8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

THE GRID SCHEME EFFECT ON HEMODYNAMICS NUMERICAL ANALYSIS IN THE ILIAC ARTERIES

***Filipa Carneiro[1], Vasco G. Ribeiro[2], José C.F. Teixeira[1] and Senhorinha F.C.F Teixeira[3]**

[1] University of Minho,
Department of Mechanical Engineering
4800-058 Guimarães
Portugal
{afcarneiro;jt}@dem.uminho.pt

[2] Centro Hospitalar de Vila Nova de Gaia
4430-502 Vila Nova de Gaia
Portugal
vasco@chvng.min-saude.pt

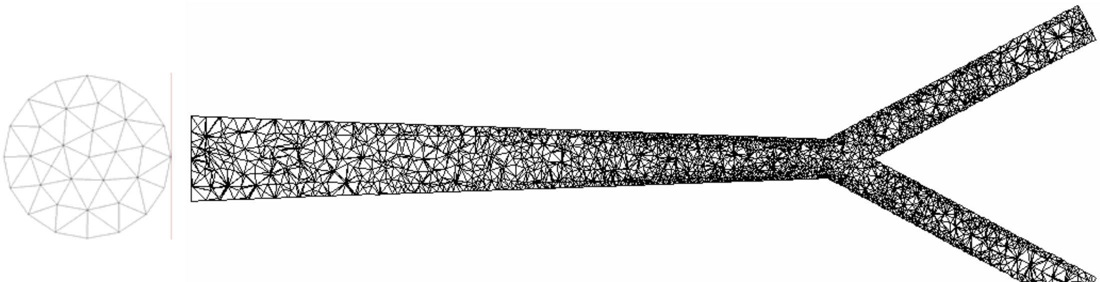
[3] University of Minho
Department of Production and Systems
4800-058 Guimarães
Portugal
st@dps.uminho.pt

Key Words: *Cardiovascular modelling, CFD, mesh optimization.*

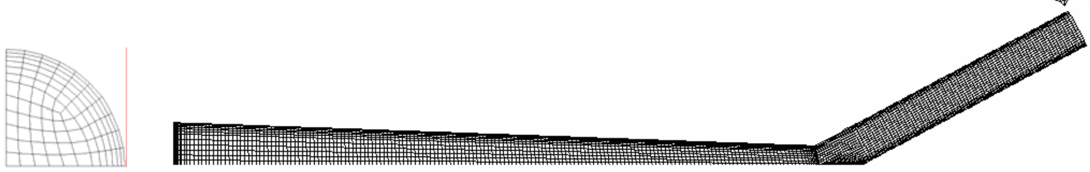
ABSTRACT

Blood flow patterns in abdominal aorta have been studied *in vitro* and numerically due to their important role in atherosclerosis localization. The reliability of any computational fluid dynamics (CFD) analysis is very dependent on an adequate domain discretization, and the element and scheme type will determine the results. The grid generation should capture the flow details in special the areas with high velocity gradients. The grid optimization is a very important step to subsequent undependable results (Liu et *al.*, 2004). This paper presents a comparison between two grids schemes upon the numerical analysis of a steady blood flow in a bifurcation model simulating the abdominal aorta.

The finite volume method requires the discretization of the domain into control volumes. The geometry and grid generation was performed with GAMBIT (Figure 1).



Grid 1



Grid 2

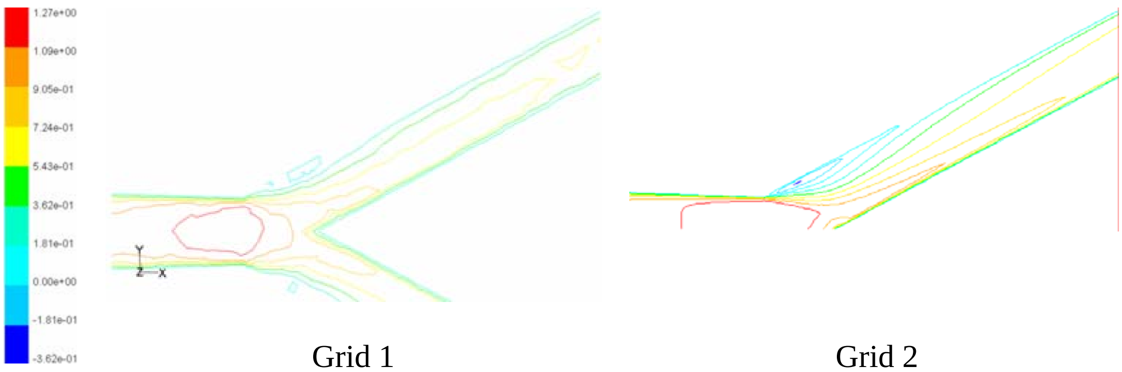
Figure 1. Computational grids for the domain in GAMBIT.

The aorta section is 165 mm long with an inlet and outlet radius of 11 and 5 mm,

respectively. The iliac arteries diverge with an angle of 56º and have a constant cross section of 5 mm in radius and a length of 75 mm. They both have the same geometric characteristics and mimic *in vitro* data obtained by this working group.

The first grid (Grid 1) was easily meshed with 32075 tetrahedral elements using the *Tgrid* sheme that can be easily applied to all types of geometry. The second grid (Grid 2) was created with 34960 hexahedral elements using the *Cooper* scheme, which projects the mesh of a face all over the geometry. This requires a better understanding of the method and the problem physics. In grid 2, boundary layers and refinements were created, where high velocity gradients are expected: near the wall throughout the entire domain and in the bifurcation region. To reduce the computational time, symmetry is assumed in the planes y=0 and z=0, reducing the computational domain by 75%. The inlet cross section and a z=0 section of these meshes of the iliac bifurcation at abdominal aorta are shown in Figure 1.

The FLUENT software was used to solve the 3D computational rigid model. A uniform inlet velocity of 0.234 m/s was assumed and the generated turbulence was simulated by the k-ε model with enhanced wall treatment. The blood was modelled as a Newtonian and incompressible fluid, obeying the Navier-Stokes equations. The comparison of the two grid schemes was based on the obtained velocity fields. The x velocity contours of grids 1 and 2 are shown in Figure 2, at a detailed plane z=0.



Grid 1 Grid 2

Figure 2. A detailed view of x velocity contours, at the plane z=0, in FLUENT.

The discontinuity in data along the axial direction, shown in Grid 1, suggests the need of further refinements and grid sensitivity studies, to guarantee mesh independence results. The hexahedral elements of the structured grid appear to guarantee a more detailed flow description. The velocity gradients near the wall seem more pronounced at grid 2. The extra effort in obtaining a more refined structured grid appears to achieve more realistic results, consistent with *in vitro* experiments.

REFERENCES

[1] Y. Liu, K. Pekkan, S.C. Jones, A.P. Yoganathan, “The effects of different mesh generation methods on computational fluid dynamic analysis and power loss assessment in total cavopulmonary connection”, *Journal of Biomechanical Engineering*, Vol. **126**, pp. 594–603, (2004).