

Design and Virtual Performance Testing of the Accessory Equipment for a Left Ventricular Assist Device

Sinemis Z. ÇELİK

Dept. of Electrical and Electronics Eng.,
Dogus University, Istanbul, Türkiye
scelik@dogus.edu.tr

Kamuran KADIPAŞAOĞLU

Dept. of Biomedical Engineering
Yildiz Technical University, Istanbul, Türkiye
kamuran@yildiz.edu.tr

Abstract—Cardiovascular diseases are the leading cause of global mortality, and heart failure remains a major consequence. While heart transplants are the gold standard for end-stage heart failure, donor shortages necessitate alternatives like left ventricular assist devices (LVADs). This study aimed to design and optimize the inlet and outlet cannulae and housing for an LVAD to minimize thrombosis risk and enhance device efficiency. Key design parameters included the diameters, lengths, and bend angles of the cannulae, ensuring smooth blood flow and reduced turbulence. Through iterative design modifications and computational fluid dynamics analysis, optimal configurations were identified. The results demonstrated that specific configurations significantly improved flow dynamics, reducing stagnant zones and wall shear stress. This study contributes to the development of a safer and more effective LVAD, addressing the critical need for mechanical circulatory support in patients with severe heart failure.

Keywords—Left Ventricular Assist Device, computational fluid dynamics, cannulae design, thrombosis risk, design optimization, hydraulic performance.

I. INTRODUCTION

Cardiovascular diseases, involving the heart and blood vessels, are the leading cause of global mortality, accounting for 32% of deaths in 2019 (17.9 million) [1]. Heart transplants are the best treatment for end-stage heart failure but are limited by donor shortages; only 2,000 transplants occur annually in the US [2]. In Turkey, in 2017, only 75 heart donors were available for 521 waiting patients, with 214 dying while awaiting transplantation [3]. Mechanical circulatory support (MCS), including ventricular assist devices (VADs) and total artificial hearts (TAHs), is a critical therapy for advanced heart failure. VADs, such as left ventricular assist devices (LVADs) and right ventricular assist devices (RVADs), initially served as a "bridge-to-transplant [4]." Now, they also provide long-term "destination therapy" and lifelong support for patients who are not suitable for transplants [5]. LVADs comprise a turbine, an inlet cannula (IC), an outlet cannula (OC), a driveline, and housing. The inlet cannula connects the left ventricle (LV) to the pump, while the outlet cannula links the pump to the aorta. Proper cannulae design is vital for steady blood flow. The driveline connects the pump to an external

mechanism, and the housing contains the pump. Thrombosis is a primary complication of VADs, requiring patients to undergo anti-coagulation therapy [6]. Factors affecting VAD performance include cannulae geometry [7], inlet cannula insertion angle [8] and length [9], post-operative malposition [10], and pressure from the outflow graft on the right ventricle (RV) [11]. Optimizing LVAD's accessories involves iterative computational fluid dynamics (CFD) simulations to predict blood flow. These simulations solve fluid dynamics equations numerically, providing insights into velocity fields, pressure distributions, and shear stresses.

II. MATERIALS & METHODS

A. Cannulae Design Parameters

The concept design for the turbine housing and the inlet and outlet cannulae was initiated. The main design parameters for the cannulae were inner diameter, bend angles, and tangent lengths (Fig. 1, a and c). For the inlet cannula, a step diameter reduction angle was considered to allow the larger diameter of the housing to meet the smaller diameter at the left ventricular entry site (LV apex) without disturbing the flow. The bend angle of the inflow cannula ranged from 55° to 65°, while the bend angle of the outlet cannula ranged from 100° to 110°. These variations influenced the formation of stagnant zones. For the housing design, the only parameter selected was the distance between the cannulae and the turbine (Fig. 1, b). The turbine length was fixed at 82.5 mm [12] and the housing length varied depending on this distance, ranging from 14 mm to 26 mm in addition to the turbine length.

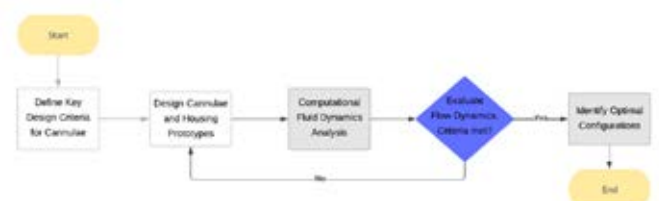


Fig. 1: Schematic diagram of the methodology workflow.

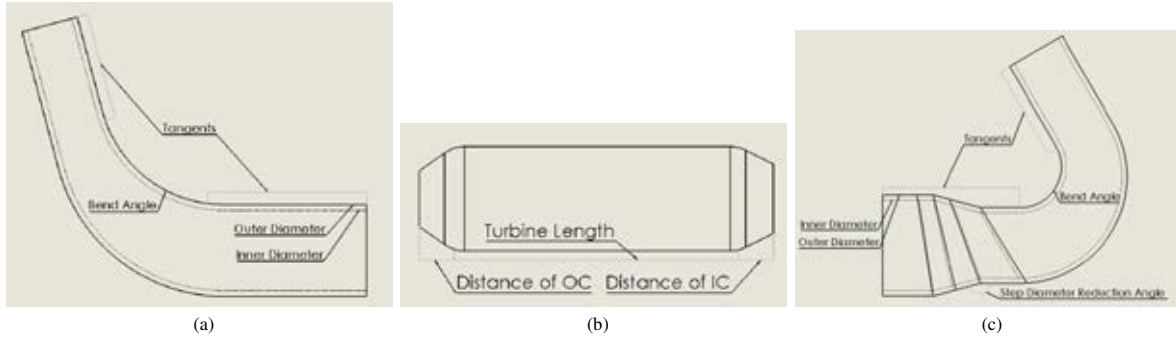


Fig. 2: Design Parameters of the Device: (a) Outlet cannula, (b) Housing, (c) Inlet cannula.

B. Cannulae Performance Analysis

Stagnant zones, where the velocity is below 5 mm/s, were identified as potential sites for thrombosis formation due to low shear rates [13]. Wall shear stress (WSS) was analyzed on the walls of both the inlet and outlet cannulae. WSS values below 0.3 Pa were associated with an increased risk of thrombosis formation, values between 0.3 Pa and 9 Pa were considered physiological and ideal for preventing both thrombosis and blood cell damage, and values above 9 Pa could trigger Von Willebrand factor activation, potentially causing blood cell damage [13].

C. Computational Fluid Dynamics Analysis

The CAD-designed models were used as the base geometry for CFD analysis. The meshing process started using two methods to improve mesh quality. The inflation mesh generation method, used to create a fine mesh near walls to observe boundary layer effects, was applied to three critical regions: the outer housing, the inducer blades, and the diffuser blades. In areas where inflation was not applied, the sizing mesh generation method was used, applied to all surfaces except the outer housing. The viscous k-epsilon model with enhanced wall treatment was used. Blood was defined as a fluid with a density of 1050 kg/m³ and viscosity of 0.0035 kg/ms. Boundary conditions included setting the inlet pressure as left ventricular pressure and the outlet pressure as aortic pressure. Rotor walls were defined as rotational moving walls to accurately simulate motion within the fluid. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was used for pressure-velocity coupling, ensuring mass conservation and determining pressure fields.

D. Post-Processing Analysis

Stagnant zones (velocity < 5 mm/s) were identified in specific cross-sections of the inlet and outlet cannulae. For the inlet cannula, due to the diameter change through step diameter reduction, two areas were selected. IC Area-1 was the region just before the cannula meets the housing, where the diameter is larger. IC Area-2 was the region before the diameter change occurred, where the cannula's diameter is smaller than in IC Area-1. OC Area was the region located after the housing, at the beginning of the outlet cannula. Wall shear stress was analyzed along the walls of the cannulae. Hydraulic efficiency is calculated to evaluate the performance of the combined system of the turbine and cannulae.

$$\eta_h = \frac{Q \cdot \Delta P}{T \cdot \omega} \quad (1)$$

where Q is the flow rate (L/min), ΔP is the pressure rise across the inlet and outlet (mmHg), T is the torque (mN · m), and ω is the rotational speed (rad/s).

III. RESULTS

Cannulae and housing geometric iterations were performed under the same physiological condition, just before aortic valve opening, which corresponds to LV pressure of 0 mmHg and aortic pressure to 84 mmHg, with an angular speed set to 550 rad/s. As the initial model, the housing had an inner diameter of 24.5 mm and a length of 82.5 mm, which was also the length of the turbine. The housing diameter was reduced to match the inlet and outlet cannulae diameters with a 10 mm fillet. The inlet cannula had an inner diameter of 20 mm, bend tangents of 25.51 mm (upper) and 20 mm (lower), a distance from the turbine of 7 mm, and a bend angle of 60°. A single-step diameter decrease was used between the housing and the

TABLE I: Housing Configurations and Performance Results

NO	Geometry			Performance				Stagnant Zone (mm ²)			WSS (Pa)	
	Dist. to IC (mm)	Dist. to OC (mm)	Length (mm)	Q (L/min)	P rise (mmHg)	Torque (mN.m)	Efficiency (%)	IC Area-1	IC Area-2	OC Area	IC Walls	OC Walls
v1	7	10	99.5	3.3242	84.6595	5.2330	21.7315	2.1578	0.9611	0.2183	3.6338	5.5535
v2	7	13	102.5	3.1527	84.2169	5.1920	20.6643	292.4740	143.4360	0.2107	15.4729	-8.4249
v5	7	7	96.5	3.1939	84.3069	5.2388	20.7695	2.2425	0.9660	0.2140	3.6179	5.4532
v12	10	10	102.5	3.4420	84.9895	5.3340	22.1616	0.6819	0.7482	0.2107	4.1214	6.0010

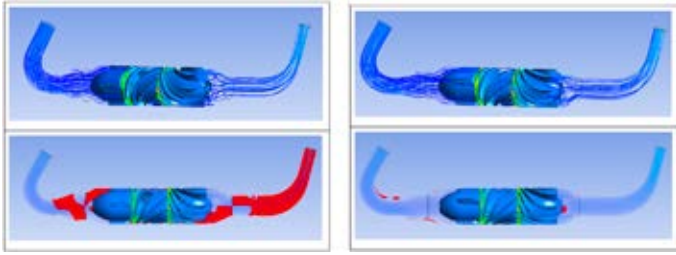


Fig. 3: Flow Visualization of Model 1 (left) and Model 12 (right): Streamlines (top) and Stagnant Zones (bottom). Red regions in the stagnant zone figures indicate flow velocities below 5 mm/s.

bend, with a reduction angle of 155° . The outlet cannula had an inner diameter of 14 mm, bend tangents of 25 mm (lower) and 24.77 mm (upper), a distance from the turbine of 10 mm, and a bend angle of 105° .

A. Housing

The length of the housing was adjusted based on the distance between the cannulae and the turbine. Outlet cannula distances of 7 mm, 10 mm, and 13 mm were tested. The inlet cannula with a 13 mm distance was not tested because preliminary analysis of the outlet cannula showed a significant increase in stagnant flow area. Inlet cannula distances of 7 mm and 10 mm were iterated. Based on these cannula distances, the housing length was determined to be 102.5 mm.

B. Outlet Cannula

The inner diameter was initially set at 14 mm. Any change in this diameter by ± 1 mm resulted in a decrease in efficiency. Therefore, an inner diameter of 14 mm was selected. The tangent before the bend (lower) was initially set at 20 mm but was later reduced to 15 mm without a significant improvement in efficiency. The bend angle was tested at 100° and 110° , with 105° determined to be optimal.

C. Inlet Cannula

The initial step diameter reduction angle at the housing entrance was modified to a smooth transition, which reduced efficiency and flow rate. A double-step reduction also produced low efficiency. For single-step reduction, the initial angle of 155° varied to 170° and 165° . The most uniform flow was observed at 165° . The initial inner diameter was 20 mm and was reduced in increments to 18 mm and 16 mm. An inner

diameter of 18 mm was selected for subsequent tests. The tangent lengths before (upper) and after (lower) the bend were adjusted and the bend angles of 50° and 60° were tested.

IV. DISCUSSION & CONCLUSION

The housing geometry was found to improve both hydraulic performance and flow. This optimization revealed the importance of the distance between the turbine and cannulae in preventing flow separation. As can be seen in Table 1, the optimal design was observed when the distance between the turbine and the cannulae was 10 mm (Model 12). This configuration achieved the highest flow rate (3.4420 L/min) among all the other models, while also producing minimal stagnant zones in IC Area-1 and IC Area-2 (0.6819 mm^2 and 0.7482 mm^2 , respectively). When the OC-turbine distance was increased to 13 mm (Model 2), it did not cause significant decreases in flow rate and efficiency, but the stagnant zone in IC Area-1 increased by 400 times compared to the optimal configuration. This result indicates that increasing the distance between the turbine and the cannulae may lead to the formation of low-velocity recirculation zones that can cause thrombosis. The WSS analysis also supported these results. In Model 2, high shear stress on the IC walls (15.4729 Pa) and negative stress on the OC walls (-8.4249 Pa) indicate reverse flows and flow separation. In contrast, the WSS values in Model 12 (4.1214 Pa on the IC walls and 6.0010 Pa on the OC walls), which is the optimal housing configuration, indicate more stable flow patterns.

It was observed that the performance of the outlet cannula varies depending on the inner diameter and bend angle. The optimal configuration, Model 17, featuring a 14 mm diameter, 25 mm lower tangent, 26.48 mm upper tangent, and a 100° bend angle, achieved the highest efficiency (21.4751%), low stagnant zones (0.2261 mm^2), and moderate WSS (5.7229 Pa). Increasing the diameter to 15 mm (Model 3) slightly decreased efficiency (20.6625%) but increased stagnant zones (0.3733 mm^2). This increased diameter accelerated flow out of the housing, suggesting that this acceleration could cause flow separation near the wall. Reducing the diameter to 13 mm (Model 4) reduced stagnant zones (0.1599 mm^2), a decrease of about 57% compared to the Model 3. However, it did not increase flow rate, suggesting a potential trade-off in adjusting the diameter. Bend angle was another influencing parameter. Models closer to 105° (Models 14 and 17) provided higher

TABLE II: Outlet Cannula Configurations and Performance Results

NO	Geometry				Performance				Stagnant Zone (mm^2)	WSS (Pa)
	Inner Diameter (mm)	Upper Tangent (mm)	Lower Tangent (mm)	Bend Angle (deg)	Q (L/min)	P rise (mmHg)	Torque (mN.m)	Efficiency (%)	OC Area	OC Walls
v3	15	25	24.77	105	3.1818	84.2844	5.2446	20.6625	0.3733	6.9969
v4	13	25	24.77	105	3.2106	84.3519	5.1778	21.1356	0.1599	5.8998
v14	14	15	27.36	105	3.3095	79.9941	5.3036	20.1712	0.2201	6.1817
v17	14	25	26.48	100	3.3037	84.6070	5.2596	21.4751	0.2261	5.7229
v18	14	25	23.27	110	3.2718	84.5094	5.2195	21.4058	0.2066	5.6539

flow rates (> 3.30 L/min), and models deviating from this angle showed a decrease in efficiency and an increase in WSS. Model 18, with a 110° bend angle, exhibited high efficiency, but also showed an increase in stagnant zones compared to Model 17. This indicated that higher bend angles could produce localized recirculation patterns. Overall, the findings indicate that bend angles between 100 and 105° yield more favorable results.

Finally, the inlet cannula geometry was optimized. A single-step diameter reduction angle of 165° yielded the most favorable performance. Model 16, with this reduction angle and a 50° bend angle, achieved the highest flow rate (3.3574 L/min) and highest efficiency (21.7353%). While the stagnant zones (IC Area-1 and IC Area-2) in this model were not the lowest among the other models (0.6057 mm² and 0.7920 mm², respectively), they still remained at moderate levels, indicating the overall performance of this model. In Model 10, a slight decrease in efficiency and flow rate was observed when the inner diameter was reduced from 18 mm (Model 16) to 16 mm. Changes to the bend angle showed that reducing the angle from 60° to 50° (Model 16) increased the flow rate without changing the WSS. Initially, smooth connections between the inlet and outlet of the IC were tested, but this resulted in low efficiency and reverse flow. Therefore, diameter reduction angle were implemented to solve this problem. The smooth and double-step reduction models (Models 6 and 8) produced the lowest flow rates and efficiencies (9.1774% and 8.5007% , respectively), which is less than half of the efficiency achieved in the optimal model (21.7353%). WSS was generally stable (4 Pa), except in Models 6 and 8 which can be due to flow jet at the housing entrance.

This study aimed to optimize the inlet and outlet cannulae of an LVAD accessories using iterative CAD and CFD analyses. This optimization reduced stagnant flow regions and improved flow stability and hydraulic efficiency. These design improvements have the potential to reduce thrombosis formation and anticoagulation requirements in patients and improve long-term device durability. This approach aims to develop a safer and more reliable LVAD for clinical use. However, the results of this study were limited to computational simulation without *in vitro* validation. Future work includes prototyping the optimized models, performing benchtop tests with the LVAD turbine model used in this study and performing PIV-based flow visualization in a circulatory mock circuit to compare

and converge the simulation results.

ACKNOWLEDGMENT

The LVAD turbine model used in this study was designed within the scope of the TUBITAK 1003 project (No. 118S098) and the Yildiz Technical University Scientific Research Projects Coordination Unit (BAP) project (No. FBA-2024-6491) at the Physiological Control Laboratory of the Department of Biomedical Engineering, Yildiz Technical University (YTU).

REFERENCES

- [1] World Health Organization, "Cardiovascular diseases (CVDs)," Available at: <https://www.who.int/news-room/fact-sheets/detail/cardiovascular-diseases-%28cvds%29> (Accessed: 13 Aug. 2025).
- [2] M. J. Everly, "Cardiac transplantation in the United States: an analysis of the UNOS registry," *Clin. Transplants*, pp. 35–43, 2008.
- [3] European Committee on Organ Transplantation, *Newsletter Transplant*, 2017.
- [4] R. C. Starling *et al.*, "Results of the post-U.S. Food and Drug Administration-approval study with a continuous flow left ventricular assist device as a bridge to heart transplantation: a prospective study using the INTERMACS," *J. Am. Coll. Cardiol.*, vol. 57, no. 19, pp. 1890–1898, 2011.
- [5] Cleveland Clinic, "Left ventricular assist devices (LVADs)," Available at: <https://my.clevelandclinic.org/health/treatments/17192-left-ventricular-assist-devices-mechanical-circulatory-support-mcs> (Accessed: 13 Aug. 2025).
- [6] Y. Yaacoub *et al.*, "Do you need antiplatelets and anticoagulation for HeartMate 3," *J. Am. Coll. Cardiol.*, vol. 81, no. 8 (Suppl.), p. 2982, 2023.
- [7] S. Liao *et al.*, "Numerical prediction of thrombus risk in an anatomically dilated left ventricle: the effect of inflow cannula designs," *Biomed. Eng. Online*, vol. 15, Suppl. 2, p. 136, 2016.
- [8] V. K. Chivukula *et al.*, "Left ventricular assist device inflow cannula angle and thrombosis risk," *Circ. Heart Fail.*, vol. 11, no. 4, p. e004325, 2018.
- [9] K. May-Newman *et al.*, "Reducing regional flow stasis and improving intraventricular hemodynamics with a tipless inflow cannula design: an *in vitro* flow visualization study using the EVAHEART LVAD," *Artif. Organs*, vol. 43, no. 9, pp. 834–848, 2019.
- [10] J. Sacks *et al.*, "Utility of cardiac computed tomography for inflow cannula patency assessment and prediction of clinical outcome in patients with the HeartMate II left ventricular assist device," *Interact. Cardiovasc. Thorac. Surg.*, vol. 21, no. 5, pp. 590–593, 2015.
- [11] R. M. Adamson *et al.*, "Principles of HeartMate II implantation to avoid pump malposition and migration," *J. Card. Surg.*, vol. 30, no. 3, pp. 296–299, 2015.
- [12] M. Yigit, "Design, production and performance testing of a novel axial-flow, axial-flux left ventricular assist device (LVAD) with feedback-controlled physiologic flow and two-way wireless communication," *Heart Failure* 2025, 2025.
- [13] M. Ghodrati *et al.*, "The influence of left ventricular assist device inflow cannula position on thrombosis risk," *Artif. Organs*, vol. 44, no. 9, pp. 939–946, 2020.

TABLE III: Inlet Cannula Configurations and Performance Results

NO	Geometry					Performance				Stagnant Zone (mm ²)		WSS (Pa)
	Inner Diameter (mm)	Lower Tangent (mm)	Upper Tangent (mm)	Diameter Reduction Angle (deg)	Bend Angle (deg)	Q (L/min)	P rise (mmHg)	Torque (mN.m)	Efficiency (%)	IC Area-1	IC Area-2	IC Walls
v6	20	19.45	20	Smooth	60	2.1462	81.9742	7.7464	9.1774	0.1778	0.1161	13.4400
v8	20	25.79	20	Double 150	60	2.1553	79.9941	8.1958	8.5007	0.0900	0.1834	9.6312
v10	16	23.60	20	165	60	3.1955	84.3144	5.2450	20.7571	0.2943	0.2521	4.5011
v16	18	27.33	20	165	50	3.3574	84.7495	5.2899	21.7353	0.6057	0.7920	3.9552