OPTIMIZED SAMPLE SHAPE FOR ROTATING CAGE FLOW-ACCELERATED CORROSION EXPERIMENTS WITH LEAD: A CFD STUDY

Abdelmagid E. A. Ali, Andrea Cioncolini, Dominique Laurence and Hector Iacovides
Department of Mechanical, Aerospace and Civil Engineering
University of Manchester
George Begg Building, Sackville Street, M1 3BB Manchester, United Kingdom
abdelmagid.emadabdelmagidali@manchester.ac.uk; andrea.cioncolini@manchester.ac.uk;
dominique.laurence@manchester.ac.uk; h.iacovides@manchester.ac.uk

ABSTRACT

In this work, the rotating cage (RC) setup, a well-known flow-accelerated corrosion testing system, was optimized for lead-cooled nuclear reactor applications using Computational Fluid Dynamics (CFD) simulations. The experimental setup is comprised of a fixed cylindrical vessel filled with a corrosive fluid and a rotating cage, which features testing samples manufactured from the material of interest. During operation, the relative motion between the samples and the fluid induces friction on the samples’ surfaces, thereby reproducing conditions found in real applications (e.g. pipes, heat exchangers and pumps). The samples normally used in the RC setup have a blunt shape with rectangular cross section. CFD analysis shows that the complex largely detached flow around this blunt shape causes a high form drag force and the power required to spin the cage becomes prohibitively large for cases with lead as a working fluid at sample speeds of 20 m/s ($Re = 2.5 \times 10^7$). A more streamlined shape of the sample, with a simple to manufacture semi-circular nose and triangular trailing edge, reduces the required power by 56% and also suppresses the unsteady massive flow separations from the corners of the sample. This makes the wall shear stresses more uniform and simplifies the interpretation of corrosion measurements.

KEYWORDS

Rotating cage, Flow-accelerated corrosion, Lead, Liquid metal, CFD

1. INTRODUCTION

Liquid-metal-cooled nuclear reactors are nuclear systems where the fluid that cools the reactor core is a liquid metal, such as sodium, lead-bismuth eutectic alloy (45% lead and 55% bismuth), or pure lead. The excellent heat transfer properties and high boiling temperatures of liquid metals, and the possibility to operate liquid metal reactors at ambient pressure and very high temperature, are the main advantages of liquid-metal-cooled reactors over water-cooled reactor systems. These advantages result in smaller, safer, and more efficient reactor designs, particularly suited for small-modular construction. Moreover, the absence of a moderator in these reactor designs yields a fast neutron spectrum in the reactor core, which in turn allows operating liquid-metal-cooled reactors as fast breeders. The associated closed fuel cycle is particularly effective at exploiting the energy value of uranium, thereby increasing the sustainability of these systems.
The growing interest for lead-cooled reactors is motivated by the advantages that lead presents in comparison with sodium and lead-bismuth. As compared to sodium, lead and lead-bismuth have significantly higher boiling temperature, which improves the safety of the system by reducing the risk of void formation in the core in case of overheating. Unlike sodium, the chemical reactivity of lead and lead-bismuth with water and air does not require the use of a sealed coolant system, thereby simplifying the system design and reducing the plant footprint. Unfortunately, bismuth is expensive and, as a consequence of neutron irradiation, it produces polonium, which is volatile, radioactive and highly toxic. Pure lead, therefore, combines the advantages of a higher boiling temperature and a milder reactivity with water and air without the burden of polonium production. On the other hand, compared to sodium, lead-bismuth and lead have much higher densities (which complicates the seismic design of pool-type reactor systems) and have a more corrosive nature.

The major design challenge of lead-cooled reactors is the development of materials and/or corrosion-prevention strategies for all reactor components immersed in lead, which would allow operating temperatures to exceed 500°C. One of the corrosion-prevention strategies currently used is environmental control (the chemical modification of the aggressive lead environment to decrease the corrosion rate). As a representative example, the reference materials selected for the reactor vessel, the reactor internals, the pumps and the steam generators of the ALFRED reactor (see [1]) include alumina-forming austenitic (AFA) stainless steels (see [2]). These contain chromium and aluminium, which form chromium oxide (Cr$_2$O$_3$) and aluminium oxide (Al$_2$O$_3$) protective layers next to the metal surface that prevent further corrosion. In addition, the lead environment in the ALFRED reactor will be conditioned with active oxygen control, a technique already successfully implemented in lead-bismuth-cooled reactors developed for submarine propulsion (see [3]). A more extensive account of reference materials and corrosion mechanisms at play in liquid-metal-cooled reactors can be found in [4].

Clearly, corrosion testing plays a central role in the development of new materials and in the assessment of corrosion-prevention strategies for lead-cooled reactors. The present study, in particular, is focused on the rotating cage setup, a simple laboratory methodology that is extensively used for flow-accelerated corrosion and erosion-corrosion testing (see [5]; [6]). The attention is here restricted to flow-accelerated corrosion, also referred to as particle-free erosion-corrosion, which is the mode of corrosion observed when the corrosive fluid is not loaded with suspended particles and is in motion with respect to the exposed metal surface: this is the case for most of the reactor components exposed to lead. The rotating cage setup comprises a fixed cylindrical vessel, a rotating cage where the testing samples are connected, and an electric motor connected to the cage via a shaft. Once the vessel is filled with the liquid of interest the cage is set into rotation, thereby creating a relative motion between the samples and the liquid. Whilst not directly simulating pipe flow, flow-corrosion data generated with the rotating cage setup are considered indicative of actual pipe/channel flow conditions, provided that the wall shear stress acting on the rotating samples matches that on the pipe/channel wall (see [7]; [8]). The rotating cage setup is a standardized methodology for investigating the corrosion of metals under pipe flow conditions: applicable ASTM (American Society for Testing and Materials) standards include (see [9]; [10]; [11]) ASTM G170-06 (Standard guide for evaluating and qualifying oilfield and refinery corrosion inhibitors in the laboratory), ASTM G184-06 (Standard practice for evaluating and qualifying oilfield and refinery corrosion inhibitors using rotating cage), and ASTM G202-12 (Standard test method for using atmospheric pressure rotating cage). Despite having rather wide applicability in terms of fluids and operating conditions, these standards do not cover flow-corrosion testing in high-temperature lead.

Using computational fluid dynamics (CFD), we show that the extrapolation of these standards to high-temperature lead is not feasible. More specifically, in the standard rotating cage setup the testing samples are plates with rectangular cross-section. This results in unsteady flow around the testing samples, with pronounced flow separation and recirculation that, combined with the high density of lead, yield a substantial drag force and associated flow resistance, resulting in unfeasibly large power input
requirements. Using CFD, we show that this problem can be circumvented by streamlining the testing samples, i.e. by making the cross-section of the testing samples more aerodynamic. To the best of our knowledge, this type of optimization of the rotating cage setup has not been previously considered. Perhaps even more important, the flow around the streamlined samples is steady and attached, and therefore more representative of actual pipe/channel flow conditions.

The rest of this paper is organized as follows: the preliminary design (based on the available standards) of the rotating cage setup for testing in high temperature lead is presented in Section 2. The CFD methodology is described in Section 3, whilst the limitations of the preliminary rotating cage design are discussed in Section 4. The improved design with streamlined testing samples is presented and discussed in Section 5.

2. PRELIMINARY DESIGN OF THE ROTATING CAGE SETUP

The simplified rotating cage geometry used for the CFD simulations is shown in Fig. 1 (CAD provided by Prof. Jung-Kun Lee of the University of Pittsburgh, USA, where the facility will be realized and operated). It consists of two rotating disks to which four samples are attached using 8 alumina isolators. The disks and samples are spun using a central shaft. Four baffles are fixed to the inner wall of the vessel. The dimensions of the geometry are provided in Appendix A.

The rotation-induced free-surface shape, discussed in [12], [5] and [13], is not represented because: 1) as a slip surface it does not significantly affect the torque that is to be predicted; 2) the large baffles herein reduce bulk fluid rotation; 3) our focus is on optimizing the coupon shape, whose hydrodynamics is isolated from the free surface by the upper rotating disk.

The rotating cage geometry studied here is different from the ASTM-standardized geometry. Even though the sample dimensions are approximately the same, the latter includes 8 samples (i.e. double the number of samples in the present geometry). In addition, the diameter of the disks in the ASTM geometry is 80 mm, which is 2.7 times smaller than the disk diameter in this work. Furthermore, the ASTM geometry features a hole in each disk that is radially adjacent to one of the samples to supposedly increase turbulence levels. This hole was ignored here as it can complicate the flow and can introduce differences in the flow behaviour between the regions surrounding the different samples.

A bigger tank facilitates maintaining a more uniform temperature at high temperatures, as required for studying molten lead. In addition, a bigger cage allows using the facility for other types of tests beside erosion-corrosion, such as developing ultrasound imaging for under-lead viewing.

Using a bigger tank and a smaller number of samples (for a given sample size) also results in a bigger spacing between the samples and thus a better flow recovery between the successive passages. If the flow does not recover enough between successive passages, then the samples are exposed to a disturbed flow, which would be facility-dependent and this would compromise the very objective of the test (to generate data that can be extrapolated to general flow conditions). Moreover, keeping the number of samples low has the additional advantage of minimizing the torque and therefore the power rating of the electric motor.
Figure 1. Simplified geometry for CFD simulations. The fine details that can complicate the study (e.g., the baffles' holders) were removed.

3. CFD METHODOLOGY AND COMPUTATIONAL SETUP

All the simulations of this work were carried out using the commercial unstructured finite volume code Star-CCM+ v.14.06. The complexity of the geometry studied here necessitated the unstructured polyhedral grid shown in Fig. 2. In Fig. 2 (b), the angle convention adopted here for the different angular positions of the samples is shown. Note that the samples are taken to rotate in an anticlockwise manner here.

The inner rotating domain containing the shaft, disks and samples and meshed with a fine resolution is delimited by the straight lines (Fig. 2 (a)) and circles (Fig. 2 (b)) along which it slides in solid body rotation while the outer domain for baffles and vessel walls is fixed. Rectangular prism layers are inserted in the near-wall regions to better capture the wall-normal sharp gradients. The cell count is about 0.64 million cells in the stationary region and 5.6 million cells in the rotating region. Grid independence was confirmed by running a simulation using a coarser grid with a 4.5 million total cell count, which returned very similar power rating values to the ones predicted with the finer grid.

The grid generated for the improved design (see section 5) is similar to the finer grid used with the original design except that the prism layers surrounding the samples, the disks and the alumina isolators are given a reduced near-wall grid spacing of $3.3 \times 10^{-7} m$ (as opposed to $5.0 \times 10^{-7} m$ in the original design) to account for the thinner boundary layers present in the improved streamlined design. The total cell count for this grid was 8.1 million cells.

The flow was simulated for a temperature of 500°C, which corresponds to a density of 10452 kg/m³ and a kinematic viscosity of $v = 1.7365 \times 10^{-7} m^2/s$ (see [14]). Disk peripheral velocity values of 5, 10, 15 and 20 m/s were tested. The 20 m/s maximum velocity corresponds to an angular speed of $\omega = 185.2$ radian/s (i.e. 1768 rpm). This speed yields a ‘global’ Reynolds number $Re_\omega = \frac{\omega RD}{v} = 2.5 \times 10^7$ (R and D are the radius and diameter of the disks, respectively) and a ‘sample’ Reynolds number $Re_{Sample} = \frac{\omega R_{Sample}}{v} \left ( L \right )$ (L is the length of each sample and $R_{Sample}$ is the radius measured from the centre of the shaft to the centre of the samples) of $3.9 \times 10^6$.

---

1 The difference between the total cell counts in the two designs is mainly because of the much finer grid generated by StarCCM+ near the leading edge in the improved design.
Since the flow is turbulent for all the rotational speeds studied here, the simulations were URANS (unsteady Reynolds-averaged Navier stokes) simulations for which the $k - \omega$ SST eddy viscosity model of [15] was chosen. The ‘all $y^+$’ near-wall treatment, which is suitable for all $y^+$ values at the near-wall node, was employed here.

With regards to the temporal and spatial discretization of the governing equations, the former was done using the 1st order Euler implicit discretization scheme. The time step was chosen such that it corresponds to the time the sample takes to rotate by 1° in the case with a 20 m/s peripheral disk velocity. The spatial discretization was achieved using the second order upwind scheme. Before extracting any results, the simulations were allowed to run for a sufficient time for the torque to exhibit a regular periodic behaviour.

Figure 2. The computational grid used to simulate the preliminary design. (a) a vertical plane that crosses one of the baffles as well as two samples. (b) A horizontal plane that passes through the mid-span of the samples.
4. CFD RESULTS FOR THE PRELIMINARY DESIGN

In this section, the results obtained for the preliminary design in which the samples have a rectangular cross section are analyzed by discussing the power requirements and the flow physics. In particular, the flow around the samples is explained as it is important not only from a power requirement perspective but also due to its direct effect on the erosion-corrosion phenomenon taking place on the surfaces of the samples.

4.1. Predictions of the Power Requirements

Fig. 3 shows the predictions of the power ratings for the four simulated preliminary design cases plotted against the peripheral disk velocity. The required power input increases significantly with the disk velocity and becomes excessive in the 20 m/s case. To explain this behaviour, the contribution of the different moving parts to the total torque at the point in time when one of the samples is at 0° (see Fig. 2 (b)) is shown in Fig. 4. It can be observed that most of the torque is caused by the drag force acting on the samples, which is mainly due to pressure differences on the samples’ surfaces (i.e. form drag). This can be attributed to the blunt profile of the samples, which also causes massive separations of the flow from the surfaces of the samples. The flow behaviour around the samples is explained in further detail in the following subsection.

Figure 3. Predictions of the maximum power input values required to achieve different disk peripheral velocities when the preliminary design is used.

Figure 4. A pie chart showing the contributions of the different parts of the preliminary design to the total torque at the angle of 0° (see Fig. 2 (b)).
### 4.2. Insight into the Flow Physics

To gain a better understanding of the physics of the flow in the cage, a closer look is taken here at the case with a disk peripheral velocity of 20 m/s. The streamlines near the samples, of the flow field, relative to a frame of reference which rotates at the same speed as the cage, have been plotted. The points in time when the samples, as shown in Fig. 2 (b), are at angular positions of 0° and 30° have been chosen to illustrate how the relative flow behaves on the plane located at the mid-span of the samples. The streamlines were constrained to this plane as the flow near the samples is three dimensional.

The variations of the friction coefficient \((C_f)\) along the sample length at the angular locations of 0° and 30° have been plotted here in Fig. 6 with \(C_f\) defined as:

\[
C_f = \frac{\tau_w}{0.5 \rho u^2}
\]

where \(\tau_w\) is taken as the magnitude of the wall shear stress vector. Note that the magnitude of this vector was taken to account for all the components of the wall shear stress in this 3D flow.

Fig. 5 clearly shows that the blunt profile of the samples causes the flow to feature massive separations from the surfaces of the samples. The 3D nature of the separation bubbles can be observed from the \(C_f\) contours on the sample’s surface that are shown in Fig. 7.

At 0° (i.e. when the sample is aligned with one of the baffles), the flow separation is more pronounced over the inner surface, because of the constraining influence of the baffle over the outer surface. From 0° to 30°, however, the separation bubble along the outer side grows, as the sample moves away from the baffle. On the inner side the large vortex initiated at the leading edge at 0° has been shed and swept backwards at 30° and the spiralling streamlines behind the body, as if there was a mass-flow sink, actually indicate flow escaping in the 3rd direction, normal to this 2D plot. Although not shown here, the characteristics of the separation bubbles were also observed to change from 30° to 60°.

One can thus conclude that the flow features and, consequently, the local wall shear stress undergo strong changes as the sample rotates over a 90° segment. It is anticipated that this will complicate the task of interpreting measurements of sample corrosion and finding correlations that relate corrosion to the wall shear stress. In addition, the conditions in the experiment would deviate from the ones found in turbulent flows in straight pipes and channels as situations with flow separations are characterized by weak wall shear stresses. On the contrary, mean flow unsteadiness may be an enhancing flow-accelerated corrosion factor, more similar to pulsating jets causing large pressure and wall shear stress fluctuations (see [16]).

Note that separation bubble characteristics should only be considered qualitative in this study; they are difficult to predict by CFD, and both mesh and model are rather crude in this preliminary study.
Figure 5. Constrained streamlines of the relative flow around the mid-section of the blunt sample at (a) 0° (b) 30°.

Figure 6. Friction coefficient along the mid-span of the blunt sample at (a) 0° (b) 30°. x represents the distance from the leading edge and L is the length of the sample.

Figure 7. Friction coefficient values on the outer and inner surfaces of the blunt sample (left and right, respectively) at the angular location of 0°.
5. IMPROVED DESIGN OF THE ROTATING CAGE SETUP

In order to alleviate the large torque exerted by the flow on the samples, one can change the profile of the samples in a way that makes them streamlined bodies rather than bluff bodies. This section reports how the samples were streamlined here and the results obtained when the flow inside the improved design was simulated.

The streamlining strategy adopted here is that of a sample profile with a semi-circular nose and a triangular tail, closer to simple fan blades rather than airfoil profiles, to keep large planar surfaces for corrosion inspection. The resulting profile is therefore similar to one of the flat plate models used in [17] to assess the noise generated by different trailing edge fan geometries. The front semi-circle is centered here at the sample’s mid-thickness and is tangent to the leading edge of the original sample (i.e. the radius is equal to half the thickness of the sample). The sample’s tail is an isosceles triangle whose apex is located at the sample’s mid-thickness. This symmetric shape ensures manufacturing simplicity, although the circular rather than rectilinear trajectory in the RC would suggest slightly tilting inwards the nose and tail to further improve the hydrodynamics.

The improved design is shown in Fig. 8. The angle of the triangular tail ($\phi$) is chosen to be 26°, which is much larger than the value of 12° chosen in [17]. This angle prevents most of the flow detachment from the rear part of the samples and at the same time maximizes the length of the unaltered flat inner and outer surfaces of the samples for a given chord length. This has been arrived at with the help of 2D simulations in which the flow behaviour on the plane located at the middle of the sample length has been simulated (see Fig. 2 (b)). The reader can refer to [18] for details about these 2D simulations. In order to confirm the suitability of the $\phi = 26^\circ$ choice made using the 2D simulations, full 3D simulations of the improved design were carried out.

![Figure 8. The cross section of the streamlined sample used in the improved design.](image)

The flow behaviour in the improved design for the case with a peripheral disk velocity of 20 m/s is presented and discussed in Figs. 9 and 11 by referring to the conditions on the plane located at the sample’s mid-span. The constrained streamlines of the relative flow around the sample are shown in Fig. 9 for the point in time when the sample is at the angular location of 0°. These streamlines are coloured here by the pressure coefficient ($C_p$), which was calculated as:

$$C_p = \frac{p-p_{ref}}{0.5 \rho v^2}$$

where $p_{ref}$ was taken as the maximum pressure on the surface of the sample. This maximum pressure occurs in the region where the relative flow impinges on the sample (this impingement region can be observed from the $C_f$ contours shown in Fig. 10). In addition, the 0° and 30° profiles of the friction coefficient on the walls of the sample are shown in Fig. 11.
Figure 9. Streamlines of the flow around the streamlined sample at 0° as they appear in a reference frame that rotates at the angular velocity of the samples. The streamlines are coloured by the pressure coefficient.

Figure 10. Contours of the friction coefficient $C_f$ on the wall of the sample at 0°.

Figure 11. Friction coefficient along the mid-span of the streamlined sample at (a) 0° (b) 30°. $x$ represents the distance from the leading edge and $L$ is the length of the sample.

In accord with the 2D simulation results, the flow remains completely attached to the samples when $\phi$ is set as 26°. Moreover, comparison between Figs. 7 and 10 shows that the flow now remains quasi-homogeneous over the full span of the sample. Although not shown here, a similar behaviour has been observed at the other angular locations. Consequently, the $C_f$ values (see Fig. 11) are on average higher than those featured in the original design, shown in Fig. 6. The absence of flow separation causes the flow...
around the samples to become more two-dimensional (see the $C_f$ contours shown in Fig. 12) and results in less fluid entrainment in comparison to that in the original design (although not shown here, this was confirmed by looking at contours of the velocity magnitude). Furthermore, the changes in the profile of $C_f$ between the different angular locations are relatively mild in comparison to the variations observed for the rectangular sample design in Fig. 6. For instance, all the $C_f$ profiles feature a plateau in the region between $\frac{x}{L} = 0.1$ and $\frac{x}{L} = 0.4$ here. The weak interaction between the boundary layers around the samples and the fluid outside of them, in the new design, seems to weaken the impact of the baffles on the shear stress distribution over the samples. This is also reflected in the differences between $C_f$ over the inner and the outer surfaces of the samples. The distribution of the friction factor $C_f$ is now similar over both sides and at all angular locations, with the levels over most of the inner side being moderately higher.

![Image of friction coefficient](image)

**Figure 12.** Friction coefficient values on the outer and inner surfaces of the streamlined sample ((a) and (b), respectively) at the angular location of 0°.

The power rating values versus peripheral disk velocities in Fig. 13, follow again a cubic function with a lower constant, reduced by more than a factor of 2, for the improved design at all velocities. It is worth noting that the moment coefficient over the range of Reynolds numbers that correspond to all disk velocities was found to be independent of the Reynolds number, with a value considerably lower than that computed for the original design.

The Pie chart in Fig. 14 shows the moving walls’ shares of the total torque in the improved design. The samples now contribute only to 17% of the total torque, most of which is due to shear forces. The situation here is thus quite different from the original case, where the samples contribute to 76% of the total torque, nearly all of which is due to pressure drag forces.
Figure 13. Predictions of the maximum power input values required to achieve different disk peripheral velocities when the preliminary and improved designs are used.

Figure 14. A pie chart showing the contributions of the different parts of the improved design to the total torque at the angle of 0° (see Fig. 2 (b)).

6. CONCLUSIONS

This study focuses on the analysis of liquid lead flow inside the rotating cage test rig, a rig used for erosion-corrosion testing, through CFD simulations. The simulations reported here utilize the RANS framework to model the effects of turbulence on the ensemble-averaged flow. The less strict grid requirements of this framework facilitated testing four rotational speeds and two different rig designs.

It has been observed that the original design (i.e. the design with blunt cuboid samples used in the literature) requires significant power to achieve high peripheral disk velocities when lead is used as a fluid. The rectangular cross-section of the samples leads to extensive flow separation, which, in the case of the high-density liquid lead especially, results in significantly large form drag acting on the samples. The flow separation also leads to weak shear stresses acting on the surfaces of the samples. In addition, it is found here that the flow behaviour and hence the shear stresses on the inner and outer surfaces of the samples change significantly with the angular location of the sample. These reasons motivated the development of an alternative design in which streamlined samples with a relatively simple shape are used. The difference
between the profile of these samples and the original profile is the presence of a semi-circular front and a triangular tail in the former.

It has been demonstrated that the improved design results in a flow that remains entirely attached to the surfaces of the rotating samples throughout the revolution. A decrease of around 56% in the power required to maintain the rotation of the cage is consequently achieved. Additionally, the profiles of the friction coefficient \( C_f \) on the surfaces of the streamlined samples are found to change very little with angular location. They also feature plateaus over about one third of the sample length. These two features can simplify the process of developing meaningful correlations between the corrosion/erosion rates and the shear stress on the samples. The relatively large \( C_f \) values present in the improved design also imply that the samples will be corroded faster than the blunt ones. This in turn can reduce the time needed to run the experiments and can better emulate the actual flow conditions in pipes and channels. The erosion-related advantages offered by the new design make it useful not only with lead but also with other corroding fluids.

ACKNOWLEDGMENTS

The authors wish to acknowledge support from the Engineering and Physical Sciences Research Council (EPSRC) for funding this work (grant EP/T003359/1). The authors are also thankful to Prof. Jung-Kun Lee of the University of Pittsburgh (USA) for providing the CAD files of the rotating cage setup and for the insight into the material-related aspects of the design. The authors would also like to thank Prof. Brian Launder and Dr. Tim Craft of the University of Manchester (UK) for their valuable help and suggestions.

REFERENCES


APPENDIX A. GEOMETRY DIMENSIONS

The cylindrical container’s radius and height are 140 mm and 339 mm, respectively. The shaft has a radius and height of 19 mm and 305 mm, respectively and its bottom face is located 15.7 mm above the bottom of the cylindrical container. The disks’ diameter and thickness are 216 mm and 5 mm, respectively. The distance from the top of the upper disk to the top wall of the container is 194 mm, whereas the distance from the bottom of the lower disk to the bottom wall of the container is 59 mm. Regarding the samples, the radius of the sample centreline is about 102 mm and their length, thickness and span are 19 mm*3 mm*77 mm. The alumina isolators are 25.4 mm long, 6.4 mm wide and 4 mm high. In addition, the radius and height of the hub were 38 mm and 22.2 mm, respectively. Furthermore, the baffle dimensions are 19 mm*3 mm*305 mm and their lower faces are 15.7 mm above the bottom wall.

APPENDIX B. TURBULENCE MODEL SENSITIVITY

The sensitivity of the results to the adopted turbulence model was assessed in [18] by comparing the SST model’s predicted power rating values with those estimated using the BL v2k (see [19]) and the two-layer k – ε (see [20] and [21]) models. The maximum difference between the predictions was 23% and 10% for the original and improved designs, respectively. The elimination of flow separation in the improved design makes the results less sensitive to the turbulence model. To further illustrate this, here, we present the flow behaviour that the BL v2k model predicts. Fig. 15 shows contours of the friction coefficient on the surfaces of the samples. In line with what was observed with the SST (Fig. 12), deviation from the 2D state occurs only close to the locations where the sample is connected to the alumina isolators. To further illustrate the similarity between the predictions, the pressure and friction coefficient profiles predicted by both models are compared in Fig. 16. The modelling of the turbulent viscosity seems to have almost no effect on $C_p$ but does affect the $C_f$ results. However, both models predict similar $C_f$ trends. The higher friction predicted by the SST model might be related to the minimum wall distance limiter that is used in StarCCM+, which clips the wall distance to $10^{-6}$ m by default in the turbulence models’ equations, thereby lowering $\omega$ near the
However, further investigation is required to draw clear conclusions about the differences in $C_f$ predictions.

**Figure 15.** BL v2k predictions of the friction coefficient on the outer and inner surfaces of the streamlined sample ((a) and (b), respectively) at the angular location of $0^\circ$.

**Figure 16.** SST and BL v2k predictions of the pressure and friction coefficients ((a) and (b), respectively) along the mid-span of the streamlined sample at $0^\circ$. $x$ represents the distance from the leading edge and $L$ is the length of the sample.