CHANGE IN THE CALCULUS PARAMETERS OF THE FLOW INNER OF THE ELASTIC CYLINDER

Márcio Antonio Bazani, bazani@dem.feis.unesp.br
Amarildo Tabone Paschoalini, tabone@dem.feis.unesp.br
Unesp, Avenida Brasil 56, Ilha Solteira S.P., Brasil, CEP:15385-000

Rafael Henrique Novaes, rh_novaes@gmail.com
Unesp, Avenida Brasil 56, Ilha Solteira S.P., Brasil, CEP:15385-000

Júlio Tadashi Tanaka, jttanaka@fatectq.br

Abstract. *This work shows a numerical solution of the flow inner of the elastic cylinder, considering the fluid structure interaction. The fluid is assumed incompressible, Newtonian and is governed by Navier Stokes equations. The elastic structure is assumed homogeneous, isotropic and is governed by Navier equation. The mathematical model was solved by software ANSYS?, Release 9.0, that applies the finites elements method in separated domains (fluid and structure), with special attention at the interface. The purpose of this study is to find optimal numerical result, changing parameters used in the simulation such as: relaxation factor, size of the grid and convergence. The numerical simulation results were compared with results obtained by the solution of the ordinary differential equations (The Navier-Stokes and Navier partial differential equations were transformed in ODE and solved by MATLAB) using Dormand Prince and Runge Kutta methods, and small variations were observed between the ANSYS and MATLAB results.*

Keywords*: Fluid Structure Interaction, Finites Elements Method, ANSYS*

1. INTRODUCTION

The term “fluid–structure interaction” refers to either (1) fluids contained within structures, or (2) structures surrounded by fluids. Examples of (1) are the seismic response of ground supported, cylindrical liquid storage tanks, aerospace applications such as rocket fuel tanks, and the response of a dam due to sudden acceleration into a contiguous reservoir. Examples of (2) are the transient motion of submerged structures, flow-induced vibrations, normal and abnormal operations of light water reactor and pressurized water reactor cores. Significant attention has been devoted to the development of fluid-structure interactions procedures over several decades. However, many fluid-structure interaction problems are of such complexity that the method of analysis must be numerical in nature (Liu, 1980).

The ability to solve general classes of fluid-structure interaction involving finite deformations depends upon the ability to solve the corresponding uncoupled fluid and structural problems, and also the ability to “interface” fluid and structural sub domains.

FEM Platform ANSYS version 9.0 (ANSYS, 2000) is a widely used finite element program which utilizes numerical methods and the power of modern computers to solve equations of motion and state for both solid and fluid materials and is capable of being used to solve thermal, structural, electro-magnetic, and fluid problems. It is a self-contained program which has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions and commands. Additionally, users can define problem parameters and solution methods to a very high level of specificity with the ANSYS Parametric Design Language (APDL). ANSYS is capable of coupled-field analyses in which two material realms can interact realistically allowing for users to model the interactions between flowing fluid and a deformable object which protrudes into the flow stream.

This work focuses on the analyses of several convergence parameters used in the fluid structure interaction solution through the commercial software ANSYS 9.0 that uses the Finites Elements Method.

2. Computational model

The computational model was done in two separated domains (fluid and structure) in cylindrical geometry. To obtain a simple analytical solution, only radial direction was used in transient regime. In the interaction between these domains were considered boundary conditions at interface, involving force and velocities balance. The figure 1 shows the studied problem.

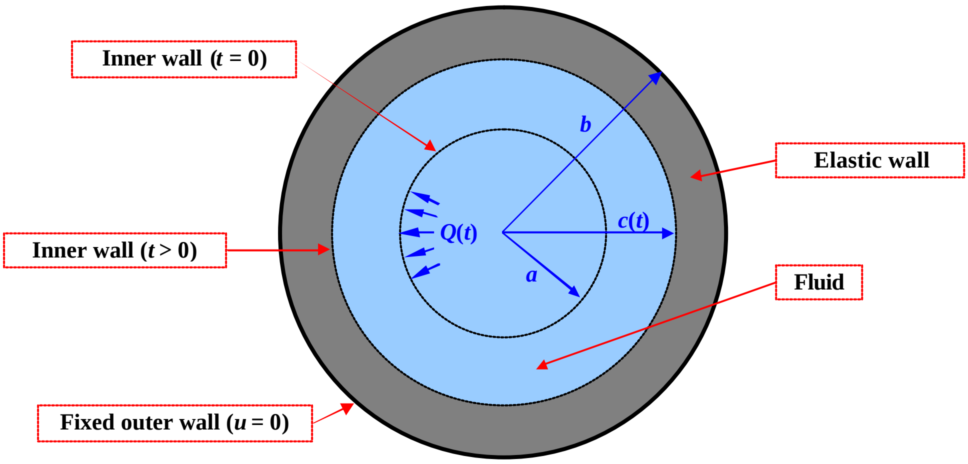

Figure 1 - Geometry of the problem.

The figure 2 shows the initial model of the problem using ANSYS. The blue areas (A1, A2, A3 and A4) represent the elastic wall, whereas the red area represents the flow fluid that interacts with the elastic wall.

After to define the control volumes, is necessary to define the material properties for each element. Ansys has a options range with information about the fluid and structure properties.

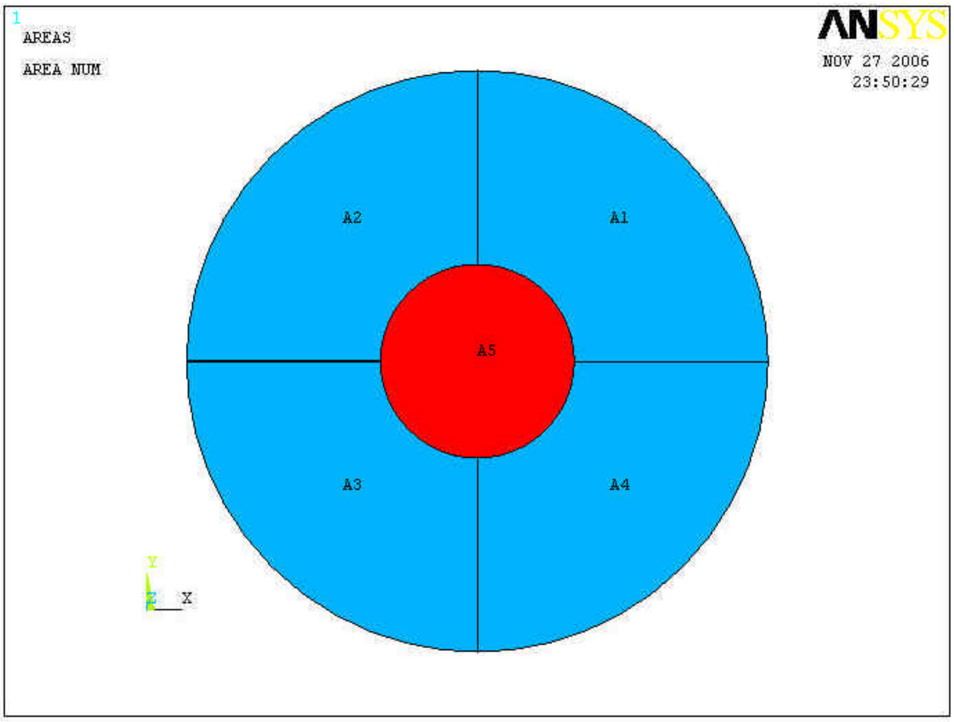

Figure 2 - Geometry of the problem in ANSYS.

2.1. Fluid dynamics in ANSYS

For this problem was used the FLUID141 that model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. Use this FLOTRAN CFD element to solve for flow and temperature distributions within a region, as opposed to elements that model a network of one-dimensional regions hooked together. You can also use FLUID141 in a fluid-solid interaction analysis. For the FLOTRAN CFD elements, the velocities are obtained from the conservation of momentum principle, and the pressure is obtained from the conservation of mass principle. (The temperature, if required, is obtained from the law of conservation of energy.) A segregated sequential solver algorithm is used; that is, the matrix system derived from the finite element discretization of the governing equation for each degree of freedom is solved separately. The flow problem is nonlinear and the governing equations are coupled together. The sequential solution of all the governing equations, combined with the update of any temperature or pressure dependent properties, constitutes a global iteration. The number of global iterations required to achieve a converged solution may vary considerably, depending on the size and stability of the problem. Transport equations are solved for the mass fractions of up to six species. The degrees of freedom are velocities, pressure, and temperature. Two turbulence quantities, the turbulent kinetic energy and the turbulent kinetic energy dissipation rate, are calculated if you invoke an optional turbulence model. For axisymmetric models, it was calculated an optional swirl - velocity VZ normal to the plane.

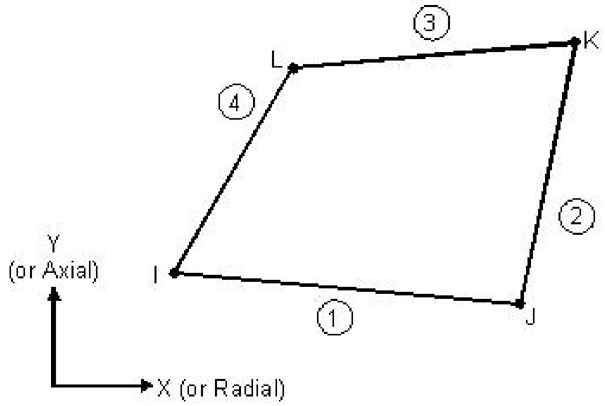

Figure 3 - Geometry of Fluid 141.

2.2. Solid Mechanics in ANSYS

The partial differential equations which govern strain deformation can be developed through the principle of virtual work (ANSYS, Inc., 2004). The method which ANSYS uses in its FEM divides the domain to be solved into several sub-regions called elements. Solutions to the differential equations for each element are developed while maintaining continuity at element boundaries. For this problem was used PLANE82 is a higher order version of the 2-D, four-node element. It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries.

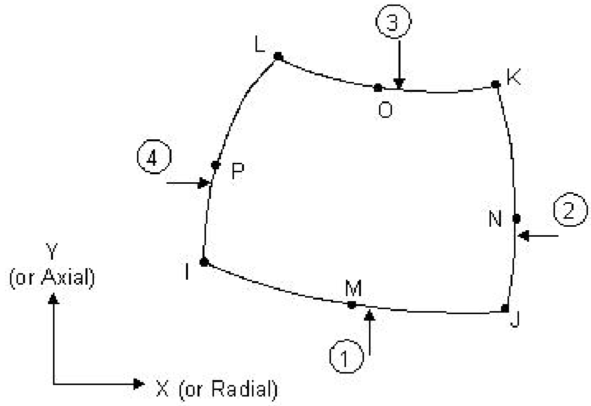

Figure 4 - Geometry of Plane 82.

The 8-node element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element or as an axisymmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Various printout options are also available.

2.3. Meshing

In any FEM, the fineness and distribution of the mesh which defines the elements is very important. In the two-dimensional FSI model, the mesh was finest near the inner wall where fluid flow was most affected by the inner wall and, as a result, special gradients of the flow field were highest. Mesh divisions were defined in the ADPL and can be altered to increase speed or accuracy. Mesh fineness was limited by the version of ANSYS which was used.

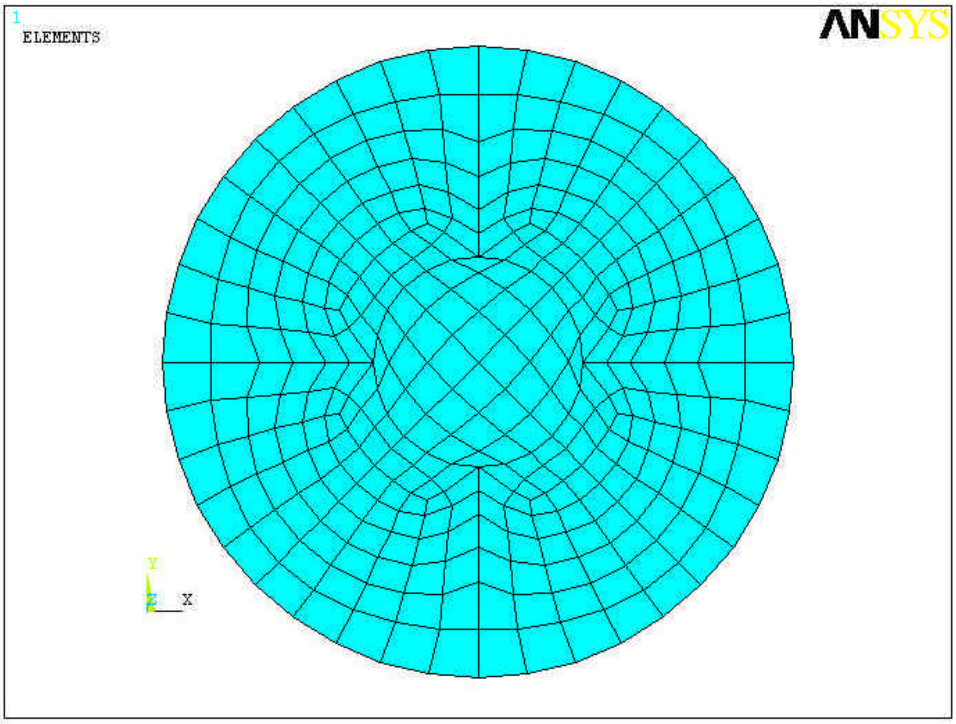

Figure 5 – Meshing of the studied model.

3. Numerical solution

A fluid-structure interaction analysis is carried out in ANSYS by creating two separate models, each governed by a specific set of material properties, and then coupling them across a defined interface. Coupling describes the process by which the solutions to the two different models are linked. An interface can be any similar or dissimilar mesh boundary between the two models. ANSYS uses a sequential coupling algorithm

When the Arbitrary Lagrangian-Eulerian (ALE) formulation is on, FLOTRAN allows the fluid nodes to move in a manner that satisfies the displacement boundary conditions. Then apply the boundary loads (pressure and shear forces) to the inner wall region. This process is carried out numerous times depending on the total time of simulation as well as the time step. Once the solid solution has converged, the fluid region is updated to reflect the new geometry of the solid. Then, the next fluid solution is determined, hence the “sequentially coupled” moniker.

The Fluid Structure Interaction tools used for this problem were: FSIN that specify the interface load transfer option for a fluid-solid interaction analysis. Non conservative formulation was used for load transfer; FSAN that activate a fluid-solid interaction analysis; FSOR that specify analysis order for a fluid-solid interaction analysis; FSTR that specify static or transient analyses for a fluid-solid interaction analysis; FSTI that set end time and load time for a fluid-solid interaction analysis and FSDT that set time step increment for a fluid-solid interaction analysis.

Some parameters were changed to compare with numerical results obtained by literature (Bazani and Tabone, 2006). These parameters are following: FSIT that set the maximum number of stagger iterations for a fluid-solid interaction analysis; FSCO set convergence values for a fluid-solid interaction analysis and FSRE that set relaxation values for a fluid-solid interaction analysis.

3.1. The studied case

The simulation of the fluid structure interaction between blood flows and arterial walls was done by ANSYS, considering that due to driven by a time-periodic pressure pulse caused by the contractions and relaxations of the heart muscle, blood flow interacts with the pulsation of arteries. When an artery become anastomosed (i.e. dilated) the affection is called aneurism. The modern goal of aneurysm surgery is to isolate the thin-walled aneurysm from arterial flow while maintaining the normal patency of the parent artery and adjacent branches. As a result, the modern neurosurgeon has a wide selection of biocompatible aneurysm clips with known closing pressure, of variable sizes and shapes, and a selection of clip applicators that do not obstruct the surgical field. Thus, this work simulates the behavior of the blood pressure in compliant arteries.

The parameter values used in the problem are following:

Outer Radius of the artery:	1.5 cm;
Inner Radius of the artery:	1.2 cm
Young's modulus:	10^5 N/m^2
Poisson ratio:	0.46
Fluid density:	1050 kg/m^2
Dynamic Viscosity :	0.004 Pa s
Initial time:	0 s
Final time:	1.2 s
Time Increments:	0.05 s

Blood flow and pressure are unsteady. The cyclic nature of the heart pump creates pulsatile conditions in all arteries. The heart ejects and fills with blood in alternating cycles called systole and diastole. Blood is pumped out of the heart during systole. The heart rests during diastole, and no blood is ejected. Pressure and flow have characteristic pulsatile shapes that vary in different parts of the arterial system going to zero when the aortic valve is closed. The aorta, the large artery taking blood out of the heart, serves as a compliance chamber that provides a reservoir of high pressure during diastole as well as systole. Thus the blood pressure in most arteries is pulsatile, yet does not go to zero during diastole. In contrast, the flow is zero or even reversed during diastole in some arteries such as the external carotid, brachial, and femoral arteries. These arteries have a high downstream resistance during rest and the flow is essentially on/off with each cycle. In other arteries such as the internal carotid or the renal arteries, the flow can be high during diastole if the downstream resistance is low. The flow in these arteries is more uniform. Thus the input pressure used in the simulations takes the following form: 9332.54*(1-cos(2*pi*t/0.6))

4. Results

The figure 6 shows the inner wall displacement due to the blood pressure. It was observed that occurs a displacement of 0,45 mm relatively to inner wall, i.e., it occurs a normal displacement about 15% of inner wall at conditions of high pressure for a human male. This result was compared with an analytic solution generated by MATLAB.

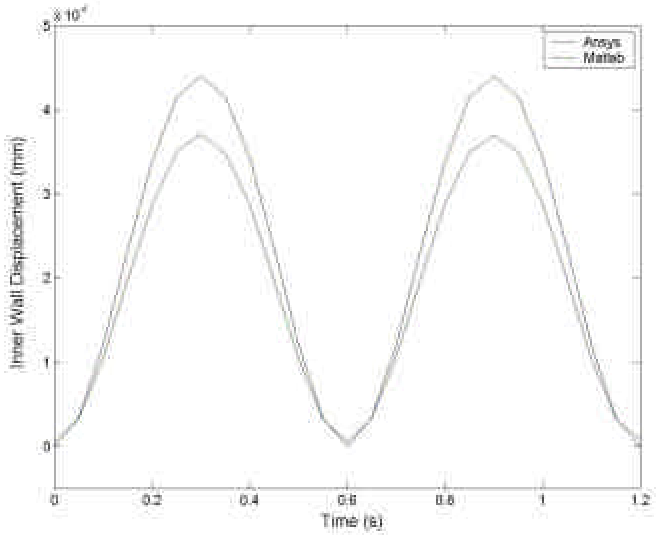

Figure 6 – Displacement calculated by ANSYS and MATLAB.

There is a dimensional difference of the inner wall displacement. This range is located between 0.36 and 0.45 mm (12% and 15%). It is important to elucidate that the visual difference is large in the figure 6 due to the magnitude of the dimensions. Thus, this real difference is very small showing that ANSYS has high accuracy. To reach this result, several calculus criterions were changed with an executions series in the simulation. The main points studied will be presented. FSCO specify convergence norm for quantities transferred across fluid-solid interface. The convergence is normalized: Conv = $\log (e_x/e_{min})/(\log e_{max}/e_{min})$, where e_{max} is specified.

ANSYS calculates the inner wall displacement and fluid displacement and at interface these values must be the same. The figure 7 plots UX fluid versus UX structure with FSCO = 0.1.

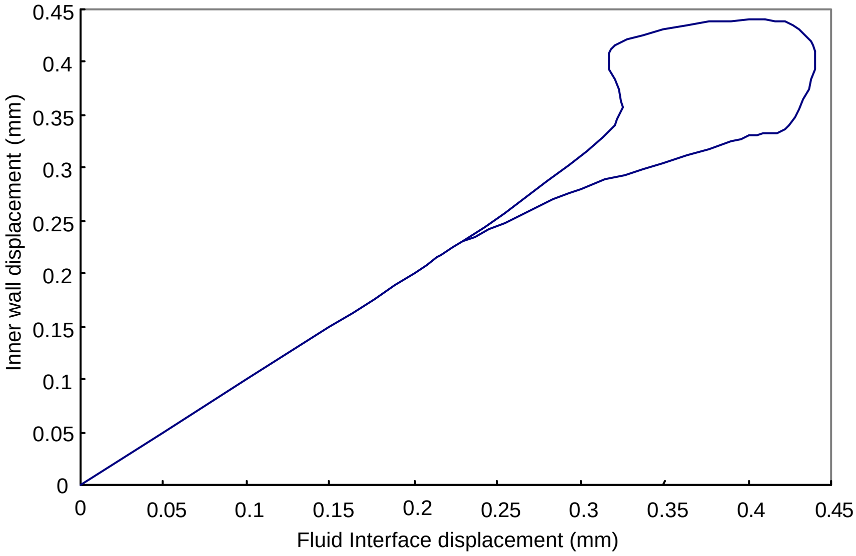

Figure 7 – displacement of interface for convergence factor 0.1.

The figure 7 presents an insufficient convergence value. There is not displacement coupled to leave from 0.23 mm.

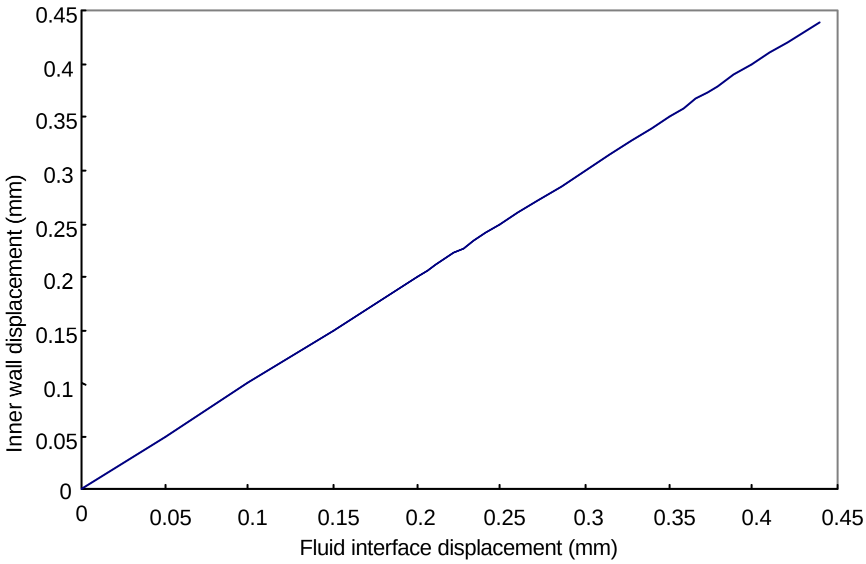

Figure 8 – displacement of interface for convergence factor 0.0001.

The figure 8 presents an unnecessary convergence value, because to exceed the minimal computational requirement to guarantee a good result, without additional improving. Thus the adopted value for FSCO is 1e-03, ten times the default value suggested by ANSYS.

Meshing is an integral part of the CAE analysis process. The mesh influences the accuracy, convergence and speed of the solution. More importantly, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. In fluid structure interaction problems, it is very common to use unstructured triangular meshes for the fluid and quadrilateral meshes for the structural problem. It is known that quadrilateral and hexahedra require less CPU time and memory whenever an edge based data structure is adopted for finite element method. Building good quality quadrilateral and mixed meshes is very challenging because quadrilaterals are very stiff from a geometric point of view. Therefore, the better and more automated the meshing tools, the better the solution.

The mesh was refined until nineteen thousand elements using a PC of RAM 512 MHz to run the simulations. It was observed by figure 11 that three thousand elements are sufficient to reach convergence.

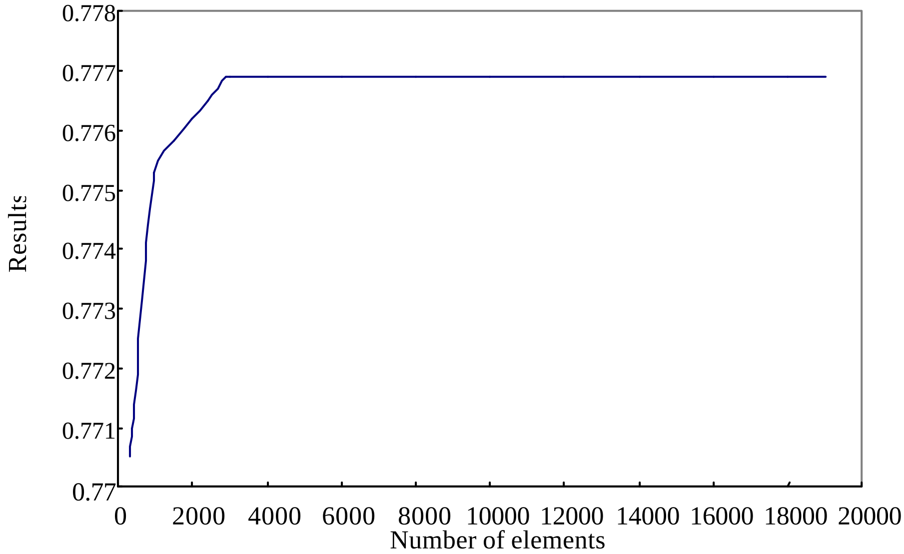

Figure 19 - Results versus number of elements

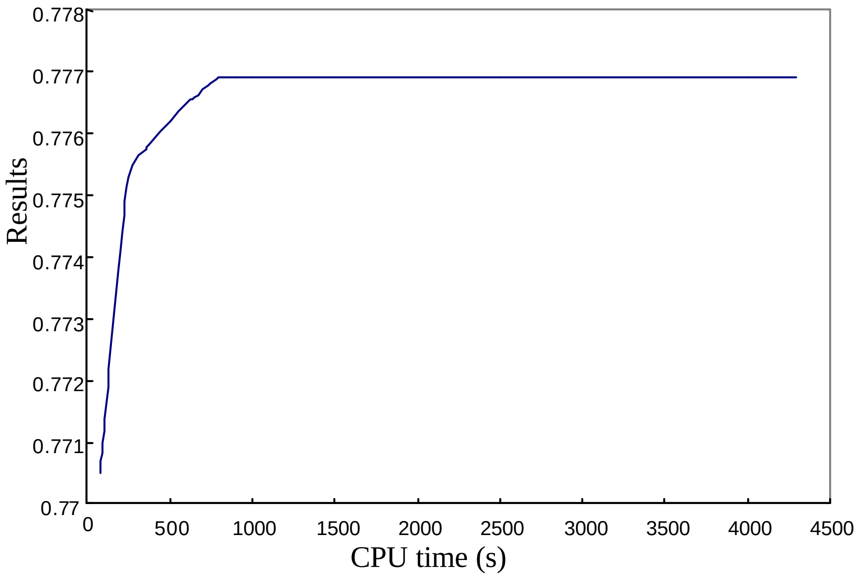

Figure 10 – Results versus CPU Time

The figure 12 shows the influence of CPU time in results. It was took 900 CPU s, time enough, to reach the convergence. One set of most significant parameters affecting the convergence of a numerical scheme is the under-relaxation factors. Some earlier work has been done to optimize these parameters. However, these previous works are restricted to special flow domains and the range of changes for under-relaxation factors and convective algorithms are limited. The values of the relaxation factors around 0.5 and 0.8 have been found to be satisfactory in a large number of fluid flow computations. However, it is not implied that these values are the optimum ones or will even produce convergence for all problems. It should be recognized that matters such as the optimum relaxation factor values are usually problem dependent. Although experience from previous computations is helpful, new problems sometimes require different relaxation practices. The displacements results are obtained changing the relaxation factor for displacement and can be compared with MATLAB results. The figures 11, 12 and show the inner wall displacement at respective relaxation factors: 0.5, 0.8 and 1.0. Finally, we can conclude that the ANSYS results are near to MATLAB solution when FSRE =1.

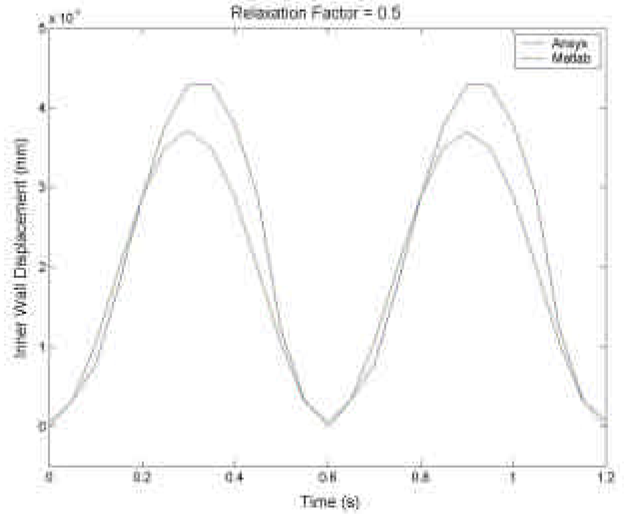

Figure 11 – FSRE = 0.5

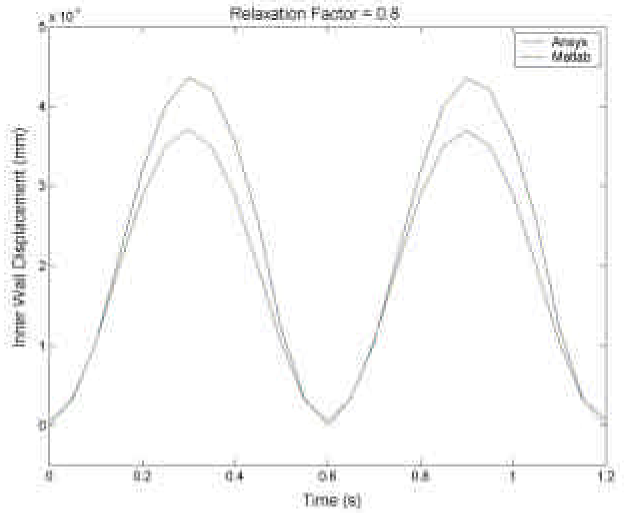

Figure 12 – FSRE = 0.8

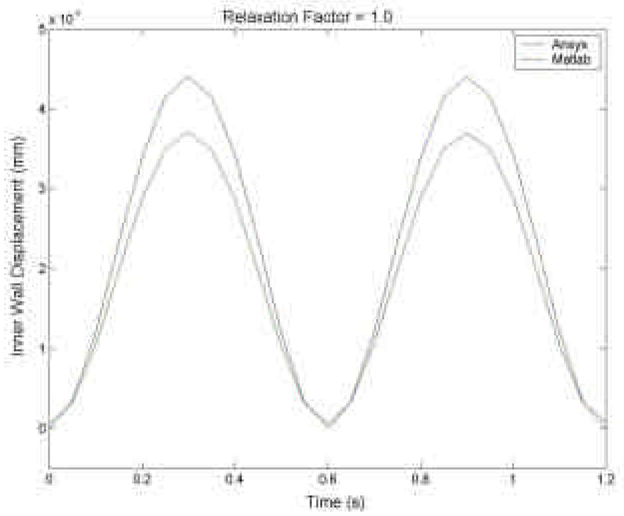

Figure 12 – FSRE = 1.0

4. Conclusion

This work has evaluated and to improved the main calculus parameters of a coupled time-dependent flow and the deformation in a cylindrical geometry in situations similar of an artery anastomosed (i.e., dilated).

The results obtained were compared with general resulting ODE system, by the Dormand-Prince and Runge-Kutta for 4^o /5^o order methods was done in MATLAB (Tabone and Bazani, 2006) and was observed small variations. It is important to note that ANSYS and Matlab use different techniques to solve the fluid structure interaction problem. With the reduced computational time, this simulation predicts local hemodynamics, wall shear stress and wall displacements to the higher order accuracy and can thus be used as a better predictor of possible pathologies in the cardiovascular function.

5. REFERENCES

ANSYS R.9.0, 2000. Ansys Inc.

Bazani, M.A., Paschoalini, A.T., Feijó, A.T., Tanaka, J.T., 2006, "Análise numérica bidimensional do escoamento no interior de cilindros elásticos", Proceedings of the XXVII CILAMCE, Recife, Brazil.

Bounaim A., 2001, "Fluid-Structure Interaction: A Simple Test Case", Scientific Computing Group, Department of Informatics, University of Oslo.

Bounaim A., 2001, "Numerical Simulation of Blood Wall Interaction in the Human Left Ventricle", Scientific Computing Group, Department of Informatics, University of Oslo.

Dormand, J. R. & P. J. Prince, P.J., 1980, "A family of embedded Runge-Kutta formulae", J. Comp. Appl. Math., Vol. 6, pp 19-26.

Liu W. K., 1980, "Development of Finite Element Procedures for Fluid Structure Interaction", A Report on Research Conducted Under Grants from the National Science Foundation and the Electric Power Research Institute, Pasadena, California.

MATLAB for Windows User's Guide, 2000. The Math Works Inc..

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.