Proceedings of the 9th World Congress on Mechanical, Chemical, and Material Engineering (MCM'23)
Brunel University, London, United Kingdom –August 06-08, 2023
Paper No. HTFF 179
DOI: 10.11159/htff23.179

Fluid Dynamics Analysis Of A Kayak Slalom Whitewater Course

Sean Ansell[1], Thomas Confrey[1], Lilibeth A. Zambrano M.[1]
[1]South East Technological University
Kilkenny Road, Carlow, Ireland
lilibeth.zambrano@setu.ie; thomas.confrey@setu.ie

Abstract **-** Kayak Slalom is a fast-paced, dynamic sport that takes place on artificial white watercourses. Irish athletes are at a distinct disadvantage as there are currently no training facilities within the country that can match the quality of the artificial whitewater courses used in international competitions, then been forced to travel abroad to these courses across Europe. The cost associated with these training camps is extremely high as athletes must transport not only themselves but also all the necessary equipment. Because of this, athletes are forced to spend less time at these courses for training camps and prior to large competitions.
The main aim of this study is to use computational fluid dynamics (CFD) analysis as a performance enhancement tool for these athletes within training facilities on their doorstep, which can be used to give the athlete an advantage before traveling to these courses; this will result in athletes needing less preparation in the lead up to these events.
Two methods are used in this study: flow visualisation and CFD analysis. CFD analysis was carried out on the London Olympic course to find the optimum path. The course was split into five sections, each of which was analysed individually. A flow visualisation test was carried out on a scaled simplified model of section five as a form of validation of the results gathered from the previous CFD analysis. The optimum path for the whitecourse was found and results for section five were coincident using both methodologies, thus proving that the results gathered from the CFD analysis are reliable enough to provide with a training plan for kayak slalom.

Keywords**:** CFD, fluid visualisation, training plan, watercourse.

1. Introduction

Science and engineering are increasingly applied to improve athletes' performance by pushing the limits of what their bodies and equipment can do. For example, in golf, engineers and scientists use biomechanics to analyse swings and develop both better club materials and improving the athlete's technique. Athletes need qualities like endurance, stamina, and mental acuity, which scientists have made more attainable through fine-tuned methods. This progress is due to the work of scientists and engineers working with athletes and improving their equipment, aiming for performance enhancement, which has led to the progression of certain sports in recent years [1].

Among the engineering tools used to improve sports, there is the use of Computational Fluid Dynamics (CFD), a widely used method of performance enhancement in sports with the main application being within motorsports. CFD was first used commercially in the 1980s in industries like aerospace, turbo machinery, and combustion. In the early 1990s, Formula One teams began using 3D inviscid panel methods to analyse their cars' aerodynamic capabilities [2, 3], allowing them to save money on wind tunnel testing. Since then, CFD software has become more commercially available and affordable, leading to dramatic increases in its capabilities. CFD has also been used in kayak sports to improve kayak design and racing performance [4, 5, 6], also to improve the racing performance by conducting biomechanics studies of the athletes' stroke technique [7] but to the best of the authors' knowledge it has not been used for flow analysis of a kayak course.

Specifically, Kayak Slalom is a fast-paced dynamic sport which is carried out on artificial whitewater courses whose design is now being carried out using computational tools [8]. For the case of kayak slalom, Irish athletes are forced to travel abroad to these courses across Europe, due to the lack of training facilities. Going to these foreign facilities with a previous knowledge of how to tackle the whitewater course would help in not only increasing the athlete's performance but also reduce the time investment needed to master the course. The primary aim of this study is to analyse the fluid flow of an artificial kayak course using CFD and Flow visualisation tests and use these results as a form of performance enhancement for athletes. Validation of the CFD analysis was carried out by conducting flow visualisation techniques, which is still considered an adequate mean of forming accurate predicts of fluid flows. Methods such as wind tunnel testing, laser sheet, smoke and tufts, surface oil and schlieren are all methods still used by NASA while analysing aerodynamic characteristics of their crafts [9]. Flow analysis studies of Whitewater courses have been carried out to validate technical performance objectives and to determine the optimal positioning for hydraulic features within the channels [10].

An optimum path for athletes competing in the London Olympic Lee Valley Whitewater Course was found. The resulting velocity of the water course was visualized, showing places were vortex affected the flow of the course and creating turbulence in the fluid. By considering the velocity magnitude and direction of the flow, the optimum path for the competitor was found. Both results, those coming from the CFD analysis and the flow visualization model match, indicating that the results gathered from the CFD analysis are reliable, from a qualitative standpoint.

2. Methodology

2.1. CFD study

CFD analysis was carried out of the 2012 London Olympic Lee Valley Whitewater Course aiming to find the optimum path for athletes to follow. The course was split into five sections, each of which was analysed individually. A 3D model was created in Solidworks using dimensions extracted from satellite images of the base of the course [11]; however, the walls' height was estimated and considered constant throughout the course, the presence of plastic bollards was neglected as well so an empty course was analysed. To get the outline shape and scale of the course, plan view imagines which were released prior to the 2012 London Olympic games along with satellite images were used, these images were analysed and measured in accordance with the reference scale bar within both sets of images (see Fig 1a & b).

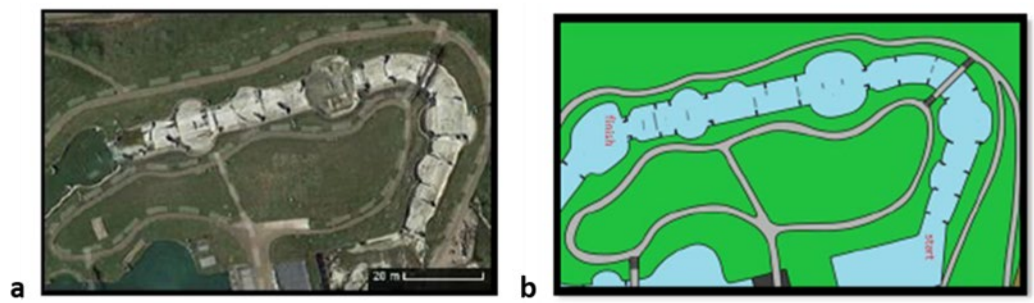

Fig. 1: 2012 London Olympic games: a) Satellite vie, b) Simplified map-view [13, 14].

A fluid volume was performed with SpaceClaim, and brought to Ansys Fluent for the analysis, where a watertight geometry workflow was employed to create a surface mesh of the geometry with no voids; mass-flow-inlet and pressure-outlet were used as boundaries of the model before meshing the entire volume (Fig 2a). The fluid material was taken as liquid water and a mass flow rate of 1300 kg/s, corresponding to the London Olympic Lee Valley Whitewater Course, was assigned. The analysis was carried out in five different sections of the course, for which a qualitative analysis of the velocity was sought (Fig 2b).

2.2. Fluid visualisation

Flow visualisation tests were carried out as a form of validation of the Ansys Fluent results. To carry out this test a scaled down model was built to match the mass flow rate which can be produced by the water pump available while carrying out the procedure. With a flow rate of 13 cubic metres per second produced by the pumps supplying the inlet of the Lee Valley Whitewater course, and only a 2.7e-4 cubic meters per second supplied by the pump being used for this procedure it was clear that the size of the model must be scaled down to a 0.002% size to match. The Model was created from wood and painted white to show up the inks during the flow visualisation tests, then it was coated in a waterproof layer of lacquer in order to match the reaction of the concreate walls of the Lee Valley Whitewater course. Once construction was complete, water was pumped down the channel and left to reach a steady state of flow. Two methods were used within this analysis, first placing of a short piece of string attached to a rod, which was used to determine the direction of the flow at specific points along section 5. The second method used was the addition of coloured dye to the water. This was used to visualise the path of the water while traveling through this section of the course; once the flow stream was steady and constant a small droplet of ink was added to the flow (only to one side of the model, due to its symmetry), upstream of any obstacles. The reaction of the ink was captured by a slow-motion recording from a digital camera which was then analysed and compared to the flow patterns generated within Ansys Fluent.

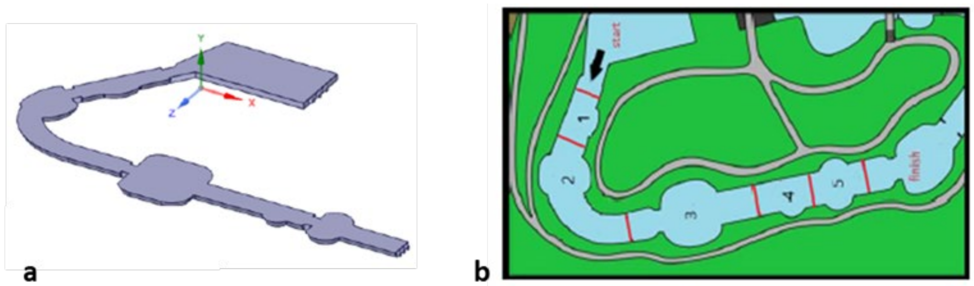

Fig. 2: 2012 London Olympic games: a) Fluid volume geometry, b) Section breakdown.

3. Results and discussions

3.1. CFD study

The resulting velocity of the water course was visualized, showing places were vortex affected the flow of the course and creating turbulence in the fluid. Each section was analysed based on flows speed and direction, combining this with plotting a path which would be achievable for the athlete and determined the final path (See Fig 3a).

The course was analysed in five sections, and the optimum path for each section was determined based on velocity, flow direction, shortest distance, and avoidance of vortices. Once the optimum path for each section was determined, they were combined to create the final optimum path for the entire course. The results and optimum paths are shown with red lines.

Section 1: This path allows the athlete to avoid the unpredictable recirculating currents of the vortices located on the athletes' left which could easily cause the athlete to capsize the kayak, while also maintaining a large enough distance away from the right-hand wall, allowing the athlete to put in maximum effort while located in a high velocity flow.

Section 2: This is the optimum path for the athlete as all unwanted flows are avoided and all strong flows are taken advantage of in order to propel the athlete in the desired direction while expending the least amount of energy possible, thus allowing the athlete to have a more significant input in a less technical section.

Section 3 is a simple section with a narrow area that creates a high-speed zone for the athlete to utilize, followed by a wide opening where vortices occur and should be avoided.

Section 4 has a smaller indentation on the athlete's right side, creating smaller vortices in a clockwise direction. As the flow reaches the bottom corner of the indentation, a strong crossflow from right to left is generated. The chosen path for this section considers athlete's fatigue and avoids putting the athlete in a position that could cause them to capsize. The optimum path involves the athlete traveling close to the left wall to generate speed from left to right and counteract the cross current. This allows the athlete room for error if the cross current is too strong and pushes them from right to left. If the athlete approaches this section without changing direction, they will be pushed up against the left wall, potentially causing them to capsize and be off-line for the next section.

Section 5 shows almost identical characteristics to that of section three, it is relatively simple which consists of a constricted area followed by a large opening. As before, this generates a large increase in velocity down the centre of the course and creates two large vortices located at either side of the large opening. This section is ideal as the last part of the course as the athlete is extremely drained after completing the previous sections of the course. The optimum path for this section is straight down the centre of the course with the main focus of the athlete being remaining straight and not making contact with the slow speed water located as the course constricts again.

By considering the velocity magnitude and direction of the flow, the optimum path for the competitor was estimated (Fig 3b).

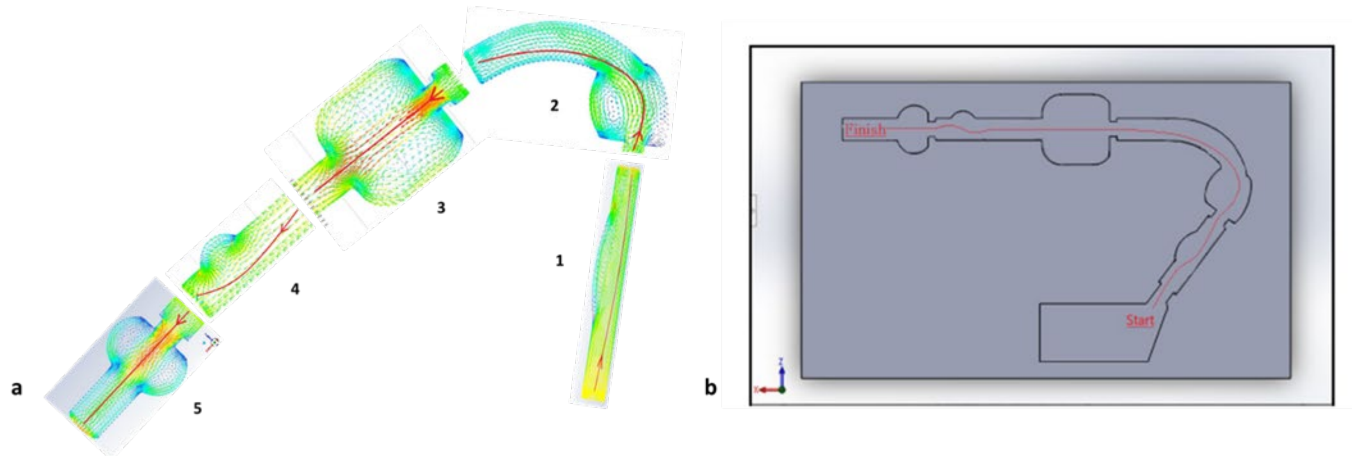

Fig. 3: Results of optimum path: a) Each section of the watercourse, b) Entire watercourse.

3.2. Fluid visualisation

Regarding the first method of flow visualisation (rod positioning), the direction of the piece of string was recorded in each position thus giving the direction of the flow. The first position being located in the centre of the wide opening, the second in the centre of the strong flow following the constriction, these results match that of the results predicted by Ansys Fluent (see Fig 4).

For the first position, the piece of string is being pulled in an upstream direction while also following the outer radius of the opening. This result shows the formation of a vortices rotating in a clockwise direction on the right-hand opening, due to the geometry being symmetrical it is clear that a matching vortex will be formed on the left-hand side but with a counter-clockwise direction. For the second position, as expected, the string follows the flow of the water. This test position was carried purely as a form of validation of this method of test.

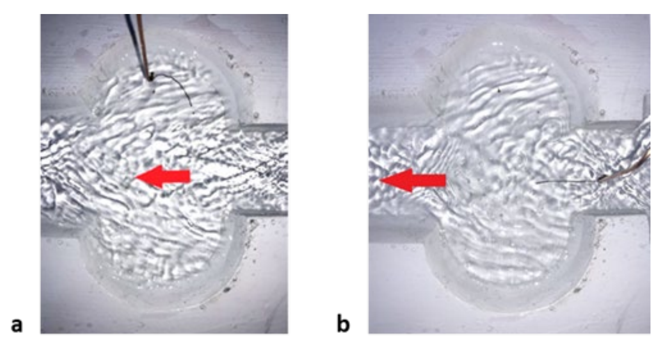

Fig. 4: Flow visualisation results showing shape of the flow in: a) piece of string in watercourse wide opening, b) piece of string in flow constriction (red arrow indicates direction of the flowstream).

For the second method of flow visualisation (colour dye test), the water could be seen to travel in a clockwise direction until one full cycle is complete where the entire vortices field contains the coloured dye. Each cycle of the fluid results in a minor loss of the coloured dye back into the main flow, this cycle will continue until all the coloured dye has been replaced by water traveling from upstream thus pushing out the dye at the downstream corner of the curve (Fig 5a).

A small amount of coloured dye remained in the centre of the vortex; this small amount of dye remained travelling in a clockwise rotation over the space of 60-80 seconds remained within the vortices, until finally getting push back into the main flow by a slight surge in the supply pump. This point is known as the vortex rotation axis line, this is the centre rotation point for all flow traveling around its axis.
Both studies (CFD and visualisation flow) show the formation of large vortices with the central axis located at the centre of the wide opening, also show the flow region divided into two regions once coming in contact with the downstream corner of the curve, the first (larger) region makes contact with the wall and begins to form the vortices, the second region simply continues downstream with a slight decrease in velocity in comparison to the main flow; this consistency indicates that the results gathered from the CFD analysis are reliable, from a qualitative standpoint (Fig 5b).

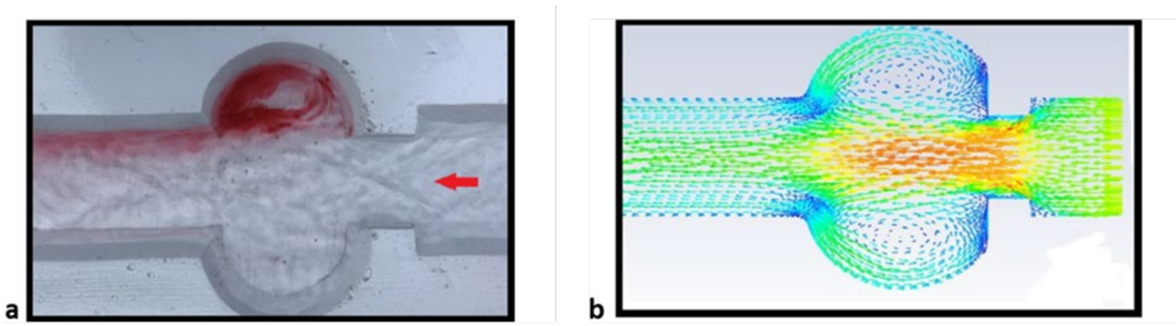

Fig. 5: Observation of similarities in vortices development in section 5 of watercourse: a) flow visualisation model, b) CFD analysis.

The main limitation of this study, regarding the CFD analysis, was related to the making of the geometry, not having precise information of the full geometry, mass flow rate used and the materials used in the whitecourse, which could not be obtained in time due to the onset of the COVID-19 pandemic, which restricted communications worldwide and forced to the simplification of the CFD model. For a more precise analysis, real dimensions, and features of the whitecourse such as plastic bollards -as in [12, 13][3 and 4]- should be included in the CFD analysis. This also influenced the construction of the scaled model for flow visualisation, as there was a disruption of the supply chain of products as well.

4. Conclusion

Performing the CFD analysis of the London Olympic Lee valley Whitewater course allowed to establish that the optimum path depends on both flow velocity and direction. To decide about the optimum path, both factors had to be considered.

Both methods employed here, CFD and flow visualisation, had key benefits. The CFD analysis is an excellent qualitative and quantitative representation of the results, which give a greater understanding of flow patterns, directions and velocities in comparison to that of purely mathematical methods. As CFD only produces results which are based on the user's inputs, flow visualisation tests are a great method of validation of these results. The main drawback associated with flow visualisation testing is the need to physically construct models of geometry being tested, unlike within CFD where it is very simply to generate new models or even modify existing models in order to carry out a new analysis.

For a more precise analysis, real dimensions, and features of the whitecourse such as plastic bollards should be included in the CFD analysis.

References

[1] D. James, “The Ethics of Using Engineering to Enhance Athletic Performance,” *Procedia Engineering ,* no. 2, pp. 3405 - 3410, 2010.

[2] Simescale, “SIMESCALE - What is CFD | Computational Fluid Dynamics?,” 2023. [Online]. Available: https://www.simscale.com. [Accessed 9 January 2021].

[3] R. K. Hanna, “CFD in Sport - a Retrospective; 1992 - 2012,” *Procedia Engineering,* vol. 34, pp. 622-627, 2012.

[4] F. L. d. R. Barros, *Study of Kayak Hull Design on Calm Water Resistance. Master Thesis,* Porto: Faculdade de Engenharia da Universidade do Porto, 2015.

[5] A. J. S. D. A. M. A. I. R. Vishveshwar R. Mantha, “Numerical Simulation of Two-Phase Flow Around Flatwater Competition Kayak Design-Evolution Models,” *Journal of Applied Biomechanics,* vol. 29, pp. 270-278, 2013.

[6] R. D. P. Q. J.-P. C. C. R. Alban Leroyer, “Toward optimization using unsteady CFD simulation around kayak hull,” *Procedia Engineering,* vol. 147, pp. 293-298, 2016.

[7] C. P. C. R. Harrison SM, “Dynamic simulation of flat water kayaking using a coupled biomechanical-smoothed particle hydrodynamics model,” *Human Movement Science,* vol. 64, pp. 252-273, 2019.

[8] S. NUMAIRO, “NUMAIRO,” 2023. [Online]. Available: https://numairo.fr/en/portfolio/kayak-en/. [Accessed 12 12 2022].

[9] N. A. a. S. A. -. NASA, “NASA - Flow Visualization,” 2023. [Online]. Available: https://www.grc.nasa.gov/www/k-12/airplane/tunvis.html. [Accessed 9 January 2021].

[10] J. s. V. S. O. J. Pollert jun, “Designing White Water Slalom Courses for Olympic Games,” *The International Archives of the Photogrammetry, Remote Sensing and Spatial Information Sciences,* pp. 177-184, Volume XLVI-5/W1-2022 2022.

[11] I. ICM, “DocPlayer,” 2023. [Online]. Available: https://docplayer.net/15178143-Designing-a-basic-white-water-canoe-kayak-course-using-infoworks-icm.html. [Accessed 12 January 2021].

[12] J. R. R. Jose A. Vasquez, “Testing river2d and flow-3d for sudden dam-break flow simulations,” in *Canadian Dam Association 2009 Annual Conference - Association Canadienne des Barrages*, Whistler, BC, 2009.

[13] H. G. S. E. H. O.-C. G. V. Selahattin Kocaman, “Experimental and Numerical Analysis of a Dam-Break Flow through Different Contraction Geometries of the Channel,” *Water,* vol. 12, no. 4, p. 1124, 2020.