A FICTITIOUS DOMAIN METHOD COMBINED WITH LOCAL MESH ADAPTATION AS A NEW TECHNIQUE FOR FLUID-STRUCTURE INTERACTION IN HEART VALVES

R. van Loon\textsuperscript{1}, P.D. Anderson\textsuperscript{2}, J. de Hart\textsuperscript{2}, F.N. van de Vosse\textsuperscript{3}, F.P.T. Baaijens\textsuperscript{3}

1. ABSTRACT

A new approach for modelling the fluid-structure interaction of heart valves is proposed. It combines a fictitious domain method with a local mesh-adaptation algorithm and is able to accurately compute displacements of a flexible solid and the velocity field around it. Using the finite element method, a Lagrangian description of a non-linear solid and an Eulerian description of a fluid are coupled using a Lagrange multiplier. Introduction of this Lagrange multiplier allows the fluid and solid mesh to be non-conform. In order to gain accuracy in the vicinity of the solid, an inexpensive mesh-adaptation algorithm is added, which enables the computation of shear stresses at both sides of a slender solid. Furthermore, the method is capable of computing pressure drop across a solid.

2. INTRODUCTION

Computational methods can be of great help in understanding heart valve pathologies. The behaviour of the valves (mitral or aortic, mechanical or biological), which should cause no resistance during systole, but need to sustain large pressure gradients during diastole, is however not easy to capture. The interaction between blood and valve cannot be neglected in the analysis of a flexible leaflet. Although many models have been proposed throughout the years in order to get better insight in the mechanical behaviour of heart valves only some incorporate this complex interaction. ALE methods combined with remeshing have been used for solving fluid-structure problems of an aortic heart valve [1]. However, the more complex the geometries for the fluid and solid domain, the more difficult and time consuming remeshing becomes. A different approach was therefore used by Baaijens (2D) and de Hart (3D) [2, 3], in which a Lagrangian solid domain is coupled to an Eulerian fluid domain by means of Lagrange multipliers, a so called fictitious domain (FD) method [4]. This way of coupling allows non-conform meshing of solid and fluid and meshes only need to be created once. However due to interpolation these methods do not allow for highly accurate descriptions of gradients in velocity field and pressure. Hence, in heart valves, where shear and substantial diastolic pressure gradients along the leaflets play an important role in their functioning, the application of FD solely to describe the interaction may not be sufficient.

Keywords: fluid-structure, heart valve, lagrange multiplier, adaptive meshing

\textsuperscript{1}PhD stud., Faculty of Biomedical Engineering, Eindhoven University of Technology, P.O.Box 513, 5600 MB, Eindhoven, The Netherlands
\textsuperscript{2}ass. prof., Faculty of Biomedical Engineering, Eindhoven University of Technology, P.O.Box 513, 5600 MB, Eindhoven, The Netherlands
\textsuperscript{3}full prof., Faculty of Biomedical Engineering, Eindhoven University of Technology, P.O.Box 513, 5600 MB, Eindhoven, The Netherlands
The goal of this work is to develop a method that is able to accurately capture stresses along the leaflet boundary. Furthermore, physiological pressure gradients across a leaflet should be computed correctly. The presented method is an extension of the FD method [2, 5], with an inexpensive adaptive meshing technique. By creating an inner fluid curve, that coincides with the solid boundary, only interpolation along (and not across) the boundary is needed, resulting in more accurate solutions.

3. METHOD

3.1 Governing equations

In this work the fluid is described using the Navier-Stokes equation and the continuity equation. The fluid is assumed to be Newtonian. For the solid an incompressible neo-Hookean constitutive law was used. The fluid domain ($\Omega_f$) is coupled to an immersed solid domain ($\Omega_s$) by the constraint that the velocity of the fluid should equal the velocity of the solid, $v_f - v_s = 0$, at a boundary ($\partial\Omega_s$) of this solid. This constraint is applied weakly by introducing a distributed Lagrange multiplier ($\lambda$). The weak forms for fluid, solid, coupling constraint and incompressibility constraints then read, respectively,

\[
\int_{\Omega_f} w_f \cdot \left( \rho \left( \frac{\partial v_f}{\partial t} + v_f \cdot \nabla v_f \right) - \nabla \cdot \tau_f + \nabla p \right) d\Omega_f + \int_{\partial\Omega_s} w_f \cdot \lambda d\partial\Omega_s = 0, \tag{1}
\]

\[
\int_{\Omega_s} w_s \cdot (\nabla \cdot \tau_s - \nabla p) d\Omega_s - \int_{\partial\Omega_s} w_s \cdot \lambda d\partial\Omega_s = 0, \tag{2}
\]

\[
\int_{\partial\Omega_s} w_\lambda \cdot (v_f - v_s) d\partial\Omega_s = 0, \tag{3}
\]

and

\[
\int_{\Omega_f} w_{pf} \nabla \cdot v_f d\Omega_f = 0 \quad \text{and} \quad \int_{\Omega_s} w_p \det(F) d\Omega_s = 1 \tag{4}
\]

The variables $w_s$, $w_f$, $w_\lambda$, $w_{pf}$ and $w_p$ are appropriate test functions. Furthermore, $\nabla$ is the gradient operator, $\rho$ the density, $F$ the gradient deformation tensor and $\tau_f$ is the viscous part of the Cauchy stress tensor. The extra stress tensor $\tau_s$ is defined as,

\[
\tau_s = G \left( F \cdot F^T - I \right), \tag{5}
\]

where $G$ is the shear modulus and $I$ the unity tensor. The set of solution variables now consists of the velocity of fluid and solid, $v_f$ and $v_s$, the pressure of fluid and solid, $p_f$ and $p_s$, and the Lagrange multiplier $\lambda$.

The same formulation is used by de Hart [5], however, the way of of applying the Lagrange multiplier is different. Consider a fluid domain, which is discretised into triangular elements, with a boundary $\partial\Omega_s$ crossing the elements. In the model problems, that de Hart presented, the Lagrange multiplier is defined along the boundary, $\partial\Omega_s$, to couple the velocity of this boundary to the velocity of the fluid elements in which the boundary
is situated [5]. Although such an approach gives satisfactory results for valve displacement and the general flow behaviour, it fails to provide an accurate description of shear stresses at either side of the valve. Furthermore, during diastolic phase of the heart cycle a pressure decrease in the left ventricle occurs, which leads to a large pressure gradient across the valve leaflets. Since the leaflets cross the fluid elements, erroneous results for the pressures are obtained. The model proposed in this work is based on the idea to create a boundary \( \partial \Omega_f \) inside the fluid domain that coincides with boundary \( \partial \Omega_s \) by performing an adaptation of the mesh in the vicinity of \( \partial \Omega_s \) as explained next.

### 3.2 Adaptive meshing

Consider a (fluid) mesh, \( \Omega_f \), with an arbitrary solid boundary curve, \( \partial \Omega_s \), crossing it (Fig. 1(a)). In order to create a boundary \( \partial \Omega_f \) in the fluid domain that coincides with boundary \( \partial \Omega_s \), first the intersections of \( \partial \Omega_s \) with \( \Omega_f \) need to be found. When all the intersections are determined a selection of fluid nodes, that ensures mesh integrity, are shifted along the intersected curves from the nodes on this curve to boundary \( \partial \Omega_s \). The repositioning of the nodes around the boundary \( \partial \Omega_s \) influences the element shapes of this boundary, which can lead to inaccurate results. Therefore, smoothing is applied in this region. Following, in case of second-order extended elements, the midside nodes and centroid are repositioned. The fluid nodes that lie on \( \partial \Omega_s \) now form a newly formed inner fluid boundary called \( \partial \Omega_f \) (Fig. 1(b)). From numerical experiments it was found that for this weakly coupled system especially the fluid is very sensitive to the amount of coupling elements. If the discretisation of the Lagrange multiplier is based on the solid discretisation [5], it is difficult to control the number of coupling elements within one fluid element. It is therefore convenient to define it based on the discretisation of the newly created inner boundary \( \partial \Omega_f \), which is now possible.

### 3.3 Solution process

For time-dependent problems one mesh is generated prior to the computations that will be adapted every time step and will be used for the computations. After every time step the solutions will be mapped. The non-linear set of equations is linearised and solved in a
Newton-Raphson iterative scheme. An implicit time-integration scheme is used for both the solid and fluid and a first-order approximation is used for the velocity. Triangular elements, $P_2^+ - P_1$, are used for the fluid and quadrilateral elements, $Q_2^+ - Q_1$, are used and for the solid. The Lagrange multiplier domain has the same discretisation as the internal fluid boundary that is obtained from mesh adaptation and is integrated using discontinuous linear interpolation functions. Since thickness and mass of the solid are negligible as far as the interaction with the fluid is considered, fluid and solid velocity are coupled at only one boundary of the solid. The CPU time needed to perform the mesh adaptation is therefore negligible compared to the time needed for solving the system. The finite element package, SEPRAN, is extended for the computations in combination with a direct HSL solver\(^1\).

4. RESULTS

4.1 Flexible slab in a pulsatile flow

An Eulerian fluid domain $\Omega_f$ is considered with an immersed Lagrangian solid domain $\Omega_s$ (figure 2). Along the walls, denoted with $\Gamma_{wall}$, a no slip condition applies and at $\Gamma_{inlet}$ the velocity is prescribed as a function of time, $v_f = \sin(2\pi t)$, over a dimensionless time period of 0.0-1.0. The solid is attached to the upper wall. As stated earlier the Lagrange multiplier domain is defined along the internal fluid boundary that is obtained from mesh adaptation and coincides with boundary $\partial \Omega_s$. For all computations concerning this model problem, time is discretised into 4000 time steps. The fluid mesh is divided into 5004 elements and the solid mesh (length/width ratio of 18:1) is divided into 40 elements. Time steps have proven to be sufficiently small, as has the element size in the spacial discretisation. A maximum Reynolds number of 1000 (height of the canal is used as characteristic length) and a maximum Strouhal of 0.1 are used.

As mentioned before, one advantage of the FD method combined with mesh adaptation compared to the FD method without it, is the possibility of gaining information about the shear stresses along the solid. Since a set of fluid edges coincides with a set of solid edges, computation of different shear stresses at either side of the solid slab is possible. In Fig. 3 the streamline plots at different time steps are plotted with their corresponding shear stress graphs. The shear stresses in the midside nodes of the fluid element edges along the solid are plotted. For sake of clarity the long inlet and outlet are not shown.

\(^1\) A collection of Fortran codes for large scale scientific computation.  
http://www.numerical.rl.ac.uk/hsl
in the figures, which leaves the most interesting middle part of the fluid domain to be presented. Note that since the tip of the solid lies inside a fluid element, no shear stress information at this position is available.

![Streamline plots and position of the solid slab with corresponding shear stress plots at both sides of the solid.](image)

**Fig. 3:** The streamline plots and position of the solid slab with corresponding shear stress plots at both sides of the solid. (a) $t=0.3$, (b) $t=0.56$, (c) $t=0.68$, (d) $t=0.76$

4.2 Solid membrane sustaining pressure in a fluid domain

The former model problem incorporated the movement of the valve, the complexity of the flow field and the determination of shear stresses. In this section the ability of the method to describe a moving pressure drop is proven. The model problem is similar to the former one except that the solid is now fixed at the upper and lower wall and at $\Gamma_{inlet}$ Neumann boundary conditions are prescribed instead of Dirichlet boundary conditions. When increasing the pressure at the inlet, erroneous results are obtained in the vicinity of the membrane (Fig. 4(a)). The error in the solution accumulates, which after several

![Pressure drop over a membrane using the fictitious domain method with (a) and without (b) mesh adaptation.](image)

**Fig. 4:** Pressure drop over a membrane using the fictitious domain method with (a) and without (b) mesh adaptation.

time steps leads to divergence of the non-linear system. The FD method combined with
adaptive meshing, on the other hand, captures the pressure drop accurately (Fig. 4(b)), using the ability of the $P_2^+ - P_1$ elements to describe pressure discontinuities across the element edges.

5. DISCUSSION

A method is presented for modelling fluid-structure problems in flexible heart valves, which is able to accurately capture shear stress along both sides of the leaflets as well as transvalvular pressure gradients. The FD method presented by de Hart [5] is extended with a computationally inexpensive adaptive meshing algorithm. Coupling of a separate Eulerian and Lagrangian mesh is established using a Lagrange multiplier, which allows for optimal choices of the discretisations of fluid and structure. The coupling between the meshes is enhanced by adapting the Eulerian mesh locally, such that an inner curve $\partial \Omega_f$ is created which coincides with a solid boundary. The use of elements with a discontinuous pressure description in combination with the inner fluid curve $\partial \Omega_f$ enables capturing of the pressure drop across the leaflets. Furthermore, accurate solutions for the shear stress at both sides of the leaflets can be obtained. Two examples are presented in which the method proves to be a significant improvement with respect to the FD method in which the Eulerian mesh is kept unchanged. Since FD methods have proven to be an interesting numerical tool for 3D analysis of fluid-structure problems in heart valves [3], extension of the presented method to three dimensions is a step of great interest.

6. REFERENCES


