Commentary: Go with the flow...dynamics!

Jay K. Bhama, MD,^{a,b} Gurjap Singh, MS,^b and Albert Ratner, PhD^b

From the ^aDepartment of Cardiothoracic Surgery & Baptist Heart Failure and Transplant Institute, Baptist Health Medical Center, Little Rock, Ark; and ^bDepartment of Mechanical Engineering, College of Engineering, University of Iowa, Iowa City, Iowa.

Disclosures: Authors have nothing to disclose with regard to commercial support.

Received for publication April 2, 2019; accepted for publication April 2, 2019; available ahead of print June 5, 2019.

Address for reprints: Jay K. Bhama, MD, Department of Cardiothoracic Surgery & Heart Failure and Transplant Institute, Baptist Health Medical Center, 9601 Baptist Health Dr, Ste 990, Little Rock, AR 72205 (E-mail: jkbhama@me.com).

J Thorac Cardiovasc Surg 2020;159:e211
0022-5223/\$36.00

Copyright © 2019 by The American Association for Thoracic Surgery
<https://doi.org/10.1016/j.jtcvs.2019.04.017>

Computational fluid/flow dynamics (CFD) is a unique branch of mechanical engineering that uses numerical analysis and data structures to analyze and solve problems involving fluid flows. CFD is applied to research and engineering problems across a broad swath of industrial and investigational fields. Its potential use in biomedical investigations is growing, especially as our field advances in the areas of percutaneous intravascular interventions and mechanical circulatory support. The benefits of CFD have largely to do with its ability to clarify the characteristics of blood flow in vascular structures that are otherwise invisible to standard experimental measurements.

Yoshida and colleagues¹ utilize CFD using a novel 3-dimensional computational model to visualize turbulent flow in the aortic root of a patient under continuous-flow left ventricular assist device (LVAD) support. Their analysis identifies retrograde spiral turbulence in the aortic root and allows for qualitative comparison of shear stress at the level of the aortic valve and its components. The authors point out the potential utility of this type of advanced patient-specific computational modeling to understand how LVAD implantation may influence the development of aortic insufficiency in patients with chronic LVAD support.

Indeed, Yoshida and colleagues¹ are not alone in having an interest in how CFD may be used to study clinically relevant problems in the continuous flow LVAD realm. Chivukula and colleagues² from the University of Washington recently utilized CFD analysis to determine the influence of inflow cannula angulation on hemodynamic and thrombogenic performance in a continuous flow-LVAD model. Using this technology, they were able to identify inflow cannula angles that would adversely influence both platelet functionality and intraventricular hemodynamic parameters. Our group at the Bhama-Ratner Artificial Heart &

Left to right: Jay K. Bhama, MD, Gurjap Singh, MS, and Albert Ratner, PhD

Central Message

Computation fluid dynamics is emerging as a unique engineering methodology to “see” what our other investigatory tools cannot.

See Article page e205.

Mechanical Circulatory Sciences Advancement Lab has a similar interest in utilizing CFD to help understand the effect of artificial pulse generation (using a novel pulse-generating device) on the flow characteristics of continuous flow LVADs.³

Yoshida and colleagues¹ are to be commended for their efforts to utilize this novel technology to help answer clinical questions regarding continuous flow LVAD technology. This study provides important insight into the tremendous value of multidisciplinary collaboration among clinicians and engineers. As technology advancements allow us to use advanced computational modeling to “see” what our other investigatory tools cannot, this type of cross-pollination of ideas will be imperative to maximizing the potential of our biomedical research endeavors. There is no doubt that when it comes to understanding what we cannot easily see, it makes sense to go with the flow...dynamics!

References

1. Yoshida S, Toda K, Miyagawa S, Sawa Y. Computational fluid dynamics visualizes turbulent flow in the aortic root of a patient under continuous-flow left ventricular assist device support. *J Thorac Cardiovasc Surg.* 2020;159:e205-7.
2. Chivukula VK, Beckman JA, Prisco AR, Dardas T, Lin S, Smith JW, et al. Left ventricular assist device inflow cannula angle and thrombosis risk. *Circ Heart Fail.* 2018;11:e004325.
3. Bhama-Ratner Artificial Heart & Mechanical Circulatory Sciences Advancement (B.R.A.H.M.A.) Lab homepage. Available at: <https://brahma.lab.uiowa.edu/article/designing-next-generation-heart-pumps>. Accessed May 3, 2019.