World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:10, No:2, 2016

Numerical Simulation of Fluid Structure Interaction Using Two-Way Method

Samira Laidaoui, Mohammed Djermane, Nazihe Terfaya

***Abstract*—**The fluid-structure coupling is a natural phenomenon which reflects the effects of two continuums: fluid and structure of different types in the reciprocal action on each other, involving knowledge of elasticity and fluid mechanics. The solution for such problems is based on the relations of continuum mechanics and is mostly solved with numerical methods. It is a computational challenge to solve such problems because of the complex geometries, intricate physics of fluids, and complicated fluid-structure interactions. The way in which the interaction between fluid and solid is described gives the largest opportunity for reducing the computational effort. In this paper, a problem of fluid structure interaction is investigated with two-way coupling method. The formulation Arbitrary Lagrangian-Eulerian (ALE) was used, by considering a dynamic grid, where the solid is described by a Lagrangian formulation and the fluid by a Eulerian formulation. The simulation was made on the ANSYS software.

***Keywords*—**ALE, coupling, FEM, fluid-structure interaction, one-way method, two-way method.

I. Introduction

THE interaction between fluids and solids is a phenomenon that can often be observed in nature, for example, the deformation of trees or the movement of sand dunes caused by wind, it can also interact with buildings, sometimes with dramatic consequences, such as the collapse of the Tacoma-Narrows Bridge in November 1940. In almost the same manner, water can also interact with structure like sloshing and slamming phenomenon. These processes can only be calculated using laws and equations from different physical disciplines. Examples like this, where the arising sub problems cannot be solved independently, are called multiphysics applications. A very important class of these multiphysics problems is fluid-structure interactions (FSI), which are characterized by the fact that the flow around a body has a strong impact on the structure, and vice versa; the modification of the structure has a non-negligible influence on the flow. Two disciplines involved in these kinds of multiphysics problems are fluid dynamics and structural dynamics, which can both be described by the relations of continuum mechanics.

Coupled systems and formulations are those applicable to multiple domains and dependent variables which usually describe different physical phenomena and in which neither domain can be solved while separated from the other and neither set of dependent variables can be explicitly eliminated at the differential equation level [1].

Solution strategies for FSI simulations are mainly divided into monolithic and partitioned methods; this paper will focus only on partitioned methods. Partitioned methods are divided into one-way and two-way coupling method.

The present paper is devoted to the simulation examples of fluid structure interaction using the two-way coupling method where we change the number of gaskets by varying the distance between them.

S. Laidaoui is PhD student in Mechanics and Civil engineering, member in FIMAS laboratory. Bechar University. Algeria (e-mail: laidaouisamira@yahoo.fr).

M. Djermane is Professor, member in FIMAS laboratory, Bechar University. Algeria (e-mail: djermane_med@yahoo.fr).

N. Terfaya, doctor, member in FIMAS laboratory, Bechar University. Algeria (e-mail: t_nazihe@yahoo.fr).

II. Mathematical Formulation

A. General Representation

The general equations of the coupled fluid–structure problem using a pressure-based formulation for the fluid written in the time domain are recalled. Fig. 1 gives a generic representation of the coupled problem [2].

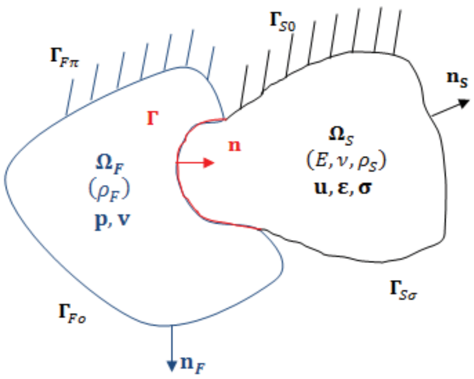


Fig. 1 General representation of a fluid–structure interaction problem [2], [3]

B. Solid Equations

Let Ω_S be the structure domain with boundary $\partial\Omega_S = \partial\Omega_{S\sigma} \cup \partial\Omega_{S0} \cup \Gamma$ where $\partial\Omega_{S\sigma}$ is the boundary part with imposed forces, $\partial\Omega_{S0}$ is the boundary part with imposed displacement and Γ is the fluid–structure interface. n_S is the outward normal on $\partial\Omega_S$, n is the inward normal on Γ. u is the structure displacement, $\sigma(u)$ is the stress tensor. ρ_S stands for the structure density. The structure problem equations in displacement formulation read

$$-\omega^2 \rho_S u_i - \frac{\sigma_{ij}(u)}{\partial x_j} = 0 \quad \text{in} \quad \Omega_S \tag{1}$$

$$\sigma_{ij}(u) n_j^S = 0 \quad \text{on} \quad \Gamma_{S\sigma} \tag{2}$$

$$u_i = 0 \quad \text{on} \quad \Gamma_{S0} \tag{3}$$

C. Fluid Equations

Let Ω_F be the fluid domain with $\partial\Omega_F = \partial\Omega_{F0} \cup \partial\Omega_{F\pi} \cup \Gamma$, where $\partial\Omega_{F\pi}$ is the boundary part with imposed normal gradient pressure (rigid wall or symmetry plane), $\partial\Omega_{F0}$ (pressure release surface or anti-symmetry plane) and Γ is the structure–fluid interface. n_F is the out ward normal on $\partial\Omega_F$, n is the out ward normal on Γ. The fluid fluctuating pressure field is denoted as p and the fluid density as ρ_F

$$\frac{\partial^2 p}{\partial x_i^2} = 0 \quad \text{in} \quad \Omega_F \tag{4}$$

$$p = 0 \quad \text{on} \quad \Gamma_{F0} \tag{5}$$

$$\frac{\partial p}{\partial x_j} n_j^F = 0 \quad \text{on} \quad \Gamma_{F\pi} \tag{6}$$

D. Coupled Problem

The variational formulation of the coupled problem in pressure displacement fields then reads

$$\int_{\Omega_S} \sigma_{ij}(u)\ \zeta_{ij}(\delta u) dV - \omega^2 \int_{\Omega_S} \rho_S u_i \delta u_i dV = \int_{\Gamma} p n_i \delta u_i dS \qquad \forall \delta u \tag{7}$$

for the structure problem and:

$$\int_{\Omega_F} \frac{\partial p}{\partial x_i}\frac{\partial \delta p}{\partial x_i} dV = -\rho_F \omega^2 \int_{\Gamma} u_i n_i \delta p dS \qquad \forall \delta p \tag{8}$$

for the fluid problem.

Discretisation of the coupling conditions yields the fluid–structure interaction matrices, calculated as

$$\int_{\Gamma} p n_i \delta u_i d\Gamma \rightarrow \delta U^T R P \tag{9}$$

$$\int_{\Gamma} u_i n_i \delta p d\Gamma \rightarrow \delta P^T R^T U \tag{10}$$

Finally, the coupled problem takes the following form [4]:

$$\begin{bmatrix} M_S & 0 \\ \rho_0 R^T & M_f \end{bmatrix}\begin{Bmatrix} \ddot{U} \\ \ddot{P} \end{Bmatrix} + \begin{bmatrix} K_S & -R \\ 0 & K_f \end{bmatrix}\begin{Bmatrix} U \\ P \end{Bmatrix} = \begin{Bmatrix} F_S \\ F_f \end{Bmatrix} \tag{11}$$

III. One-Way and Two-Way Methods

Regardless of whether one-way or two-way coupling methods are used, the solutions are based on a partitioned method where separate solutions for the different physical fields are prepared. One field that has to be solved is fluid dynamics, the other is structure dynamics. At the boundary between fluids and solids, the fluid-structure interface, information for the solution is shared between the fluid solver and structure solver. The information exchanged is dependent on the coupling method. For one-way coupling calculations, only the fluid pressure acting at the structure is transferred to the structure solver. For two-way-coupling calculations, the displacement of the structure is also transferred to the fluid solver [1].

A. One-Way Method

In Fig. 2, the solution procedure is shown for one-way coupling. Initially, the fluid field is solved until the convergence criteria are reached. The calculated forces at the structure boundaries are then transferred to the structure side. Next, the structure side is calculated until the convergence criterion is reached. Then, the fluid flow for the next time step is calculated to convergence. The solution is finished when the maximum number of time steps is reached.

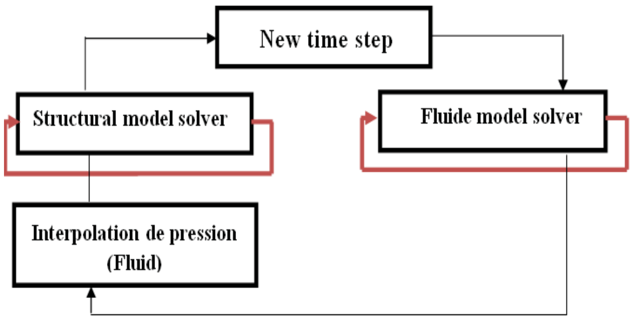


Fig. 2 Solution algorithm for one-way coupling [5]

B. Two-Way Method

The process flow chart for the strong two-way coupling algorithm is shown in Fig. 3. Within one-time step during the transient simulation, a converged solution for the flow field is required to provide the forces acting on the body. After interpolating the forces from the fluid mesh to the surface mesh of the structure, a converged solution of the structural dynamics will be attained under the effect of the acting forces. The response of the structure to the emerging load represents a displacement of the structural grid nodes. The displacements at the boundary between structure and fluid are interpolated to the fluid mesh which leads to its deformation. This step closes one inner loop of the simulation [1]. These steps are repeated until the changes in the flow forces and the structural displacements fall below a prescribed amount. Afterwards, a new time step is launched.

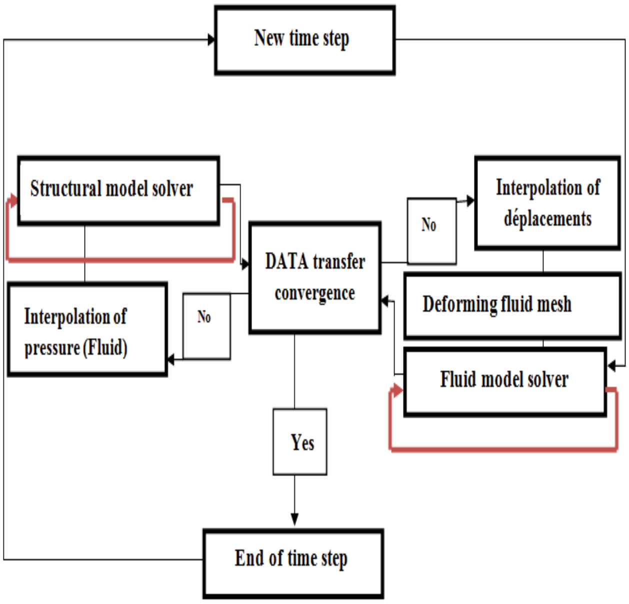


Fig. 3 Solution algorithm for two-way coupling [5]

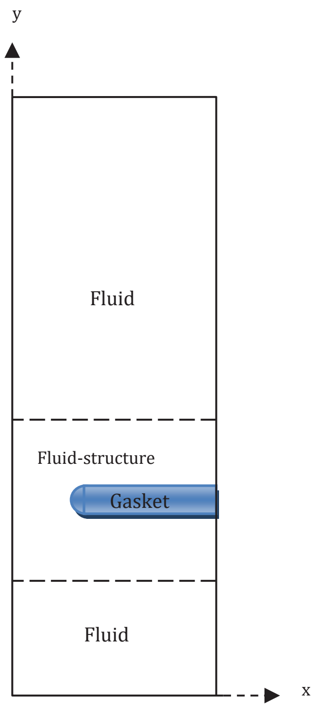


Fig. 4 Fluid and gasket regions

IV. Application Examples

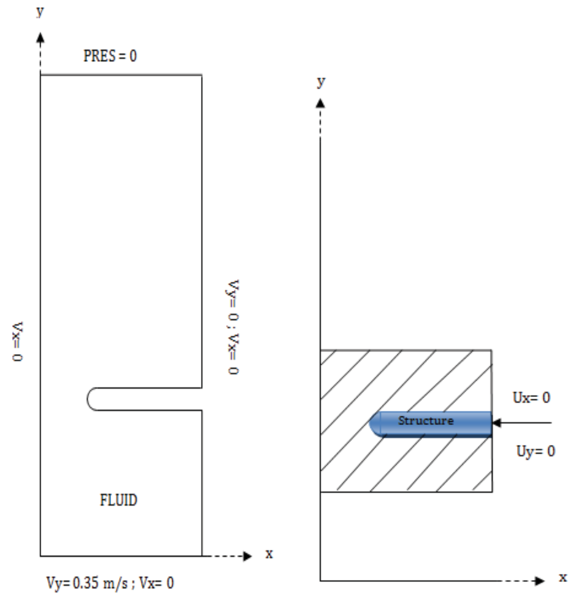
A. Example 1

The example in this section illustrates a fluid-structure interaction problem. This problem demonstrates the use of nonlinear large-deflection structural coupling for a fluid domain by using the two-way method.

A channel containing a rubber gasket is subjected to water flowing with an inlet velocity of 0.35 m/sec. (See Fig. 4). The object of the problem is to determine the pressure drop and gasket deflection under steady-state conditions [6].

TABLE I
CHARACTERISTICS OF FLUID AND STRUCTURE

Fluid	Structure
Density = 1000 kg/m3 Viscosity = 4.6E-4 kg-s/m	C10= 2.93E+005
Inlet velocity = 0.35 m/s	C01= 1.77E+005 d= 1.4043E-009



(a) (b)

Fig. 5 Nominal fluid and structure physics boundary conditions: (a) fluid, (b) structure

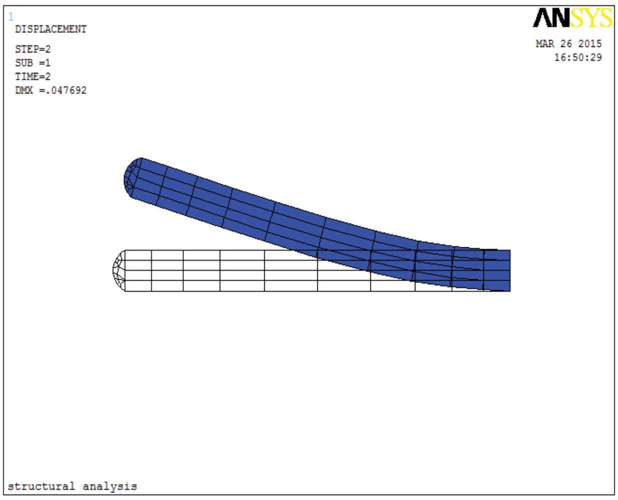
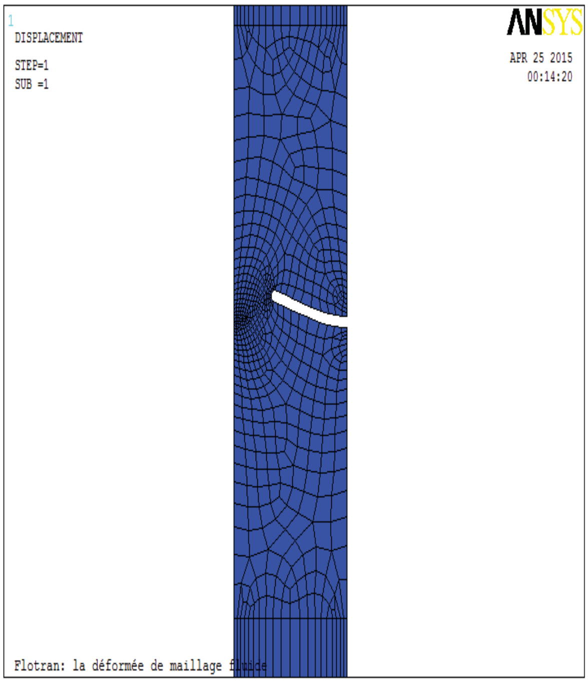
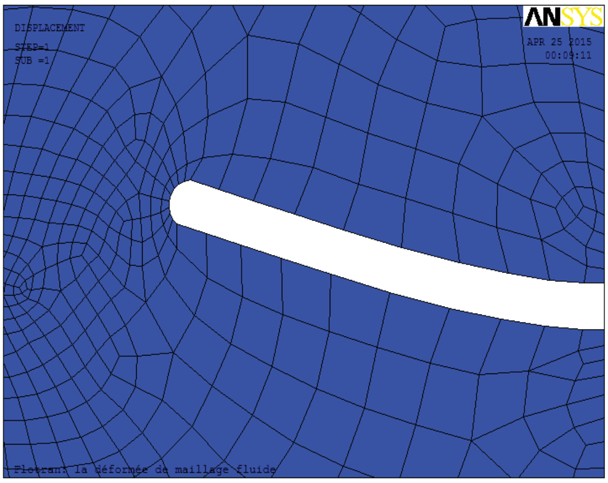


Fig. 6 Large deflection of gasket

In Fig. 6, large deflection of gasket due to the fluid pressure. The deflection may be significant enough to affect the flow field (see Fig. 7).



(a)



(b)

Fig. 7 Deformed mesh: (a) General view, (b) Detail in the region close to the gasket

In two-way coupling, when the structure imposes displacement on the fluid, it changes the characteristics of fluid flow which results mesh deformation. The mesh was moved by the full value of the displacements. The deformed mesh is shown in Fig. 7.

Fig. 8 depicts the streamlines near the gasket for the deformed geometry and Fig. 9 depicts the pressure contours.

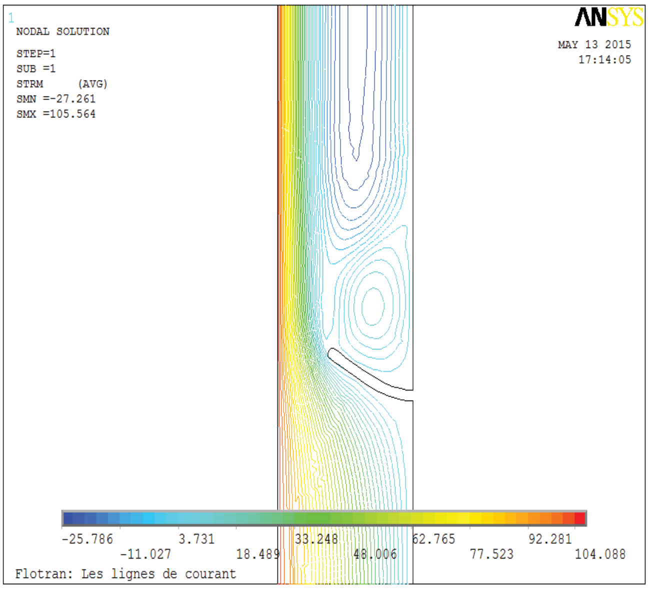


Fig. 8 Streamlines near gasket

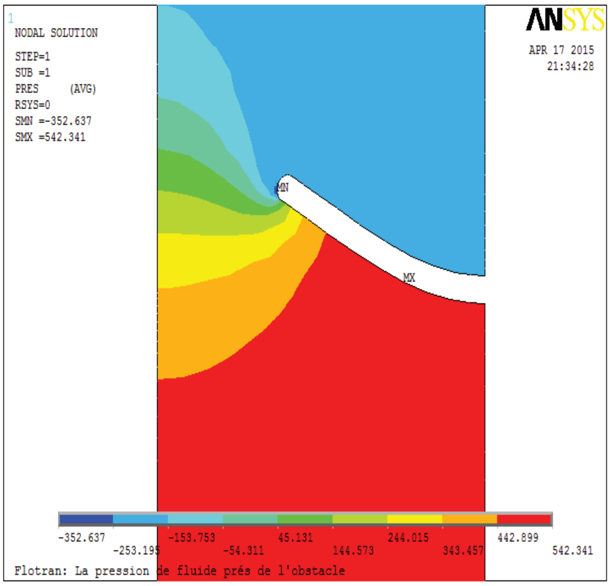


Fig. 9 Pressure contours

B. *Example 2*

In this example, we retain the same characteristics of the example 1. We add another gasket placed in an opposite way as shown in Fig. 10 [7].

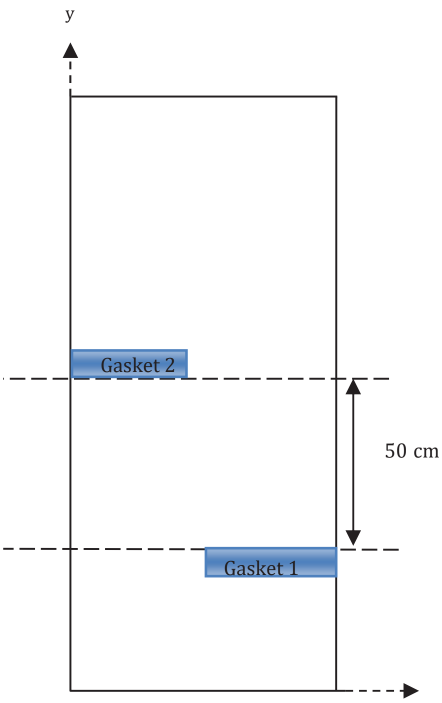


Fig. 10 Channel geometry

Fig. 11 depicts the streamlines near the gasket for the deformed geometry and Fig. 12 depicts the pressure contours.

Through these results we note the presence of the whirlpools after each gasket. According to Figs. 11 and 12, we may point out the influence of the first gasket on the second it is mounted more clearly in Fig. 13 where we may see that the displacement of second gasket is higher compared to the first this is due to the influence of depression of first gasket because the distance between the two gaskets is insufficient.

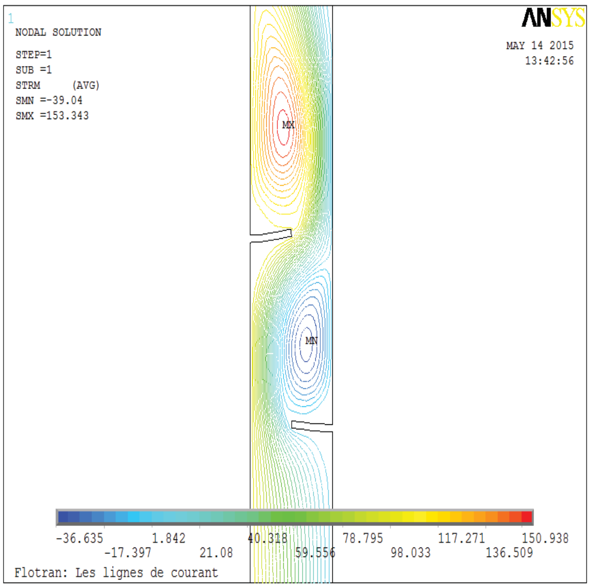


Fig. 11 Streamlines near gasket

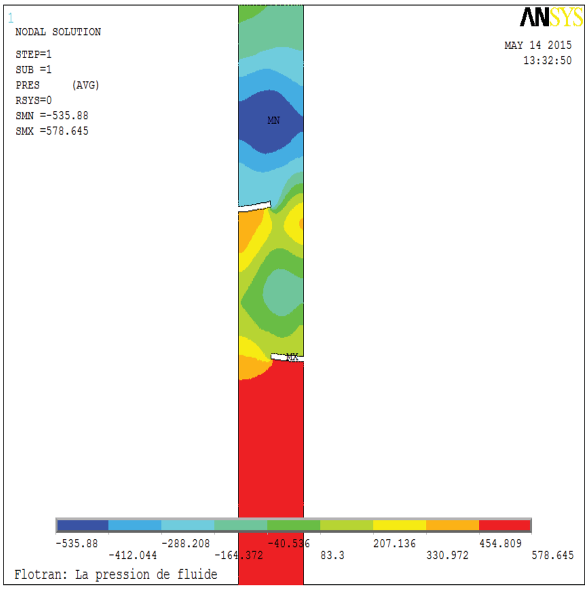


Fig. 12 Pressure contours

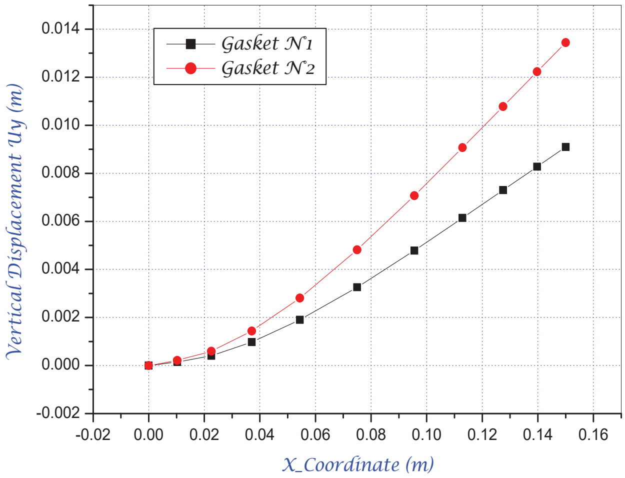


Fig. 13 Vertical displacements on the bottom of gasket

C. Example 3

To confirm the results of example 2 we increase the distance between the two gaskets up to 2 m as shown the Fig. 14 [7].

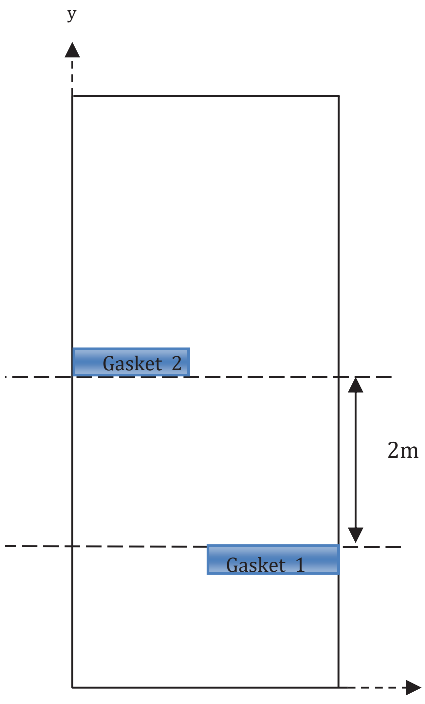


Fig. 14 Channel geometry

Fig. 15 depicts the streamlines near the gasket for the deformed geometry and Fig. 16 depicts the pressure contours.

According to the results, we note the disappearance of the influence of depression of the first gasket on the second and by consequence we find that the first obstacle moves more than the second.

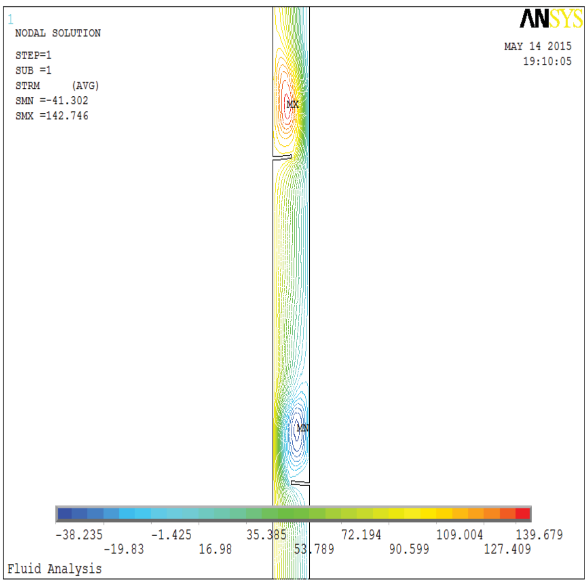


Fig. 15 Streamlines near gasket

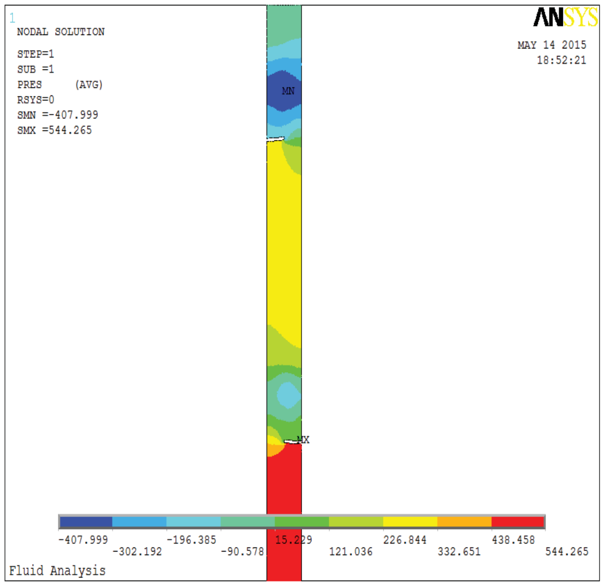


Fig. 16 Pressure contours

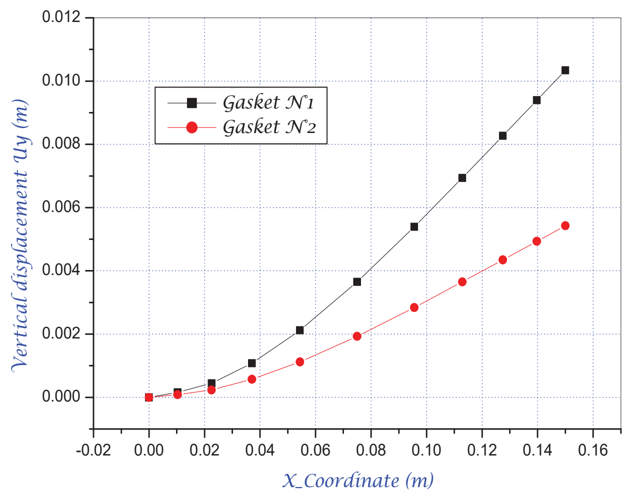


Fig. 17 Vertical displacements on the bottom of gasket

V. Conclusion

The numerical simulation of fluid structure interaction became possible mainly due to advances and increased performance of computing and considerable progress in both branches of research that are considered distinct: fluid mechanics and dynamics of structures.

In this paper, a problem of fluid structure interaction is investigated with two-way coupling method.

The formulation ALE is particularly well adapted to the study of coupled problems; it is used by considering a dynamic grid.

The examples studied in this work have shown the influence of the number of obstacles as well as the distance between them on the fluid flow on the one hand and on the behavior of the structure on the other hand.

REFERENCES

[1] 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". Journal of Applied Mathematics. 2011. Article ID 853560.

[2] J. F Sigrist, S. Garreau, "Dynamic analysis of fluid–structure interaction problems with modal methods using pressure-based fluid finite elements". Finite Elements in Analysis and Design 43 (2007) 287–300.

[3] M. Souli, J. F Sigrist, "Interaction fluide-structure modélisation et simulation numérique". Hermes. Lavoisier, 2009, pp. 25-66.

[4] J. F Sigrist, "Méthodes numériques de calculs couplés fluide/structure: Cas du fluide stagnant". 2010. Technique de l'ingénieur. BM5200.

[5] R. S. Raja, "Coupled fluid structure interaction analysis on a cylinder exposed to ocean wave loading". Master's Thesis in Solid and Fluid Mechanics. Charmers University of Technology. Göteborg, Sweden 2012.

[6] Ansys Tutorials, Example fluid structure interaction, 2009.

[7] S. Laidaoui, "Modélisation Des Interactions Fluide-Structure: Comportement D'une Structure Sous L'impact D'un Fluide". Magister thesis. 2015, Bechar University. Algeria.