Fluid - Structure Interface Modeling

Benzaigaou Ali, Abdeldjelil Faycel, Nabbou Rachid, Haddi Lakhedar

Laboratoire de mécanique: Modélisation et expérimentation (L2ME)
Bechar University, Po. Box 417, Algeria.

Abstract—The fluid - structure interaction is the behavior of a system consisting of two distinct mechanical entities: a structure (rigid or deformable) and a fluid (flowing or at rest) around or inside the structure.

In this work, we will consider:
- Simulating the behavior of a cylinder in the presence of a non-Newtonian fluid flow past the structure in an unconfined environment. The cylinder is assumed to be elastic and deformable.
- Simulating the flow of a stationary fluid that is viscous and incompressible in a channel.

Keywords: numerical simulation, coupling fluid-structure, finite element, finite volume.

I. INTRODUCTION

Numerical simulation of coupled phenomena has grown constantly in recent years. This development is particularly due to the success of the previous simulation in general, but also to the permanent increase in performance of computers. Among these phenomena are coupled fluid-structure interactions. These involve a constantly moving, rigid or deformable structure, and a liquid or gas, flowing around or against part of the structure. These phenomena are called coupled, because the evolution of each component depends on that of the other.

In most cases, numerical simulations of unsteady and nonlinear problems of fluid-structure interaction algorithms require the use of coupling, where the fluid and structure are advanced in time by different methods and different codes [1], [5], [7]. Except for simplified cases, the "monolithic" solution, which implies to write a complete formulation combining all the unknowns of the problem, is at first very heavy and very difficult (it is still possible in some cases). In addition, it requires specific developments, which are unlikely to be reused, and makes it difficult to update the platform developed when new numerical methods are proposed.

The most commonly adopted approach today, is to introduce a coupling more or less strong (at least less strong than that proposed in a monolithic approach). In this approach, the coupled system is solved subsystem by subsystem, successively or iteratively, so that the existing code can be reused, and made adaptable to new problems, and to use the latest versions and the most effective solvers used. For this, the variables are exchanged on the fluid-structure interface, directly or indirectly between the solvers via a coupler acting as the interface.

II. CONSERVATION EQUATIONS

Cartesian equations that describe the flow of a two-dimensional incompressible non-Newtonian fluid are the conservation equation of mass and momentum; These can be written as [1], [4]:

$$\nabla \cdot u = 0 \quad (1)$$

$$\rho \left( \frac{\partial u}{\partial t} + u \nabla u \right) = \nabla (-p I + 2\eta \mathbf{D} + \tau) \quad (2)$$

Where $u$ is the fluid velocity, $\rho$ is its density, $p$ is the pressure, $I$ is the unit tensor, $\eta$ is the fluid viscosity, $D = \left( \nabla u + \nabla^T u \right) / 2$ is the tensor of the deformation rate, and $\tau$ is the non-Newtonian strain tensor. For a visco-elastic fluid, $\tau$ is defined using the PTT model according to the equation:

$$G \tau^\lambda \left( \frac{\partial \tau}{\partial t} + u \nabla \tau - (L - 3\mathbf{D}) \tau - \tau (L - 3\mathbf{D})^T \right) = 2\eta_m D^\lambda$$

Where $\lambda$ is the relaxation time, $\eta_m$ is the non-Newtonian viscosity and $G = \exp (\lambda \xi/\eta_m \text{Tr}(D))$ is a function that limits the growth of the elongational viscosity. $\xi$ and $\varepsilon$ variables and are parameters that characterize the elastic behavior of materials and $\eta_{mol}$ is the molecular viscosity in the absence of shear rate.

The above equations are formulated in the Cartesian case, it is important to write them in a generalized form. For this, we use the Pope method (1978), [11]. In the latter one assumes the existence of generalized orthogonal coordinates $(\psi_1, \psi_2)$ defined by $\psi_1 = \psi_1(x_1, x_2)$ and $\psi_2 = \psi_2(x_1, x_2)$. The basis vectors of the new system are $\mathbf{g}_i = \partial \psi_i/\partial x_j \mathbf{e}_j$, $(i = 1, 2)$ and according to the definition of orthogonality, the metric tensor $g_{ij} = \mathbf{g}_i \cdot \mathbf{g}_j$ is diagonal and the scale factors are $h_i = \sqrt{g_{ii}}$. The physical velocity field $V_i$ is used here to rewrite the conservation equations by means of physical space derivatives $d\zeta = h_i d\psi_i$, $(i = 1, 2)$. Thus, the Cartesian components of the velocity $u_i$ are written as follows:
\[ u_i = \sum_{j=1}^{2} \frac{\partial x_i}{\partial y_j} \frac{V_j}{h_j}, \quad (i=1,2) \]  

(4)

Where \( \frac{\partial x_i}{\partial y_j} \) is the Jacobean of the transformation.

The operator of generalized divergence is written:

\[ \nabla_{ij}(\omega) = \frac{\partial}{\partial x_i} \sum_{k=1}^{N} H_k^j(\omega), \quad (i=1,2) \]  

(5)

Where:

\[ H_k^j = \frac{1}{h_j} \frac{\partial h_i}{\partial y_j} = \frac{1}{h_i h_j} \frac{\partial \psi}{\partial y_j}, \quad (i, j=1,2) \]  

(6)

By adopting this procedure, the conservation equations (1) and (2) in generalized coordinates can be written:

\[ \sum_{j=1}^{2} \nabla_{ij} V_j = 0 \]  

(7)

\[ \frac{\partial \psi}{\partial t} + \frac{\partial}{\partial x_i} \left( \frac{\partial \psi}{\partial t} \right) - \frac{\partial}{\partial x_i} \left( \psi \frac{\partial V_i}{\partial x_i} - \mu \frac{\partial^2 V_i}{\partial x_i^2} \right) = 0 \]  

(8)

\[ \frac{\partial}{\partial t} \left( \frac{\partial \psi}{\partial t} \right) - \frac{\partial}{\partial x_i} \left( \psi \frac{\partial V_i}{\partial x_i} - \mu \frac{\partial^2 V_i}{\partial x_i^2} \right) = \frac{2 \mu \eta \Delta_0}{1 + \Delta_0} \left( \frac{\partial \psi}{\partial t} \right) - \frac{\partial}{\partial x_i} \left( \psi \frac{\partial V_i}{\partial x_i} - \mu \frac{\partial^2 V_i}{\partial x_i^2} \right) \]  

(9)

where \( \beta = \eta_m / (\eta_m + \eta_m0) \) is the delay ratio and \( L_{ij} = \frac{\partial u_j}{\partial x_i} - \zeta D_{ij} \)

III. NUMERICAL METHOD

The conventional method of finite volume is used to obtain the discretized form of equations (7), (8) and (9). The quadratic scheme "QUICK" is used to calculate the convection terms. The acceleration terms and pressure gradient are evaluated at the center of the velocities volume control. The Euler decentered explicit scheme is adopted for temporal integration to which we associates the decoupling procedure called "Mark and Cell". At each new time step, a discretized Poisson equation for pressure is solved to verify the incompressibility constraint. The matrix of the Poisson equation is symmetric and defined as positive, the resulting linear system is solved using the direct Cholesky factorization. In this procedure, the terms of diffusion, advection, and acceleration are treated explicitly. To improve stability of the method, the Elastic Viscous Split Stress (EVSS) pattern is applied to decompose the non-Newtonian strain tensor. The mesh in generalized coordinates is obtained from the stream function (\( \psi \)) and the velocity potential function (\( \varphi \)). These are defined as follows:

\[ \psi = U_u \sin \theta \left( r - \frac{A^2}{2} \right), \quad \varphi = U_u \cos \theta \left( r + \frac{A^2}{2} \right) \]  

(10)

where \( A \) is the radius of the cylinder, \( U_0 \) is the upstream velocity, \( r \) is the polar radius and \( \theta \) is the angle. Note that the center of the cylinder coincides with the origin of the coordinate.

IV. FLUID-STRUCTURE COUPLING

The methodology implemented for the simulation of fluid-structure coupling is to model the behavior of the fluid and then the behavior of the structure by taking into account the fluid pressure as an imposed condition. The numerical methods for fluid and structure are often different and it is difficult to make them coexist in a single solver.

In this work, we used two computing codes, one for simulating fluid flow based on the finite volume method [3], and another to simulate the behavior of the cylinder by finite elements: a first analysis by the fluid code gives us the pressure around the cylinder. A second analysis with the finite element code gives the behavior of the cylinder by imposing the fluid pressure.

We used the B-splines method to calculate the pressure at finite element nodes.

V. NUMERICAL APPLICATIONS

A. APPLICATION No. 1

Figure 2 shows the curvilinear mesh with 160 ×80 cells.
Numerical simulation is made for different values of Re (Reynolds number):

1. Re = 25 <= 47 Rec: for a computation time = 40sec. Figures 4 and 5 show respectively, the deformation of the cylinder and the Von Mises strain distribution.

The cylinder is meshed with 84 triangular finite 6-node elements (T6).

Figure 2: Curvilinear mesh with $160 \times 80$ cells; (a) the full domain (b) zoom around the cylinder

We notice a greater displacement in the cylinder side that first comes into contact with the fluid following a more dominant pressure.

2. Re = 100, 9 sec:

Comparing the two deformed figures (4) and figure (6), we see that for a higher computation time, the pressure increases and thus a greater distortion of the cylinder.
In Figure (8), for Re = 100, 9sec we notice the turbulences affecting pressure in this area and leading to instability of the structure.

B. APPLICATION N°2: Case of a plane channel in the presence of a vertical obstacle

This case concerns the flow of a viscous and incompressible stationary fluid (η = 1), in a channel with dimensions (W x H = 30m x 3m), inside which is placed vertically in a flexible plate with dimensions (1xh = 1m x 0.1m), [3]. The plate, considered as deformable obstacle is located 13m from the entrance channel. The boundary conditions are (u = 10m / s, v = 0) at the entrance, (p = 1 bar) at the output (u = 0, v = 0) on the other walls.
VI. CONCLUSION

In this work, we studied the behavior of a cylinder under pressure an incompressible non-Newtonian fluid in a two-dimensional flow within an unconfined environment. We also modeled the movement of an obstacle inside a plane channel. The fluid-structure coupling presents some difficulties related to the fact that the nodes of the mesh (finite volume) in the vicinity of the cylinder does not coincide with the nodes of the mesh (finite element). For this reason, we used the b-splines method to calculate the pressure on the nodes mesh (finite element). This study showed that the behavior of a structure is influenced by the presence of fluid which shows the need to take into account the effect of fluid on structures such as bridges, boats, planes, etc.

REFERENCES


AUTHORS' INFORMATION

Name: Benzegaou Ali
Bechar University, B.P 417, 08000 Bechar, Algeria.
E-mail: benzegaou@yahoo.fr
Date of birth: 1967
Academic record:
1996: Philosophy Doctor (PhD) in Mechanical Sciences for Engineer at Compiegne University of Technology (France).
1992: Associate’s Degree Diploma (DEA) in Applied Mechanics at Compiegne University of Technology (FRANCE).
1991: Mechanical Engineer Diploma at Bechar University (Algeria)
1986: Baccalaureate in Mathematics (Algeria)

Research interest:
- Mechanical Sciences for Engineer.
- Simulation of metal forming and joining processes by FEM.
Name: Nabbou Rachid  
Bechar University, B.P 417, 08000 Bechar, Algeria.  
E-mail: nabou_r@yahoo.fr  
Academic record:  
- Magister in mechanical engineer (University of Bechar, Algeria).  
- Mechanical Engineer Diploma at Bechar University (Algeria)  
Research interest:  
- Mechanical Sciences for Engineer.  
- Finite Element Method in Nonlinear Analysis of Structures.

Name: Haddi Lakhdar  
Bechar University, B.P 417, 08000 Bechar, Algeria.  
Academic record: