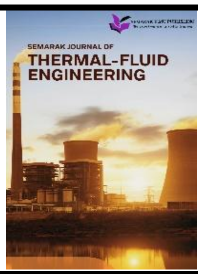
SEMARAK ILMU PUBLISHING

Semarak Journal of Thermal-Fluid Engineering

Journal homepage: https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639



CFD Analysis of Blood Flow Through a Dialyzer Hollow-Fibre Bundle

Wan Muhammad Danial Wan Zalaluddin[1,*], Thaanes Kumara Piran[1], Yashlen Pichyalagan[1], Zul Fitry Nazri[1], Tan Wei Li[1]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Malaysia

ARTICLE INFO

Article history:
Received 20 January 2026
Received in revised form 14 February 2026
Accepted 10 March 2026
Available online 31 March 2026

Keywords:
Haemodialysis; hollow fibre; blood flow; computational fluid dynamics; pressure drop

ABSTRACT

Hemodialysis is a vital medical treatment for patients with renal failure, in which metabolic waste products and excess fluids are removed from the blood using a dialyzer containing thousands of semi-permeable hollow fibres. The hydrodynamic behaviour of blood flow within these fibres plays a crucial role in determining dialysis efficiency and patient safety, as non-uniform velocity distribution, excessive pressure drop, and elevated shear stress may lead to hemolysis and platelet activation. Experimental investigation of flow behaviour inside hollow-fibre dialyzers is challenging due to the small fibre dimensions and dense packing arrangement, making computational fluid dynamics (CFD) a suitable analysis tool. In this study, CFD simulations were performed to investigate steady-state blood flow through a single hollow fibre representing a dialyzer fibre bundle. Blood was modelled as an incompressible, laminar, Newtonian fluid using ANSYS Fluent, with an inlet velocity of 0.01 m/s and physiological material properties. The results show a fully developed laminar velocity profile with a maximum axial velocity of approximately 0.037 m/s at the fibre centreline. A smooth pressure gradient was observed along the fibre length, with an overall pressure drop on the order of 0.4-1.7 Pa depending on inlet and outlet locations. The absence of flow separation, recirculation, or abrupt pressure fluctuations indicates stable hydrodynamic conditions and low shear stress levels within the fibre. Overall, the numerical findings demonstrate that the selected operating conditions produce favourable flow characteristics with minimal risk of blood damage. This study confirms the applicability of CFD as a reliable tool for analysing blood flow behaviour in hemodialysis devices and provides quantitative insight that can support safer and more efficient dialyzer design.

1. Introduction

Haemodialysis is a life-sustaining extracorporeal blood purification therapy used for patients suffering from chronic kidney failure [1]. The primary function of haemodialysis is to remove metabolic waste products and excess fluids from the blood when the kidneys are no longer capable of performing these functions effectively [1],[2]. Central to this process is the dialyzer, a biomedical device that contains thousands of semi-permeable hollow fibres arranged in a compact bundle [2,8].

* *Corresponding author.*
E-mail address: AD210114@student.uthm.edu.my

https://doi.org/10.37934/sjotfe.8.1.3137a

During dialysis, blood flows through the lumen of these fibres, while dialysate flows outside the fibres, enabling solute and fluid exchange across the membrane [3],[4].

The efficiency and safety of haemodialysis treatment are strongly influenced by the hydrodynamic behaviour of blood flow within the hollow fibres [4],[5]. Due to the small characteristic dimensions of the fibres and the low flow velocities involved, blood flow inside a dialyzer typically occurs under laminar conditions dominated by viscous forces [6],[7]. Under such conditions, velocity distribution, pressure drop, and wall shear stress become critical parameters that directly affect solute transport efficiency and blood compatibility [5],[8]. Excessive pressure loss may increase cardiovascular load, while elevated shear stress can cause haemolysis, platelet activation, and clot formation [9].

Experimental investigation of flow behaviour inside hollow-fibre dialyzers is technically challenging because of the small fibre diameters, dense fibre packing, and limited optical accessibility [10],[8]. As a result, computational fluid dynamics (CFD) has become a widely adopted approach for analysing blood flow in dialyzer systems [3],[5],[11]. CFD enables detailed visualisation and quantitative assessment of velocity fields, pressure distribution, and shear-related parameters that are difficult to obtain through experimental methods [5],[8].

Due to the parallel arrangement and geometric uniformity of fibres in commercial dialyzers, modelling a single representative hollow fibre provides an efficient and physically meaningful approach for fundamental flow analysis [12],[13]. This simplification significantly reduces computational cost while retaining the essential hydrodynamic characteristics of the system. Although numerous CFD studies have examined dialyzer flow behaviour, further investigation is required to better understand velocity and pressure distributions under clinically relevant operating conditions and simplified modelling assumptions [8],[11]. Therefore, the objective of this study is to numerically investigate blood flow through a single hollow fibre using CFD.

The study focuses on analysing the velocity distribution and pressure variation along the fibre length and assessing their implications for dialyzer performance and blood damage risk [5],[9]. The findings aim to contribute to improved understanding of internal flow behaviour in haemodialysis devices and demonstrate the applicability of CFD as a design and evaluation tool for biomedical flow systems. Figure 1 shows the diagram that depicts the inlet and outlet of the device. There are a blood inlet and a dialysate inlet along with the outlets for these two inlets. There is fluid exchange taking place through the permeable nature of the materials.

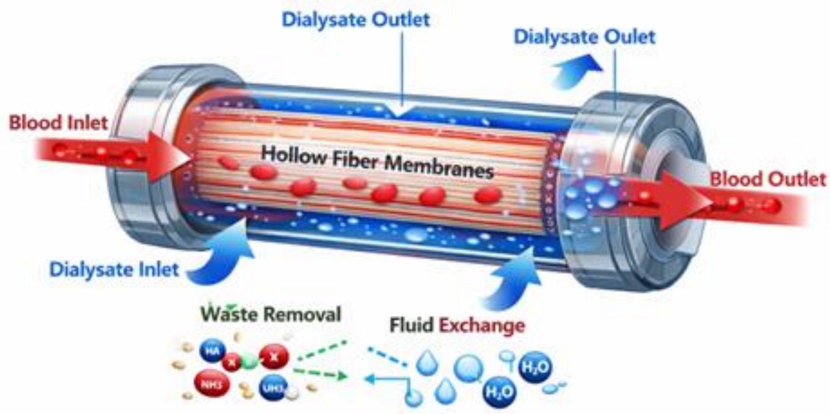


Fig. 1. Dialyser inlets and outlets along with membrane placement

2. Methodology
2.1 Geometric Modelling

The computational model was developed to represent blood flow through a single hollow fibre of a dialyzer. In commercial dialyzers, thousands of hollow fibres are arranged in a parallel and geometrically uniform manner [14],[15]. Explicitly modelling the entire fibre bundle is computationally expensive and impractical [12],[11]. Therefore, a single fibre was selected as a representative domain to capture the fundamental flow behaviour, as commonly adopted in previous CFD studies [11],[13].

The hollow fibre was modelled as a straight cylindrical tube with dimensions consistent with typical dialyzer specifications reported in the literature [13]. The fibre length was set to 200 mm, with an inner lumen diameter of 0.20 mm and an outer diameter of 0.28 mm, resulting in a membrane thickness of 0.04 mm [13]. The blood flow domain corresponded to the inner lumen of the fibre. The geometry was created using ANSYS DesignModeler and imported into ANSYS Fluent for flow simulation [16].

2.2 Mesh Generation

The computational domain was discretised using a structured mesh to ensure numerical accuracy and solution stability. Mesh refinement was applied near the lumen wall to accurately capture velocity gradients and wall shear stress, which are critical parameters in blood flow analysis [5],[17]. Inflation layers were introduced along the wall region to improve near-wall resolution [17]. A mesh independence study was conducted by progressively refining the element size and evaluating solution convergence, following standard CFD best practices [17]. The final mesh consisted of 310,650 elements with a global element size of 14.215 mm. The maximum skewness value was maintained below 0.9, indicating acceptable mesh quality for high-fidelity CFD simulation [17]. Figure 2 depicts the schematic diagram of a typical hollow fibre hemodialyzer.

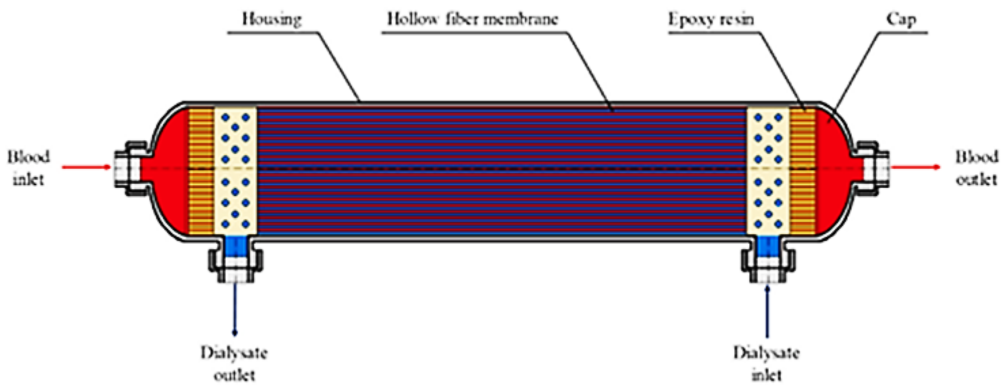


Fig. 2. schematic diagram of a typical hollow fiber hemodialyzer

2.3 Governing Equations

Blood flow within the hollow fibre was modelled as incompressible and laminar under steady-state conditions. The flow was governed by the continuity equation and the Navier–Stokes momentum equations, which describe mass conservation and the balance between inertial, pressure, and viscous forces acting on the fluid [16],[18]. These equations are widely used in CFD studies of internal laminar flow in biomedical applications [5],[8].

2.4 Material Properties and Boundary Conditions

Blood was modelled as a Newtonian fluid, which is a commonly adopted assumption for low Reynolds number flow in hollow fibres [8],[7]. The blood density was set to 1050 kg/m³, and the dynamic viscosity was taken as 0.0035 kg/ms, consistent with reported physiological values [19]. Although blood exhibits non-Newtonian behaviour, the Newtonian assumption provides reliable predictions under the shear rates typically encountered in dialyzer fibres [8],[19].

A uniform velocity inlet boundary condition of 0.01 m/s was applied at the blood inlet to represent physiological flow conditions in a hollow fibre [6],[14]. A pressure outlet condition with zero-gauge pressure was imposed at the outlet. The fibre wall was modelled as a no-slip boundary, ensuring zero velocity at the wall surface, as required for viscous flow modelling [18]. These boundary conditions resulted in a laminar flow regime with a Reynolds number close to unity, which is representative of clinical dialyzer operation [7]. Table 1 explains the parameters from the fibre length to the hydraulic diameter.

Table 1
Explanation for parameters

Parameter	Symbol	Value (mm)	Explanation
Fiber length	L	200	Typical dialyzer fiber length
Lumen diameter (inner diameter)	D_i	0.20	Blood flow diameter inside fiber
Lumen radius	R_i	0.10	Half of inner diameter
Fiber outer diameter	D_o	0.28	Includes membrane thickness
Fiber outer radius	R_o	0.14	Half of outer diameter
Membrane thickness	t_m	0.04	Semi-permeable membrane
Blood flow domain length	Lb	200	Same as fiber length
Hydraulic diameter	Dh	0.20	Circular lumen

2.5 Solver Settings

The pressure-based solver in ANSYS Fluent was used with the laminar viscous model enabled. Pressure–velocity coupling was handled using the SIMPLE algorithm, which is commonly employed for incompressible flow simulations [16]. Second-order discretisation schemes were employed for the momentum equations to improve numerical accuracy. Convergence was achieved when residuals fell below the specified tolerance levels and monitored flow variables stabilised [16].

3. Results
3.1 Velocity Distribution

The velocity streamline results reveal a smooth and fully developed laminar flow profile within the hollow fibre. The maximum velocity occurs along the centreline of the fibre, while velocity decreases gradually toward the wall due to the no-slip boundary condition. This parabolic velocity profile is characteristic of laminar flow in cylindrical tubes and is consistent with classical internal flow theory [18]. The flow remains stable along the axial direction, with no evidence of flow separation, recirculation, or turbulence. Similar laminar velocity behaviour has been reported in previous CFD investigations of hollow-fibre dialyzers operating at low Reynolds numbers [5],]8]. The absence of abrupt velocity gradients indicates favourable flow conditions that minimise shear stress and support efficient solute transport during dialysis [7],[9]. Figure 3 displays the velocity profile for the dialyzer.

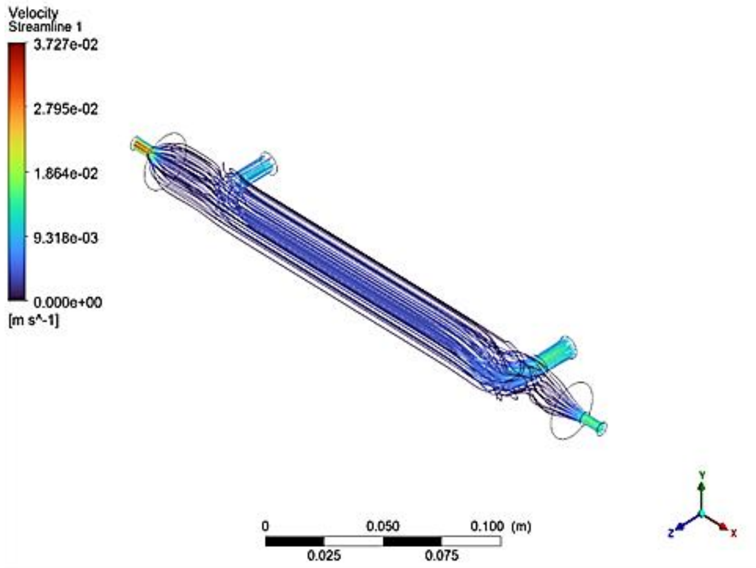
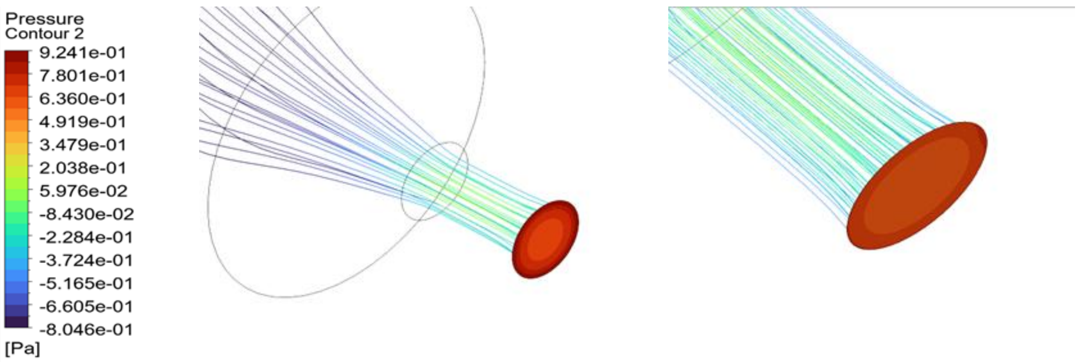


Fig. 3. Velocity profile for dialyzer

3.2 Pressure Distribution

The pressure contours show a gradual decrease in pressure from the inlet to the outlet of the hollow fibre. The pressure drop is primarily attributed to viscous resistance along the fibre wall, which is typical of laminar flow in small-diameter tubes [18]. The smooth pressure gradient indicates stable hydrodynamic behaviour without sudden losses or adverse pressure fluctuations, in agreement with reported CFD results for dialyzer flow [8],[11].

The magnitude of the pressure drop remains relatively small, suggesting that the selected operating conditions do not impose excessive hydraulic load on the cardiovascular system. Similar pressure drop magnitudes have been reported in numerical studies of blood flow through hollow-fibre dialyzers under laminar conditions [5],[13]. Figure 4 shows the pressure distribution at the blood inlet and the dialysate inlet whereas Figure 5 shows the pressure distribution at the blood outlet and the dialysate outlet.



(a) (b)

Fig. 4. Pressure distribution (a) Blood inlet (b) Dialysate inlet

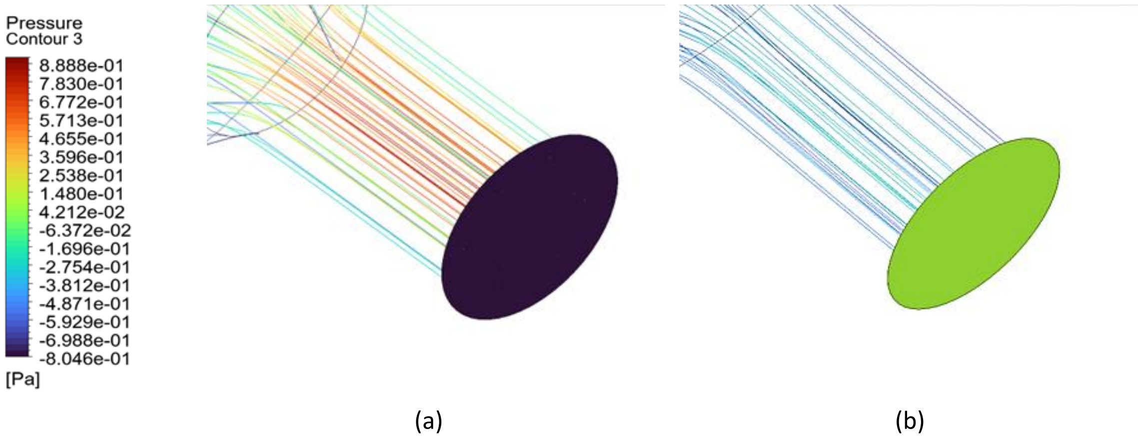


Fig. 5. Pressure distribution (a) Blood Outlet (b) Dialysate Outlet

4. Conclusions

In this study, computational fluid dynamics was employed to investigate blood flow behaviour within a single hollow fibre representative of a hemodialyzer bundle. The CFD model was developed using realistic geometric dimensions, appropriate material properties, and clinically relevant boundary conditions based on established literature [5],[8],[13]. Blood was modelled as an incompressible, laminar, Newtonian fluid, and steady-state simulations were performed using ANSYS Fluent [16]. The numerical results demonstrated a fully developed laminar velocity profile with smooth axial flow and a parabolic cross-sectional distribution, consistent with classical laminar pipe flow theory [18]. Pressure decreased gradually along the fibre length, resulting in a modest pressure drop dominated by viscous effects. These findings are in agreement with previous CFD investigations of hollow-fibre dialyzers operating under similar flow conditions [5],[20],[8].

From a biomedical perspective, the observed flow characteristics suggest low shear stress levels and minimal risk of blood damage under the prescribed operating conditions. Elevated shear stress and abrupt pressure fluctuations, which are known contributors to hemolysis and platelet activation, were not observed in the present model [9]. Although the study was limited to a single fibre representation, the results provide valuable insight into fundamental flow behaviour in dialyzer fibres. Overall, this work confirms the effectiveness of CFD as a reliable tool for analysing internal flow behaviour in hemodialysis devices. The developed model provides a solid foundation for future studies incorporating non-Newtonian blood rheology, fibre bundle interactions, and coupled mass transfer analysis to further optimise dialyzer design and performance [8],[11].

References

[1] Jin, Dong-Chan. "Dialysis registries in the world: Korean Dialysis Registry." *Kidney International Supplements* 5, no. 1 (2015): 8-11. https://doi.org/10.1038/kisup.2015.3

[2] Cheung, Alfred K. "Biocompatibility of hemodialysis membranes." *Journal of the American Society of Nephrology* 1, no. 2 (1990): 150-161. https://doi.org/10.1681/ASN.V12150

[3] Truby, Lauren, Lily Mundy, Bindu Kalesan, Ajay Kirtane, Paolo C. Colombo, Koji Takeda, Shinichi Fukuhara, Yoshifumi Naka, and Hiroo Takayama. "Contemporary outcomes of venoarterial extracorporeal membrane oxygenation for refractory cardiogenic shock at a large tertiary care center." *ASAIO Journal* 61, no. 4 (2015): 403-409. https://doi.org/10.1097/MAT.0000000000000225

[4] Ledebo, Ingrid. "Does convective dialysis therapy applied daily approach renal blood purification?." *Kidney International* 59 (2001): S286-S291. https://doi.org/10.1046/j.1523-1755.2001.59780286.x

[5] Pabby, Anil Kumar, S. S. H. Rizvi, and Ana-Maria Sastre. *Handbook of membrane separations*. Taylor & Francis Group, Boca Raton, Florida, USA: CRC Press, 2023. https://doi.org/10.1016/j.compbiomed.2014.08.010

[6] Yu, Hui, Xing Yang, Rong Wang, and Anthony G. Fane. "Numerical simulation of heat and mass transfer in direct membrane distillation in a hollow fiber module with laminar flow." *Journal of Membrane science* 384, no. 1-2 (2011): 107-116. https://doi.org/10.1016/j.memsci.2011.09.011

[7] Yu, Jian, Vipul C. Chitalia, Olukemi O. Akintewe, Aurelie Edwards, and Joyce Y. Wong. "Determinants of hemodialysis performance: Modeling fluid and solute transport in hollow-fiber dialyzers." *Regenerative Engineering and Translational Medicine* 7, no. 3 (2021): 291-300. https://doi.org/10.1007/s40883-019-00135-0

[8] Eloot, Sunny, Dirk De Wachter, Ilse Van Tricht, and Pascal Verdonck. "Computational flow modeling in hollow-fiber dialyzers." *Artificial Organs* 26, no. 7 (2002): 590-599. https://doi.org/10.1046/j.1525-1594.2002.07081.x

[9] Leverett, L. B., J. D. Hellums, Clarence P. Alfrey, and Edward C. Lynch. "Red blood cell damage by shear stress." *Biophysical journal* 12, no. 3 (1972): 257-273. https://doi.org/10.1016/S0006-3495(72)86085-5

[10] Zhongwei, Ding, Liu Liying, and Ma Runyu. "Study on the effect of flow maldistribution on the performance of the hollow fiber modules used in membrane distillation." *Journal of Membrane Science* 215, no. 1-2 (2003): 11-23. https://doi.org/10.1016/S0376-7388(02)00557-4

[11] en Numerieke, Experimentele. "Experimental and numerical modeling of dialysis." PhD diss., Ghent University, 2005.

[12] Keyvan Hosseini, Mahsa, Lei Liu, Parisa Keyvan Hosseini, Anisha Bhattacharyya, Kenneth Lee, Jiahe Miao, and Bing Chen. "Review of hollow fiber (HF) membrane filtration technology for the treatment of oily wastewater: applications and challenges." *Journal of Marine Science and Engineering* 10, no. 9 (2022): 1313. https://doi.org/10.3390/jmse10091313

[13] Petitclerc, Thierry. "Estimation of mass transfer through a hemodialyzer: theoretical approach and clinical applications." *Artificial organs* 22, no. 7 (1998): 601-607. https://doi.org/10.1046/j.1525-1594.1998.06201.x

[14] Ronco, Claudio, and William R. Clark. "Haemodialysis membranes." *Nature Reviews Nephrology* 14, no. 6 (2018): 394-410. https://doi.org/10.1038/s41581-018-0002-x

[15] Clark, William R., and Dayong Gao. "Properties of membranes used for hemodialysis therapy." In *Seminars in Dialysis*, vol. 15, no. 3, pp. 191-195. New York, US: Blackwell Science Inc, 2002. https://doi.org/10.1046/j.1525-139x.2002.00059.x

[16] Fluent, A. N. S. Y. S. "*Ansys fluent theory guide*." ANSYS Inc (2023).

[17] Martins, Nuno MC, Nelson JG Carrico, Helena M. Ramos, and Didia IC Covas. "Velocity-distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis." *Computers & Fluids* 105 (2014): 218-230. https://doi.org/10.1016/j.compfluid.2014.09.031

[18] Batchelor, George Keith. *An introduction to fluid dynamics*. Cambridge university press, 2000. https://doi.org/10.1017/CBO9780511800955

[19] Cho, Young I., and Kenneth R. Kensey. "Effects of the non-Newtonian viscosity of blood on flows in a diseased arterial vessel. Part 1: Steady flows." *Biorheology* 28, no. 3-4 (1991): 241-262. https://doi.org/10.3233/BIR-1991-283-415

[20] Daugirdas, John T., Peter G. Blake, and Todd S. Ing. *Handbook of dialysis*. Lippincott Williams & Wilkins, 2025.