Supplementary file

COMPUTATION FLUID DYNAMICS

CFD background: CFD (computational fluid or flow dynamics) is a well-established technique in bioengineering and fluid mechanics for assessment of aerodynamics of fast-moving objects including vehicles. More recently, it has been used to assess flow within heart and vascular structures.[1,2] The technique uses a computer program to calculate fluid flow parameters in a specific model geometry such as aorta, based on general fluid flow governing equations. Due to the highly complex and non-linear nature of flow problem, the equations cannot be solved analytically. Numerical methods such as finite volume method can be used to solve the complex fluid flow equation in discretise form in which the model geometry is divided into fine elements. Together with the flow parameters such as flow rate, pressure or velocity values at boundary planes as boundary conditions, the fluid flow equations can be simplified as simple, linear equations which can be solved by specific computer software. The flow parameters such as velocity and pressure at the node points of the elements are the solutions of the program. With proper interpolation calculation, the flow parameters at any special point inside the model, i.e. ventricle and aorta, can be predicted. To perform such a simulation, a model geometry and boundary conditions are an essential input.

CFD modelling and analysis: From the CCTA data, a trans-valvular flow geometry including LVOT, valve leaflets, aortic root and part of ascending aorta, was reconstructed by a purpose-built software. The 3D computational mesh (or the fine elements) was generated by a mesh builder ANSYS-ICEM 16.2 in the model. From the patient specific model, CFD simulation was performed by using a standard CFD software (ANSYS16.2-CFX, Pennsylvania, USA). To avoid influence of flow entrance and exit effects in the CFD simulation, the LVOT segment was extended and an artificial aortic arch was linked at the aortic root. The flow rate at the LVOT was derived from TTE V_{max} and CT area, and this was used as the CFD boundary condition at the inlet plane. The outflow boundary was defined as a pressure boundary with a constant pressure (zero) assigned for a steady flow simulation. Despite the pulsatile nature of AV flow, it is the maximum flow resistant caused by AV maximum opening phase that was of interest. Therefore, a steady flow simulation was used to mimic the flow situation in the maximum valve opening phase. The standard k-w turbulent model was used in the CFD simulation with high resolution on the advection scheme.

The distribution of peak velocities (CFD-AV) was evaluated at the aortic valve orifice in the post-processing analyses. A ratio of ≥ 1.2 between peak and mean velocity across the AV orifice was consisted as skewed. Besides, an area ratio between LVOT and AV was calculated similar to the velocity ratio (or dimensionless index).

References

1. Wong KK, Wang D, Ko JK, Mazumdar J, Le TT, Ghista D. Computational medical imaging and hemodynamics framework for functional analysis and assessment of cardiovascular structures. Biomedical engineering online 2017; **16**: 35.

2. Morris PD, Narracott A, von Tengg-Kobligk H, Silva Soto DA, Hsiao S, Lungu A, et al. Computational fluid dynamics modelling in cardiovascular medicine. Heart 2016; **102**: 18-28.