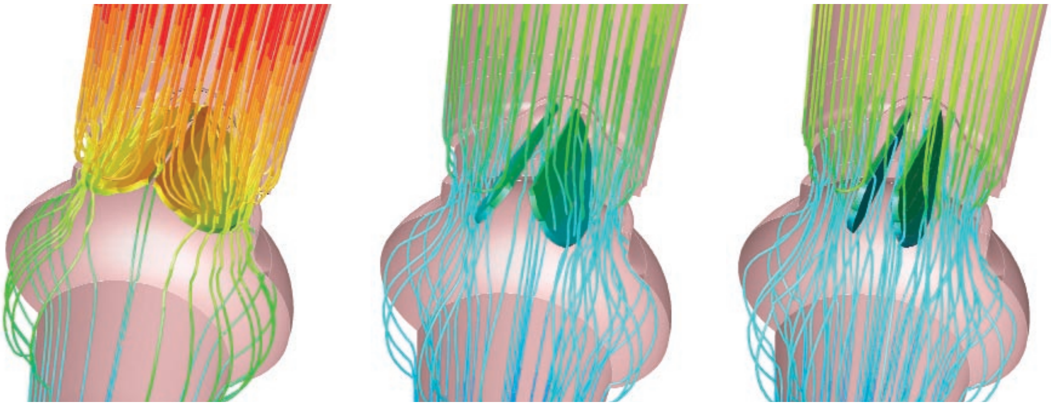
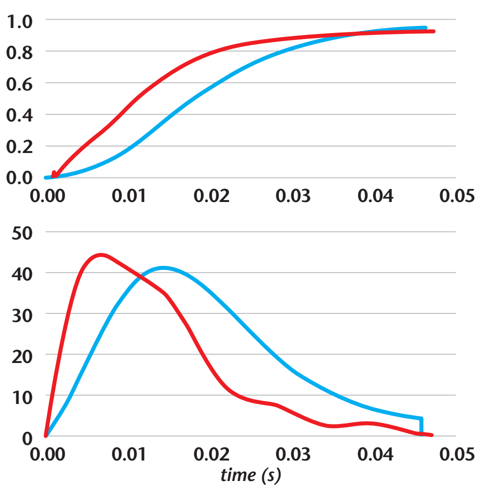
Heart Valve **Dynamics**

By Helene Bothorel[1], Emiliano Votta[1], Monica Soncini[1], Umberto Morbiducci[2], Costantino DelGaudio[2], Antonio Balducci[2], Mauro Grigioni[2], and Alberto Redaelli[1]
[1] Dept of Bioengineering, Politecnico di Milano, Italy
[2] Laboratory of Biomedical Engineering, Istituto Superiore di Sanità, Rome, Italy



Pathlines through the aortic valve during the opening phase at opening angles of 37.5°, 63° and 80°, from left to right; pathlines are colored with respect to pressure values

The "Opening Valve Project" is a collaborative effort being carried out by three organizations to study the behavior of a mechanical aortic valve. The Department of Bioengineering at Politecnico di Milano in Italy is performing CFD simulations, the Italian National Institute of Health (Istituto Superiore di Sanità, or ISS) is carrying out experimental tests, and St. Jude Medical Center in Fullerton, CA is providing technical support.



Angular displacement (above) and angular velocity (below) during the opening phase showing a comparison between experimental (red) and numerical (blue) results

To date, the investigation has focused on a St. Jude Medical HP mechanical aortic valve, 27 mm in size, mounted in a mock physiological flow loop designed by Vivitro Systems of Canada. Two rigid leaflets are mounted on hinges at opposite sides of the aortic cavity. Depending on the varying pressures upstream and downstream of the valve, the leaflets change from completely closed to fully open positions.

The dynamic mesh model in FLUENT has been used for the CFD simulations. To account for the fluid structure interaction (FSI) that is an important component of the valve motion, a user-defined function (UDF) was developed. The UDF computes the forces acting on the valve leaflets at the end of each time step, then calculates the difference between the prescribed leaflet motion and that corresponding to the computed forces. This information is used to forecast the correct motion of the leaflets in the subsequent time step on the basis of a relaxation scheme. To avoid flow field instabilities, a small relaxation constant is used. The method was first tested and refined using a 2D model, and has subsequently been applied to a 3D case.

Preliminary experimental tests were performed at ISS using ultrafast cinematography. These tests provided the transient boundary conditions (inlet and outlet pressure) for the numerical simulations. A mesh of about 320,000 cells was used along with a time step of 0.02 ms to allow for optimal remeshing and accuracy of the results. Simulations were performed on a single processor Intel Pentium PC (1.7 GHz with 2GB RAM) and took about 150 h of CPU time.

The CFD results show a reasonably good correlation with experimental measurements of leaflet motion (angular displacement) and angular velocity during the opening and closing phases. The simulated angular displacement is delayed with respect to the experimental data by about 7 ms. This value is considered acceptable with respect to the opening phase time lapse (45 ms). A better agreement with experimental data is observed for the predicted angular velocities, which are just slightly underestimated (7%). These results suggest that the dynamic mesh model, in conjunction with the FSI customization in FLUENT, is reliable for valve opening simulations when an accurate description of the valve kinematics is not required. ■