

# Indicative results and progress on the development of the unified single solution method for fluid-structure interaction problems

***Citation for published version (APA):***

Giannopapa, C. G., & Papadakis, G. (2007). *Indicative results and progress on the development of the unified single solution method for fluid-structure interaction problems*. (CASA-report; Vol. 0711). Technische Universiteit Eindhoven.

***Document status and date:***

Published: 01/01/2007

***Document Version:***

Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

***Please check the document version of this publication:***

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

***General rights***

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

[www.tue.nl/taverne](http://www.tue.nl/taverne)

***Take down policy***

If you believe that this document breaches copyright please contact us at:

[openaccess@tue.nl](mailto:openaccess@tue.nl)

providing details and we will investigate your claim.

# INDICATIVE RESULTS AND PROGRESS ON THE DEVELOPMENT OF THE UNIFIED SINGLE SOLUTION METHOD FOR FLUID-STRUCTURE INTERACTION PROBLEMS

**C. G. Giannopapa**

Dept. of Mathematics and Computer Science  
Eindhoven University of Technology  
PO Box 513, 5600 MB Eindhoven  
The Netherlands  
Email: c.g.giannopapa@tue.nl

**G. Papadakis**

Experimental and Computational Laboratory for the  
Analysis of Turbulence  
Department of Mechanical Engineering  
King's College London, Strand WC2R 2LS, UK  
Email: george.papadakis@kcl.ac.uk

## ABSTRACT

This paper presents the progress on the development of a novel unified solution method for solving strongly coupled fluid-structure interaction problems. The method has been developed and fully tested for solids in [1]. The new approach is based on continuum mechanics formulation for fluids and structures where both continua can be solved using the momentum and continuity equation. The difference between the two continua lies in the constitutive equations. In this framework a single set of equations is used for the simultaneous solution of both fluid and solid. The common equations are written such that velocity and pressure are unknown variables for both continua. The discretisation method used for the solution of the problems is finite volumes. The physical interface between the two continua is treated as an internal part of the computational domain and no explicit exchange of information is needed. The test case used to demonstrate the idea is wave propagation in liquid filled flexible vessels. The solution is fully implicit and transient. Results regarding pressure, velocity and wall distention at different times and various locations along the tube are presented. The method is stable and robust and can be used for the next step of development and validation against classical analytical and numerical models.

## NOMENCLATURE

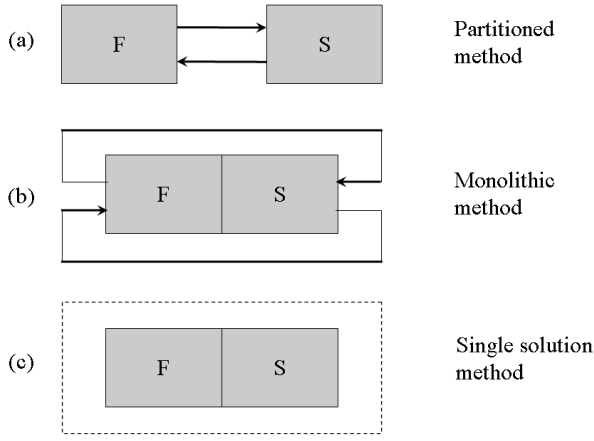
$Co$		Courant number
$c$	m/sec	wave speed
$D$	m	diameter
$f$	Hz	frequency
$h$	m	thickness
$I$		unit tensor
$K$	Pa	bulk modulus
$L$	m	length
$F$		fluid
$p$	Pa	pressure
$S$		solid

$t$	sec	time
$U$	m/sec	velocity
$\Delta t$	s	time step
$\Delta x$	m	grid interval
$\eta$	m <sup>2</sup> /s	dynamic viscosity
$\lambda$	Pa	Lamé's coefficient
$\mu$	Pa	Lamé's coefficient
$\varepsilon$		strain tensor
$E$	Pa	Young's modulus
$\nu$		Poisson ratio
$\rho$	kg/m <sup>3</sup>	density
$\sigma$	Pa	Cauchy stress tensor
$\tau$	Pa	applied end shear
$\omega$	Hz	frequency of undamped oscillations

## INTRODUCTION

Fluid structure interaction occurs in many areas of engineering (aerospace, civil or mechanical) as well as other scientific disciplines including medicine, biomechanics etc. During this interaction, the normal and shear stress due to the fluid flow act on the structure and cause it to deform which in turn affects the fluid flow and consequently the stress of the fluid. Thus, the response of the system can be determined only if the coupled problem is solved.

In the conventional approach for fluid-structure interaction problems, the fluid and solid components are treated separately and information is exchanged at their interface. According to the conventional terminology, the current numerical methods can be grouped in two major categories: Partitioned methods and monolithic methods. Both methods use two separate sets of equations for fluid and solid. In Figure 1 a schematic description of these methods is depicted.



**Figure 1: Classification of FSI methods**

### Partitioned methods

In the partitioned methods the fluid and the solid solution are partitioned. The fluid dynamics and structural dynamics equations are solved separately in a consecutive manner and the information regarding the dynamic and kinematics interface conditions is done *asynchronously*. In many cases two separate solvers are used for the solution of the fluid domain and the solid domain. Data transfer between the two solvers is done typically by the use of another algorithm. Many researchers have focused their research in optimizing the interface exchange algorithms. An overview of the advantages and the disadvantages of using such a method can be found in [2]. In cases where there is weak coupling between the fluid and solid, partitioning methods may be beneficial. On the other hand in cases where there is strong coupling partitioning leads to higher computational time and may inherently lead to loss of conservation of the properties of the continua and instabilities.

### Monolithic methods

In the monolithic methods the fluid and solid solution are still obtained using separate sets of equations for the two media in a coupled manner and information at the common interface is done *synchronously*. A single mesh description is typically used to describe the computational domain. For every time step the equations of the fluid dynamics and structural dynamics are solved simultaneously implicitly and information is exchanged in the interface synchronously. The equations are solved by sub-iteration until convergence is obtained within each time step. These methods are quite complex and computationally expensive due to the sub-iteration. These methods are suitable for highly coupled fluid structure interaction problems. Examples of these methods can be found in [3,4].

A novel unified single solution method has been developed and tested for structural dynamic problems [1]. This method is different from partitioned and monolithic methods. The unified single solution method treats both fluid and solid as a continuum, thus the whole computational domain is a single entity in a single grid. Its behavior is described by a single set of equations and is solved fully implicit.

The objective of this paper is to demonstrate the unified single solution method on a full fluid-structure interaction problem and present some preliminary results.

## **MATHEMATICAL FORMULATION**

This section is concerned with presenting the mathematical model used to describe the unified single solution method for fluid structure interaction problems as described in [1, 5-7]. This mathematical formulation is based on the continuum mechanics where the fluid flow and the solid motion for both fluid and solid can be described by the momentum and the continuity equations. In an Eulerian framework these read [8]:

*Momentum equation neglecting body forces:*

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = \nabla \cdot \boldsymbol{\sigma} \quad (1)$$

*Continuity equation or mass conservation:*

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (2)$$

The difference between the two continua lies on the constitutive equations of the stress tensor.

For Newtonian compressible fluids the constitutive equation for the stress tensor,  $\boldsymbol{\sigma}$  is given by:

$$\boldsymbol{\sigma} = 2\eta \text{dev}(\dot{\boldsymbol{\varepsilon}}) - p\mathbf{I} = \eta \left( \nabla \mathbf{U} + \nabla \mathbf{U}^T - \frac{2}{3} \text{tr}(\nabla \mathbf{U}) \right) - p\mathbf{I} \quad (3)$$

where  $\boldsymbol{\varepsilon}$  is the strain tensor,  $\dot{\boldsymbol{\varepsilon}}$  is the strain rate tensor and  $\text{dev}(\cdot)$  is the deviatoric part of the tensor.

For linear elastic or Hookean solids the constitutive equation for the stress tensor,  $\boldsymbol{\sigma}$  is given by:

$$\boldsymbol{\sigma} = 2\mu \boldsymbol{\varepsilon} + \lambda \text{tr}(\boldsymbol{\varepsilon})\mathbf{I} \quad (4)$$

In order to obtain a single formulation for both fluid and solid the constitutive equation of the solid are rewritten such that velocity and pressure appear as prime variables in the stress tensor. This can be obtained if the stress tensor is split into its deviatoric and hydrostatic part. This way of formulating the stress tensor has been previously used in [9]. The stress tensor reads:

$$\boldsymbol{\sigma} = \text{dev}(\boldsymbol{\sigma}) - p\mathbf{I} \quad (5)$$

In [9] a displacement-pressure formulation is used. In this work we obtain a velocity-pressure formulation by using the integral relationship of velocity and displacement:

$$\mathbf{D} = \int_{t_n}^{t_n + \Delta t} \mathbf{U} dt \quad (6)$$

Any integration method can be used. In the present paper the trapezoid rule is used.

The momentum equation of the unified single solution method for both fluid and solid reads:

$$\frac{\partial \rho_c \mathbf{U}}{\partial t} + \nabla \cdot (\rho_c \mathbf{U} \mathbf{U}) = \nabla \cdot (\phi_c \nabla \mathbf{U}) + \nabla \cdot (\phi_c \nabla \mathbf{U}^T) - \frac{2}{3} \phi_c \text{tr}(\nabla \mathbf{U}) + \nabla \cdot \text{dev} \Sigma^c - \nabla p \quad (7)$$

Here  $\phi$  is the function that describes both fluid and solid coefficients:

$$\phi = (1-c)\eta + c \frac{\Delta t}{2} \mu \quad (8)$$

and  $c$  defines the consecration of the fluid or solid, e.g.  $c=1$  in the solid domain and  $c=0$  in the fluid domain. In the case where

$c=0$  the term  $\nabla \cdot dev \Sigma^*$  will not be present as it would have not been accumulated from previous information obtained.

An implicit expression is obtained for pressure using the barotropic relationship and the continuity equation. For small variations of pressure about a reference pressure  $p_0$  the barotropic relationship is linearised as:

$$\rho = \rho_0 [1 + (p - p_0) / K] \quad (9)$$

In the case where incompressibility is assumed, the density is constant and the second term on the right hand side is zero.

$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$
$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$	$\rho = \rho_s$ $c = 1$ $\phi = \mu \Delta t / 2$
$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$
$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$
$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$	$\rho = \rho_f$ $c = 0$ $\phi = \eta$

**Figure 2: The different properties distribution in the single mesh for solving fluid-structure interaction problems with the unified solution method.**

In Figure 2 a schematic of how the unified single solution method can be applied in a single grid is shown. A single grid is considered and the two phases of the continuum can be distinguished by the different coefficients. The physical interface between the two continua is not explicitly identified and now exchange of information is performed during computations. This idea can also be used in case of large deformations and move the grid in an Arbitrary-Langragian-Eulerian (ALE) fashion.

## DISCRETISATION METHOD

The finite volume method was used for the discretisation of the equations, but the principal idea is not restricted to finite volumes. The system of momentum and continuity equations needs to be solved implicitly for velocity and pressure. The solutions of these equations is complicated because they are highly coupled.

The momentum equation in a semi-discretised form reads:

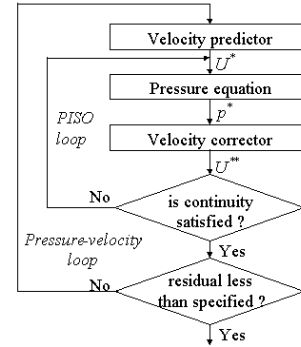
$$\alpha_p U_p = H(U) - \nabla p \quad (10)$$

where  $H(U)$  contains the contributions of the surrounding nodes, previous time step as well as all other source terms except pressure. Substituting Equation (9) in equation (2) we obtain the following equation for pressure:

$$\frac{\partial(\psi p)}{\partial t} + \nabla \cdot [(\rho_0 - \psi \rho_0) \mathbf{U}] + \nabla \cdot (\psi \rho \mathbf{U}) = 0 \quad (11)$$

Where  $p_0$  is the reference pressure for which  $\rho(p_0) = \rho_0$  and  $\psi = \rho_0 / K$ .

The velocity pressure coupling is achieved using the Pressure Implicit with Splitting of Operators (PISO) algorithm [10]. The PISO algorithm is most suitable for transient problems and can be used for compressible flows in an iterative manner. The PISO algorithm was used for the first time for solids in [1,6] and proved to be a good choice for a velocity pressure formulation for solids. Here it is used for the solution of the full fluid-structure interaction problem. A PISO algorithm flow chart can be seen in Figure 3. The method is not restricted to using the PISO algorithm, e.g. the Simple algorithm can be used as well.



**Figure 3: PISO algorithm flow chart**

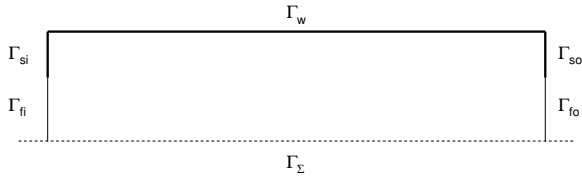
The momentum equation is solved in order to obtain the predicted values for the velocity  $U^*$  field at the new time step. Using these values and the pressure equation (11) a new estimated pressure field  $p^*$  is obtained. This pressure field is correct when the PISO loop has converged after a couple of iterations. Finally, using the new estimated pressure fields the velocity field  $U^{**}$  is updated, as well as the density. For the evaluation of velocities at the faces of the control volumes, the Rhie and Chow interpolation method is used [11]

Issues related to the stability of the numerical method for the solid part can be found in [1,7].

## SIMULATION MODEL

The wave propagation of a wave in an elastic straight tube was simulated to demonstrate the unified single solution method. This case was chosen because experimental measurements have been previously obtained for flexibles tubes and will be used in the future for the full validation of the method.

The tube has a length of 100 mm and an inner diameter of 25 mm and outer diameter of 27 mm. An EPDM rubber tube was considered with the Young's modulus of the tube being 2 MPa. The Poisson's ratio is 0.3. The tube is filled with water. The pressure applied at the inlet was 1.73 kPa. The tube is considered axi-symmetric, thus the problem can be solved in 2D and only half of the tube needs to be modeled. A grid of 3010 cells was used where 70x3 were used for the solid and 70x40 were used for the fluid.



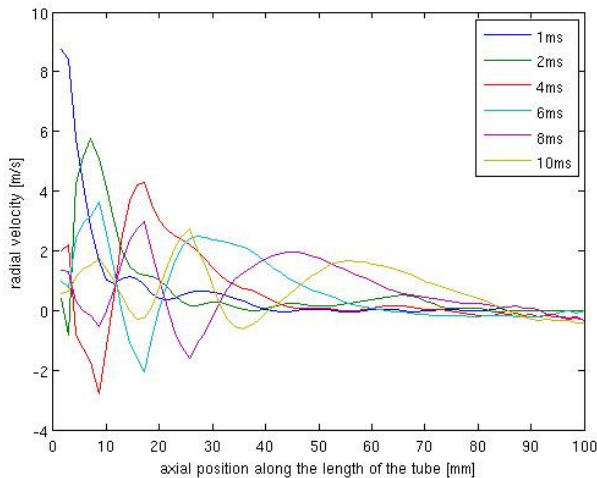
**Figure 3: Boundary settings.**

The schematic of the boundary description of the domain concerned is shown in Figure 3 with  $\Gamma_\Sigma$  at  $r=0$  being a symmetry boundary ensuring that there is no flow across the axis and that the gradients of all variables in the radial direction are locally zero. Symmetry or slip boundary conditions were also applied on the left and right part of the wall in  $\Gamma_{si}$  and  $\Gamma_{so}$ . The inlet and outlet of the fluid domain  $\Gamma_{fi}$ ,  $\Gamma_{fo}$  was fixed pressure and zero gradient. It should be noted that as we use a single set of equations, thus the interface between fluid and solid is part of the inner computational domain and no explicit exchange of information is needed.

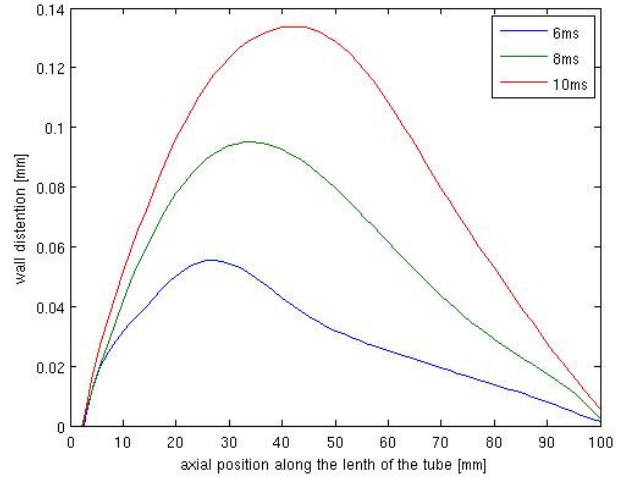
## RESULTS

The time integration scheme used was first order accurate Euler implicit. The time step used for the calculations presented were  $2e-7$  and grid size was  $1.4e-3$  corresponding to a Courant number of 0.0013. Convergence was achieved using two PISO loops and two external velocity-pressure loops for each time step.

The solution of the pressure along the symmetry axis of the tube at different times and can be seen in Figure 4. The axial position is measured from the wave entrance at the left side of the tube.



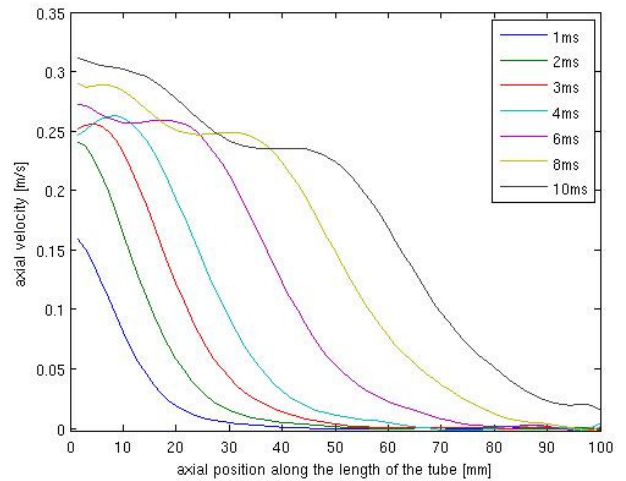
**Figure 4: Velocity along the tube's axis of symmetry.**



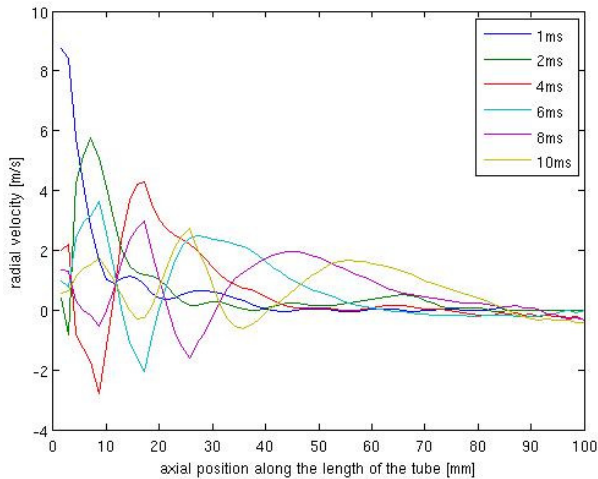
**Figure 5: Wall distention measured at the top of the tube.**

The wall distention is measured at the top of the tube along its length at different times and is presented in Figure 5.

Axial and radial velocities were measured at different times. The axial velocity was measured at the center of the tube and can be seen in Figure 6, The radial velocity was measured at the top of the tube wall and can be seen in Figure 7.



**Figure 6: Axial velocity along the tube's axis of symmetry.**



**Figure 7: Radial velocity at the top of the tube.**

## CONCLUSIONS

The idea of the unified solution method was presented and illustrated solving a fluid structure interaction problem in flexible tubes. The method is based on the idea that a single set of equations for both fluid and solid can be used for the solution of fluid structure interaction problems without the need to exchange information at the physical interface of the two continua. The two phases can be distinguished by their different local coefficients. The equations are describing 3 D and are solved in a fully implicit manner. This method gives an advantage over potential instabilities and additional computational cost that may appear using conventional monolithic and partitioned methods.

It is believed by the authors that the method in itself is general and it is not restricted to the type of the discretisation method used; both finite elements and finite volumes can be used. The method was presented for compressible continua without restricting its applicability to incompressible cases. The equations are for elastic solids can be further extended to include viscoelasticity. The test case used was taking into account small deformations. Large deformation handling can also be further included using an arbitrary-Lagrangian-Eulerian large strain formulation for the continuum.

The next step of development of this method will be to validate it against analytical wave propagation models [1,12] and compare it to other computational methods well established in the field. The method will be extended it for viscoelastic materials and compared against experimental measurements [1].

## ACKNOWLEDGMENTS

The authors would like to thank Marie Currie Research Training Networks Fellowships, European Commission for funding this project.

## REFERENCES

- [1] Giannopapa, C. G., 2004, *Fluid-Structure interaction in flexible vessels*, PhD thesis, University of London.
- [2] Felippa, C. A., Park, K. C. and Farhat, F., 2001, "Partitioned analysis of coupled mechanical systems. Computer methods in applied mechanics and engineering," **190**, pp. 3247-3270.
- [3] van Brummelen, E. H. and Koren, B., 2003, "A pressure-invariant conservation godonov-type method for barotropic two-flud flows," *Journal of computational physics*, **185**, pp. 289-308.
- [4] de Hart, J., Peters, G.W.M. Schreurs, P.J.G., Baaijens, F.P.T., 2000, "A two-dimensional fluid structure interaction model of the aortic valve" *Journal of Biomechanics*, **33**, pp.1079-1088.
- [5] Giannopapa, C. G. and Papadakis, G., 2004, "A new formulation for solids suitable for a unified solution method for fluid-structure interaction problems," *ASME PVP 2004*, San Diego California, July 2004, PVP-Vol. **491-1**, pp. 111-117.
- [6] Giannopapa, C. G. and Papadakis, G., 2006, "New formulations of the dynamic equations of elastic solids suitable for a unified methodology for fluid-structure interaction problems", *Computer Methods in Applied Mechanics*, submitted.
- [7] Giannopapa, C. G., and Papadakis, G., "Linear stability analysis and application of a new solution method of the elastodynamic equations suitable for a unified fluid-structure-interaction approach", *Journal of Pressure Vessels Technology*, in press.
- [8] Malvern, L.E., 1969, *Introduction to the mechanics of a continuous medium*, Prentice Hall, Englewood Cliffs, New Jersey.
- [9] Bathe, K. J., 1997, *Finite element procedures*. Prentice Hall, Englewood Cliffs, New Jersey.
- [10] Issa, R. I., 1986, "Solution of the implicit discretised fluid flow equation by operator-splitting", *Journal of Computational Physics*, **62**, pp 40-65.
- [11] Rhie, C. M., Chow, W. L., 1983, "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," *AIAA Journal*, Vol. 21, pp. 1527-1532.
- [12] Lighthill, J. 1975, "Waves in fluids", Cambridge University Press, Cambridge, UK.