. For the LPW case which has a more complex mesh, one would apply the same mesh spacing used in the mesh independent flat plate case to the 2D LPW domain, which results in a mesh of several million elements. The 2D case currently uses a mesh of about 100,000 elements, which takes several days to solve for the multiple blade impacts seen in figure (4). The 3D case uses a mesh of slightly over one million elements (a very coarse grid when compared to the 2D mesh spacing) which takes several weeks to solve. An LPW result with minimum uncertainty would also require extending the flow time beyond that performed in this study in order to get a steady repeating impact force curve. The average of that repeating force curve would theoretically match closer to the experimental result. With the current computational resources available it is estimated each refined 2D case would take over a month to solve. For an approximation of the CFD LPW case, the results presented with uncertainty are sufficient.

In figure (4) thrust force is seen to decrease to a negative value at one blade impact. This is due to the bow splash or flow being pushed forward by the wheel at certain intervals. It has also been seen in experimental results. At certain immersion depths and velocity ratios this causes negative average thrust. This unsteady nature of the flow is the reason no "steady state" or repeating force pattern has been modelled to date.

Conclusion

A 2-D and 3-D model of a lifting paddlewheel has been presented with lift force compared to experimental results of such a wheel. The curve trends observed in the experiments were re-created with CFD. Within the known operating zone of lifting paddlewheel operation the simulation agrees with experimental results within a "CFD uncertainty" (which comes from a time averaged approximation of an unsteady force) of about 50%. The CFD error is consistently greater than experimental which could theoretically be reduced by running a simulation of more refined mesh for a longer flow time, and could be explored as future research.

Acknowledgements

The authors would like to thank Technix Group Limited for assisting in funding the cost of this research.

References

[1] Alexander, K.V., The Lifting Paddlewheel – a non-buoyant wheel enabling a high speed amphibious craft to run on the water surface, PhD Thesis, University of Canterbury, 1983

[2] Alexander, K.V. Prototype Model Tests of a High Speed Wheeled Amphibious Vehicle, 8th AFMC, 1983, 4c10-4c13

[3] Chuang, S.L., Experiments on Flat Bottom Slamming, Journal of Ship Research Vol. 10 No. 1, March, 1966, 10-17