Development of a blood flow model and validation against experiments and analytical models

EINDHOVEN UNIVERSITY OF TECHNOLOGY
Department of Mathematics and Computer Science

CASA-Report 11-48
September 2011

Development of a blood flow model and validation against experiments and analytical models

by

M.H.A. van Geel, C.G. Giannopapa, B.J. van der Linden

Centre for Analysis, Scientific computing and Applications
Department of Mathematics and Computer Science
Eindhoven University of Technology
P.O. Box 513
5600 MB Eindhoven, The Netherlands
ISSN: 0926-4507

DEVELOPMENT OF A BLOOD FLOW MODEL AND VALIDATION AGAINST EXPERIMENTS AND ANALYTICAL MODELS

M.H.A. van Geel* **C.G. Giannopapa†** **B.J. van der Linden‡**
Dept. of Mathematics and Computer Science
Eindhoven University of Technology
PO Box 513, 5600 MB Eindhoven
The Netherlands
Email*: m.h.a.v.geel@student.tue.nl
Email†: c.g.giannopapa@TUE.nl
Email‡:B.J.v.d.Linden@TUE.nl

ABSTRACT

In cardiovascular research, FSI is expressed by the interaction of the blood with the vessel or the heart. FSI plays a crucial role when the deformation of the boundary, in this case the vessel wall, cannot be neglected. Arterial blood flow and wave propagation in liquid filled vessels has been investigated by many researchers. Their work comprises computational, theoretical and experimental investigations and will be outlined below.

This paper presents the development and validation of an arterial blood flow model. The model has been developed using finite elements and the fluid and the solid are coupled using the ALE method. This method allows moving boundaries without the need for the mesh movement to follow the material. In this paper both straight and tapered aortic analogues are included in the investigation. The pressure, pressure gradient, fluid flow and wall distension obtained from the finite element model is compared with an unique experimental data set and analytical theory. There is a good agreement between the computational, analytical and experimental results.

INTRODUCTION

Fluid-structure interaction (FSI) occurs in many areas of engineering, aerospace, civil or mechanical, as well as other disciplines including medicine. The term FSI is used to describe the influence of fluid properties, like pressure or temperature, on solid properties and vice verse. FSI plays a crucial role and cannot be neglected when the deformation of a solid boundary affects significantly the fluid behavior and crucially vice versa.

FSI is an important modeling aspect in cardiovascular research in order to understand arterial blood flow. In FSI terms the blood is the fluid and the vessel or heart is the structure. This interaction becomes important when the heart beats and a volume of blood is introduced into the vessel. The vessel then has to expand to accommodate for the change in volume. Due to this expansion of the vessel, the fluid boundaries are being altered which leads to changes in the velocity and pressure of the fluid.

In liquid filled flexible vessels FSI plays a significant role in determining the wave propagation since the fluid normal and shear stresses act on the structure which leads to deformations. Wave propagation in liquid filled vessels has been studied over 200 years by performing experiments and developing analytical and computational models. Experiments have been performed in order to understand wave propagation and to validate analytical and experimental models. The experiments can be categorized as *in-vivo* [1–3] and *in-vitro* [4–9] experiments.

Analytical models have been derived to investigate wave propagation theoretically. Witzig and Womersley have been the pioneers in the field of analytically modeling pulsating blood flow according to Cox [10] but also Korteweg [11] has performed work which is widely referenced in this area. Korteweg already used fluid properties to calculate the wave speed in an elastic tube in the nineteenth century. This equation is now known as the Moens-Kortweg equation.

FSI equations were computationally solved for the first time in the 1970s with the introduction of computers. Complicated two-dimensional and three-dimensional problem were solved using finite element or finite volume methods [12–17].

The numerical solution of FSI involves solving two distinct problems. One for the fluid and another for the solid. Their interaction can be accounted for in various methods [18]. Figure 1 show the following approaches:

Non-iterative over all time In this method the fluid and solid equations are solved separately for the whole time domain. First the fluid is solved and then the solid is solved using the pressure from the fluid solution as a boundary condition.

Iterative over all time This method is similar to the non-iterative over all time method. But in this case the solution of the solid is now used to determine the boundary condition for the fluid. The fluid is now solved again and from the solution a new pressure at the boundary of the solid is obtained and used as a boundary condition in turn to solve again the solid equations. This process is repeated until convergence.

Iterative over each time step In a single time step the fluid equations are solved. The pressure solution now becomes the boundary condition for solving the solid equations. After the solid equations are solved, the solution obtained is returned as a boundary condition for the fluid. The fluid equations are solved again. This process is repeated until the system converges for this time step and further proceed with the next time step.

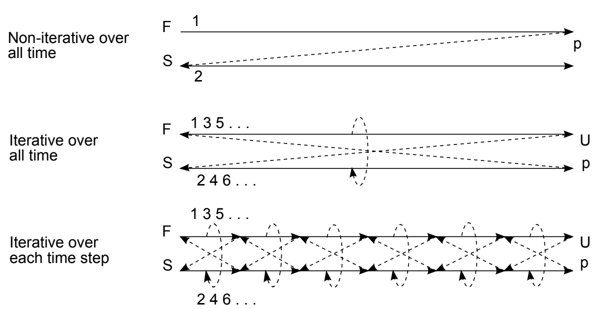


Figure 1. Solution procedure of three FSI methods.

Comsol is a finite element modeling package used for the simulation of physical processes that can be described with partial differential equations. The aim of this paper is to present a model developed in Comsol that is able to simulate wave propagation in the aorta. In Comsol the *Iterative over each time step* method is used. The geometrical dimensions and physical properties for both the fluid and solid are the same as the ones used in [6]. This will allow the comparison of the finite element model developed in this paper with the analytical and experimental data presented in [6].

EXPERIMENTAL SET-UP

The experimental data of Giannopapa [18] will be used to validate the computational model that has been derived. In Fig. 2 a schematic drawing of the experimental set-up can be found. Here C is a closed container at a constant pressure of $+1$ bar, D and E are open containers and F is a polyurethane tube which is pre-strained axially to 3% to keep the vessel straight when it is filled with water. The three way valve B is opened by a computer (PC) and valve A is operated manually. The vessel is fixed to container E and to the three way valve B. When the valve is not engaged the water column level inside the open tank D prescribes the initial pressure inside the vessel. By engaging the solenoid valve it opens for 50ms and generates a pulse. This pulse was taken as short as possible to be able to distinct backward and forward traveling waves and to keep the stationary pressure rise during the experiment as low as possible. The pressure gradient was measured simultaneously by two pressure sensors which were placed 17mm apart and the pressure was measured at 10 positions along the tube, $z = [0.03, 0.05, 0.1, 0.15, 0.2, 0.25, 0.3, 0.35, 0.4, 0.446]$ m. The volumetric flow was measured using a flow rate sensor, Q, and the wall distention was measured using an ultrasound probe, W.

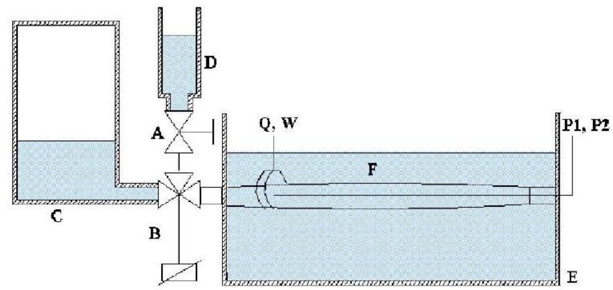


Figure 2. The experimental set-up [18].

The vessels

In this experimental set-up three different polyurethane vessels, two straight ones and one tapered one, are used. The vessels were designed to be analogues of the human aorta and were manufactured manually by spin coating. In Fig. 3 the three vessel can be found. The density of the vessels is $880\,kg/m^3$.

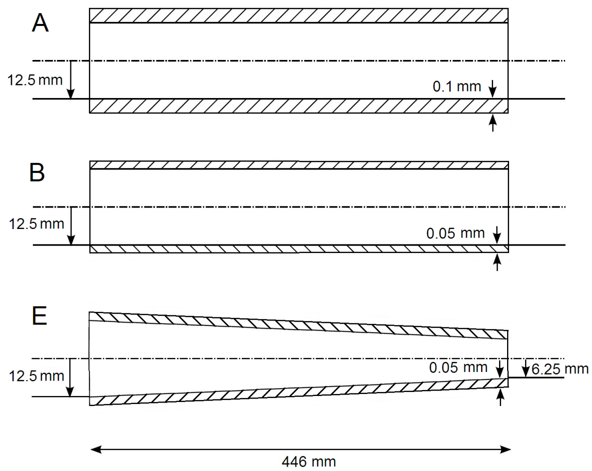


Figure 3. The polyurethane vessels

MATHEMATICAL FORMULATION

Wave Propagation in Flexible Vessels

Wormersly was one of the pioneers and mostly referenced in analytically modeling of wave propagation. His work is now known as the Womersley theory and will be recapitulated below using [6, 19].

The momentum equation and the continuity equation can be solved in the frequency domain for an elastic tube with an axi-symmetric flow. The fluid is considered to be incompressible and Newtonian.The momentum equation is

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho\nabla\cdot(\mathbf{uu}) = \nabla\cdot(\mu\nabla\mathbf{u}) - \nabla p \tag{1}$$

and the continuity equation

$$\nabla\cdot\mathbf{u} = 0 \tag{2}$$

Here $\mathbf{u}$ is the velocity field, μ the dynamic viscosity and p is the pressure. The diameter of the tube, $2R$, is assumed to be small compared to the wave length λ of the disturbance. By performing a dimensional analysis it appears that the convective terms and the derivatives in the axial direction can be neglected [19]. The Navier-Stokes equations can now be reduced to

$$\rho\frac{\partial u_r}{\partial t} + \frac{\partial p}{\partial r} = \mu\left(\frac{\partial^2 u_r}{\partial r^2} + \frac{1}{r}\frac{\partial u_r}{\partial r} - \frac{u_r}{r^2}\right) \tag{3}$$

$$\rho\frac{\partial u_z}{\partial t} + \frac{\partial p}{\partial z} = \mu\left(\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r}\frac{\partial u_z}{\partial r}\right) \tag{4}$$

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{\partial u_z}{\partial z} = 0 \tag{5}$$

Appropriate boundary conditions must be specified in order to integrate over a tubes's cross section. At the wall the no-slip, no-leak conditions and axi-symmetry hold. Further it is assumed that the is no axial movement. This results in the following boundary conditions

$$u_z|_{r=R} = 0 \quad u = 0 \quad \frac{\partial u_z}{\partial r}|_{r=0} = 0 \tag{6}$$

In linear theory p, u_r and u_z can be expressed as a combination of harmonics

$$p = \mathrm{Re}\left(\hat{p}e^{i(\omega t-\kappa z)}\right) \quad u_r = \mathrm{Re}\left(\hat{u}_r e^{i(\omega t-\kappa z)}\right)$$
$$u_z = \mathrm{Re}\left(\hat{u}_z e^{i(\omega t-\kappa z)}\right) \tag{7}$$

with ω and κ the angular frequency and wave number, respectively.

The work of Womersley is describing the theory for a vessel that is infinitely long and needs modification to be able to use it for a finite length vessel with closed ends. In [6] is described how reflecting waves can be determined. Here a tube of length L is considered starting at $z = -L_0$ and ending at $z = L_1$, see Fig. 4. An input pressure p_i in positive direction is applied at $z = 0$, which results in a traveling wave that propagates in positive direction. When the wave reaches $z = L_1$ the wave will be reflected with reflection parameter Γ_1 and it will travel partially towards $z = L_0$ where the wave will be reflected again with parameter Γ_0, this process will continue. Using Eqn. (7) the expression for the pressure, in the case of total reflection, can be written as

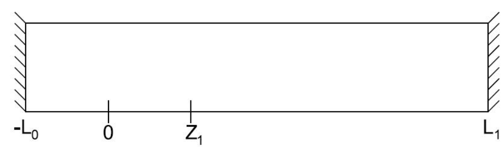


Figure 4. Schematic drawing of the tube with closed ends.

$$p(\omega, z_1, t) = \mathrm{Re}\left(p_i(\omega, 0)e^{i(\omega t-\kappa L_0)}\frac{\cos((L_1 - z_1)\kappa)}{i\sin(\kappa L)}\right) \tag{8}$$

The experimental data and the analytical data, using Eqn. (8), will be used to validate the model made in Comsol.

Modeling Blood Flow using Comsol

Comsol [20] is a modeling package based on finite elements and can be used to model the behavior of blood flow through a blood vessel. The *iterative over each time step* method is used to solve FSI. For the coupling of the fluid flow with the structural mechanics a moving mesh application is used to capture the changes of the fluid domain. This moving mesh is the Arbitrary Lagrangian Eulerian (ALE) method. The fluid flow application mode is defined on the ALE frame and the structural mechanics application mode is defined on a reference frame [20].

The Fluid Flow The fluid flow inside the vessel is described by the incompressible Navier-Stokes equations for the velocity field $\mathbf{u}$ and the pressure p. In the spatial moving coordinate system Eqn. 2 is the continuity equation and the momentum equation is now written as

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \left[-p\mathbf{I} + \mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)\right] + \rho((\mathbf{u} - \mathbf{u}_m) \cdot \nabla)\mathbf{u} = \mathbf{F} \quad (9)$$

Here μ is the dynamic viscosity, ρ is the density, $\mathbf{u}$ is the velocity field, $\mathbf{u}_m$ the coordinate system velocity, p is the pressure, $\mathbf{I}$ the unit diagonal matrix and $\mathbf{F}$ is the volume force affecting the fluid. To compare the results of Comsol with the experimental and analytical data, there is no gravitational or other volume force affecting the fluid, therefore $\mathbf{F} = 0$. At the entrance of the vessel, a normal inflow velocity pulse is defined. This velocity pulse satisfies the no-slip condition for the wall. At the outflow, the end of the vessel, the normal outflow velocity is set 0, that means that the vessel is closed. On the solid walls the velocities are equal to the deformation rate.

The Solid Domain The structural deformations are solved using a visco-elastic wall by using a standard linear solid model. This model can be found schematically in Fig. 5. Here

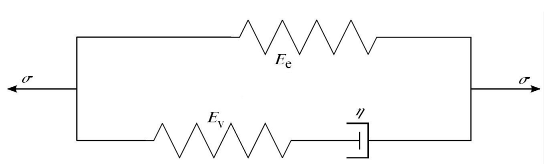


Figure 5. Standard linear solid model representing the viscous elastic model.

η is the dashpot's coefficient, E_e is the Young's modulus of the elastic part and E_v is the Young's modulus of the viscoelastic part. The stress, σ, and strain, ε, are related through

$$\sigma + \tau\dot{\sigma} = 2E_e\left(\varepsilon + \tau\left(1 + \frac{E_v}{E_e}\right)\dot{\varepsilon}\right) \quad (10)$$

where τ is called the relaxation time. The boundary experiences a load from the fluid, given by

$$\mathbf{F}_T = -\mathbf{n} \cdot (p\mathbf{I} + \mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)) \quad (11)$$

where $\mathbf{n}$ is the normal vector to the boundary. This load represents a sum of pressure and viscous forces [20].

Moving Mesh The *Lagrangian method* is often used in structural mechanics. In this method the mesh follows the material during motion. Unfortunately this method typically cannot follow large distortions of the computational domain without remeshing. The *Eulerian method* is often used in fluid mechanics. In this method the mesh is fixed and the fluid moves with respect to the grid. This method can handle large distortions but it typically cannot take moving boundaries into account. The *ALE method* combines the best features of the Lagrangian and Eulerian method. ALE is very helpful when the the structure undergoes large deformations since the method allows to have a flexible grid and a grid that allows for material to flow through it [21]. In this model the *ALE method* is used to couple the fluid and the solid.

RESULTS AND DISCUSSION

The wave propagation of a wave in the aorta was simulated and this model has been compared with experimental and analytical data available. The simulations for the straight vessels A and B and the tapered vessel E are performed using the fluid and solid parameters given in Table 1.

Table 1. Parameters used for the simulations

parameters	Vessel A	Vessel B	Vessel E
ρ [kg m^{-3}]	998	998	998
μ[kg m^{-1}s^{-1}]	1.04×10^{-3}	1.04×10^{-3}	1.04×10^{-3}
K [kg m^{-1}s^{-2}]	337×10^6	337×10^6	337×10^6
E_e [kg m^{-1}s^{-2}]	7.5×10^6	7.2×10^6	7.5×10^6
E_v [kg m^{-1}s^{-2}]	3.7×10^6	3.8×10^6	3.8×10^6
τ [kg m^{-1}s^{-1}]	6.58×10^{-5}	4.61×10^{-4}	1.97×10^{-3}

At the inlet a velocity pulse was applied. The vessel is considered to be axi-symmetric, therefore the problem can be solved in 2D. A structured grid of 12000 elements was used for vessels A, B and E, 600×15 for the fluid and 600×5 for the solid.

The system of equations was solved using the direct solver PARDISO, Parallel Sparse Direct Linear Solver . PARDISO is a highly efficient and direct solver for solving large sparse symmetric, structural symmetric or non-symmetric linear systems of equations on shared memory multiprocessors [20,22]. The initial time step is set to $0.0001s$ and the maximum time step is $0.01s$. Figure 6 shows the deformation of the wall and the pressure inside the vessel at $t = 0.05s$. The pulse inside the vessel travels through the vessel and is reflected at the closed ends of the tube. The results of Comsol are compared with both analytical and experimental data available for the pressure, pressure gradient, fluid flow and the wall displacement.

In Fig. 7,8 (vessel A), Fig. 9,10 (vessel B) and Fig. 11, 12 (vessel E) the experimental (solid line) and analytical (dashed

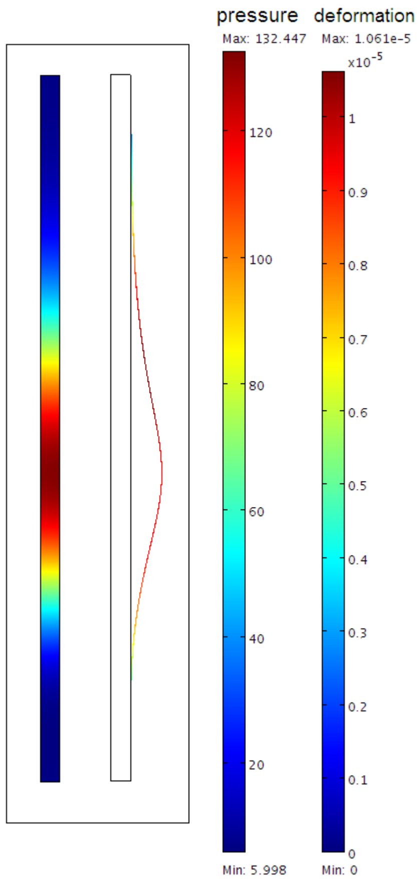
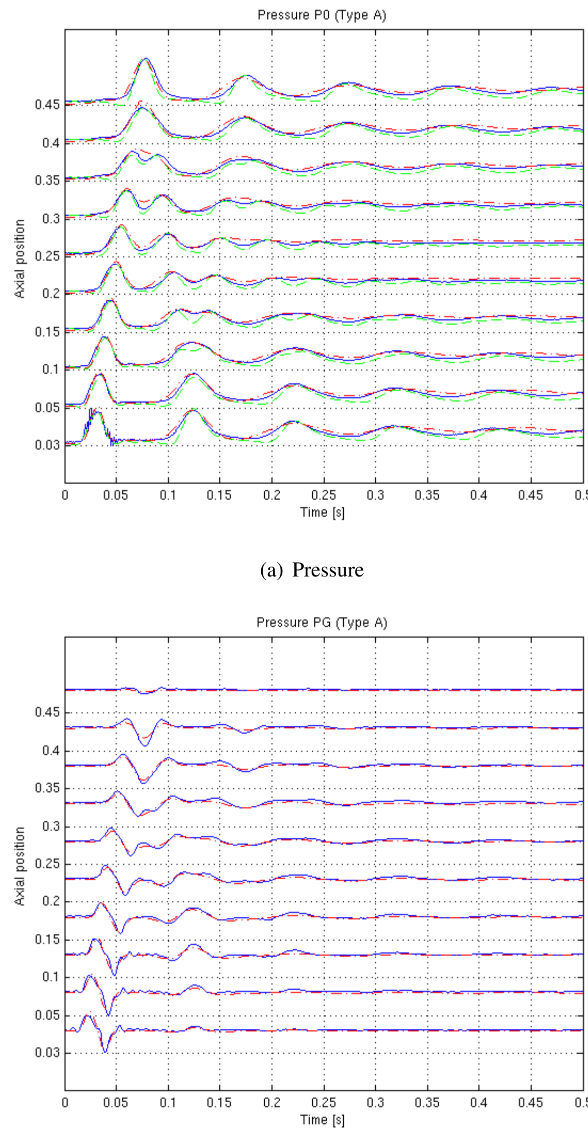


Figure 6. The flow pressure (left) and the deformation of the wall (right) at $t = 0.05s$. Here the deformation is scaled.

line) data are compared with the computational model (dash dotted line). Comparison of the data for the pressure shows that the computational data appears to be in good agreement with the experimental and analytical data for both vessels A and B. The comparison for vessel E shows that the computational data for the pressure experiences more damping at the end of the vessel than the experimental data does. However, the computational data is matching well the experimental data. In [23] the experimental data of vessel E has been compared with the multiple section analytical theory and gave good agreement which gives confidence in the quality of the experimental data to use it for the validation of the computational model.

For all three vessels the experimental data and the computational data for the pressure gradient have discrepancies at the beginning of the tube. The experimental data shows some inaccuracies at the beginning of the vessel. These inaccuracies are likely caused by noise in the acquisition of the data by the pressure sensor at the first measurement point. In spite of this, the computational and experimental data appear to be in good agreement.



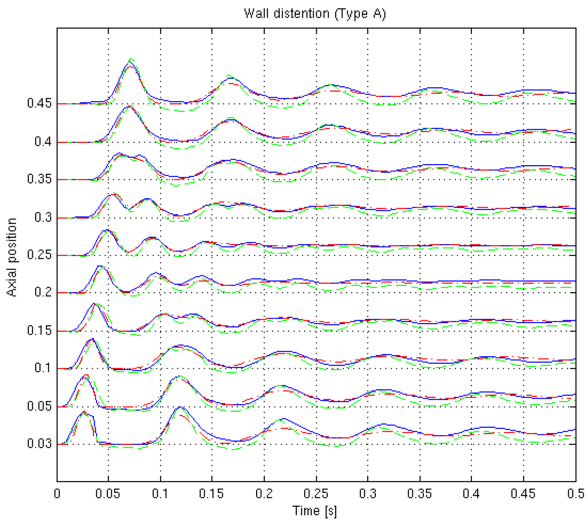
(a) Pressure

(b) Pressure gradient

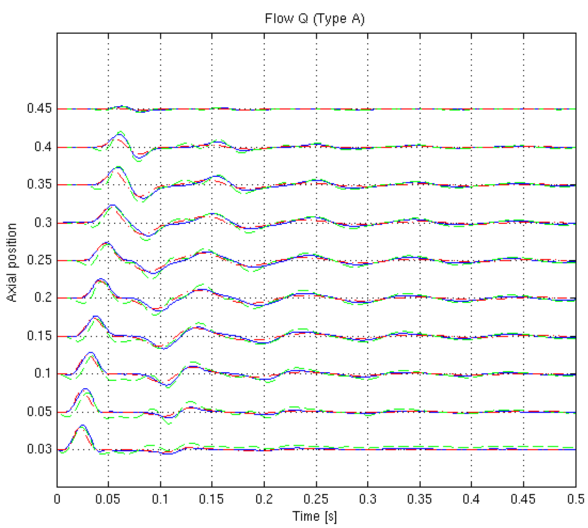
Figure 7. Comparison of experimental, analytical and computational data for vessel A.

The computational data obtained for the wall distention are in good agreement with experimental and analytical data for all three vessels. For vessel E however, there are discrepancies between the computational and experimental data for the first axial position. These discrepancies are due to the fact that the measurement is performed very close to the pulse entry point and additional noise has been introduced in the acquainted data by the sensor. Overall, by ignoring the first axial position, the conclusion is that the data are in good agreement.

The computational data for the fluid flow appears to be in good agreement with the experimental and analytical data for all three vessels.

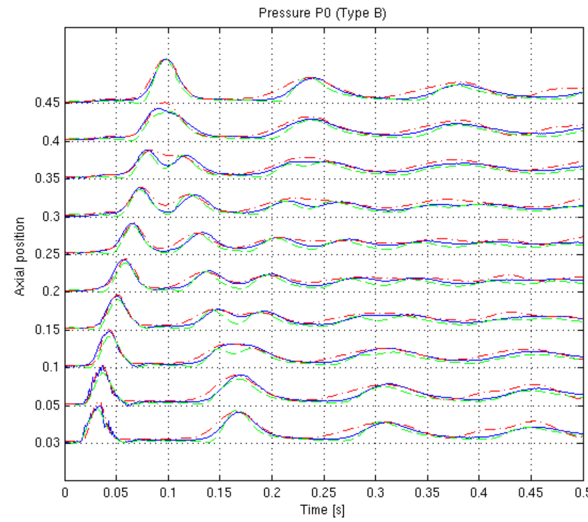


(a) Wall distention

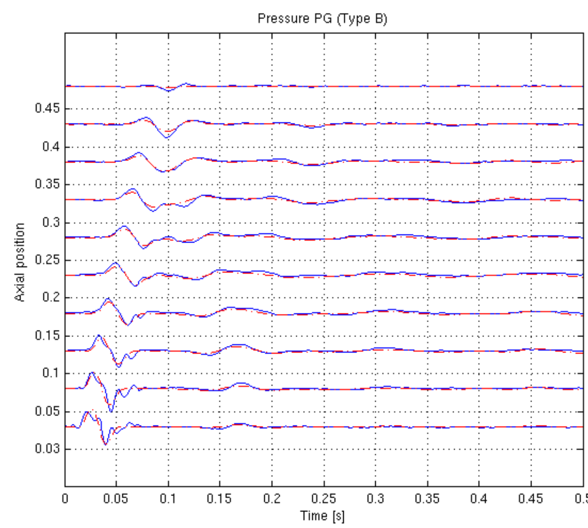


(b) Flow

Figure 8. Comparison of experimental, analytical and computational data for Vessel A.

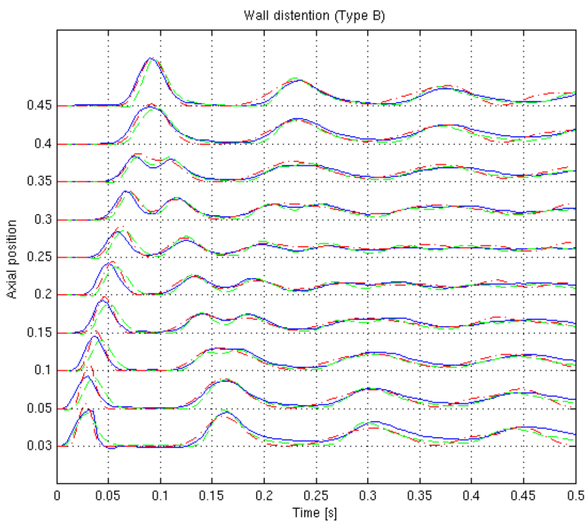


(a) Pressure

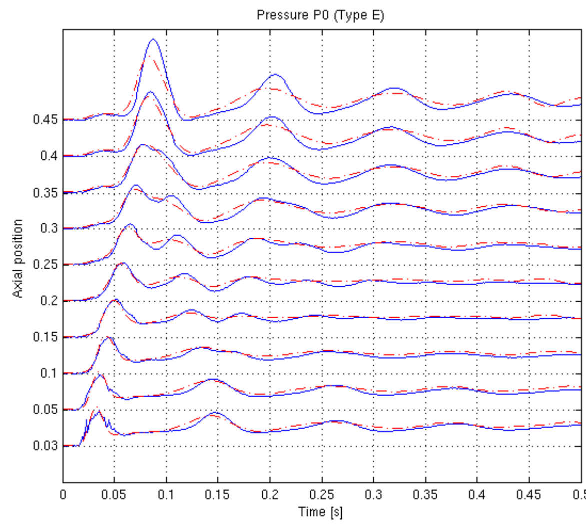


(b) Pressure gradient

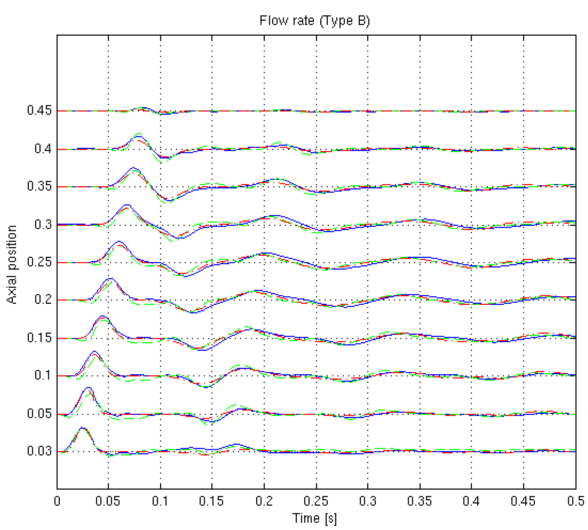
Figure 9. Comparison of experimental, analytical and computational data for vessel B.



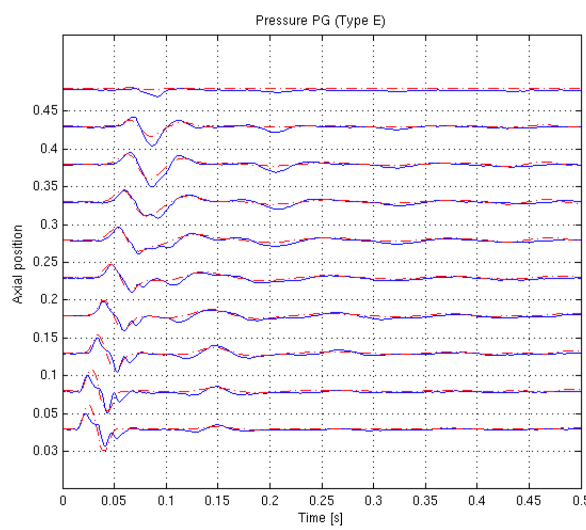
(a) Wall distention



(a) Pressure



(b) Flow



(b) Pressure gradient

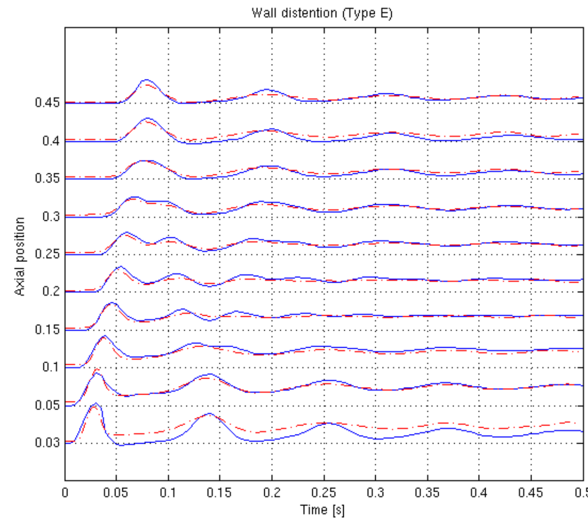
Figure 10. Comparison of experimental, analytical and computational data for vessel B.

Figure 11. Comparison of experimental, analytical and computational data for vessel E.

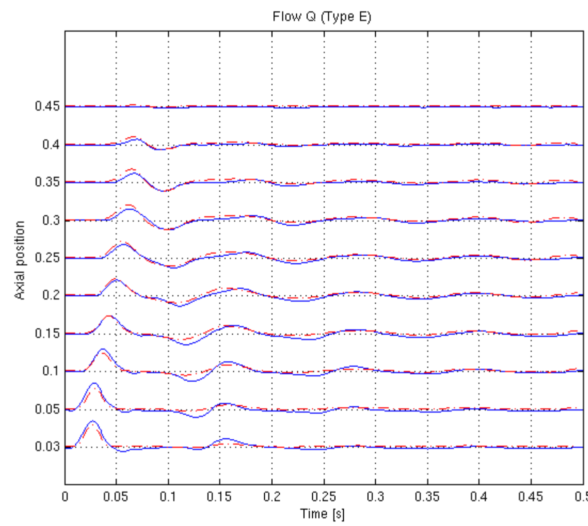
CONCLUSION AND FUTURE WORK

A computational model, using finite elements, for modeling arterial blood flow has been developed. This model was compared with available experimental and analytical data for both straight and tapered vessels with constant wall thickness. The computational model appears to be in good agreement with both the experimental an analytical data. Therefore this model can be used for predicting waves in aortic analogues and wave propagation in liquid filled flexible vessels that also exhibit geometric variations like tapering.

The computational model has been extended with hypergravity. For validating this model analytical or experimental data is needed which is currently not available in the literature as far as the author is concerned. Therefore the next step is to perform experiments in a Large Diameter Centrifuge with an experimental set-up like the experimental set-up used by Giannopapa [18] to obtain data which enables us to validate the computational model with hypergravity.



(a) Wall distention



(b) Flow

Figure 12. Comparison of experimental, analytical and computational data for vessel E.

REFERENCES

[1] Bertram, C., Gow, B., and Greenwald, S., 1997. "Comparison of different methods for determination of the true wave propagation coefficient, in rubber tubes and in canine thoracic aorta". *Medical Engineering & Physics,* ***19***, pp. 212 – 222.

[2] Milnor, W., and Bertram, C., 1978. "The relation between arterial viscoelasticity and wave propagation in the canine femoral artery in vivo". *Circulation research, journal of the American heart association,* ***43***, pp. 870 – 879.

[3] Moore, J., Rutt, B., Karlik, S., Yin, K., and Ethier, C., 1999. "Computational blood flow modeling based on in vivo measurements". *Annals of Biomedical Engineering,* ***27***, pp. 627 – 640.

[4] Bessems, D., Giannopapa, C. G., Rutten, M. C., and van de Vosse, F. N., 2008. "Experimental validation of time-domain-based wave propagation model of blood flow in visoelastic vessels". *Biophysical Journal,* ***41***, pp. 284 – 291.

[5] Čanić, S., Mikelić, A., and Tambača, J., 2005. "A two-dimensional effective model describing fluid-structure interaction in blood flow: analysis, simulation and experimental validation". *Comptes Rendus Mecanique,* ***333***, pp. 867 – 883.

[6] Giannopapa, C., Kroot, J., Tijsseling, A., Rutten, M., and van de Vosse, F., 2010. "Wave propagation in thin-walled aortic analogues". *Journal of Fluids Engineering*, pp. 021104–1 – 021104–6.

[7] Kanyanta, V., Ivankovic, A., and Karac, A., 2009. "Validation of a fluid-structure interaction numerical model for predicting flow transients in arteries". *Journal of Biomechanics,* ***42***, pp. 1705 – 1712.

[8] Matthys, K., Alastruey, J., J.Peiró, Khir, A., Segers, P., Verdonck, P., Parker, K., and Sherwin, S., 2007. "Pulse wave propagation in a model human arterial network: Assessment of 1-d numerical simulations against *in vitro* measurements.". *Journal of Biomechanics,* ***40***, pp. 3476 – 3486.

[9] Reuderink, P., Sipkema, P., and Westerhof, N., 1988. "Influence of geometric tapering on the derivation of the true propagation coeffcient using a three point method". *Journal of Biomechanics,* ***21***, pp. 141 – 153.

[10] Cox, R. H., 1968. "Wave propagation through a newtonian

fluid contained within a thick-walled, viscoelastic tube". *Biophysical Journal,* ***8***, June, pp. 691 – 709.

[11] Korteweg, D., 1878. "Ueber die fortplanzungsgeschwindigkeit des schalles in elastischen röhren". *Annalen der Physik und Chemie,* ***5***, pp. 525 – 542.

[12] Perktold, K., and Rappitsch, G., 1995. "Computer simulation of local blood flow and vessel mechanics in a compliant carotid artery bifurcation model". *Journal of Biomechanics,* ***28***, pp. 845– 856.

[13] Taylor, C., Hughe, T., and Zarins, C., 1998. "Finite element modeling of bood flow in arteries". *Computer Methods in Applied Mechanics and Engineering,* ***158***, pp. 155 – 196.

[14] Bathe, M., and Kamm, R., 1999. "A fluid-structure interaction finite element analysis of pulsatile blood flow through a compliant stenotic artery". *Journal of Biomedical Engineering,* ***121***, pp. 361 – 369.

[15] Figueroa, C., Vignon-Clementel, I., Jansen, K., Hughes, T., and Taylor, C., 2006. "A coupled momentum method for modeling blood flow in three dimensional deformable arteries". *Computer Methods in Applied Mechanics and Engineering,* ***195***, pp. 5685 – 5706.

[16] Giannopapa, C., and Papadakis, G., 2008. "Linear stability analysis and application of a new solution method of the eastodynamic equations suitable for a uniefied fluid-structure-interaction approach". *Journal of Pressure Vessel Technology*, August, pp. 031303–1 – 031003–8.

[17] Giannopapa, C., 2007. Development of the monolithic unified single solution method for fluid-structure interaction problems. Tech. rep., ASME Pressure vessels and piping division, July.

[18] Giannopapa, C., 2004. "Fluid structure interaction in flexible vessels". PhD thesis, King's college London.

[19] Reuderink, P., van de Vosse, F., van Steenhoven, A., van Dongen, M., and Janssen, J., 1993. "Incompressible low-speed-ration flow in non-uniform distensible tubes". *International Journal for Numerical Methods in Fluids,* ***16***, pp. 597 – 612.

[20] *COMSOL Multiphysics Quick Start and Quick Reference*, November 2008.

[21] Donea, J., Huerta, A., Ponthot, J.-P., and A.Rodríguez-Ferran, 2004. *Encyclopedia of Computational Mechanics: Arbitrary Lagrangian-Eulerian Methods*, Vol. 1:Fundamentals. John Wiley & Sons.

[22] Schenk, O., Grtner, K., Fichtner, W., and Stricker, A., 2001. "Pardiso: a high-performance serial and parallel sparse linear solver in semiconductor device simulation". *Future Generation Computer Systems,* ***18***, pp. 69 – 78.

[23] Giannopapa, C., and Kroot, J., 2009. "Multiple reflection and transmission theory for wave propagation in the aorta". *Proceedings of PVP2009*, pp. 1 – 9.

PREVIOUS PUBLICATIONS IN THIS SERIES:

Number	Author(s)	Title	Month
11-44	P.I. Rosen Esquivel J.H.M. ten Thije Boonkkamp J.A.M. Dam R.M.M. Mattheij	Wall shape optimization for a thermosyphon loop featuring corrugated pipes	Sept. '11
11-45	J. de Graaf	A complex-like calculus for spherical vectorfields	Sept. '11
11-46	C. Cancès I.S. Pop M. Vohralík	An a posterior error estimate for vertex-centered finite volume discretizations of immiscible incompressible two-phase flow	Sept. '11
11-47	M.H.A. van Geel C.G. Giannopapa B.J. van der Linden J.M.B. Kroot	Development of a blood flow model including hypergravity and validation against an analytical model	Sept. '11
11-48	M.H.A. van Geel C.G. Giannopapa B.J. van der Linden	Development of a blood flow model and validation against experiments and analytical models	Sept. '11

Ontwerp: de Tantes,
Tobias Baanders, CWI