

The 2014 conference of the International Sports Engineering Association

A CFD analysis of flow around a disc

R.A. Lukes^a, J.H. Hart^a, J. Potts^b, S.J. Haake^{a*}

^a*Centre for Sports Engineering Research, Sheffield Hallam University, Sheffield, S1 1WB, UK*

^b*Department of Engineering and Mathematics, Sheffield Hallam University, S1 1WB UK*

Abstract

Discus, disc golf and ultimate Frisbee are all sports that utilise rotating disc projectiles. The aim of this study was to use computational fluid dynamics (CFD) to examine the flow over a disc and in doing so determine which CFD model is most suited to examining the aerodynamics of sports disc projectiles. All of the CFD analysis was carried out using ANSYS-Fluent V6.3. The study initially compared experimental and CFD data of flow around a simple non-rotating parametric disc, with a thickness to diameter ratio of 0.1. Aerodynamic coefficients and surface flow visualisations were compared. Three models within ANSYS-Fluent gave steady converged solutions; standard $k-\epsilon$, realizable $k-\epsilon$ and standard $k-\omega$. The standard $k-\omega$ model gave unrealistic aerodynamic coefficients and was discounted. The two $k-\epsilon$ models were used for the second stage of the study, simulating the flow around a disc golf disc, called a Floater. The geometry of the disc was captured using a 3D non-contact laser scanner. Aerodynamic coefficients and surface flow visualisations showed good agreement between the two $k-\epsilon$ models. The standard $k-\epsilon$ model was deemed the most suitable for a study of flow around a sports disc, mostly due to its more accurate simulations of the parametric disc. The results were used to further examine the flow, looking at separated flow on the disc surface with increasing angle of attack and the structure of the wake. The study identified which CFD model is most suited to the simulation of flow over a sports disc and in doing so provided a platform for using CFD for future disc aerodynamics studies.

© 2014 Published by Elsevier Ltd. Open access under [CC BY-NC-ND license](https://creativecommons.org/licenses/by-nc-nd/4.0/).

Selection and peer-review under responsibility of the Centre for Sports Engineering Research, Sheffield Hallam University

Keywords. Sports disc; Frisbee; Computational fluid dynamics; Aerodynamics.

* Corresponding author. Tel.: +44 (0) 1142252255; fax: +44 (0) 1142254356.

E-mail address: john.hart@shu.ac.uk

1. Introduction

Rotating disc projectiles are utilised in a number of sports including, discus, disc golf and ultimate Frisbee. These projectiles rely on their aerodynamic characteristics and an applied spin rate to achieve a stabilised flight path. A variety of disc profiles are used in these sporting applications, particularly so in disc golf, where during one game the competitor can use multiple discs in three distinct styles. Goff (2013) provided interesting insight into the broader aerodynamics of sports projectiles, including a section on discus and discs. Potts (2006) used experimental data to develop a theory of flight for spinning discs. Kamaruddin (2011) furthered this work by exploring the influence of geometrical variations on the disc's aerodynamic characteristics. This work aims to use computational fluid dynamics (CFD) software ANSYS-Fluent, alongside experimental data, to examine the flow over a disc.

This paper presents a comparison between experimental and CFD data for two non-rotating discs. The comparison was used to determine the most suitable ANSYS-Fluent CFD model for the analysis of disc aerodynamics. The first of the two discs is a simple parametric geometry, with the thickness to diameter ratio (t/d) of 0.1. The second is a disc golf disc called the Floater, manufactured by Dynamic Discs. Despite all sporting disc projectiles stabilising in flight due to an imparted spin rate, previous experimental studies have first examined non-rotating discs before considering the rotating case. This study will also examine non-rotating discs with the broader aim of continuing to the rotating case in future.

2. Method

2.1. Geometry acquisition

The geometry of the simple parametric disc was created within pre-processing software Gambit. The thickness of the disc, t , was 0.02 m and the diameter, d , 0.2 m, giving a t/d of 0.1. The geometry of the Floater disc was acquired from a physical disc using a 3-dimensional non-contact laser scanner. The cross-section of the Floater disc is shown in Fig. 1.



Fig. 1. Floater disc cross-section.

2.2. Mesh generation

The mesh generation process was identical for both the parametric and Floater disc. The pre-processing software, Gambit, was used to create a flow domain and apply the initial boundary mesh. The disc was placed 1.5 m from the inlet of a domain, 1 m x 1.4 m x 5 m, see Fig. 2. An unstructured boundary mesh was applied to the disc and domain with a 0.001 m and 0.08 m node spacing, respectively.

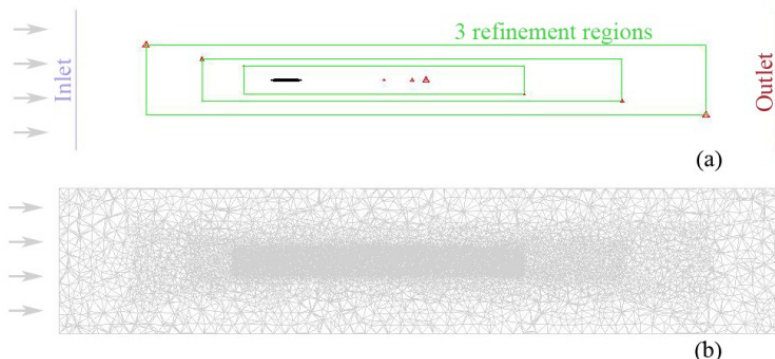


Fig. 2. (a) The flow domain showing local refinement regions; (b) A cross section of the completed mesh.

The boundary mesh was exported to further mesh generation software, ANSYS-TGrid. In TGrid 10 layers of prisms were grown from the disc surface, with an initial height that ensured the flow in the boundary layer was modelled as accurately as possible. Three regions of local refinement were defined to ensure adequate mesh resolution around the mesh and in its wake. These three regions are shown in Fig. 2.

The above mesh generation process was used to create ten meshes, with resolutions ranging from 250,000 cells to 6,200,000 cells. Running simulations with these 10 meshes showed that a mesh with around 3,500,000 cells gave a grid-independent solution. This mesh was chosen for use in the study.

2.3. Simulation

Simulations were run on ANSYS-Fluent V6.3. Inlet velocity was 27.5 ms^{-1} and turbulence was characterized by turbulence intensity (0.5%) and hydraulic diameter (0.1 m), to correspond as closely as possible with the experimental conditions. Simulations were run with a variety of turbulence models and near wall treatments. Second-order discretization was used. Convergence was monitored by examining the residuals as well as force on the disc and flow velocity at 3 downstream points. The simulations were deemed converged when the residuals were sufficiently reduced and stable and the force and velocity monitors were stable.

3. Results and discussion

3.1. Parametric disc

A variety of turbulence models were studied at a range of angles of attack, 0° , 5° , 10° and 15° . Not all of the turbulence models used produced converged simulations. The standard k- ϵ , realizable k- ϵ and standard k- ω models were robust, resulting in consistent and converged simulations. The SST k- ω model and the Reynolds Stress Model would not converge satisfactorily, and were disregarded from the study. The Reynolds Stress Model is very sensitive to mesh construction and is best suited to fully structured meshes, therefore its non-convergence with this unstructured mesh is understandable. The problems encountered with the SST k- ω model would suggest it was predicting strong separation and unsteady flow. The RNG k- ϵ model produced unrealistic flow phenomena. Like the Reynolds Stress Model, RNG k- ϵ is temperamental and highly reliant on mesh construction. It is believed that the RNG k- ϵ model did not resolve satisfactorily due to a mesh issue and was disregarded from the study.

In terms of the near-wall treatment, it has already been stated that the mesh was constructed to be used with a wall functions approach. It was found that the standard wall functions model gave identical results to the non-equilibrium wall functions model, so the standard wall functions model was used.

The three turbulence models that provided data that could be confidently compared to the experimental data were standard k- ϵ , realizable k- ϵ and standard k- ω . Aerodynamic coefficients for lift, drag and pitching moment (C_l , C_d , and C_m , respectively) have been plotted alongside the experimental data of Kamaruddin (2011), see Fig. 3. In this figure experimental results are shown by the lines and the CFD results by the individual data points.

Further comparison between experiment and CFD was also made for the 5° angle of attack case by looking at the flow on the disc surface, Fig. 4. In the experimental case, Kamaruddin (2011), oil flow on the disc's surface was examined. This image shows a region of separated flow almost immediately downstream of the leading edge, and reattachment at approximately 0.1 chord length. Similar images were produced from the CFD results by examining pressure coefficient on the disc's surface. In this case the exact values of pressure coefficient are not significant, instead it is important to note that separated flow is represented by the solid dark blue regions.

Examining initially the aerodynamic coefficients, it is apparent that the two k- ϵ models produce largely similar results for C_l and C_d . In terms of C_l , the three turbulence models are in agreement up to and including 10° . Both the k- ϵ models and the experimental data show a modest drop in C_l between 10° and 15° , whereas the k- ω model predicts an increase. The k- ω model also predicts a C_d profile that does not correspond to the experimental or other CFD data, particularly so at 15° . It has been previously shown that the k- ω model tends to over predict drag forces especially for bluff bodies, as flow separation becomes more dominant (Lukes *et al.*, 2005). The reason for this is suggested to be due to an over prediction of stagnation pressure and wake size, both of which have been

previously shown to be an issue with the $k-\omega$ model formulation within ANSYS-Fluent (Lukes, 2006). The $k-\omega$ model is in agreement with the standard $k-\epsilon$ model as far as C_m is concerned, both models over predicting at 10° and 15° . Contrastingly, the realizable $k-\epsilon$ model under-predicts C_m at these angles.

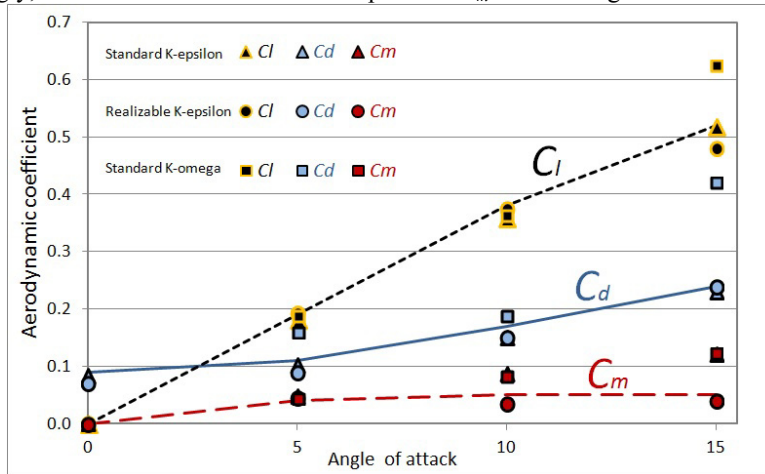


Fig. 3. Aerodynamic coefficients of the parametric disc.

Scrutinising the surface flow visualisation in Fig. 4 builds a further understanding of each model's performance. To re-iterate, the experimental image shows the flow detaching from the surface almost immediately downstream of the leading edge, and reattaching shortly after, creating a crescent of detached flow. In the CFD images, detached flow is shown by the dark blue areas. The most striking result is the realizable $k-\epsilon$ case which has little to no region of detached flow, therefore not corresponding at all to the experimental result. The standard $k-\epsilon$ model provides the best representation of detached flow, although not as large an area as the experiment suggests, it is the best of the three CFD results. The inability of the realizable $k-\epsilon$ model to correctly predict flow separation ties in with the under-prediction of C_m at higher angles of attack.

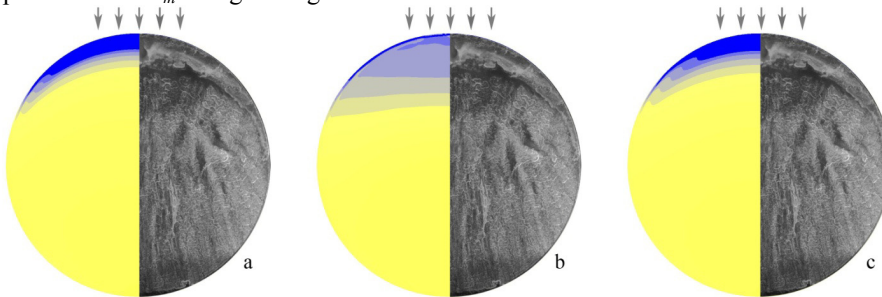


Fig. 4. 5° disc surface flow visualisation (a) Standard $k-\epsilon$; (b) Realizable $k-\epsilon$; (c) Standard $k-\omega$.

Despite the $k-\omega$ model giving a reasonable representation of separated flow, the predictions of C_l and C_d were significantly different to the experimental results. For this reason the $k-\omega$ model was deemed to be an unsuitable choice for modelling the flow around a disc. Both $k-\epsilon$ models provided similar and reasonably accurate predictions of C_l and C_d , although neither produced accurate C_m values for higher angles of attack. Even though there is a discrepancy between the two models' surface flow visualisation, both will be used for the study of the Floater disc.

3.2. Floater disc

Simulations were run for both $k-\epsilon$ turbulence models and aerodynamic coefficients were plotted against angle of attack, Fig. 5. The results show good agreement between the two $k-\epsilon$ models. There is no experimental data available for direct comparison, but the results do seem characteristic of a more efficient wing. The results show greater lift generation at all angles of attack, and no decrease in lift at 15° .

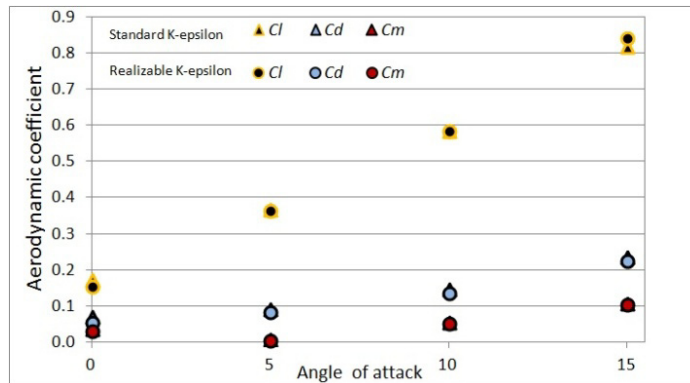


Fig. 5. Aerodynamic coefficients of the Floater disc.

As with the parametric disc, regions of separated flow were examined on the surface of the disc. The realizable k- ϵ model did predict regions of separated flow on the upper surface of the Floater disc. These regions were again smaller than those predicted by the standard k- ϵ model, although the difference between the two models was not as significant as with the parametric disc.

The lack of a direct experimental comparison for the Floater disc means that the choice of the most suitable turbulence model for a CFD study of disc aerodynamics is mostly governed by the results for the parametric disc. The conclusion is therefore that the k- ϵ model is most suitable for this and future CFD studies.

In this instance the standard k- ϵ model can be used to further examine the Floater disc. Fig. 6 shows pressure coefficient on the surface of the disc for the 5° , 10° and 15° case. In this image, as before, the dark blue areas show regions of separated flow. These images show the significant growth in the size of the separated flow region, with increasing angle of attack. The accurate modelling of the separated flow in the parametric disc simulations gives confidence that these images are a reasonable representation of flow structure.

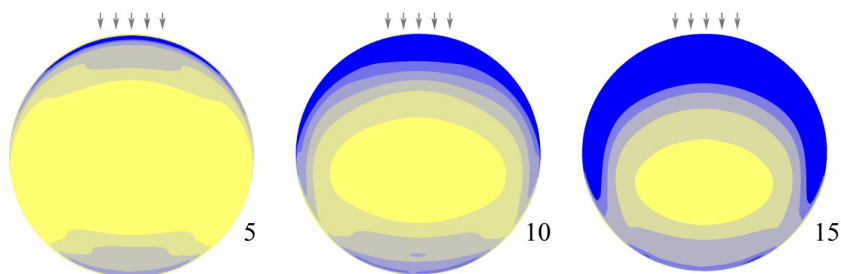


Fig. 6. Pressure coefficient on the Floater disc at 5° , 10° and 15° angle of attack.

Potts (2005) used extensive wind tunnel flow visualisations to develop an understanding of flow structure around a Frisbee disc. In Fig. 7 two schematics from this work are compared to the current work for the 10° case. The CFD images are both coloured by velocity magnitude, with the image on the left showing pathlines and on the right velocity vectors on a vertical plane through the centre of the disc. The comparison shows similar generic flow structure. The Floater disc produces a less defined wake and less recirculation in the cavity due to its streamlined profile. The current study can be used to further examine the structure and diffusion of the wake. Fig. 8 examines velocity vectors coloured by velocity magnitude for the 15° case on 5 downstream planes. The image in the top left of Fig. 8 illustrates the location of the planes, and the perspective of each individual image is viewed from the inlet looking towards the outlet. The vectors show the initially strong presence of tip vortices immediately aft of the disc and their movement down and reduction in magnitude as the flow progresses.

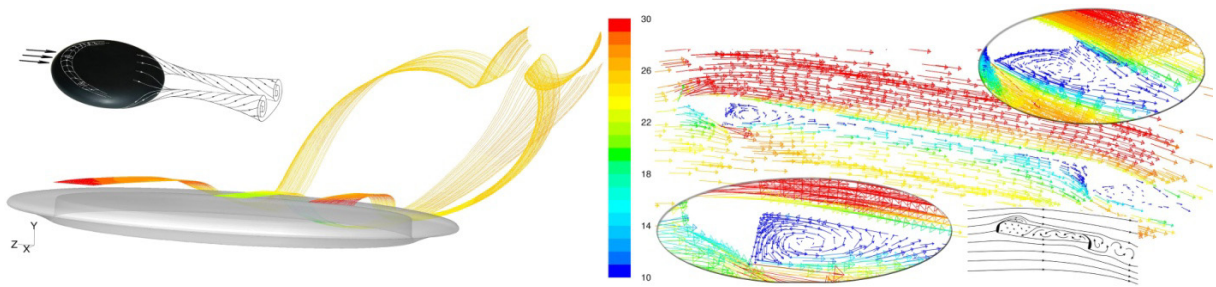


Fig. 7. Flow visualisations for 10° disc with flow topology from Potts (2005).

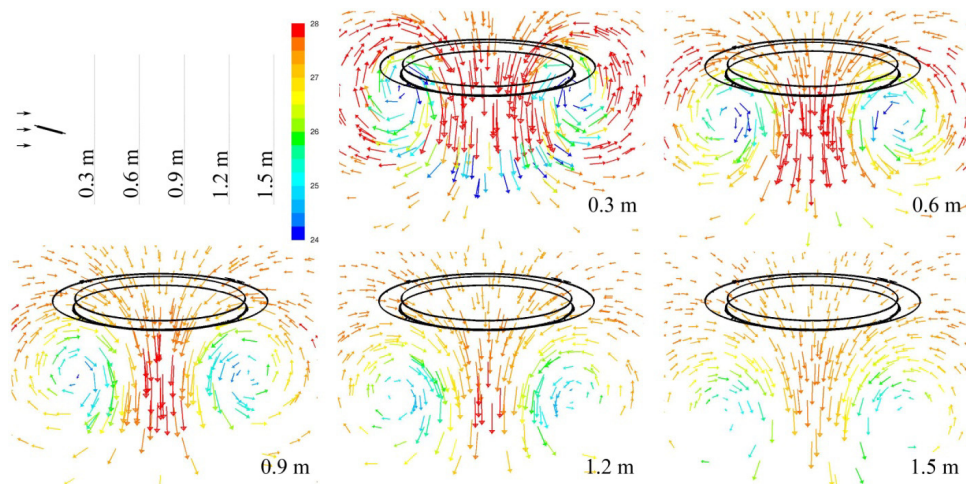


Fig. 8. Velocity vectors on vertical planes downstream of the 15° disc; the numbers represent the horizontal distance from the disc centre.

4. Conclusions and future work

The study shows that CFD can be used as a tool for examining flow over sports disc projectiles. It has also shown the importance of comparing experimental data, theoretical understanding and CFD results in order to ensure the most suitable CFD model is chosen. It was found that the standard $k-\epsilon$ model, with standard wall functions, gave robust solutions, a good representation of surface flow structure and reasonable agreement with experimental aerodynamic coefficients. Therefore the $k-\epsilon$ model is the most suitable for examining the flow over a disc.

This validation of the CFD model will be used as the foundation for future CFD disc flow studies. Beyond examining the current simulations in greater detail further work will investigate; a broader range of angles of attack, the effect of spin and the impact of geometrical modifications.

References

- Goff, J.E., 2013. A review of recent research into aerodynamics of sport projectiles. *Sports Engineering* 16:137-154
- Kamaruddin, N., 2011. Dynamics and Performance of Flying Discs. PhD thesis. The University of Manchester, UK.
- Lukes, R.A., Hart, J.H., Chin, S.B. & Haake, S.J., 2005. Validation of computational fluid dynamics applied to aerodynamic flows in sport with specific application to cycling. In: *Proceedings of the Asia-Pacific Congress on Sports Technology*, Tokyo Institute of Technology, Japan, pp. 286-291.
- Lukes, R.A., 2006. Improving Track Cycling Performance Using Computational Fluid Dynamics. PhD thesis. The University of Sheffield, UK.
- Potts, J., 2005. Disc-wing Aerodynamics. PhD thesis. The University of Manchester, UK.