# FLOW IN AN ARTERIOGRAPHY-BASED CORONARY ARTERY NETWORK . COMPARAISON BETWEEN CFD AND SIMPLIFIED MODELS

**L. Dumortier*, F. Wilquem*, C. Hirsch** and J.M. De Smet*****

** Numeca Int., av. F. Roosevelt 5, 1050 Brussels, Belgium.*
***Dept. Fluid Mechanics, Vrije Universiteit Brussel,*
*Pleinlaan 2, 1050 Brussels, Belgium.*
**** Erasme Hospital, Université Libre de Bruxelles*
*Route de Lennik 808, 1070 Brussels, Belgium .*

Heart diseases and in particular arterial blockage are among the main causes of deaths in the western countries. A surgical procedure that consists in including bypass grafts is usually used in order to keep a proper irrigation of the heart. This solution is not perfect and failures may occur. It is commonly admitted that eventual problems are strongly related to the blood flow structure in the arteries In particular, reduced or negative wall shear stress, which can be related to vortices, are favorable for the development of atheromatous plaques. It is therefore fundamental to be able to track precisely such fluid phenomena. A biphasic and predominantly diastolic pattern characterize physiological flows in this context. More complex issues are related to the competition between flows in the graft and the stenosed native artery, the size discrepancy or the angle between the graft and the artery.
The aim of the present study is to compare the steady and unsteady numerical results obtained on a real coronary artery network to reported theoretical works carried out on simplified geometries. This ensures that the numerical simulation is able to deal precisely with the physiological flow structures. The configuration model, shown in Fig.1, includes a native artery, which is completely obstructed. The bypass is made of a saphenous vein graft (SVG) attached proximally end-to side to the aorta and distally with an end-to-side anastomosis to the coronary artery. The SVG is also subjected to a severe stenosis at 2 cm from its distal end.

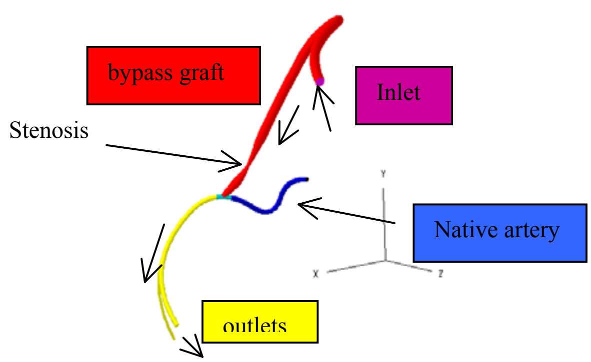


**Figure 1: Computational domain.**

## MATERIALS AND METHODS

### Numerical method

Computational Fluid Dynamics (CFD) methods and in particular the commercial code FINE from NUMECA has been used. The geometry and mesh have been generated using the

interactive grid generator block structured IGG™ v. 4.2 on the basis of an arteriography provided by the ERASME Hospital in Brussels. The governing Navier-Stokes equations of the blood flow have been solved using a second order cell-centered finite volume approach. The results have been extracted using the visualization tool CFView™.
A set of time-accurate physiological conditions that reproduce the flow in the vessels of the patient has been selected. For the SVG, a velocity profile has been set at inlet on the basis of a Berne and Levy profile[1]. The mean and maximum Reynolds numbers at this location are 106 and 203, respectively. The Womersley frequency parameter is set to 5.58, based on the bypass inlet diameter, the inlet maximum velocity, the mean velocity in the bypass and the heart pulsation. At the outlet, a static pressure profile is imposed as a reference. Turbulent calculations have been carried out in order to track the generation of turbulence in the stenosis area within the SVG.

**Comparison with experimental data**
Reported studies[2-4] include both experimental and theoretical approaches to the flow in vessels. The experiments were carried out using water in vessel models of clear polyester resin. The pressure drop across the constriction was measured by the piezometer columns connected to wall taps 12 in. apart z=±6 in. The streamlines, separation and reattachment points were visualized by injecting aniline blue dye in the separation region.

## RESULTS

Among the numerous results extracted, the flow at the stenosis is shown to be in agreement with theoretical and experimental[2-4] flow predictions, regarding the flow physics (Fig.2), turbulence intensity in the steady case.

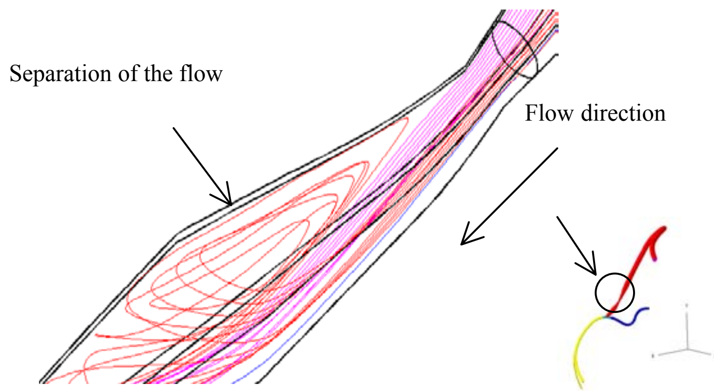


**Figure 2: Streamlines at the stenosis (steady case).**

The pressure drop across the constriction has been compared to the theoretical equation[2] for a similar shape (mild axisymmetric stenosis).

| | Pressure drop |
|---|---|
| Theoretical (Young & Tsai) | 59.36 |
| Calculated | 63.95 |

**Table 1: Comparison of the theoretical & calculated pressure loss through the stenosis.**

The difference of 7% can be mainly explained by the geometrical difference of the stenosis shapes and also by the location of section for pressure calculation downstream the stenosis in the post-processing with CFView$^{TM}$.

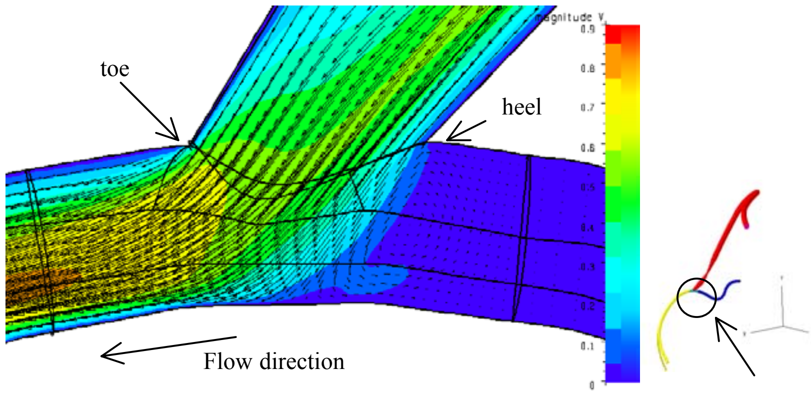


**Figure 3:Velocity field and magnitude (range 0-0.9 m/s) at the anastomosis (plane in the middle of the vessel).**

At the anastomosis and further downstream in the native artery, the flow path in the artery shows strong 3-dimensional counter-rotative structures. These structures are progressively damped because of viscosity effects before appearing again due to the curved shape of the artery. In addition, the unsteady calculations outlined the evolution of the separation regions in time enabled to nicely capture the natural phase-lag in accordance with the behavior described by Bartolotti[4].

## CONCLUSIONS

CFD simulations of the unsteady flow through an arteriography-based coronary artery network have been carried out using the FINE$^{TM}$/Turbo software developed by NUMECA. The flow structures induced by the full 3D nature of the network, including stenosis and anastomosis singularities, have been computed. The main flow characteristics outlined in simplified configurations have been compared and found in satisfactory agreement with the results of this study. Those preliminary steps in characterizing the flow disturbances in physiological situations should help and translate data already developed in theoretical and in vitro models to actual clinical situations. Optimization of the surgical techniques derived from this information should include selection of the size of the grafts, the location and angulation of the anastomosis in respect to the native stenosis, or the possibility of constructing multiple distal anastomoses with a single graft.

## REFERENCES


1. Berne, R. M., and Levy, M. N., *Cardiovascular Physiology*, Mosby, St. Louis, MO, p 201, 1967.
2. Young, D. F., and Tsai, F. Y., Flow Characteristic in model of arterial stenosis – I Steady flow, *J. Biomechanics*, Vol 6, Pergamon Press, pp. 395-410, 1973.
3. Young, D. F, Fluid Mechanics of Arterial Stenoses, *J Biomechanical Engineering*, Vol. 101, ASME,pp 157-175,1979.
4. Bartolotti, C., Modèles Numérique et Expérimental des Ecoulements dans les Pontages Coronaires, *PhD Thesis*, Université de la Méditerranée Aix-Marseille II, Ecole Supérieure de Mécanique de Marseille, 1999.