Cite this article

Marimuthu S and Chinnathambi D (2020)
Computational analysis of biomimetic butterfly valve.
Bioinspired, Biomimetic and Nanobiomaterials 9(4): 223–232,
<https://doi.org/10.1680/jbibn.20.00027>

Research Article

Paper 2000027
Received 07/04/2020; Accepted 19/10/2020
Published online 07/12/2020

Keywords: biomimetic materials/
computational studies/surface
modification

ICE Publishing: All rights reserved

Computational analysis of biomimetic butterfly valve

1 Siva Marimuthu MEng

Assistant Professor, Department of Aeronautical Engineering,
Dhanalakshmi Srinivasan College of Engineering and Technology,
Tamilnadu, India (Orcid:0000-0001-6296-4081)
(corresponding author: sivavins@gmail.com)

2 Dhavamani Chinnathambi PhD

Professor, Department of Mechanical Engineering, Mahendra
Engineering College, Tamilnadu, India

Butterfly valves are most commonly used in aerospace and mechanical industries to regulate fluid flow. Although butterfly valves are known for their low pressure drop functions, still instabilities occur when operating the valves from closed to open positions. Disturbances like the skin friction and turbulence kinetic energy of the disc greatly reduce the outlet velocity of the fluid. Although there are many techniques to address the disturbances, studies on the sharkskin pattern gained a spotlight in solving both aerodynamic and hydrodynamic problems in internal flows. In this computational work, the performance of the butterfly valve is improved with the help of the implemented biomimetic pattern on the surface. It was noted from the computational analysis that the skin friction coefficient, turbulence kinetic energy and outlet velocity of the butterfly valve could be enhanced by up to 83.24, 45.54 and 3.42%, respectively. The net mass flow rate of the pipe is also improved with the help of the biomimetic butterfly valve. Computational results were validated to ensure its steadiness. This computational research is proof of the concept ‘fluid flow over a surface with little roughness is better than a smooth surface’.

Notation

A	area
d	diameter
R_{ij}	Reynolds stress tensor ($= -\rho \overline{u_i' u_j'}$)
V	velocity
μ	viscosity
ρ	density

1. Introduction

Butterfly valves (BVs) are known for their easy operational control, minimum area occupancy and faster installation capability. Consequently, they are used widely in many industries like aerospace, mechanical, oil and gas to regulate fluid flow, and have also replaced many conventional valves. Many studies have been carried out to enhance the performance and to extend the applications of BVs. McPherson *et al.*¹ noted that the characteristics of fluid flow were influenced by either the blades' shape or the near-opening or closing angles of the disc, when it was tested with turbulent flow over a two-dimensional (2D) BV. Similarly, when Sarpkaya² investigated the 2D BV in laminar flow, he was able to approximate the hydrodynamic torque, cavitation and flow coefficients for three-dimensional (3D) BVs using semi-empirical equations. On the other hand, Kimura *et al.*³ and Ogawa *et al.*⁴ noted in their theoretical analysis that it was incorrect to predict the performance of a 3D BV with the values

of a 2D valve. Likewise, Addy *et al.*⁵ concluded in their research work that it could be possible to estimate the valve's performance characteristics when the valve flow coefficient is known for a particular pressure ratio. Over decades, only a few researchers managed to perform experimental work on BVs. Nevertheless, when computational fluid dynamics (CFD) was introduced in the late nineteenth century, innovators were impressed by the idea of time- and cost-efficient tools, which also motivated Song *et al.*,^{6,7} who noted in their CFD study that simulation data were not highly similar when compared to the actual results at 20° and above angles of opening (AOP). Huang and Kim⁸ verified their computational results with the experimental values of Blevins⁹ and identified that their computational values were in good agreement with experimental data at 45° AOP, while the rest lacked agreement. This impact of CFD changed the course of research on BVs by adapting the use of this technique. In another research, Lin and Schohl¹⁰ compared their computational results with Hoerner's¹¹ experimental data and opined that it was difficult to match the parameters of a numerical model and an experimental setup. Within a few years, they overcame this issue with a 3.66 m dia. BV that could be matched using the CFD technique. In addition to that, Leutwyler and Dalton^{12,13} carried out their CFD study on 2D and 3D BVs over a range of pressure ratios and favoured the turbulence models for their low computational costs.

On another instance, Henderson *et al.*¹⁴ concluded in their CFD analysis using Ansys software that the flow downstream from the BV was dominated greatly by vortices. These disturbances were widely noticed at 20° and above AOP. Additionally, Cheiworapuek *et al.*¹⁵ examined the two different BVs of 150 and 300 mm dia. using the CFD code Fluent to predict the incompressible turbulent flow past a BV that resulted in the formation of vortices near the valve tips and also identified that computational results lacked agreement with experimental values. Besides, Ghaleb *et al.*¹⁶ performed a numerical analysis over a 38 mm dia. BV and suggested that the performance of BVs is way low at near-closed angles. Likewise, Arun *et al.*¹⁷ highlighted increased form drag and pressure loss at small AOP from their research work on BVs using CFD. After continuous research, Adam¹⁸ also mentioned in his research that CFD is able to predict the performance factors of BVs in agreement with experimental performance values, at low AOP. In addition, Feng *et al.*¹⁹ found in their CFD research that a double eccentric BV had improved dynamic response and was self-sealing in comparison with a single or no offset BV. Besides, Marius *et al.*²⁰ confirmed in their analysis that increased disc thickness would lead to an increased mass and would contribute to the decrease in natural frequency values.

The above two paragraphs describe the transition of research methods from experimental to computational and the inclusion of various BV designs. In addition to that, they also explain the disturbances found in the fluid flow regime. In 1988, Eom²¹ developed a new design called perforated valve and identified the similarities between a perforated BV and a non-perforated BV, and the results were quite surprising. The performance of a perforated disc was higher than that of a solid disc. This patterned model was the motivation of this research. However, when Miyazaki *et al.*²² indicated that the morphological non-uniformity in sharkskin denticles likely plays a critical role in passively

controlling local turbulent flow in aquatic vehicles as well as other fluid machinery, the focus towards biomimetic became intriguing. Biomimetics applies principles and strategies from biological systems to engineering and technological products, processes and design in general.²³ Furthermore, Xiao *et al.*²⁴ mentioned that recirculation transformed the original fluid–wall sliding friction into the fluid–fluid rolling friction, effectively reducing the skin friction. For instance, the structure of the short-fin mako shark skin had been found to reduce 8% of skin friction drag.²⁵ Throughout the years, many models have been proposed, but none has been universally reliable. Indeed, the flow over rough surfaces is a complex field of research, and much is still to be discovered.²⁶ Similarly, the morphology of the sharkskin surface was made clear, but its mechanism of drag reduction is not yet clear, and to unveil its mechanism is still a hot topic of research.²⁷ Moreover, Nzombo²⁸ strongly recommended in his research thesis on the CFD study over a BV that further studies are needed to be carried out to identify and improve its performance. This warrants the necessity to improve the performance of BVs with a biomimetic pattern.

2. Objectives

Disturbances noted at the end of this literature review were verified with the computational analysis performed over a BV. Figure 1 clearly indicates the occurrence of instabilities like vortices and high-pressure regions around the BV disc. Hence, the main purpose of this research is to reduce the turbulence kinetic energy and skin friction coefficient, which would improve the mass flow rate and outlet velocity of the BV in pipelines. These objectives were achieved by the computational implementation of the bio-inspired pattern over the surface of the valve disc.

3. Methodology

With the advancement of computational power, there has been much focus on using CFD to simulate turbulent flows over rough

Figure 1. Pressure path lines past the BV disc

surfaces.²⁹ Therefore, in this research work, a BV and a fluid pipeline were designed and analysed in water flow, using computational tools like SolidWorks and Ansys. Biomimetics provides us with a unique way forward to enable drag reduction without the use of complicated solutions.³⁰ The skin of a fast-swimming shark has revealed its riblet structure that could help reduce the shark's skin friction drag, enhancing its efficiency and speed while moving in water.³¹ This bio-inspired pattern was applied and has successfully reduced the drag force, which suggests that the longer-scale structures actively influence the near-wall features.^{32,33} Therefore, the identified biomimetic pattern was applied on the disc and analysed in the flow domain, computationally.

3.1 Computational design

In this internal water flow research work, two main elements were designed. They are the pipeline and the BV disc. The standard dimensions required to design these two essential parts of the

study were selected from the research work of Adam.¹⁸ The selected dimensions of the pipeline, BV disc and hollow patterned biomimetic butterfly valve (HPBBV) disc are tabulated in Table 1. In the Ansys DesignModeler, a circular disc was created on the X - Y plane with the help of sketch tools. That 2D image was extruded into a 3D object and filleted. Similarly, another circular sketch was drawn on the same plane to develop a pipeline of $2.5D$ upstream and $15D$ downstream lengths. From previous research work, the dimensions of the shark-inspired 'raw riblet (RR)' pattern was chosen.³⁴ The 3D image of the RR pattern designed in SolidWorks was imported into Ansys and implemented on the BV disc. Figures 2 and 3 clearly display the 3D image of the BV and HPBBV discs, respectively.

3.2 Computational analysis

The designed BV and HPBBV models were imported onto the working area of the mesh. The values of display, quality and inflation were set to default, and a linear mesh was generated. The purpose of mesh generation was to divide the whole computational design into a group of grids. It can also be stated as cells with a definite and minimised control domain. Additionally, those grids would enable the commercial code solver to help in solving the governing equations of motion at every point. The mesh was checked for its fineness and irregular curves. The nodes and grid lines were eminently examined to identify the accumulation of improper mesh. More precisely, the mesh was looked for possibilities of any errors. Boundary definitions like inlet, outlet, disc and pipeline wall were defined at the respective faces using the named-selection and box-face select tools. Then the updated mesh was updated and noted for number of elements and nodes.

The brought-in meshed 3D models were converted to polyhedral mesh as illustrated in Figure 4 with the help of the double-precision and pressure-based Fluent 19.2 solver. The polyhedral mesh has reduced computational cost and time. Then its scale and

Table 1. Dimensions of the pipeline, BV disc and HPBBV disc

Specification	Pipeline	BV disc	HPBBV disc
Diameter: m	1.2383	1.2192	1.2192
Length of the pipe: m	17.6784	NA	NA
Upstream length: m	3.0480	NA	NA
Downstream length: m	14.6304	NA	NA
Fillet radius of the disc: m	NA	0.0500	0.0500
Thickness of the disc: m	NA	0.0500	0.0500
Length of the pipe: m	17.6784	NA	NA
Number of patterns	NA	NA	16.0000
Depth of the cut: m	NA	NA	0.0127
Distance between front and rear patterns (vertical): m	NA	NA	0.02071
Distance between side patterns (horizontal): m	NA	NA	0.0109

NA, not applicable

Figure 2. Three-dimensional image of the BV disc

Figure 3. Three-dimensional image of the HPBBV disc

Figure 4. Polyhedral mesh of the pipeline and disc

quality report were noted. Among the many viscous models, Reynolds-averaged Navier–Stokes models are the most widely used approaches in calculating industrial flows and include several effects that are not easily handled by eddy viscosity models.³⁵ They discard the eddy viscosity hypothesis and directly solve the transport equations for Reynolds stresses, which means that, in total, they solve seven transport equations in a 3D problem.³⁶ The stabilisation of the turbulence capability of the Reynolds stress model (RSM) assisted the authors to study the wall stress and skin friction coefficients. Wall boundary conditions and wall reflection effects for linear pressure–strain models make the RSM more suitable for full-swirl flows. The RSM has given information not only on the time-averaged flow statistics but also on the shear stress correlations.³⁷ Moreover, the research conducted at NASA has evidenced that the RSM is capable of predicting significant reduction in turbulent kinetic energy.³⁸ In addition, the RSM is seen to perform better than the $k-\epsilon$ model in terms of both mean flow quantities (friction factor and mean velocity) and the Reynolds stress predictions.³⁹ Hence, the RSM was chosen for its effectiveness in internal fluid flow

problems. Water flow in pipe obeys the continuity equation mentioned in Equation 1.⁴⁰

$$1. \quad \rho_1 A_1 V_1 = \rho_2 A_2 V_2$$

where ρ , V and A are the density, velocity and area, respectively. The subscripts 1 and 2 denote the inlet and outlet of the pipe, respectively. The RSM is based on the Reynolds averaged momentum equation as shown in Equation 2.³⁵

$$2. \quad \rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_k \frac{\partial \bar{u}_i}{\partial x_k} \right) = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_k} \right) + \frac{\partial R_{ij}}{\partial x_j}$$

where $R_{ij} = -\rho \overline{u'_i u'_j}$ is defined as the Reynolds stress sensor. RSMs are modelled as per the flow characteristics in order to close the system of governing equations. The control volume of the pipeline was defined as fluid. The boundary conditions at the

inlet and outlet were defined. The inlet velocity and hydraulic diameter were set to 0.914 m/s and 1.23 m, respectively. The Reynolds number of the flow is calculated using the Equation 3.⁴⁰

$$3. \quad \frac{\rho V d}{\mu}$$

where ρ , V , d and μ are the density, velocity, diameter and viscosity, respectively.

The Reynolds number of the flow is 1 126 837, which confirmed the occurrence of turbulence. The turbulence intensity, kinetic energy and dissipation values of the flow were set to default. Simple second-order upwind discretisation schemes were used to increase the accuracy of the solution. The standard initialisation of the inlet was carried out, and the solution was iterated in the fluid domain. When the solution was converged, the results were noted. For instance, the solution was converged at the 168th iteration as shown in Figure 5, when HPBBV was analysed at 10° AOP.

4. Results and discussion

4.1 Skin friction coefficient

The surface of the BV disc has high interaction with fluid particles and was noted to be very close to the vortices in the flow at 10° AOP as indicated in Figure 6(a). This interaction between the fluids and the disc was reduced by the implementation of RR patterns on the disc surface. This action has moved the vortices away from the disc as shown in Figure 6(b). The skin friction

coefficient could be reduced between the range 40.24 and 83.24%.

4.2 Turbulence kinetic energy

The turbulence created near the smooth surface of the BV disc is large as shown in Figure 7(a). With the help of the RR surface pattern in HPBBV, the concentration of turbulence for longer distances is reduced as indicated in Figure 7(b). The turbulence kinetic energy of the HPBBV disc is lower than the BV disc at all AOP. The turbulence kinetic energy could be reduced between the range 5.41 and 45.54%.

4.3 Outlet velocity

Figure 8(a) clearly indicates the lower velocity values of the BV disc at 20° AOP in fluid flow. The reduced pressure in the HPBBV disc resulted in improved velocity as shown in Figure 8(b). The magnitude of velocities very close to the HPBBV disc surface is greater than that of the BV. The outlet velocity could be increased between the range 0.46 and 3.42%, at all AOP.

4.4 Net mass flow rate

The net mass flow rate is defined as the difference between the inlet and outlet mass flow rates of the fluid flow in the pipeline. It depends on the pressure variations in the fluid flow. The HPBBV has reduced values of pressure when compared to the BV. Due to the increased low-pressure region in the HPBBV, more mass flows through the pipe than in the BV. The net mass flow rate of the pipeline with the HPBBV is greater than that with the BV at 20°, 30°, 50°, 60° and 70° AOP.

Figure 5. Residual plot of HPBBV at 10° AOP

Figure 6. Skin friction contour at 10° AOP: (a) BV and (b) HPBBV

Figure 7. Turbulence kinetic energy contour at 40° AOP: (a) BV and (b) HPBBV

Figure 8. Velocity vectors at 20° AOP: (a) BV and (b) HPBBV

4.5 Discussion

Due to the implemented hollow biomimetic pattern on the smooth surface of the BV, the interaction between the fluid and the surface of the disc was minimised. The fluids were circulated at the sides, reversed against the flow direction and lifted off; then they left the pattern along the flow direction, when they past the hollow biomimetic pattern, as shown in Figure 9. This cultivated the formation of vortices and eddies in the far wake region of the

fluid flow. Most of the instabilities that accumulated in the flow were contained within the hollow section of the biomimetic RR pattern. Thus, reduction in flow problems led the fluids to flow through the disc more freely in HPBBV than in BV. As a result of lessened eddies in the fluid flow, the turbulence kinetic energy of the disc was reduced in HPBBV, when compared to that in BV. The minimised eddies in the flow was due to the linearities in the flow and lessened separation. The domination of the biomimetic

Figure 9. Velocity vectors past the HPBBV disc

pattern in the fluid flow was quite convincing and stipulated the requirement of the pattern on the smooth-walled surface. Concurrently, this behaviour of fluids with the hollow RR pattern was very similar to that of sharkskin, which enabled the HPBBV to perform better than the BV.

5. Validation

Grid refinement test was carried out to check the consistency of the results, which also helped to improve the mesh quality of the

computational design. The designed BV model at 90° AOP was imported onto the mesh area, and the mesh size was varied from 0.25 to 2 m. Outputs like turbulence kinetic energy, net mass flow rate and skin friction coefficient were parameterised. At the end of the computational analysis, similar results were noted at 0.5, 0.75, 1, 1.25 and 1.5 m, as shown in Table 2. This test has confirmed the stability of results in the absence of experimental data. The chosen 1 m mesh size was used at all simulations to attain consistent results.

Table 2. Grid refinement test of the BV at 90° AOP

Mesh element size	Turbulence kinetic energy: m ² /s ²	Skin friction coefficient	Net mass flow rate: kg/s
0.25	0.004445	3.0496	0.007146
0.5	0.004483	3.0684	0.000621
0.75	0.004483	3.0684	0.000621
1	0.004483	3.0684	0.000621
1.25	0.004483	3.0684	0.000621
1.5	0.004483	3.0684	0.000621
1.75	0.004258	2.9316	0.003769
2	0.003971	2.8046	0.007951
Consistent values	0.004483	3.0684	0.000621

6. Conclusion

Based on this computational study, we can conclude that the HPBBV is more suitable for pipes than is the BV. Figure 10 demonstrates the improved performance factors in percentage. The main outcomes of this research are as follows.

- The skin friction coefficient of the disc reduced up to 83.24% at 10°.
- The turbulence kinetic energy of the disc reduced up to 45.54% at 40°.
- The outlet velocity of the pipe increased up to 3.42% at 20°.
- The net mass flow rate of the pipe increased up to 0.0255 kg/s at 20°.

Figure 10. Improved values of skin friction, turbulence kinetic energy and outlet velocity in percentage

Challenging problems in the internal fluid flow over a BV were addressed, and detailed technical analysis were carried out appropriately with the help of the complicated RSM turbulence model. The influence of the newly constructed RR biomimetic pattern in the fluid flow has helped in attaining all the objectives and paved the way for the betterment of engineering designs. This biomimetic design has given proficient results in both the internal and external fluid flows. The applied RR pattern is relatively larger when compared to the sharkskin pattern in previous studies. This would make way for easy maintenance and longer use. The implementation of the biomimetic pattern in valves emphasised the concept ‘fluid over a surface with little roughness is better than smooth surface’. This research would extend the application of BVs beyond its current limitations. The HPBBV model with reduced disturbances would further minimise the operational cost of the valves. Although the main objectives are achieved, still pressure-related problems are yet to be solved at a few AOP. Further analysis with high-end facilities is required to help the researchers develop more efficient solutions that are applicable at all AOP. These research findings would motivate the upcoming researchers to develop more bio-inspired solutions. The focus of researchers on biomimetic inventions would keep on thriving because of the vast availability of feasible solutions in the environment.

REFERENCES

- McPherson MB, Strausser HS and Williams JC (1957) Butterfly valve flow characteristics. *Journal of the Hydraulics Division* **83**(1): 1–28.
- Sarpkaya T (1961) Torque and cavitation characteristics of butterfly valves. *ASME Journal of Applied Mechanics* **28**(4): 511–518.
- Kimura T, Tanaka T, Fujimoto K and Ogawa K (1995) Hydrodynamic characteristics of a butterfly valve – prediction of pressure loss characteristics. *Elsevier Science, ISA Transactions* **34**(4): 319–326.
- Ogawa K and Kimura T (1995) Hydrodynamic characteristics of a butterfly valve – prediction of torque characteristics. *Elsevier Science, ISA Transactions* **34**(4): 327–333.
- Addy AL, Morris MJ and Dutton JC (1985) An investigation of compressible flow characteristics of butterfly valves. *ASME Journal of Fluids Engineering* **107**(4): 512–517.
- Song XG, Wang L and Park YC (2008) Fluid and structural analysis of large butterfly valve. *AIP Conference Proceedings* **1052**: 311–314.
- Song XG and Park YC (2007) Numerical analysis of butterfly valve – prediction of flow coefficient and hydrodynamic torque coefficient. In *Proceedings of the World Congress on Engineering and Computer Science, October 24–26* (Ao SI, Douglas C, Grundfest WS, Schruben L and Wu X (eds)). Newswood Limited, San Francisco, USA, pp. 759–763.
- Huang C and Kim RH (1996) Three-dimensional analysis of partially open butterfly valve flows. *ASME Journal of Fluids Engineering* **118**(3): 562–568.
- Blevins RD (1984) *Applied Fluid Dynamics Handbook*. Van Nostrand Reinhold, New York, NY, USA, chap. 6.
- Lin F and Schohl GA (2004) CFD prediction and validation of butterfly valve hydrodynamic forces. In *World Water and Environmental Resources Congress, 26 April 2012*. American Society of Civil Engineers, Salt Lake City, Utah, USA, pp. 1–8.
- Hoerner S (1958) *Fluid-Dynamic Drag: Practical Information on Aerodynamic Drag and Hydrodynamic Resistance*. Hoerner Fluid Dynamics, Bakersfield, CA, USA.
- Leutwyler Z and Dalton C (2006) A computational study of torque and forces due to compressible flow on a butterfly valve disk in mid-stroke position. *ASME Journal of Fluids Engineering* **128**(5): 1074–1082.
- Leutwyler Z and Dalton C (2008) A CFD study of the flow field, resultant force, and aerodynamic torque on a symmetric disk butterfly valve in a compressible fluid. *ASME Journal of Pressure Vessel Technology* **130**(2): 021302.
- Henderson AD, Sargison JE, Walker GJ and Haynes JH (2008) A numerical prediction of the hydrodynamic torque acting on a safety butterfly valve in a hydro-electric power scheme. *WSEAS Transactions on Fluid Mechanics* **1**(3): 218–223.
- Chaiworapuek W, Champagne J, El Hajem M and Kittichaikan C (2010) An investigation of the water flow past the butterfly valve. *AIP Conference Proceedings* **1225**: 249–262.
- Ibrahim G, Al-Otaibi Z and Ahmed HM (2013) An investigation of butterfly valve flow characteristics using numerical technique. *Journal of Advanced Science and Engineering Research* **3**(2): 151–166.
- Azad A, Baranwal D, Arya R and Diwakar N (2014) Flow analysis of butterfly valve using CFD. *Journal of Modern Engineering Research* **4**(6): 50–56.
- Del Toro A (2012) *Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors*. Master’s thesis, Utah State University, Logan, Utah, USA, paper 1456.
- Feng W, Xiao G and Song L (2009) Numerical simulation of the flow field to the double eccentric butterfly valve and performance prediction. In *IEEE PES Asia-Pacific Power and Energy Engineering Conference (APPEEC)*. IEEE, Wuhan, China, pp. 1–4.
- Draghiciu M, Korka ZI and Gillich GR (2017) Modal-based design improvement of a butterfly valve disc. *Analele Universității “Eftimie Murgu” Reșița Anul XXIV*(1): 80–87.
- Eom K (1988) Performance of butterfly valves as a flow controller. *ASME Journal of Fluids Engineering* **110**(1): 16–19.
- Miyazaki M, Hirai Y and Moriya H (2018) Biomimetic riblets inspired by sharkskin denticles: digitizing, modeling and flow simulation. *Journal of Bionic Engineering* **15**: 999–1011.
- Wanieck K, Fayemi PE, Maranzana N, Zollfrank C and Jacobs S (2017) Biomimetics and its tools. *Bioinspired, Biomimetic and Nanobiomaterials* **6**(2): 53–66.
- Xiao WS and Ming XZ (2019) Turbulent drag reduction characteristics of bionic non-smooth surfaces with jets. *Applied Sciences* **9**: 1–18.
- Marquez MC (2019a) Spilling the secret to a mako shark’s speed. *Forbes*, 4 March. See <https://www.forbes.com/sites/melissacristinamarquez/2019/03/04/spilling-the-secret-to-a-mako-sharks-speed/> (accessed 04/06/2019).
- Harun Z, Abbas AA, Nugroho B, Leon C and Mat S (2018) Surface roughness effects studies in transportation industries. *Jurnal Kejuruteraan SI* **1**(7): 87–90.
- Abbas AA, Ghopa WAW, Mat S et al. (2018) Surface roughness effects on turbulent boundary layer structure of NACA 0026 Airfoil. *International Journal of Engineering & Technology* **7**(3–17): 254–259.
- Nzombo DN (2017) *A Numerical Investigation of Cavitation in Valves and Techno-Economic Analysis of Pinewood Solvent Liquefaction*. Master’s thesis, IOWA State University, Ames, IA, USA, 15593.
- Azman FN, Lotfy ER, Abbas AA and Harun Z (2018) Flow structure in modern cities: wind tunnel investigation. *Jurnal Kejuruteraan* **30**(1): 39–46.
- Bhatia D and Wang J (2019) Biomimetics – a potential solution to drag reduction in modern aerodynamics. *Global Journal of Engineering Sciences* **3**(4), <https://doi.org/10.33552/GJES.2019.03.000568>.
- Ibrahim MD, Amran SNA, Yunos YS et al. (2018) The study of drag reduction on ships inspired by simplified shark skin imitation. *Applied*

- Bionics and Biomechanics*, 1–11, <https://doi.org/10.1155/2018/7854321>.
32. Harvard JA (2018) Using shark scales to design better drones, planes and wind turbines. *ScienceDaily*, 7 February. See <https://www.sciencedaily.com/releases/2018/02/180207090135.htm> (accessed 04/06/2019).
 33. Dengke C, Yang L, Huawei C and Deyuan Z (2018) Bio-inspired drag reduction surface from sharkskin. *Biosurface and Biotribology* **4**(2): 39–45.
 34. Siva M, Megalingam Murugan A, Sivasathya U and Dharmalingam S (2015) Biomimetic in turbulence reduction – recent developments. *Journal of Applied Sciences Research* **11**(19): 123–134.
 35. ANSYS, Inc. (2019) *ANSYS Fluent, Release 19.2, Help System, Fluent User's Guide*. ANSYS, Inc., Canonsburg, PA, USA.
 36. Safayet HMD and Ishtiaque HMD. (2018) Computational investigation of turbulent flow development in 180 channel with circular cross section. *European Journal of Engineering Research and Science* **3**(12): 98–105.
 37. Vijapurapu S and Cui J (2010) Performance of turbulence models for flows through rough pipes. *Applied Mathematical Modelling* **34**: 1458–1466.
 38. Olsen ME (2017) Reynolds-stress and triple-product models applied to a flow with rotation and curvature. *American Institute of Aeronautics and Astronautics Aviation Conference, Washington DC, USA, 13 June 2016*, pp. 1–20.
 39. Resende PR, Pinho FT and Cruz DO (2006) Performance of the k-ε and Reynolds stress models in turbulent flows with viscoelastic fluids. In *Proceedings of ENCIT 2006 – ABCM, Curitiba, Brazil, 5–8 December 2006*. Paper CIT06-0805. See https://www.researchgate.net/publication/242221979_PERFORMANCE_OF_THE_k-e_AND_REYNOLDS_STRESS_MODELS_IN_TURBULENT_FLOWS_WITH_VISCOELASTIC_FLUIDS/stats.
 40. White FM (1974) *Viscous Fluid Flow*. Mc-Graw Hill Book Company, New York, USA.

How can you contribute?

To discuss this paper, please submit up to 500 words to the journal office at journals@ice.org.uk. Your contribution will be forwarded to the author(s) for a reply and, if considered appropriate by the editor-in-chief, it will be published as a discussion in a future issue of the journal.

ICE Science journals rely entirely on contributions from the field of materials science and engineering. Information about how to submit your paper online is available at www.icevirtuallibrary.com/page/authors, where you will also find detailed author guidelines.