Excerpt from the Proceedings of the COMSOL Conference 2009 Boston

A Study of Planning Hydrodynamics

Nathaniel D. Barnett,
General Dynamics – Electric Boat
1657 South Rd Kingston RI 02881
natbarnett@gmail.com

Ernesto Gutierrez- Miravete
Rensselaer at Hartford,
275 Windsor St, Hartford CT 06120
gutiee@rpi.edu

Abstract: This paper report on a study of the hydrodynamics of skimboards and surfboards using the computational fluid dynamics (CFD) module in COMSOL. The study analyzes the flow in a thin water layer underneath a skim board in a 2-D Cartesian coordinate. Three different sets of boundary conditions were employed and one of them produced the best agreement with previous findings.

Keywords: Fluid Flow, surfing, planning, lift coefficient, lift force

1. Introduction

A skimboard is a flat board approximately 1-2 m in length and 1m in width that is used to plane on shallow water along a shoreline for short distances (10 m max). Boards are normally used in less than 2 inches (50mm) of water. The experience is somewhat similar to using a surfboard, or powerboat, except it takes place in very shallow water. This study investigates the flow in the water layer under a skimboard using the CFD module in COMSOL, and validates the analysis by comparison with prior work by Tuck and Dixon [1] and Sugimoto [2]. An analysis of this type may be helpful to skim board designers who are interested in prototype testing of design ideas with fast turnaround time.

Previous studies of skim boards can be found [1]-[4]. All three studies neglect gravity and assume one dimensional flow under the board. In this study, the flow is assumed to take place in a two dimensional Cartesian system of coordinates, that the water surface is smooth surface (no waves) and that the water flow is negligible. In practice, the individual using a skimboard uses the board on the flattest part of the water surface in order to gain the best advantage.

The skim board user faces tradeoffs during the operation of the board. The angle of attack should be as small as possible to reduce drag. However, the angle must be greater than zero to remain afloat. The properties of lift disappear with the introduction of water above the board. This can be seen in surfing - as soon as the leading edge digs into the water, the rider takes a swim. Another tradeoff is the length versus the potential speed. As the vehicle slows down the wetted area must increase in order to remain afloat. Hence, an unpowered skimboard can only go a few meters from the start point. Sugimoto also argues that posture and wind resistance pay a part in the ability to travel on a skimboard. He concludes that a person standing up straight to provide the best leverage for balance will forfeit some advantage with bluff body friction against the surrounding air. Although the weight of a board is implicitly taken into account in this study, no attempt is made to model the wind resistance.

The main objective of the study was to compare the previously obtained results using simplified models, to those obtained using the CFD module in COMSOL. Another goal was to demonstrate the development of simple and easily solvable models to gain meaningful information about the flow during underneath a skim board.

A full three dimensional CFD model often requires months to program, model and solve. Resolution on a scale that provides useful information requires an enormous amount of nodes and corresponding fluid elements. For example, modeling the rudder and propeller area of a power boat may require a $3m^3$ computational box divided up into 3000^3 nodes, with the propeller and free surface modeled in place. For this size a finite element approach starts to be impractical and finite difference methods are used. In addition, the free surface has to be calculated. This requires that special programming be added into the standard models available in CFD packages.

A small manufacturing company or a hobbyist cannot afford the time and resources required to create such models. They are thus more likely to make a best “guess” and then build a prototype to try. For such companies, it would be desirable to be able to run some quick calculations over a short period of time to determine if their design ideas are on the right track before going to the added expense of building a prototype. The determination of the viability of simplified CFD models for the use in investigating the flow behavior under a planning type craft, here in the case of a skimboard, can be very helpful to designers

2. Theory and Model Formulation

2.1 Tuck and Dixon and Sugimoto Analyses [1], [2]:

Figure 1 is a schematic of the system, where a board is shown standing over a water layer that moves from left to right to left producing a jet at the leading edge. This arrangement is entirely equivalent to the board moving from right to left over a stagnant water layer.

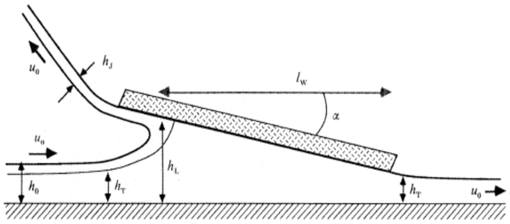

Figure 1: Sketch of the flow under a skim board [1]

The skim board will be assumed to be a planing surface of length l_w on the water and the original depth of the water is h_o. Moreover, the board is at a very low angle of attack.

Since the flow upstream of the board tends to pile up prior to any spray back we can apply Bernoulli's equations to the flow. Incorporation of conservation principles yields the following expression for the pressure underneath the skimboard

$$p(x) = p_0 + \frac{1}{2}\rho[u_0^2 - u(x)^2] = p_0 + \frac{1}{2}\rho u_0^2\left[[1 - \frac{h_T^2}{h(x)^2}\right] \quad (1)$$

Since the jet only covers a minimal portion of the surface, the lifting force component of the jet is assumed to be minimal. The lifting force is then based on the pressure of the wetted area downstream from the entry point and is given by:

$$F = \int_{x_L}^{x_T}(p - p_0)\,dx = \frac{1}{2}\rho u_0^2 \int_{x_L}^{x_T}\left[[1 - \frac{h_T^2}{h(x)^2}\right]dx \quad (2)$$

The moment about the trailing edge is then found by multiplying the force with the distance from the trailing edge, i.e.

$$M = \frac{1}{2}\rho u_0^2 \int_{x_L}^{x_T}(x - x_T)\left[[1 - \frac{h_T^2}{h(x)^2}\right]dx \quad (3)$$

Both Tuck and Dixon and Sugimoto reviewed a paper by Edge [3]. Sugimoto compared the results from Edge paper to those by Tuck and Dixon paper and concluded that the latter paper provided a better model for a skimboard. Sugimoto solved the lift using an alternate method and obtained agreement with the results of Tuck and Dixon. Sugimoto then went further and determined the point at which the skimboard will cease to float and accounted approximately for wind resistance by assigning a shape factor of 0.72 for a 70 kg individual and a bluff body drag coefficient of 1.0.

2.2 Finite Element Model:

Figure 2 shows a sketch of the system and labels the boundary conditions used in this study. A two-dimensional Cartesian system of coordinates is used.

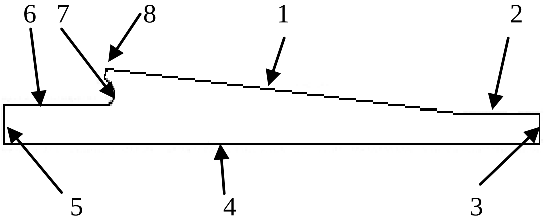

Figure 2: Sketch of the system being modeled and description of domain boundaries.

Boundary 1 represents the underside of the skim board while boundary 4 is the sea floor. Boundaries 2, 6, and 7 are free surfaces, where the shape of 7 is based on the geometry given by Sugimoto. Vertical boundary 5 is the entry channel and boundaries 3 and 8 are the outflow channels, respectively, the trailing edge and the jet. COMSOL Multiphysics was used to create a model of the above system. COMSOL solves the governing equations for a laminar, incompressible, Newtonian fluid, namely, the equation of continuity and the momentum balance (Navier-Stokes) equations.

$$\nabla \cdot \mathbf{U} = 0\,, \qquad (4)$$

$$\rho \frac{DU}{Dt} = \rho g - \nabla p + \mu \nabla^2 \mathbf{U}\,. \qquad (5)$$

Regarding boundary conditions, no-slip, non-moving wall boundary conditions are designated as u = 0. Free surfaces are assumed to be symmetry boundaries, and are defined by having no shear, as in $\mathbf{n} \cdot \mathbf{u} = 0$. For moving walls, the nodes on the boundary are given the velocity of that boundary ($u = u_w$). For sliding walls, the nodes are given the tangential velocity and it is assumed to have no-slip.

The following specific cases were considered to determine which scenario produced best agreement with the results of prior workers:

- Case 1: With the bottom boundary of the water layer held fixed, the board is made to move tangentially along the angle of attack. The boundary condition in this case will be as a conveyor belt moving.
- Case 2: Here 2 the board is held fixed and an entrance velocity is introduced. The lower boundary is moved with the same velocity as the incoming fluid, in order to maintain the relative velocities of the solid objects. Moreover, the upper boundary (skim board) is fixed.
- Case 3: The board moves as a rigid body at constant velocity along the negative x direction. One problem with this case is that it fails to take into account how the board will have lift and float on the water while the board is in motion above the terminal velocity, i.e. the point at which the board does not have enough lift to rise above the bow wake.

It is noted that through modifications to the program, such as adding special boundary conditions for the free surfaces, and special dimensional changes for board movement, a much better model could be achieved. However, since the aim of the project was to verify the use of simplified models, the above approaches were considered reasonable.

3. Solution Methodology

The geometry described previously (Figure 2) was created using COMSOL Multiphysics. The specific dimensions were selected so as to match, or closely represent the magnitudes noted in Sugimoto. The geometry was then meshed using the generic automatic meshing tool available in the software. The element shape is of a 2-D planar tetrahedron with nodes at the apex. The mesh was made finer with one level of refinement using the tool available in the software. The coarse mesh had 366 elements, and the refined mesh had 1464 elements. A comparison of calculation time between the two meshes showed negligible difference in calculation time and results. Since the purpose of the project is to show a simple CFD method for developing planing craft, this general approach for creating the model was chosen.

Figure 3 below shows the Sugimoto geometry and the mesh. Note the concentration of nodes near the leading edge. Since a lot is happening to the flow in this location the software decided that added nodes were needed in order to capture the detail in the flow. It should be noted the shape of the jet is entered into the geometry prior to calculation and is not an output of the calculation. Thus, when using the present approach, some prior knowledge of the flow is needed. Calculation of the free surface shape and the spray jet would require a more extensive study.

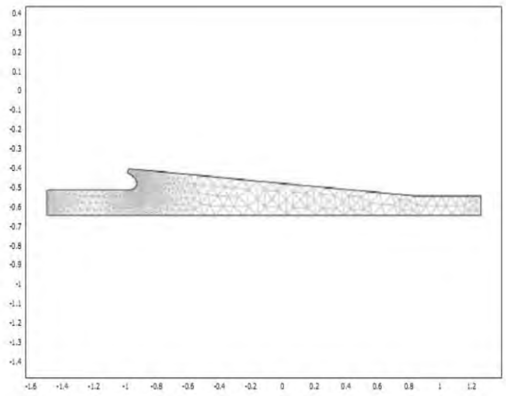

Figure 3: Sugimoto geometry and mesh produced using COMSOL

The model shown in figure 3 is then solved using the COMSOL solver with the default solution parameters and the boundary conditions noted previously, specifically:

- Case 1: The left hand side (boundary #5) is designated an entryway with a minimal velocity of 0.3 m/s. The trailing edge (#3) and the jet (#8) are designated as exits, with the pressure set to zero. The free surfaces (# 2, 6, and 7) are set to symmetry. The skim board (boundary #1) is set as sliding wall. The induced motion (between 3 and 6 m/s) is that of a conveyor belt.
- Case 2: The water at the entrance channel is given a velocity equal to that of the lower boundary (between 3 and 6 m/s). The free surfaces and exits are given the same boundary conditions as in case 1. The board (#1) is held fixed in space with a no-slip condition.
- Case 3: The board (#1) is then given a velocity ranging from 3 to 6 m/s in the negative x direction. A no-slip condition is then entered for the lower boundary (#4). The entryway (# 5) is given a velocity of 0.3 m/s. The free surfaces and exit channel are given the exact same boundary conditions as cases 1 and 2.

Steady state conditions were assumed for all solutions. This was justified since the velocity field changes little from one small fraction of a second to the next.

Computed velocities, streamlines, pressure distributions and the like were all easily generated by the program and they were useful in determining the appropriateness of the selected boundary conditions. Other post-processing tools available in the software were then used to investigate the computed field variables along selected cut lines. Specifically, the pressure distribution under the board was plotted against the length of the board. The computed pressure distribution was then integrated using a trapezoidal method in order to obtain the lift forces. The computed forces were finally compared against the results obtained by prior workers.

4. Results

The COMSOL finite element model was run for the three cases mentioned above and the results obtained in each case were carefully examined for comparison with previous solutions. Figure 4 shows an example of the computed flow for an induced velocity of the board at 3m/s with the boundary set as a sliding wall (case 1). The axes of the velocity contours are distances in meters. The model clearly shows the action of the flow around the jet. It should also be noted that when the board is moved in this manner in the model the fluid tends to move with the board, dragged along by the no-slip condition at the boundaries.

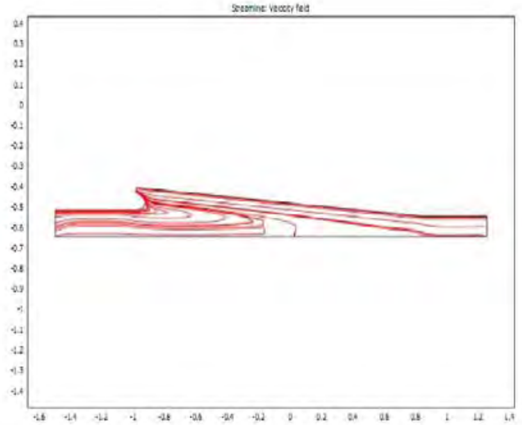

Figure 4: Velocity streamlines: Case 1

The corresponding pressure profile on the underside of the skimboard is shown in figure 5. In addition to the profile graph, the software can export out data points. These points can easily be entered into a spreadsheet or similar program to integrate and find the total lift force on the wetted area. The same line of pressure was used in all of the CFD cases. The pressure found is assumed to be the pressure on the underside of the board. The data was obtained from points in space very close (less than 0.005 m) to the nodes on the board. Therefore, very little error is induced as the program will output a proportional average of the nearest nodes. Thus the great majority of the data comes from the nodes touching the boundary, or adjacent to it.

The computed pressure profile is shown in figure 5. When this function is integrated numerically, the resulting lift force is 584N, which is enough force to provide lift for a 60 kg individual. The sliding wall boundary condition assumes that the wall is moving tangentially, in the direction of the angle of attack. While this may be the case occasionally while riding a skimboard, it does not produce similar results to the theoretical analysis. The small blip near the leading edge of all three cases appears to be an artifice of the model.

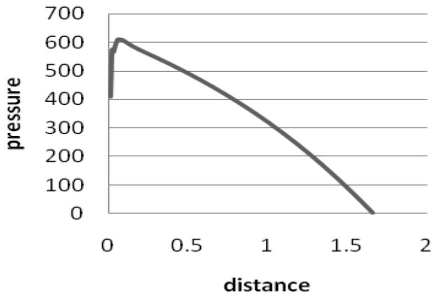

Figure 5: Pressure profile (Pa): Case 1

4.2 Case 2:

The streamlines calculated in this case are shown in figure 6 while the pressure profile can be seen in figure 7. Note that the pressure never drops below zero (atmospheric nominal) using the second method. Also, the center of pressure can be seen to be more towards the center of the board. The lift force using this method almost doubles to 1100 N. This is closer to the values from Tuck and Dixon, but still underestimated by a factor of four. The streamlines near the leading edge match closely those obtained by Tuck and Dixon.

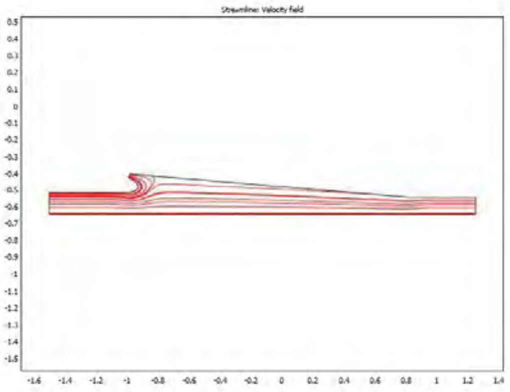

Figure 6: Velocity Streamlines: Case 2

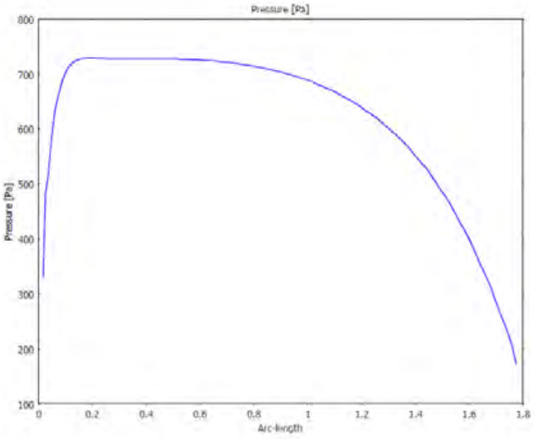

Figure 7: Pressure profile (Pa): Case 2

4.3 Case 3:

Figures 8 and 9 show respectively the resulting streamlines and pressure distribution obtained in this case. The spray jet can be seen to be developed with a divided flow. The pressure curve was found to be more uniform. Integration of the pressure yielded the total lift force to be 2821N, much closer to the expected values from Dixon. Based on the streamline plot, it appears as though the flow is almost sheared off by the dimension of the jet. The streamlines do not match those from the Tuck and Dixon paper. The same model was also run for 4 and 5 m/s. For 4 m/s the lift is 3715N and for 5 m/s it is 4622 N.

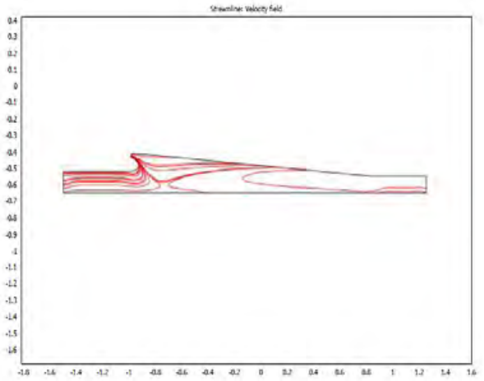

Figure 8: Velocity streamlines: Case 3

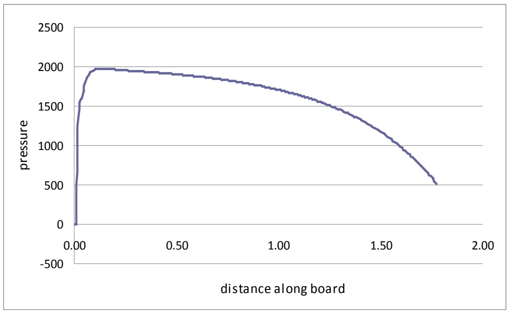

Figure 9: Pressure profile (Pa): Case 3

5. Conclusions

Useful information about the flow underneath skim boards can be obtained from models readily built using the CFD module in COMSOL Multiphysics. Of the three cases considered in this study, only in the third it was found that the lift generated by the computational methods was in good agreement with the lift values noted in previous theoretical studies. Case 1 was found to produce the lowest lift, note quite enough to lift a 70 kg individual. Since a 70 kg person has been witnessed and observed on a board this solution was discounted.

Some knowledge of the particular flow and forces was vital in being able to determine the viability of the data obtained. Without the benefit of the previous studies it may have been difficult to determine which of the boundary conditions produced the best results. Certainly from an aesthetic standpoint case 2 produces the prettiest picture. However, the forces in this case were not the closest match to the theoretical. Observations in the field show that the closest representation to the flow is case 3.

Although some error is introduced in both the computational and empirical methods, for a designer working on a project, the visualization of the flow using commercially available software can be an invaluable tool to assist with rapid prototyping of design concepts.

6. References

1. Tuck, E.O. Dixon, A. *Surf skimmer hydrodynamics*. Journal of Fluid Mechanics. 205, 581-592, 1 February 1989.
2. Sugimoto, Takeshi. *Mechanics of a surf skimmer revisited*. American Journal of Physics. 71,144-149, February 2003.
3. Edge, R D. *The surf skimmer*, American Journal of Physics. 36, 630-631, July 1968.
4. Green, A.E. *The Gliding of a plate on a stream of finite depth*. Proceedings of the Cambridge Philosophical Society. 31, 589-603, 1936.