Computational fluid dynamics study of blood flow in aorta using OpenFOAM

ABSTRACT

Understanding of flow pattern behaviour inside the aorta contributes significantly in diseases treatment artificial design. Objective of present study is to simulate the blood flow in patient specific aorta using open source computational fluid dynamics (CFD) platform OpenFOAM. The real geometry was obtained from real male Malaysian patient. There are not much data available in literature incorporate real geometry of aorta due to complex geometry. The validation is done against existing experimental result of the 90 degree curve tube model. It was shown that our method is able to capture complex flow in the curve tube like secondary and separation flow that responsible for development of wall shear stress at the tube wall. These flow physics could have similarity in aorta blood flow. Finally, we apply our method with anatomy human aorta with pulsatile inlet condition. Further comparison is made with unstructured boundary fitted mesh. The final result shows that the detailed flow physics can be captured in an aorta.

Keyword: Computational fluid dynamics; OpenFOAM; Blood flow; Aorta