A multiscale fluid structure interaction model derived from Koiter shell equations

A Chierici¹, L Chirco¹, V Giovacchini¹, S Manservisi¹ and G Santesarti¹

¹University of Bologna, DIN, Lab. of Montecuccolino, Via dei Colli 16, Bologna 40136, Italy
E-mail: andrea.chierici4@unibo.it

Abstract. In this work we propose the numerical simulation of fluid structure interaction (FSI) problem by using a membrane model, derived from the Koiter shell equations. With this approach the thickness of the solid wall can be neglected, with a meaningful reduction of the computational cost of the numerical problem. The fluid structure problem is then reduced to the fluid equations on a moving mesh together with a particular Robin boundary condition imposed on the surface corresponding to the solid moving wall. Furthermore an artificial absorbing outflow boundary condition has been implemented in order to reduce the damping and reflections of the pressure waves at the domain’s outlet. This model is implemented and solved with an in-house finite elements code, and tested through axisymmetric cases that show the robustness of the developed algorithm. Finally, we report a comparison of the implemented model with results of a FSI monolithic model, based on non-linear incompressible structure.

1. Introduction
In last years the numerical simulations of fluid-structure interaction (FSI) problems have gained popularity and interest in the research community thanks to the large variety of possible applications, ranging from wind turbines and aircrafts to hemodynamics. In particular, in hemodynamics the FSI mechanisms are important for the pressure wave propagation in the cardio-circulatory system. Several different models have been implemented in order to represent the behavior of the interaction between a fluid and a solid, and a large variety of articles and books is available on this topic [1, 2].

Most software packages for the resolution of such problems uses the so-called partitioned approach, which decouples the problem into two separate sub-problems and uses dedicated algorithms for each different region. The coupling is then achieved by enforcing continuity conditions along the fluid-solid interface [3, 4]. However, using explicit partitioned algorithms can lead to unbalances on the interface when large displacements are taken into account. In order to achieve better stability properties one could implicitly enforce coupling conditions that solve simultaneously the fluid and structure unknowns, with monolithic algorithms [5, 6, 7], that are stable and robust but also CPU-time expensive.

In recent years many works focused on reducing computational costs of such algorithms, and various techniques have been developed for this goal. These algorithms are generally based on velocity-pressure splitting preconditioners, that keep the original number of degrees of freedom and preserve exact boundary conditions [8, 9, 10]. In the framework of the reduction of
computational cost in the fluid-structure simulations, some studies are based on the reduction of the dimensionality of the solid. In this work we use such approach.

A model based on the Koiter shell equations [11] for the solid modeling, together with the assumption that the structure is thin and deforms mainly in the normal direction to the mean surface have been implemented. In order to couple the fluid and the structure domains, the Koiter shell equations are embedded into the fluid equations as a Robin boundary condition [12]. The stability of this numerical scheme is preserved since the coupling fluid-structure conditions are automatically treated in an implicit way. This model have many applications in cases where a fluid interacts with a thin membrane. It is particularly used for hemodynamic applications.

While being simple, this model allow the reproduction of important fluid-structure mechanisms, such as the propagation of pressure waves. Proper absorbing boundary conditions should be prescribed to have a correct numerical representation of the the pressure field at the outlet of the studied domain. In fact spurious reflections occurs when a Dirichlet boundary condition on the pressure is imposed at the outlet. In this work we follow the general approach to artificial boundary conditions in [13], and we apply the simplified model considered in [12]. Here the three-dimensional fluid-structure problem is coupled with a reduced one-dimensional model, acting as a nonreflecting boundary condition which is applied directly to the fluid problem.

All the described techniques are implemented in a finite element code, in order to reduce the computational cost of FSI simulations and to obtain proper outflow conditions for the propagation of pressure waves. Results are compared with simulations of a FSI monolithic model implemented and described in past works [14].

2. Physical model

2.1. The reduced structural model

The structural model is based on the Koiter shell approach that considers the model of an elastic thin membrane with the hypotheses of small displacements, negligible bend and shear stresses, normal displacements to the shell’s surface and linear elastic constitutive law with a homogeneous and isotropic material [11]. Furthermore we consider the presence of pre-stresses in the structure as happens in the blood vessels for hemodynamic applications [15]. We denote the density and the thickness of the shell with $\rho_s$ and $h_s$, the outer surface and the volume of the structure with $\Gamma_s$ and $\Omega_s$, the displacement and the external surface forces vectors with $\eta$ and $f_s$ and the elasticity, the change of metric and the Cauchy stress tensors with $E^{\alpha\beta\lambda\delta}$, $\gamma_{\alpha\beta}$ and $\sigma$, respectively [12]. The weak form of the shell equation results

$$
\int_{\Gamma_s} \rho_s h_s \frac{\partial^2 \eta}{\partial t^2} \cdot \psi \, d\Gamma + \int_{\Gamma_s} h_s E^{\alpha\beta\lambda\delta} \gamma_{\alpha\beta}(\eta) \psi_{\lambda\delta} \, d\Gamma + \int_{\Omega_s} \nabla \eta \sigma : \nabla \psi \, d\Omega = \int_{\Gamma_s} f_s \cdot \psi \, d\Gamma ,
$$

for appropriate test functions $\psi$ belonging to the functional space $M$ to be determined on the basis of the imposed boundary conditions.

Now by imposing the hypothesis of only normal displacements and neglecting the terms in function of the thickness coordinate we obtain the following pre-stressed model

$$
\rho_s h_s \frac{\partial^2 \eta_3}{\partial t^2} - \nabla \cdot (\sigma \nabla \eta_3) + \beta \eta_3 = f_s \quad \text{in} \quad (0, T) \times \Gamma_s , \quad \eta_3 |_{t=0} = \eta_0 , \quad \frac{\partial \eta_3}{\partial t} |_{t=0} = \eta_v \quad \text{in} \quad \Gamma_s ,
$$

where $\beta$ is the reactive elastic coefficient and $\eta_3$ is the displacement normal to the surface [12].

In particular when one considers cylindrical geometries and analyzes only longitudinal stress $\sigma^{zz}$ the model (2) becomes

$$
\rho_s h_s \frac{\partial^2 \eta_3}{\partial t^2} - \sigma^{zz} \frac{\partial^2 \eta_3}{\partial z^2} + \beta \eta_3 = f_s \quad \text{in} \quad (0, T) \times \Gamma_s ,
$$
with

\[ \beta = \frac{h_s E}{1 - \nu^2 R^2}, \]  

(4)

where \( R \) is the cylinder radius. Equation (3) is similar to the equation of a linear harmonic oscillator, with the inertial, elastic and forcing terms and the added prestress \( \sigma^{zz} \).

### 2.2. The fluid-structure multiscale coupling

For the fluid model we consider a Newtonian, homogeneous and incompressible fluid described by the following system of equations in ALE form [1, 12, 16, 17]

\[
\begin{align*}
\rho_f \frac{\partial \mathbf{u}}{\partial t} \bigg|_{A} + \rho_f ((\mathbf{u} - \mathbf{w}) \cdot \nabla) \mathbf{u} - \nabla \cdot \mathbf{\sigma} &= 0 & \text{in} & (0, T) \times \Omega_f, \\
\mathbf{u}|_{\Gamma_{D,f}} &= \mathbf{\tilde{u}} & \mathbf{\sigma} \cdot \mathbf{n}|_{\Gamma_{N,f}} &= \mathbf{\tilde{h}} & \text{in} & (0, T),
\end{align*}
\]

(5)

where \( \rho_f \) and \( \mathbf{u} \) are the density and the velocity vector of the fluid, \( \mathbf{\sigma} \) the Cauchy stress tensor written as

\[
\mathbf{\sigma} = -p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T),
\]

(6)

where \( p \) and \( \mu \) are the pressure and the dynamic viscosity of the fluid, respectively. Then \( \Omega_f \) is the fluid domain, \( \Gamma_{D,f} \) and \( \Gamma_{N,f} \) are the boundary’s portions where the Dirichlet and Neumann conditions are applied, respectively, and \( \mathbf{w} \) is the ALE velocity that determines step by step the position of fluid domain’s nodes

\[
\mathbf{x}_f(t) = \mathbf{x}_0 + \int_0^t \mathbf{w} d\tau.
\]

(7)

Now we can couple the two sub-systems (3) and (5). First we introduce the following functional spaces

\[
\begin{align*}
V^0 &= \{ \phi \in H^1(\Omega_f) : \phi|_{\Gamma_{D,f}} = 0 \}, \\
Q^0 &= \{ q \in L^2(\Omega_f) \}, \\
M^0 &= \{ \psi \in H^1(\Gamma_s) \}, \\
V^t_g &= \{ v \in H^1(\Omega(t)) : v|_{\Gamma_{D(t)}} = g \land (v \cdot \mathbf{t}_i)|_{\Gamma(t)} = 0 \}
\end{align*}
\]

(8)

with the test functions \( \phi \), \( q \) e \( \psi \) associated to the unknown solutions \( \mathbf{u} \), \( p \) and \( \eta \), respectively. \( \mathbf{t}_i \) are the tangent unit vector on the surface \( \Gamma(t) \) are denoted by \( \mathbf{t}_i \).

It is important to consider the continuity condition of test functions \( \phi \cdot \mathbf{n} = \psi \) over the interface surface \( \Gamma_s \) in the coupled system involving (3) and (5). This introduces the new functional space resulting from their union as

\[
W^0 = \{ (\phi, \psi) \in V^0 \times M^0 : \phi \cdot \mathbf{n} = \psi \quad \text{over} \quad \Gamma_s \}.
\]

(9)

We can now derive the weak form of the coupled final system

\[
\begin{align*}
\rho_f \mathbf{u} &\quad \frac{\partial u}{\partial t} |_{A} + \rho_f ((\mathbf{u} - \mathbf{w}) \cdot \nabla) \mathbf{u} - \nabla \cdot \mathbf{\sigma} = 0 & \text{in} & (0, T) \times \Omega_f, \\
\mathbf{u} |_{\Gamma_{D,f}} &= \mathbf{\tilde{u}} & \mathbf{\sigma} \cdot \mathbf{n} |_{\Gamma_{N,f}} &= \mathbf{\tilde{h}} & \text{in} & (0, T), \\
\mathbf{\sigma} &\quad \mathbf{\nabla} \phi \mathbf{d} \mathbf{x} - \int_{\Gamma_{N,f}} \mathbf{h} \cdot \phi \mathbf{d} \Gamma + \\
&\quad + \int_{\Gamma_s} \rho_s h_s \frac{\partial^2 \eta}{\partial t^2} \psi \mathbf{d} \Gamma + \int_{\Gamma_s} \mathbf{\sigma} \mathbf{\nabla} \eta \psi \mathbf{d} \Gamma + \int_{\Gamma_s} \beta \eta \psi \mathbf{d} \Gamma = 0, \\
\mathbf{\nabla} \cdot \mathbf{u} &\quad q = 0,
\end{align*}
\]

(10)

for all \( (\phi, \psi) \in W^0 \), \( q \in Q^0 \). The functional spaces of the unknowns are \( \mathbf{u} \in V^t_g \), \( p \in L^2(\Omega(t)) \), \( \chi \in H^1(\Gamma^0) \).
2.3. Numerical modeling with FEM
We use a finite element technique in order to obtain the discrete weak formulation of (10). Following the work in [12], we threat explicitly the position of the fluid domain, and consider an implicit discretization of the coupling conditions. With this approach the structural equation can be incorporated in the fluid equations as a boundary condition (Robin scheme). The structural equation (2) can be discretized as

\[ \frac{\rho_s h_s}{\Delta t^2} \eta^{n+1} - 2\eta^n + \eta^{n-1} = f_s^{n+1}, \]

where \( \eta^{n+1} \) is the unknown at the a given iteration, and \( \eta^n \) and \( \eta^{n-1} \) are the solution at the last and second-last iteration, respectively. In the following, we will maintain this notation.

Now we can consider the discrete problem as: find \( u_{h_i}^{n+1} \in V^t_{g,h} \subset V^t_g \) and \( p_{h_i}^{n+1} \in Q^h \subset L^2(\Omega(t)) \) such that

\[
\frac{1}{\Delta t}(u_{h_i}^{n+1}, \phi_h)_n + \left( (u_{h_i}^{n} - w_{h_i}^n) \cdot \nabla \right) u_{h_i}^{n+1}, \phi_h)_n + \mu (\nabla u_{h_i}^{n+1} + (\nabla u_{h_i}^{n+1})^T, \nabla \phi_h)_n
- (p_{h_i}^{n+1}, \nabla \phi_h)_n + \int_{\Gamma_2} \left( \left( \frac{\rho_s h_s}{\Delta t} + \beta \Delta t \right) u_{3,h_i}^{n+1} \phi_{3,h_i} + \Delta t \tilde{\sigma} \nabla u_{3,h_i}^{n+1} \cdot \nabla \phi_{3,h_i} \right) d\gamma
= \frac{1}{\Delta t}(u_{h_i}^{n}, \phi_h)_n - \int_{\Gamma_2} \left( \left( \frac{\rho_s h_s}{\Delta t^2} + \beta \right) \eta_{h_i}^{n} + \frac{\rho_s h_s}{\Delta t^2} \eta_{h_i}^{n-1} \right) \circ (x_i^n)^{-1} \phi_{3,h_i} d\gamma
- \int_{\Gamma_2} \tilde{\sigma} \nabla (\eta_{h_i} \circ (x_i^n)^{-1}) \cdot \nabla \phi_{3,h_i} d\gamma + \int_{\Gamma_n} h \cdot \phi \ d\gamma
\]

\[
(\nabla \cdot u_{h_i}^{n+1}, q_h)_n = 0,
\]

where \( \tilde{\sigma} \) is the pre-stress tensor in the reference configuration expressed in Cartesian coordinates, \( \phi_h \in V^h \subset W_0 \), and \( q_h \in Q^h \subset Q^0 \). Furthermore \( u_{3,h_i} \) and \( \phi_{3,h_i} \) are referred to the normal component of the vector to the outer surface and \( x_i^n \) is taken from equation (7) and maps each point of the simulated domain from the starting to the current configuration.

Once the velocity and the pressure field are computed, the displacement field can be obtained from

\[ \eta_{h_i}^{n+1} = \Delta t (u_{3,h_i}^{n+1} \circ x_i^n)|_{t_0} + \eta_{h_i}^n. \]

Then the numerical problem is closed with the mesh motion, which is performed through a moving mesh algorithm based on a multilevel Arbitrary Lagrangian Eulerian method [14]. This technique allows to couple in an implicit way the interface conditions.

2.4. Boundary conditions for pressure and flux in bounded domains
Now we introduce suitable outflow boundary condition in order to avoid spurious reflections of the pressure waves. In fact even if the fluid is described by parabolic equations, FSI systems have some hyperbolic behavior. As introduction of the mathematical and numerical problem, we consider the Navier-Stokes equation and we follow the method described in [13].

Let \( \mathbf{b} \) be the extension of a prescribed Dirichlet boundary values into the whole numerical domain \( \Omega \). Since we are considering the finite element method, this can be achieved by prescribing the appropriate nodal values along the boundary. Consider also \( \Gamma \) as the boundary walls (no-slip condition) and \( S_i \) as inlet and outlet boundaries. It is required that \( \mathbf{u}(t) = \mathbf{b} + \mathbf{v}(t) \), where \( \mathbf{v}(t) \in V^t_1(\Omega) = \{ \varphi \in H^1(\Omega) : \varphi|_\Gamma = 0 \} \) and \( p(t) \in L^2(\Omega), \forall t \). We have

\[ \nu(\nabla \mathbf{u}, \nabla \varphi) + (\mathbf{u} \cdot \nabla \mathbf{u}, \varphi) - (p, \nabla \cdot \varphi) = 0, \quad \forall \varphi \in V^t_1(\Omega). \]
Pressure conditions can be implicitly derived from (14), in particular the mean pressure on each free section $S_i$ is zero: $\int_{S_i} p \, ds = 0$. Now we formulate problems more generally in terms of prescribed pressure drops. This can be achieved simply considering the equation (14) such that for any prescribed $P_j$, the integral of the pressure gives $\int_{S_i} p \, ds = P_j(t)$.

Now we want to find prescribed differences between the mean pressures across the various $S_i$ (inlets and outlets). The variational problem is to find $u(t) \in V^*_i(\Omega)$ and $p(t) \in L^2(\Omega)$ such that

$$
\nu(\nabla u, \nabla \varphi) + (u_t + u \cdot \nabla u, \varphi) - (p, \nabla \cdot \varphi) = -\sum_j P_j(t) \int_{S_i} \varphi \cdot \hat{n} \, ds, \quad \forall \varphi \in V^*_i(\Omega),
$$

Using the variational formulation (15) we can now derive a system of artificial boundary conditions. All the considerations made for the Navier-Stokes equations can now be applied to the studied Koiter-FSI problem, considering that we actually deal with the fluid equations together with an embedded Robin boundary condition. For this reason the extension of the procedure (15) to the studied model is straightforward. Indeed one may wish to find the pressure drops that are required to achieve a desired net flux through each of various ducts. Since in this work we use prescribed inlet pressure, in our case this search is limited to the definition of the reference pressures at the outlets $P_j$.

2.5. Absorbing boundary condition

We prescribe an absorbing boundary condition by coupling the 3D model with a 1D reduced one [18], in order to obtain a consistent value of pressure $P_j$ to be imposed at the outlets of our domain. The guess of pressure field at the outlet is obtained from the simplified 1D model. For this purpose we consider a cylinder whose length is $L$. The simplified 1D model can be obtained by integrating at each time $t$ the Navier–Stokes equations over each section $S$ normal to the axis of the cylinder. For each $t > 0$ and $0 < z < L$ the 1D model is

$$
\begin{cases}
\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial z} = 0, \\
\frac{\partial Q}{\partial t} + \frac{\partial}{\partial z} \left( \frac{Q^2}{A} \right) + \frac{A}{\rho_f} \frac{\partial P}{\partial z} + K_R \frac{Q}{A},
\end{cases}
$$

where $Q$ is the flow rate through $S$, $A$ the area of $S$, $P$ the mean pressure over $S$, $K_R$ a resistance parameter which accounts for the fluid viscosity and $\alpha$ accounts for the shape of the velocity profile over $S$ [19]. For the closure of system (16), a third equation is provided through a pure algebraic wall model, relating the radial displacement to the mean pressure in a section $p = \frac{\beta(\sqrt{A} - \sqrt{A^0})}{\pi}$, where $A^0$ is the area of the surface $S$ at $t = 0$, $h_s$ the wall thickness, $E$ the Young modulus of the solid wall, $\nu$ the Poisson coefficient and $\beta$ is given by (4).

Following the method exposed in [12], the system now turns out to be hyperbolic, and it possesses two distinct eigenvalues. The absorbing outflow boundary condition is derived by imposing that the characteristic variable entering the 3D computational domain be zero, meaning that no information is entering. In particular, we impose

$$
W_2|_{\Gamma^\text{out}} = \left[ \frac{Q}{A} \pm \frac{2\sqrt{2}}{\sqrt{\rho_f}} \left( \sqrt{P + \beta \sqrt{A^0}} - \sqrt{\beta \sqrt{A^0}} \right) \right] |_{\Gamma^\text{out}} = 0,
$$

obtaining

$$
P|_{\Gamma^\text{out}} = P_j(t) = \left[ \frac{\sqrt{\rho_f} Q}{2\sqrt{2} A} \pm \sqrt{\beta \sqrt{A^0}} - \beta \sqrt{A^0} \right] |_{\Gamma^\text{out}}.
$$

By replacing $P_j(t)$ obtained in (18) into (15) we can obtain suitable Neumann inhomogeneous outflow boundary conditions that allow us to treat multidimensional phenomena along the studied domain.
3. Results
Some numerical results are reported in order to test the presented algorithm and the absorbing boundary conditions. We consider an axisymmetric cylinder of radius \( r = 0.05 \) m and length \( l = 1 \) m, taken from [20]. The outer wall of the cylinder is deformable. It has thickness \( h_s = 0.0025 \) m, density \( \rho = 1000 \) kg/m\(^3\), Young modulus \( E = 3 \times 10^6 \) Pa and Poisson coefficient \( \nu = 0.5 \). The cylinder is crossed by a Newtonian fluid of density \( \rho_f = 1000 \) kg/m\(^3\), and kinematic viscosity of 0.001 m\(^2\)/s. The considered time-step is \( \Delta t = 10^{-3} \) s.

![Figure 1: Pressure along the cylinder axis at different time-steps: \( t = 0.03 \) s (left top), \( t = 0.06 \) s (right top), \( t = 0.09 \) s (left bottom), \( t = 0.12 \) s (right bottom).](image1)

![Figure 2: Displacement of the structure at different time-steps: \( t = 0.03 \) s (left top), \( t = 0.06 \) s (right top), \( t = 0.09 \) s (left bottom), \( t = 0.12 \) s (right bottom).](image2)

The first simulation is carried out imposing an impulsive pressure at the inlet, represented by a step function \( p_{inlet} = 1500 \) Pa for \( t \leq 0.03 \) s and \( p_{inlet} = 0 \) Pa otherwise. At the outlet section we impose absorbing outflow boundary condition. On the outer wall Robin boundary
condition is imposed. Two different models are reported: one case with the pre-stress term ($\tilde{\sigma} \neq 0$), and another where pre-stress effects are neglected ($\tilde{\sigma} = 0$). The numerical results of the pressure along the cylinder axis and the displacement near the outer wall are reported in Figure 1 and 2, respectively. The results show that the pre-stressed simulation is smoother but with similar behavior. The imposed absorbing outflow boundary conditions allow the wave propagation in the considered domain. In particular, as one can see in Figure 3, the setting of standard outflow boundary conditions (such as zero-pressure Dirichlet boundary condition or null normal stress) is the cause of spurious wave reflections at the outlet of truncated computational domains. In fact, at the first considered time-step the two simulations show similar behavior.
Then, as soon as the pressure wave crosses the outlet boundary, wave reflection occurs with standard outflow conditions. With absorbing outflow boundary condition the wave cross the outlet without interference, so the numerical solution respects the physics of the problem.

In order to validate the algorithm and to further analyze the absorbing condition, a sinusoidal inlet pressure has been considered: \( p_{in}(t) = 7.5 \left(1 + \cos(12.5t)\right) \) [kPa]. In Figure 4 pressure field along the domain is reported for different in-phase times, with period \( T = 0.08 \) s. Since the problem has a periodic nature then a periodic solution is expected. The solution with absorbing condition shows periodic behavior, since solutions with in-phase time steps are similar to each other. On the other hand, the solution with standard outflow boundary conditions shows a non-periodic behavior where reflected waves interfere with upcoming ones.

The simulations of the axisymmetric cylinder with Robin (on the outer wall) and absorbing (on the outlet) boundary condition show overall good results and stability both with sinusoidal and impulse inlet pressure.

4. Conclusions
In this work a simple structure with thin thickness, which behaves like a membrane and deforms mainly in the normal direction, has been considered as a fluid structure interaction problem. This structural model has been embedded in a Newtonian fluid region. Thanks to this approach it is possible to reduce the computational cost of the fluid structure simulations, since the dimensionality of the solid is reduced by one. We have also introduced absorbing boundary condition in order to reduce the spurious wave reflections at the outlet of truncated computational domains. The model has been implemented on a finite element code and tested on axisymmetric geometry to show the results both with an impulse inlet pressure and a sinusoidal one. These tests have shown good stability properties of the model and remarked the importance of suitable outflow absorbing boundary conditions for truncated domains in order to avoid spurious wave reflections and the possible interference with upcoming waves.

References
[1] Formaggia L, Quarteroni A & Veneziani A. 2009. Cardiovascular Mathematics I, Springer, Heidelberg.
[2] Bazilevs Y, Takizawa K & Tezduyar T E. 2013. John Wiley & Sons.
[3] Tian F B, Dai H, Luo H, Doyle J F, Rousseau B. 2014. J. Comput. Phys. 258 451–469.
[4] Nobile F, Pozzoli M, Vergara C. 2014. J. Comput. Phys. 273 598–617.
[5] Causin P, Gerbeau J F, Nobile F. 2005. Comput. Methods Appl. Mech. Eng. 194 4506–4527
[6] Michler C, Huilshoff S J, Van Brummelen E H, De Borst R. 2004. Comput. Fluids 33 839–848.
[7] Hron J & Turek S. 2006. Fluid-structure interaction, 146-170, Springer, Berlin, Heidelberg.
[8] Guermond J L, Minev P, Shen J. 2006. Comput. Methods Appl. Mech. Eng. 195(44) 6011–6045.
[9] Nobile F & Vergara C 2008. SIAM Journal on Scientific Computing, 30(2), 731-763.
[10] Heywood J G, Rannacher R & Turek S. 1996. Int. Journal for Numerical Methods in Fluids, 22(5), 325-352.
[11] Cerroni D, Manservisi S & Menghini F. 2015. Int. Journal of Mathematical Models and Methods in Appl. Sciences, 9, 227-234.
[12] Formaggia L, Gerbeau J F, Nobile F & Quarteroni A. 2001. Comput. Methods Appl. Mech. Eng., 191(6-7), 561-582.