Experimental and numerical simulation study of the near-hinge flow field of a bileaflet mechanical heart valve

Yan Li[1], Guanglei Wang[2], Calin Neamtu[1], Dan Rafiroiu[1], Giuseppe D'Avenio[2], Mauro Grigioni[2]

[1]*Technical University of Cluj-Napoca, Romania,* [2] *Istituto Superiore di Sanita, Italy*

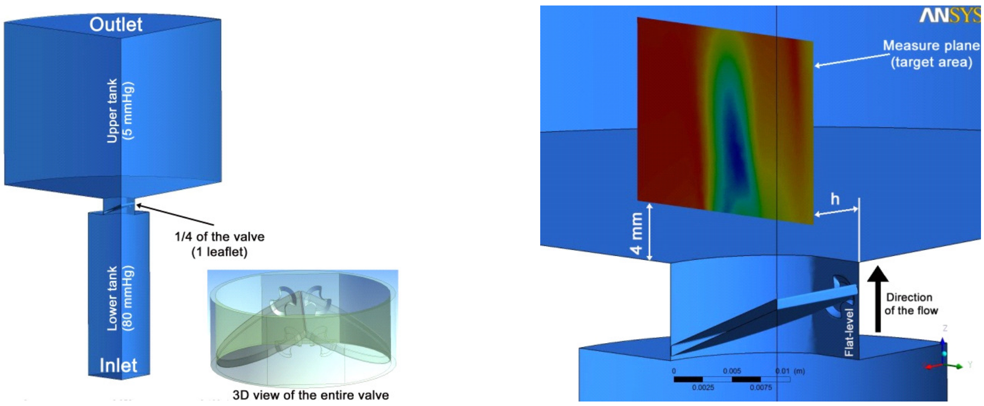
Correspondence: ally_ly_3@hotmail.com, Str. Memorandumului nr. 28, 400114, Cluj-Napoca, Romania

Introduction

The most widely implanted prosthetic valve design is the mechanical bileaflet, the proper functioning of which requires the use of a hinge mechanism [1]. However, clinical experiences illustrate the importance of understanding the leakage jets and the flow structures generated within the hinge. Numerous experimental studies of the hinge flow fields have been done in the past [2] and still continues nowadays [3]. Less numerous seem to be the 3D simulation studies of the hinge flows and their correlation with the experimental studies [4]. The purpose of our study was to attempt a cross-validation between the numerically evaluated near-hinge-flow fields of a bileaflet valve and the Particle Image Velocimetry (PIV) data acquired by Istituto Superiore di Sanita (ISS), Italy, on the same valve [3]. That could be a proof of the accuracy that the Computer-aided Design (CAD) model of the valve has been reconstructed with. A reasonably good accuracy would allow for a later Computational Fluid Dynamics (CFD) insight into the hinge flow field itself, where the PIV experiment didn't.

Materials and Methods

To investigate its leakage flow field, a 24 mm diameter bileaflet valve was mounted in the steady-flow experimental in vitro chamber designed by ISS. The 75 mmHg of pressure applied across the valve kept it tidily closed. Using PIV, the corresponding leakage flow velocity was measured across several 2D slices situated 4 mm away from the flat plane of the hinge and at different heights h above it (fig. 1b). The whole experimental setup is described in [3]. After taking the PIV measurements, the valve was dismantled and scanned, thus getting an accurate 3D representation of its geometry. The CAD model of the valve was completed with the in-vitro chamber's model and then a steady-state flow model was implemented in ANSYS-CFX. Only one quarter of the model has been simulated. The density and dynamic viscosity of the fluid were $\rho = 1g/cm^3$ and $\mu = 1cP$ respectively.



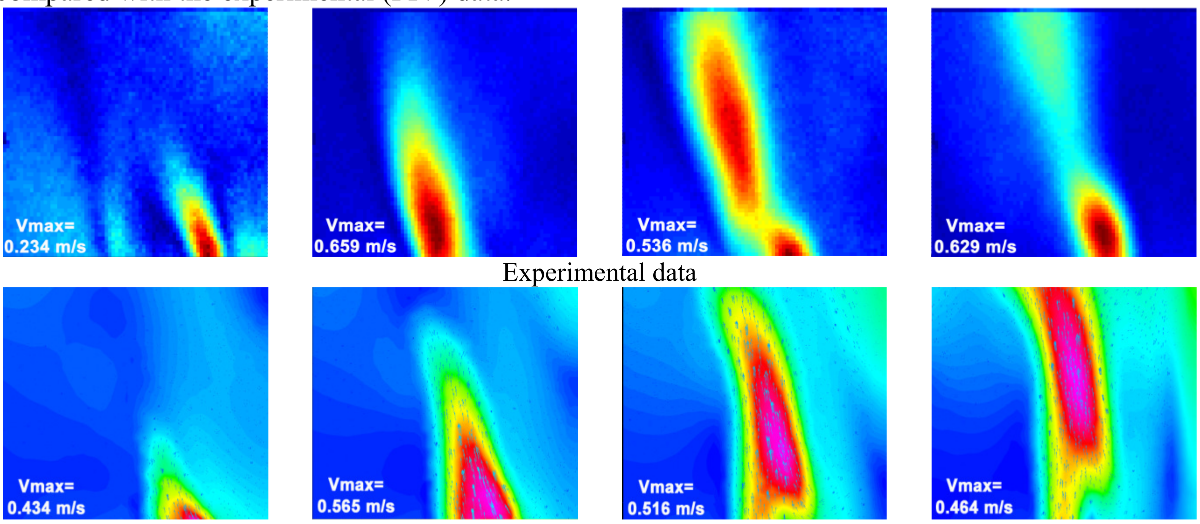
a) b)

Figure 1: a) geometry and boundary conditions, b) detail of the flat-plane region of the hinge and the position of the target (measure) area relative to the hinge

Given the narrow gaps between the bottom of the hinge recess and the tip of the leaflet ear (150 µm) and between the two leaflets in the b-datum plane (10 µm), Reynolds number values greater than 2000 and a fully turbulent flow is expected to have there. Therefore, one of the turbulent models available under CFX v13 had to be chosen. Similarly to the use of the FDA Benchmark Nozzle model for supporting the validation of CFD simulations [5], an ISS Benchmark Nozzle model was used for selecting the best suited turbulence model. Two of the turbulence models available under CFX v13, k- ε and SST (Shear Stress Transport) have predicted the velocity field with errors less than 10%. Eventually, the k-ε model with 5% inlet turbulence intensity was used for the valve simulation. Figure 1 shows ¼ of the CAD model of the experimental chamber with the valve included and a detail of the hinge area (the total number of elements in the entire mesh was of 6,062,146). Fig. 1.a shows the applied boundary conditions (75 mmHg at inlet and 0 at the outlet) whereas fig. 1.b shows the position of the target (measure) area relative to the hinge.

Results and Discussion

Numerical simulation data in the form of the velocity distributions within the target areas situated at different heights *h* above the flat plane level and corresponding to the left side of the valve are compared with the experimental (PIV) data.



h=1 mm h=3 mm h=5 mm h=7 mm

Figure 2 Comparison between the experimental and simulation data at different distances to the flat plane level (different *h*). The maximum in-plane velocity of each slice is indicated

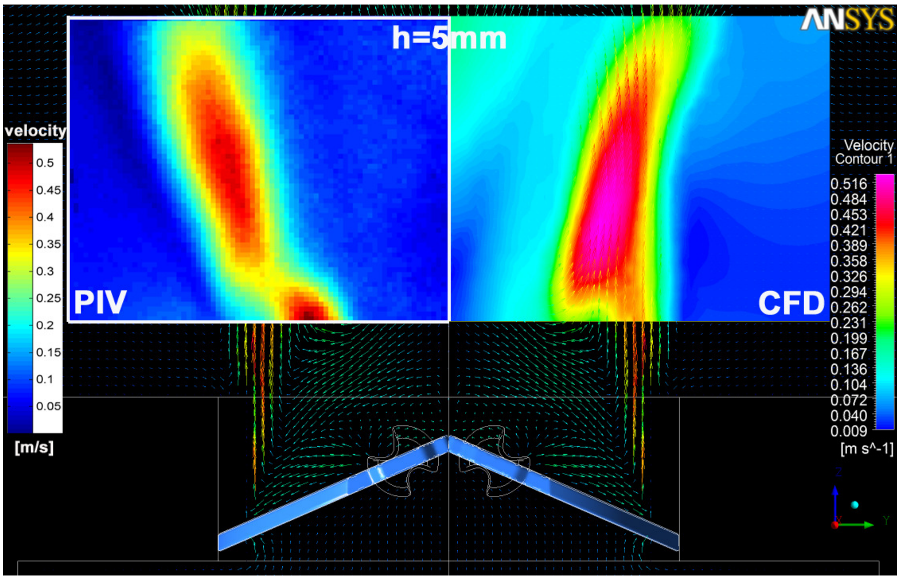


Figure 3 Contour and vector plots of the velocity at 5 mm above the hinge plane

Both the PIV and the CFD data show a single leakage jet. The exact source of this jet is yet unknown but, most probably it is the combined effect of the three gaps (critical zones) identified by Manning [6]. Some differences were found between the spatial distributions of the jets and between the maximum PIV and CFD velocities. As the CFD analysis gives us more flexibility, we tried to see how close to the close position our “numerical valve” was compared to the “real valve”. Initially, the only degree of freedom of the leaflets during their movement that we considered in our CFD analysis was the rotation around a pre-defined axis. Now, we added two more degrees of freedom (displacements along y and z). That allowed us to get a better match between the numerical and the experimental data. The overall mismatch between the CFD and the PIV results lies between 4% and 85%, depending on the distance *h* and the position within each slice. The greatest errors were found in the first slice (*h*=1mm). Nevertheless**, t**he CFD analysis allowed us to better trace the source of the leakage jets and to make a correlation between the measured and the calculated velocities. Despite the differences between the experimental and numerical values of the velocity, figure 3 shows a reasonably good correlation between the velocity vector plot around the target area and in the target area itself. The most prominent seems to be the peripheral leakage jet and there is no central leakage jet.

Conclusion

The differences still existing between the CFD and the PIV results might have been caused by the differences between the experimental and the numerical closed position of the leaflets. Therefore, our numerical model is currently improved by adding more degrees of freedom to the leaflets, not just the rotation and the two displacements but also the other three degrees of freedom as well. We specify that both the PIV measurements and the CFD analysis have been done in steady-state so that any inertial effects could be ignored. A full CFD-FSI which is currently under go will be able to quantify the inertial effects.

Acknowledgements

The research leading to these results has received funding from the European Union Seventh Framework Programme (FP7/2007-2013) undergrant agreement n° 238113 (project 'MeDDiCA' - 'Medical Devices Design in Cardiovascular Applications').

References

1. Ellis JT, Yoganathan AP. A comparison of the hinge and near-hinge flow fields of the St. Jude Medical hemodynamic plus and Regent bileaflet mechanical heart valves. *J Thorac Cardiovasc Surg* 2000;119:83-93

2. Simon HA, Leo HL, Carberry J, Yoganathan AP. Comparison of the hinge flow fields of two bileaflet mechanical heart valves under aortic and mitral conditions. *Ann. of Biomed. Eng,(32)/12, 2004; 1607-1617*

3. Wang GL, D’Avenio G, Daniele C, Grigioni M. Spatial Correlation in Mechanical Heart Valve Leakage jets. *IFMBE Proc.* (36)/2:128-135, 2011

4. Simon HA, Ge L, Sotiropoulos F, Yoganathan AP. Simulation of the three-dimensional hinge flow fields of a bileaflet mechanical heart valve under aortic conditions. *Ann. of Biomed. Eng,(38)/3, 2010; 841-953*

5. Hariharan P, et al, Multilaboratory particle image velocimetry analysis of the FDA benchmark nozzle model to support validation of computational fluid dynamics simulations. *J. Biomech. Eng. 2011, Apr; 133(4) :041002*

6. Manning KB, Kini V, Fontaine AA, Deutsch S, Tarbell JM. Regurgitant flow field chatacteristics of St. Jude bileaflet mechanical heart valve under physiologic pulsatile flow using particle image velocimetry. *Arti*ficial *Organs, 2003; 27(9):840-846*