Comparative Analysis of Computational Methods in Fluid-Structure Interaction:

Temporal discretization and coupling techniques

Endashaw T. Woldemariam and Hirpa G. Lemu
Dept. of Mechanical and Structural Engineering and Materials Science
University of Stavanger,
Stavanger N-4036, Norway
Endashaw.t.woldemariam@uis.no, Hirpa.g.lemu@uis.no

Abstract* — Fluid-structure interaction problems occur in a wide variety of science and engineering fields. In solving such problems, two domains of different characteristics shall be modeled and analyzed through separate kinematic equations. The complexity in the computation of such kind of problems arise mainly from the modeling and analysis of the problem at the interface. Thus, the interfacing condition is also treated as an independent part. A range of computational techniques based on spatial and temporal discretization schemes have been proposed and being employed in solving diverse FSI problems. Monolithic and partitioned approaches are the two broad classes of FSI numerical analysis approaches under the temporal discretization scheme. The monolithic approach requires development of complex mathematical model to represent the whole FSI domain collectively. The partitioned approach, unlike the monolithic approach, make good use of the existing developed and advanced tools with segregate analysis. Nevertheless, the latter requires an additional external code for coupling at the interface. Based on the behavior of the coupling conditions at the interface, FSI computational methods are also categorized under either one-way or two-way coupled approaches. Moreover, the formulation scheme chosen to describe mesh motions, spatial discretization and more other approaches determine classification schemes of FSI solution techniques. In line with the broader classifications based on temporal discretization and coupling schemes, simulation results on a benchmark problem employing selected computational approaches have been analyzed and discussed. Results from a strongly coupled two-way partitioned approach, from an open source code, is compared against one-way coupled partitioned approach utilizing ANSYS code (academic research mechanical). The strongly coupled portioned approach shows a more realistic result than the one-way coupled system.

Keywords—Fluid-Structure interaction; temporal discretization; monolithic and partitioned method; one and two-way coupling

I. INTRODUCTION

Various fields of science and engineering applications face challenges from the occurrence of fluid structure interaction (FSI) problem. For instance, the interaction of rotating and stationary parts in pumps and turbines against water in hydrodynamics and in offshore constructions [1, 2], the interaction of pulsating blood flow against the wall of cardio vascular vessels in medical science[3, 4], and others can be mentioned. In such FSI problems, two different domains, as shown in Fig. 1, are involved and analyzed by different kinematic equations. Furthermore, their interface is considered and analyzed separately. Thus, three basic components should be modeled mathematically in solving FSI problems: (i) fluid, (ii) structure and (iii) the interface.

The computational challenges in numerically solving FSI problems, perhaps, arise mainly in dealing with how the interfaces have to be treated in modeling the coupling of the two domains and the discretization techniques that should be employed. As a result, a number of research works in diverse applications based on different coupling conditions and discretization schemes are emerging and reported [5-7].

Navier-Stokes equations in fluid dynamics and dynamic equilibrium equations in structural continuum mechanics based on the conservation laws of momentum, mass and energy are among the governing kinematic equations in modeling FSI problems. Coupling of the two models are undertaken through well-known coupling conditions, called Dirichlet and Neumann coupling conditions [8]. Moreover, there is another coupling condition based on Robin's condition [6] that modifies the former two conditions and addresses both conditions through a single model. Algorithms of the coupling conditions are discussed in detail latter in this paper. In the former coupling conditions, the fluid domain at the interface obtains the velocity resulting from the structural analysis, referred to as Dirichlet condition. On the other hand, the resulting traction load from computational fluid dynamics (CFD) analysis of the fluid model is transferred to the structure domain at the interface, referred to as Neumann condition.

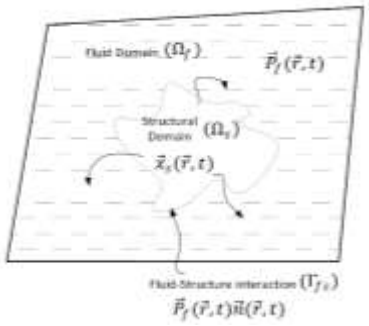


Fig. 1. Fluid-Structure interaction model of a simple problem

Based on how the coupling conditions are enforced at the interface, coupling approaches of computational methods are categorized as a *one-way* or *two-way* coupled computational methods. Moreover, depending on the intensity of the coupling, the two-way coupling may also be stated as weak or strong. Apart from that, computational methods are broadly classified as *monolithic* or *partitioned* approach based on their temporal discretization scheme [8-10]. Available commercial codes often incorporate the partitioned based concept, for instance ANSYS utilizes immersed boundary and domain partitioned methods, for 2D and 3D problems analysis respectively, for both one-way and two-way coupled FSI problems [11].

Most FSI computational techniques employ finite element method (FEM) to analyze and simulate the structural model as it is a well-developed tool. However, with the recent development on *Isogeometric Analysis* (IGA) method a considerable number of studies reported to show a better performance compared to FEM, especially IGA analysis results of FSI problems that involve large deformations [4, 12]. Similarly, finite volume (FV) based analysis methods are utilized mostly to solve the fluid model.

In this article, concepts of FSI computational methods categorized based on temporal discretization scheme and coupling conditions are discussed. Moreover, simulation results of computational approaches on a benchmark problem utilizing ANSYS 17.0 and an open source codes (Open FOAM) are analyzed and discussed. In general, the paper is organized as follows. In the next section, mathematical models of governing kinematic equations are discussed followed by discussion of coupling conditions and computational techniques based on the spatial and temporal discretization approaches in Section III. In Section IV, the comparative simulation results are presented and analyzed. Finally, in Section V, the drawn conclusions are given.

II. MATHEMATICAL MODELS: GOVERNING EQUATIONS IN FSI PROBLEMS

Both the fluid and structural domains in any FSI numerical analysis processes are described by governing kinematic equations derived from the conservation laws, which highly determine the mathematical models.

Moreover, the basic mesh motion formulations adopted to represent both in fluid and structural domains largely determine the performance of any computational approach. In most cases, either Eulerian or Lagrangian formulation techniques, which are the classical mesh motion formulations in continuum mechanics, are employed to describe mesh motions. The basic difference in the concept of Eulerian and Lagrangian formulation is that, in the Lagrangian formulation, the model equations are represented with respect to the reference frame of the original domain, i.e. with no deformation. In the Eulerian formulation, however, it is represented with respect to the reference frame of the deformed domain, by tracking the particles in the domain. Mostly, structural models are perfectly formulated using Lagrangian description while the fluid models are best formulated using Eulerian description for Lagrangian description does not constitute a convective term in it, which may lead to an element entanglement in cases of large flow deformation.

Arbitrary Lagrangian-Eulerian (ALE) mesh motion formulation is introduced in order to benefit from the positive features of both formulations. This formulation approach incorporates the material, spatial and referential descriptions of the structure and fluid domains, which enables solving the discretized governing equations in structural and fluid dynamics [13]. The approach is formulated and applied in various incompressible and viscous hydrodynamic flow problems [14 - 16]. This formulation technique is popular in numerical methods in the monolithic class and most other FSI computation techniques as well [8, 10].

Depending on the problem that one is dealing with, the models of the governing kinematic equations will have different characteristics that the governing equations would have to be modified accordingly. For instance, based on the fluid flow condition the fluid could be modeled as either turbulent or laminar, and either viscous or inviscid based on the property of the fluid medium. Similarly, depending on the boundary and loading conditions, and the characteristic property of the material in the problem, structural models could be modeled as either rigid or flexible, static or moving, or a combination of any of the aforementioned characteristic models.

A. Governing Kinematic equations in structural mechanics

Applying the conservation of momentum in continuum structural mechanics, based on the D'Alembert's principle, the governing equation of the structure domain is defined. Considering a continuum solid model, shown in Fig 1, that occupy finite volume, Ω_s, bounded by a closed surface, Γ_s, the governing equation is expressed as

$$\rho_s \ddot{\vec{x}}_s(\vec{r},t) - \nabla \cdot \bar{\bar{\sigma}}_s = \vec{f}_s(\vec{r},t); \ \forall \vec{r} \in \Omega_s \tag{1}$$

where: $\bar{\bar{\boldsymbol{\sigma}}}_{\boldsymbol{s}}$ is Cauchy stress tensor given by:

$$\bar{\bar{\sigma}}_s = \lambda_s \mathrm{Tr}(\bar{\bar{\varepsilon}}_s)\bar{\bar{I}} + 2\mu_s \bar{\bar{\varepsilon}}_s \tag{2}$$

And the strain tensor, $\bar{\bar{\boldsymbol{\varepsilon}}}_{\boldsymbol{s}}$ is given by

$$\bar{\bar{\varepsilon}}_s = \frac{1}{2}\left(\overline{\overline{\nabla \vec{x}_s}} + \left(\overline{\overline{\nabla \vec{x}_s}}\right)^T\right) \tag{3}$$

where ρ_s- is density of the solid medium, $\vec{f}_s$ -element force on the solid domain, $\ddot{\vec{x}}_s$ - double time derivative of displacement vector of the solid domain, λ_s-Lame's Constant for the solid medium, Tr-traction function, $\bar{\bar{I}}$-Identity matrix, μ_s-poisons ratio of the solid material, $\vec{x}_s$ -displacement vector of the solid medium, and $\vec{x}_s(\vec{r},t)$- time dependent displacement vector of the solid medium.

B. Governing Kinematic Equations in fluid dynamics

Generally, fluid dynamics models can be expressed by the general transport equation (4). In this formulation, the equations from the conservation of mass (continuity equation) and momentum (Navier-Stokes equation), can be obtained, for both steady state and transient problems based on the problem type and the assumptions considered, by simply replacing corresponding variables.

The general transport equation in fluid dynamics is expressed by

$$\underbrace{\frac{\partial}{\partial t}(\rho\psi)}_{Transient\,term}+\underbrace{\frac{\partial}{\partial \vec{u}}\left(\rho\vec{v}\psi\right)}_{Convention\,term}=\underbrace{\frac{\partial}{\partial \vec{u}}\left[\Gamma_{\phi}\frac{\partial\psi}{\partial \vec{u}}\right]}_{Diffusion\,term}+\underbrace{S_{\psi}}_{Source\,term} \tag{4}$$

As indicated in Eq. (4), the first and last terms in the left hand side of the equation are the transient and the convection terms respectively, while the right hand side of the equation contains the diffusion term and the source terms. The relations of the governing equations with the general transport equation are given by substituting some variables in the general equation. These substituting variables are given in Table I.

The simplified continuity and Navier-Stokes equation for an incompressible fluid are expressed as:

Continuity equation

$$\nabla\cdot\vec{v}=0 \tag{5}$$

Navier-Stokes equation

$$\frac{\partial}{\partial t}\vec{v}+\vec{v}\cdot\nabla\vec{v}-\vec{v}\cdot\mu_0\nabla\vec{v}=-\nabla\vec{P} \tag{6}$$

The Navier-Stokes equation can also be described in stress tensor form as a function of velocity and pressure as:

$$\rho^{f}\frac{\partial}{\partial t}\vec{v}-\nabla.\overline{\overline{\boldsymbol{\sigma}}}^{\boldsymbol{f}}(\vec{v},\vec{p})=0 \tag{7}$$

For anisotropic Newtonian fluid, the viscous stress tensor of the fluid is expressed as:

$$\overline{\overline{\boldsymbol{\sigma}}}^{\boldsymbol{f}}=2\mu_o\overline{\overline{\varepsilon}}_f+\lambda_f\,\mathrm{Tr}\,[\overline{\overline{\varepsilon}}_f]\overline{\overline{\mathrm{I}}} \tag{8}$$

and the rate of strain tensor of the fluid domain, $\overline{\overline{\dot{\varepsilon}}}_f$, is given by:

$$\overline{\overline{\dot{\varepsilon}}}_f=\frac{1}{2}(\nabla\vec{v}+[\nabla\vec{v}]^{T}) \tag{9}$$

where $\overline{\overline{\boldsymbol{\sigma}}}^{\boldsymbol{f}}$- is Cauchy stress gradient in the fluid domain, λ_f- is Lame constant of the fluid medium, μ_o- is viscosity of the fluid medium, $\vec{p}$ - applied pressure density ratio vector on the fluid domain, $\vec{v}$ - velocity vectors of the fluid particles, $\overline{\overline{\dot{\varepsilon}}}_f$- time derivative of strain vector of the fluid domain.

TABLE I. RELATIONS BETWEEN GENERAL TRANSPORT EQUATIONS AND GOVERNING EQUATIONS IN FLUID DYNAMICS

Equation type	**Variables substitution**		
	Ψ	$\Gamma_{\emptyset}$	S_{Ψ}
Continuity equation (mass conservation equation)	1	0	0
Navier-stokes equation (momentum equation)	$\vec{v}$	$\boldsymbol{\mu_o}$	$\partial\vec{p}/\partial\vec{u}$

The variables in the table are defined as follows:

μ_o is the viscosity coefficient,

$\vec{p}$ is field pressure tensor and

$\vec{v}$ is velocity vector of fluid particles.

III. FLUID-STRUCTURE INTERFACE COUPLING CONDITIONS AND NUMERICAL APPROACHES

Neumann and Dirichlet conditions are well-practiced interface conditions in solving FSI problems that are also known as explicit coupling schemes. These conditions are being adopted in most FSI computational methods in order to hold the no-slip conditions at the interface. They enable the fluid and structure models to not only share a common location but also hold the same velocity and normal stress at the interface. To advance the computational solutions the standard Dirichlet condition, most often, imposes the common velocity to the fluid field at the interface, which maintain the no-slip condition along the fluid structure interface. However, these conditions are known to be unconditionally unstable in problems with the fluid density close to the density of the structure, as it leads to large added mass effect [17, 18].

A. Coupling conditions

Based on the Neumann-Dirichlet conditions, the nodal velocities of the fluid and structure at the interface, in Fig 2, should be equal and are defined by

$$\dot{\vec{x}}_i(x_i,t)=\vec{v}_i \tag{10}$$

Therefore, through time integration of Eq. (10), the common interface location condition perhaps is imposed by

$$x_i=\vec{u} \tag{11}$$

On the other hand, the Neumann standard condition imposes the common normal stress equivalent to the static pressure to the structural field. Thus, the normal stresses at each nodes at the interface are equal, and given as

$$\sigma_{ij}^{s}.\boldsymbol{n}=\sigma_{ij}^{f}.\boldsymbol{n} \tag{12}$$

where σ_{ij}^{s}- and σ_{ij}^{f} are stress components at ij location of the fluid interface, $\boldsymbol{n}$ is the normal vector at the interface and $\vec{u}$ - is deformation vector of fluid particles at the interface.

Considering $\Omega_f \in R^d$ $(d=2,3)$, $\Omega_s \in R^d$ and $\Gamma_f, \Gamma_s \in R^d$, shown in Fig 2, for a time step $\Delta t>0$, time size of$t_n = n\Delta t, where\ n \in N$, the first order backward difference in time denotes velocities as

$$\dot{x}^n=v^n=\frac{\partial}{\partial t}x^n=\frac{x^n-x^{n-1}}{\Delta t} \tag{13}$$

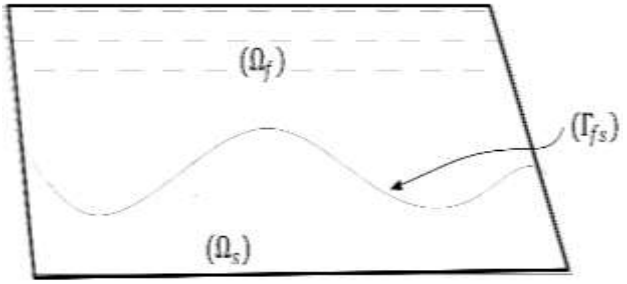


Fig. 2. Simple fluid-structure interface model

Another interface condition known as Robin transmission condition [5], which is an implicit condition scheme, is introduced to replace the Dirichlet and Neumann's individual conditions following the conventional block Gauss-Seidel scheme. The method is a weighted linear combination of the Dirichlet and Neumann conditions. While implementing this condition, the weighted coefficients should carefully be evaluated in order to hold the stability of the method. Robin-Robin method was initially introduced as an alternative stable explicit coupling method [19]. Among many investigations carried out using the transmission conditions by researchers [5], on a simplified fluid-structure model, it is observed that best convergence property is achieved using the Robin-Neumann algorithm, which imposes the Robin transmission condition on the fluid field and the Neumann condition on the structure field.

B. Numerical Approaches Based on Different Coupling Conditions and Discretization Schemes

Various taxonomies of computational methods are available in FSI problem based on spatial and temporal discretization schemes, coupling conditions and others. Classification of approaches based on temporal discretization and coupling conditions are discussed in this subsection.

Computational approaches based on temporal discretization scheme: In computational analysis of FSI problems, computational approaches decide whether to solve the individual models separately in a separate time marching or solve the whole domain collectively as a single model in each time step throughout the entire time marching period. These choices have led to the classification of computational approaches based on temporal discretization, first introduced back in 1983 [20]. Thus, depending on the choice, computational methods are categorized under either monolithic or partitioned approach [8-10], which describes the broad classification under temporal discretization. Of the two approaches, the former urges one to deal with complex mathematical models that represent the entire domains. Numerically conquering FSI problems, however, has been developed since the early 1970th with the partitioned based approach [20]. This approach make use of the existing FEM and CFD codes to solve the structural and fluid parts respectively. Moreover, regardless of the growing computational power and development of different analysis techniques, the monolithic approach remained undeveloped due to the complexity in the mathematical modeling, especially at the interface. As it is shown in Fig. 3 (a), under the partitioned approach, the analysis of individual domain vectors are undergone in a separate time marching (as t_n^f), where as in the monolithic approach, Fig. 3 (b), each domain vector is analyzed through a similar time marching (as t_n).

Most introduced spatial discretization and interface treatment techniques employ the partitioned approach in order to utilize the well tested and developed separate computational methods from both fluid and structural dynamics.

Computational approaches based on coupling scheme: Apart from the broad classifications discussed above, computational method in FSI are also broadly classified as a one-way coupled or two-way coupled methods [7, 10]. This classification is based on whether the analysis result from the structural domain is imposed on to the fluid domain at the interface, the Dirichlet condition, or not. In the one-way coupling approach, Fig. 4 (a), the responses from the structural analysis at the interface are not transferred to the fluid domain. However, in the two-way approach, Fig. 4 (b), responses from the analysis of each domains are transferred to one another at each time step.

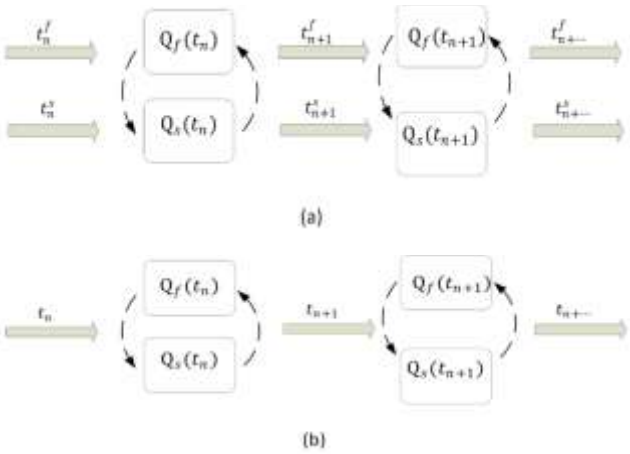


Fig. 3. Temporal discretization approaches (a) Partitioned approach and (b) Monolithic approach

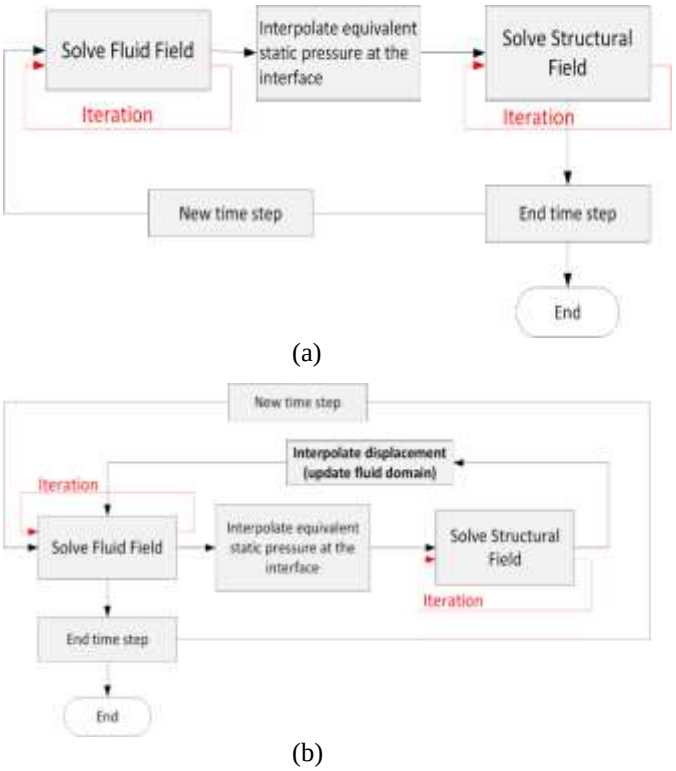


Fig. 4. (a) One-way and (b) Two-way coupled FSI solution algorithm flow charts, adapted from [7]

IV. COMPARATIVE STUDY ON A BENCHMARK CONFIGURATION

Study results of various numerical approaches under the discretization and coupling schemes applied on a benchmark problem are presented and discussed in this section. The standard benchmark configuration with a proposed initial and boundary conditions and material properties as proposed by [21] serves as test tool for numerical methods in FSI. It is a channel flow of incompressible laminar fluid over an elastic object attached to a cylinder, whose configuration is shown in Fig. 5 below. The results of the computational methods are compared in a broader sense, but more focus is given on computational methods classified based on temporal discretization and interface coupling schemes. The values of the geometric and material parameters for this proposed FSI model are given in Table II and Table III respectively. The reference response values from the proposed benchmark [21] are also taken for comparison purpose.

As an initial condition, a parabolic velocity profile has been applied at the left channel in flow of the configuration, with an empirical formulation of Eq. (14), which is a function of the y-coordinate value along the height of the channel at the inlet. Furthermore, a stress free boundary condition at the outlet boundary, at the right side of the channel, is assumed in all analyses.

The empirical equation for the inlet velocity condition is given by,

$$v_f(0,y) = 1.5\overline{U}\, y(H-y)/(H/2)^2 \qquad (14)$$

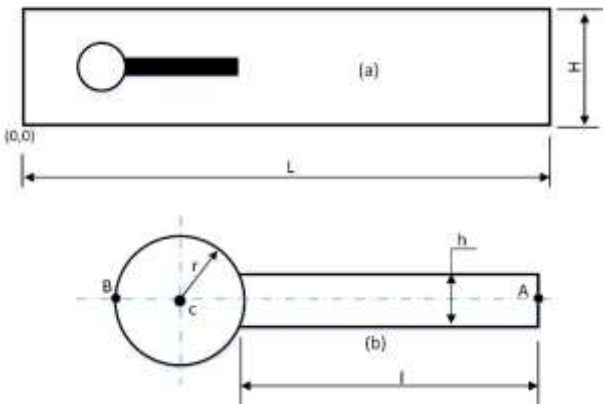


Fig. 5. HronTurek fluid-structure interaction benchmark configuration (a) the computational domain (b) Magnified detail of the structural part

TABLE II. THE GEOMETRIC CONFIGURATION PARAMETER VALUES

Geometric parameters		Value [m]
Channel length	L	2.5
Channel width	H	0.41
Cylinder center position	C	(0.2, 0.2)
Cylinder radius	r	0.05
Elastic structure length	l	0.35
Elastic structure thickness	h	0.02
Reference point (at t = 0)	A	(0.6, 0.2)
Reference point	B	(0.15, 0.2)

In order to induce vibration on the structure body the inflow velocity is applied to increase gradually with time. Thus, the general inlet condition is given by equation (15) below, as

$$v_f(t,0,y) = \begin{cases} v_f(0,y)[1-\cos(\pi t/2)]/2 & if\ t < 2.0 \\ v_f(0,y) & otherwise \end{cases} \qquad (15)$$

A. Computational techniques applied

For the test purpose, simulation of one-way and strongly coupled two-way partitioned based approaches are applied on the benchmark configuration, which is a one cell thickness 3D model. For the one-way coupled approaches analysis an ANSYS 17.0 Inc. code is utilized, CFD (Fluent) and Transient Structure tools for the fluid and structure respectively. A laminar transient incompressible fluid flow model with fluid cell number of 973 is solved applying a SIMPLE solver algorithm for the pressure velocity coupling. Least square cell based method for gradient discretization and second order UPWIND method for moment discretization techniques are applied in all one-way testes. Time step size of 0.00001 is set over the 6 s test period. For the structural part, a transient structure model with varying nonlinear solver and material properties are simulated on a structured meshing type and total number of 20 cells. The traction load from the fluid analysis is transferred to the structural model.

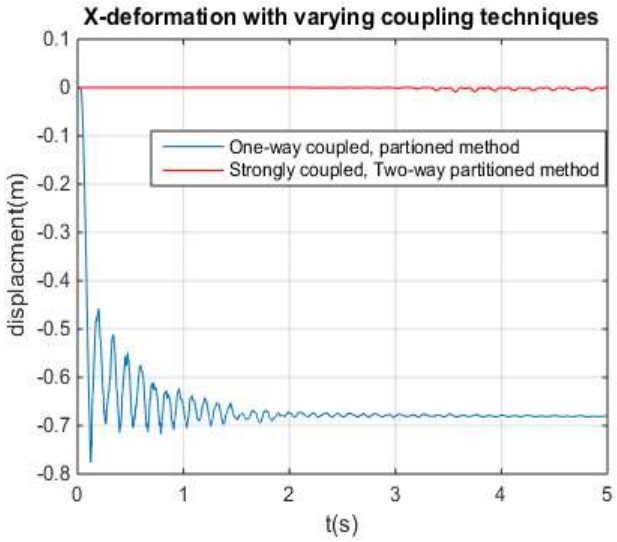
The results of the strongly coupled two-way partitioned approach are obtained utilizing the model in an open source code of Open FOAM's fsiFoam solver on the benchmark HronTurek FSI case in foam-extended 3.1 platform. In this analysis, a one cell thickness structured mesh model with spline curve setting of the lateral edge of the structure is employed. The total mesh cell number for the fluid domain is 4706 and for the structure is 630. Time step size of 0.01 is used over the 6 s simulation period.

B. Results and discussion

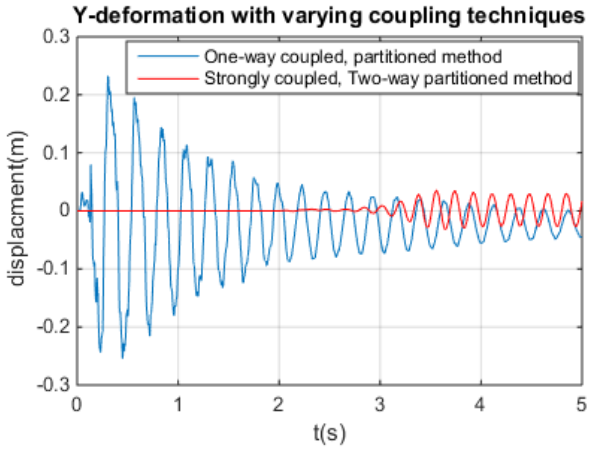
Responses in the local x-and y- axis direction at point A, from the geometric configuration are examined for one-way coupled and two-way strongly coupled partitioned numerical approaches as shown in Fig 6 (a) & (b) respectively. As it can be seen from Fig 6 (a), the y-response graph from the one-way coupled approach simulation is less realistic than the strongly coupled two-way approach.

TABLE III. PROPERTIES OF MATERIAL USED IN THE PROPOSED BENCHMARK FSI MODEL WITH TWO DIFFERENT INFLOW SPEED INITIAL CONDITIONS

Parameter		Values
Solid density	$\rho^s[10^3\ kg/m^3]$	1
Poison's ratio	v^s	0.4
Shear Modulus	$\mu^s[10^6\ kg/ms^2]$	2
Fluid density	$\rho^f[10^3\ kg/m^3]$	1
Kinematic viscosity	$v^f[10^{-3}\ m^2/s]$	1
Mean velocity	$\overline{U}(m/s)$	2



(a)



(b)

Fig. 6. Responses at point A along the (a) x and, (b) y- local coordinate axes of two coupling techniques

Given the intensity of the initial condition, which is time dependent that only reaches its maximum from zero at (t = 2 s), the response behaved unrealistic unlike the response from the other approach. Similarly, the method's x-response also demonstrate unrealistic behavior. The maximum and minimum values of the responses are also depicted for comparison purpose in Table IV.

From the qualitative results obtained from both coupling approaches, it is clear to see that the unrealistic behavior of the one way coupling response graph is due to the fact that the fluid is solved throughout the time period without considering the deformation at the interface and fail to undergo re-meshing when necessary. On the contrary the two-way method do consider the deformation, hence shows expected response behavior.

TABLE 4: ONE-WAY COUPLED ANALYSIS RESULT, ANSYS 17.0, USING CDD (FLUENT) AND TRANSIENT STRUCTURE TOOLS

Structure Shear modulus	Fluid cell no	Structure cell	Max/min y-responses at the tip (A),	Max/min x-responses at the tip (A)
2E+6	974	150	0.23243, -0.2541	-5.422E-7, -0.7771
2E+7	974	150	0.3009, -8.661E-2	-4.463E-7, -0.61799
2E+8	974	150	-3.838E-7, -2.522E-3	8.403E-5, -2.346E-5
2E+6	4706	630	0.03591, -0.03196	0.000309, -0.00881

The y-responses' graphs behavior of the one-way approach from slightly changing the value of one of the material property parameter of the structure have been closely examined as shown in Fig. 7. From the qualitative comparison, the amplitude of the vibration reduces as the material stiffness property improves, which is realistic. However, the deformation of the second test, with modulus value of 20, demonstrated unexpected results at the beginning and also along the whole simulation period as it should be having a mean response close to zero. But, one can see from the graphs that varying the material property would not shed the effects from manifesting drawbacks of the coupling techniques.

Moreover, from the study results of the one-way coupling approach by setting different nonlinear solvers in the transient structure tool, as shown in Fig. 8, the Modified Newton Rapson and Full Newton Rapson (with line search method on) solver algorithms give similar results than the Full Newton Rapson method with line search method set program controlled. The first two methods converged by far faster than the other. However, the later method gives a response value closer to the two-way approach at the upper boundary for the period roughly 3.5 to 5 than the other two methods.

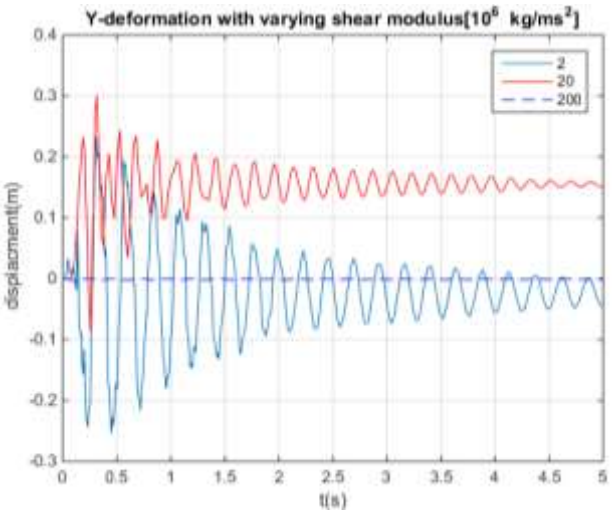


Fig. 7. One-way coupling approach y-response at point A of the FSI model with varying values of a material property parameter.

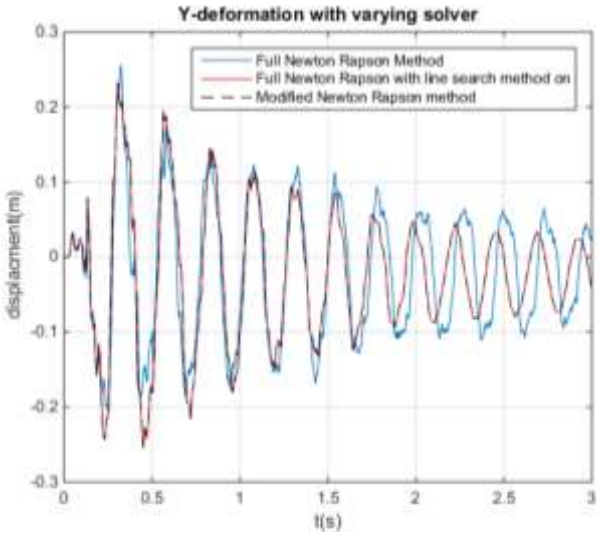


Fig. 8. Y-responses of one-way coupling approach with varying nonlinear solver of the structural analysis tool

V. SUMMARY AND CONCLUSION

Solving FSI problems is challenging because two distinct domains with different governing kinematic equations are involved. Often, the challenge arises in relation to how the interfaces at the two domains are treated about applying the appropriate coupling and discretization (both spatial and temporal) techniques. However, Regardless of the interaction problem large range of methods have been emerging to solve FSI problems. Based on the review and study in this article, development of FSI analysis methods flow in two broad classes of approaches, the Monolithic and Partitioned approach. In the first approach, an integrated mathematical model has to be developed in order to solve both domains as a single continuum model and it enables a strong coupling of the domains at the interface. Majority of the computational techniques developed in the monolithic approach domain utilize the ALE formulation so as to benefit from the positive features of both Lagrangian and Eulerian techniques. However, the approach is relatively complex, inefficient and computationally expensive compared to the partitioned approach. On the other hand, the later approach benefits from utilizing the existing developed computational codes. Nevertheless, coupling the solvers at the interface needs an additional coupling code; hence, this limits a strong coupling capability. Apart from that, the study results on one-way coupled partitioned approaches on the benchmark problem demonstrates unrealistic results because it fails to consider the structural deformation at the interface regardless of the material property or nonlinear solver techniques employed at the interface. However, the strongly coupled two-way partitioned approach result on the benchmark demonstrates very realistic result due to its consideration of the structural deformation at the interface and re-meshing algorithm employed.

REFERENCES

[1] R.S. Raja, “Coupled fluid structure interaction analysis on a cylinder exposed ocean wave loading,” Master thesis, Department of Applied Mechanics, Chalmers University of Technology, Gothenburg, Sweden, 2012.

[2] Y. Zhang, S. Hu, Y. Zhang, and L. Chen, “Optimization and Analysis of Centrifugal Pump considering Fluid-Structure Interaction,” The Scientific World J., vol. 2014, Article ID 131802, s http://dx.doi.org/10.1155/2014/131802.

[3] Y. Bazilevs, J.R. Gohean, T.J.R. Hughes, R.D. Moser and Y. Zhang, “Patient-specific isogeometric fluid–structure interaction analysis of thoracic aortic blood flow due to implantation of the Jarvik 2000 left ventricular assist device,” Comput. Methods Appl. Mech. Eng. vol. 198(45–46), pp. 3534-3550, 2009.

[4] D. Kamensky, M.-C. Hsu, D. Schillinger, J.A. Evans, A. Aggarwal, et al., “An immersogeometric variational framework for fluid–structure interaction: Application to bioprosthetic heart valves,” Comput. Methods Appl. Mech.Eng., vol. 284, pp. 1005-1053, 2015.

[5] S. Badia, F. Nobile and C. Vergara, “Fluid–structure partitioned procedures based on Robin transmission conditions,” J. Comput. Phys., vol. 227(14), pp. 7027-7051, 2008.

[6] S. Badia, F. Nobile and C. Vergara, “Robin–Robin preconditioned Krylov methods for fluid–structure interaction problems,” Comput. Methods Appl. Mech. Eng., vol. 198(33–36), pp. 2768-2784, 2009.

[7] F.-K. Benra, H.J. Dohmen, J. Pei, S. Schuster, and B. Wan, "A comparison of one-way and two-way coupling methods for numerical analysis of fluid-structure interactions," J. Appl. Math., vol. 2011, pp. 16, 2011.

[8] G. Hou, J. Wang, and A. Layton, "Numerical methods for fluid-structure interaction — a review," Comm. Comput. Phys., vol. 12(02), pp. 337-377, 2012.

[9] J. Degroote, "Partitioned simulation of fluid-structure interaction," Arch Computat. Methods Eng, vol. 20(3), pp. 185-238, 2013.

[10] T. Richter, "Numerical methods for fluid-structure interaction problems," (unpublished online document on Research gate), 2010.

[11] "ANSYS web page," Available fat: www.Ansys.com. (Last visited: 2016-06-30).

[12] Y. Bazilevs, V.M. Calo, Y. Zhang, and T.J.R. Hughes, "Isogeometric fluid–structure interaction analysis with applications to arterial blood flow," Comput. Mech., vol. 38(4-5), pp. 310-322, 2006.

[13] E. Lawrence, Malvern, "Introduction to the mechanics of continuous medium," Prentice-Hall Inc., vol.1, 1969.

[14] L. WK, "Finite element procedures for fluid-structure interactions and application to liquid storage tanks," Nucl. Eng. Des., vol. 65, pp. 221-238, 1981.

[15] H. TJR, Liu WK, and Z. TK., "Lagrange-eulerian finite element method for interaction for viscous flows," Comput. Methods Appli. Mech. Eng., vol. 29, pp. 329-349, 1981.

[16] W.K. Liu and J. Gvildys, "Fluid-structure interaction of tanks with an eccentric core barrel," Comput. Methods Appl. Mech. Eng., vol. 58, pp. 51-77, 1986.

[17] P. Causin, J.F. Gerbeau, and F. Nobile, "Added-mass effect in the design of partitioned algorithms for fluid–structure problems," Comput. Methods Appl. Mech. Eng., vol. 194(42–44), pp. 4506-4527, 2005.

[18] C. Förster, W.A. Wall and E. Ramm, "Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows," Comput. Methods Appl. Mech. Eng., vol. 196(7), pp. 1278-1293, 2007.

[19] M.A. Fernández, "Coupling schemes for incompressible fluid-structure interaction: Implicit, semi-implicit and explicit," SeMA J., vol. 55(1), pp. 59-108, 2013.

[20] C.A. Felippa, K.C. Park, and M.R. Ross, "A classification of interface treatments for FSI," in Fluid structure interaction II, H.-J. Bungartz, M. Mehl, and M. Schäfer Eds. Springer Berlin Heidelberg. pp. 27-51, 2010.

[21] S. Turek and J. Hron, "Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow," Fluid-Structure Interaction: modelling, Simulation, optimizaiton LNCSE. Springer, vol. 53, 2006.