Two-way strongly coupled fluid-structure interaction simulations with OpenFOAM

Benjamin Doulcet\textsuperscript{1}, Christophe Devals\textsuperscript{1}, Bernd Nennemann\textsuperscript{2}, Maxime Gauthier\textsuperscript{2}, François Guibault\textsuperscript{1} and Jean-Yves Trépanier\textsuperscript{1}

\textsuperscript{1}École Polytechnique de Montréal, Montréal, Québec, Canada
\textsuperscript{2}Andritz Hydro Canada Inc., Pointe-Claire, Québec, Canada

E-mail: benjamin.doulcet@polymtl.ca

Abstract. Predicting fluid-structure interaction phenomena is an ongoing challenge in numerical simulations. While we look for fast algorithms to solve the equations, some problems need a high accuracy in order to correctly predict results. Hence, controlling every aspect of the algorithm gives the possibility to optimise specific parts, potentially leading to greater accuracy. Developing such a numerical methodology can be achieved on an open source software such as OpenFOAM.

1. Introduction

For a structurally sound design of hydraulic turbines, it is important to consider a multitude of fluid structure interaction phenomena. One of them is rotor stator interaction between the guide vanes and the runner. In extreme cases and if not properly considered, runner cracking and premature failures can occur \cite{1, 2, 3}. While numerical methods to account for fluid structure interaction in hydraulic turbines have been in use for many years, full two-way fluid structure interaction simulations still pose many numerical challenges. This paper explores the development and validation of an integrated FSI solver based on OpenFOAM fluid and structural analysis capabilities to perform FSI simulations using a fully coupled formulation. The ultimate goal is its application to full hydraulic turbines. In an intermediate step, validation with the damping measurements of a hydraulic profile in a flow channel \cite{4} as well as comparison with results from a commercial solver \cite{5} is planned. However, for now the work has been limited to the implementation of the integrated fluid-structure solver and its verification against analytical test cases.

2. Algorithm methodology

In order to solve our coupled problem, we must insure a two-way communication between the fluid and the solid. Thus, several approaches may be considered. We can consider a partitioned algorithm, which consecutively solves the fluid domain and the solid domain. Fluid pressure and stress data must be transferred to the solid, and the solid displacement to the fluid, at each time step. We represent this algorithm in figure 1.

Otherwise, we can solve both domains at the same time with a monolithic algorithm. This will imply a greater accuracy and a lower CPU time \cite{6} than the previous method. However, this algorithm would be harder to implement. Indeed, this algorithm needs to solve fluid and solid
domains in a unique convergence loop. Hence, it necessitates a monolithic fluid solver within which we add the solid domain resolution. This algorithm is illustrated in figure 2.

![Figure 1. Representation of partitioned algorithm](image1)

![Figure 2. Representation of monolithic algorithm](image2)

In this study, we only consider the partitioned method. We implement it with existing solvers (respectively *pimpleFoam* for the fluid and *solidDisplacementFoam* for the solid) and add the coupling ensuring a two-way communication.

The algorithm passes through a convergence loop at each time step. We consider the time step as converged if the ratio between the displacement during the last iteration and the characteristic solid size is below a value chosen by the user. The characteristic solid size can be chosen as the dimension of the solid along the displacement axis. Then, for each iteration in the convergence loop, we perform five major operations.

1) First, we solve the fluid domain with the incompressible Navier-Stokes equations. During this calculation, the solid is considered as a wall.

2) Then, we have our first transfer between the fluid domain and the solid domain. Indeed, the resolution of the fluid domain gives the velocity and pressure fields at the fluid-structure interface. Hence, the pressure induces an isotropic pressure stress

\[ p \]

and the velocity induces an anisotropic viscous stress on the solid \( \bar{\tau} \). These stresses are calculated and summed as the resulting stress \( \bar{\sigma}_{\text{total}} \):

\[ \bar{\sigma}_{\text{total}} = -p \cdot I + \bar{\tau} \]  

(1)

With the viscous stress given by:

\[ \bar{\tau} = \nu \cdot \left[ \nabla \bar{U} + \nabla^T \bar{U} \right] - \frac{2\nu}{3} \cdot \text{div}(\bar{U}) \cdot I \]  

(2)

Now, we have the symmetric and deviatoric parts of a tensor \( \bar{T} \) defined as:

\[ \text{symm}(\bar{T}) = \frac{\bar{T} + \bar{T}^T}{2} \quad \text{dev}(\bar{T}) = \bar{T} - \frac{1}{3}(\text{tr}(\bar{T}))I \]

And the divergence operator of the velocity field:

\[ \text{div}(\bar{U}) = \text{tr}(\text{symm}(\nabla \bar{U})) \]
This gives us the simplified expression of the viscous stress tensor:

\[ \bar{\tau} = 2 \cdot \nu \cdot \text{dev}(\text{symm}(\nabla \bar{U})) \]  

(3)

This stress corresponds to the dynamic condition at the fluid structure interface. As it is calculated on the fluid side of the interface, we have to transfer the stress to the solid faces via interpolations to the interface points.

3) At this point, the solid is solved with the fluid stress and an eventual external force as a boundary condition. The output is then the displacement in the solid domain.

4) We can now apply some relaxation on the solid equation if needed in order to improve the convergence of the results. The applied relaxation is chosen by the user and can be either a fixed constant relaxation, or the Aitken dynamic relaxation. The Aitken relaxation is described by Naseri et al. [7].

5) Finally, we have to transfer back the displacement to the fluid to respect the kinematic condition at the interface. To ensure this, we update the solid and the fluid meshes at the interface with the calculated displacement and in the fluid domain. The fluid domain mesh is displaced with a magnitude inversely proportional to the distance to the wall over a fixed chosen distance.

The convergence condition is then verified and the simulation continues until the last time step. These transfers between the fluid and solid domains and the convergence condition ensures the two-way communication in the solver.

3. Verification

3.1. Test case definition and analytical solution

In order to verify our solver, we consider a clamped-free square base beam (figure 3) with a length \( L = 1 \) m, a thickness \( h = 0.06 \) m and a width \( b = 0.06 \) m. The density of the beam is \( \rho_S = 2550 \) kg \( \cdot \) m\(^{-3}\), the Young modulus is \( E = 2.5 \cdot 10^5 \) Pa and the Poisson coefficient is \( \nu_{\text{Poisson}} = 0.35 \). The mesh in figure 3 corresponds to the M1 solid mesh with a tetrahedral fluid mesh used for the simulations in the next section.

This geometry was initially described by M. Glück et al.[10], and K. Namkoong et al.[11] in 2001 and 2005 and was studied as a verification case by T. Liaghat [12].

![Coupled test case geometry and tetrahedral M1 mesh](image)

**Figure 3.** Coupled test case geometry and tetrahedral M1 mesh

Analytically, the first bending modes are given by Naudascher and Rockwell in section 2.6 of [8]:

\[ \omega_N = \frac{c_N^2}{L^2} \sqrt{\frac{EI}{\rho_A A}} \quad \text{with} \quad I = \frac{b \cdot h^3}{12} \quad \text{for a rectangular section and} \quad A = h \cdot b \]  

(4)
The first modal coefficient for a clamped-free beam is \( c_1 = 1.875 \), which gives the natural frequency for first mode of a clamped-free beam: \( \omega_1 = 1.907 \text{rad} \cdot \text{s}^{-1} \).

Then, for the solid immersed in a fluid, we can calculate the natural frequencies by taking into account the added mass of the beam (section 3.2 of [8]):

\[
\omega_{N, \text{fluid}} = \omega_N \cdot \sqrt{\frac{1}{1 + \frac{M_A}{\rho_S A}}} \tag{5}
\]

with \( M_A \) the added mass given by \( M_A = \rho_F \cdot \pi \cdot \left( \frac{b}{2} \right)^2 \) and the fluid density \( \rho_F \).

We perform simulations on the solid with the solidDisplacementFoam solver for several hexahedral mesh refinements to calculate the pulsation of the beam. We apply a nominal force of 10 \( \text{N} \cdot \text{m}^{-1} \) on the left side of the solid during 1 s and study the displacements of the point B for several meshes. With our results of the two finest refinements \( \varphi_1 \) and \( \varphi_2 \), we can find the limit pulsation value \( \omega_R \) with a second order Richardson extrapolation [9]. Given the size ratio \( r \) between \( \varphi_1 \) and \( \varphi_2 \), this pulsation \( \omega_R \) is:

\[
\omega_R = \omega_{\varphi_1} + \frac{\omega_{\varphi_1} - \omega_{\varphi_2}}{r^2 - 1} \tag{6}
\]

For the solid-only simulations, we have found a pulsation \( \omega_R = 1.899 \text{rad} \cdot \text{s}^{-1} \) which corresponds to a 0.42\% relative error compared to the analytical value \( \omega_1 = 1.907 \text{rad} \cdot \text{s}^{-1} \).

### 3.2. Verification of the coupled solver: low-density fluid

We simulate a case with the free-clamped beam immersed in a low density fluid, \( \rho_F = 1 \text{kg} \cdot \text{m}^{-3} \). The fluid is meshed with tetrahedral and hexahedral elements. Both hexahedral and tetrahedral meshes have the same cell density, and thus the cell sizes are of the same order. Also fluid and solid cells have the same size at the interface. Equation 5 approximates the analytical first natural frequency of the beam in the low-density fluid: \( \omega_{1, \text{low-density}} = 1.907 \text{rad} \cdot \text{s}^{-1} \).

Initially, we impose a uniform nominal force of 40 \( \text{N} \cdot \text{m}^{-1} \) on the beam for 1 s, then we remove the force and the structure is free to oscillate. Several mesh refinements are considered for our simulations, and pulsation results for each refinement are shown in table 1. From results in table 1 we can calculate the limit pulsation with the Richardson extrapolation from the 2 most refined meshes as shown in equation 6. This gives \( \omega_{R, \text{tet}} = 2.012 \text{rad} \cdot \text{s}^{-1} \) for the tetrahedral mesh, and \( \omega_{R, \text{hex}} = 1.942 \text{rad} \cdot \text{s}^{-1} \) for the hexahedral mesh.

### Table 1. Mesh characteristics and results for a low-density fluid with second order time discretisation

| Mesh | Solid mesh elements on X / on Y | Fluid mesh elements | Fluid mesh elements |
|------|--------------------------------|--------------------|--------------------|
| M1   | 2 / 36                         | 20 082             | 2.121              |
| M2   | 4 / 71                         | 36 420             | 1.925              |
| M3   | 9 / 143                        | 95 733             | 1.988              |
| M4   | 17 / 286                       | 313 773            | 2.007              |
| M5   | 34 / 571                       | 1 156 410          | 2.011              |
| Extrapolated | -                          | 2.012              | -                  |
| Analytical | -                  | 1.907              | 1.942              |
Figure 4 presents the solid displacement at the free end of the beam (point B in figure 3) for the mesh M5 with the tetrahedral fluid mesh.

Hence, with a tetrahedral mesh for the fluid domain, we are able to predict the first analytical pulsation with a relative error of 5.51% compared to the extrapolated pulsation.

For the hexahedral mesh, an error of 1.85% is obtained. However, we could not achieve convergence for the most refined meshes, due to instabilities brought on by the decrease of the time step forced by the decrease of mesh size, as discussed in the following section.

These results suggest that the hexahedral mesh gives more accurate results than the tetrahedral mesh. However, simulations with the hexahedral mesh are less stable.

3.3. Verification of the coupled solver: high-density fluid

We next performed simulations with the same configuration, but for a high-density fluid ($\rho_F = 1000 \text{kg} \cdot \text{m}^{-3}$). Equation 5 approximates the first analytical natural pulsation of the beam in the high-density fluid: $\omega_{1, \text{high-density}} = 1.856 \text{rad} \cdot \text{s}^{-1}$. However, we were unable to achieve convergence using the 2nd order time discretization, because of stability issues. These added-mass instabilities were described by Förster et al. [13] and the conclusion was that they strongly depend on the time discretisation. These instabilities appear sooner in the simulation when the time step decreases. This effect limits the range of possible time steps for our simulations. We used a 1st order time discretization to avoid these stability issues.

Furthermore, we only ran calculations using tetrahedral meshes because calculations are unable to converge with hexahedral meshes.

We obtained pulsation results for the high-density fluid in table 2. The Richardson’s extrapolation gives a limit pulsation $\omega_R = 1.999 \text{rad} \cdot \text{s}^{-1}$.

Figure 5 presents the solid displacement at the free end of the beam for mesh M3 for the high-density fluid.

| Mesh   | $\omega$ [rad $\cdot$ s$^{-1}$] |
|--------|--------------------------------|
| M1     | 2.041                          |
| M2     | 1.926                          |
| M3     | 1.995                          |
| Extrapolated | 1.999                      |
| Analytical  | 1.856                      |

Table 2. Pulsation results for a high-density fluid with a tetrahedral mesh and a first order time discretisation.
The error for the high-density fluid and the tetrahedral mesh is now 5.83%, which is of the same order as the error we had in the low-density fluid with the tetrahedral mesh. The slight difference could be explained by the use of the first order discretisation, which is more stable but less accurate.

4. Conclusion and Outlook

After these verifications on the coupled solver, we observe that we have errors in the prediction of theoretical values of the natural frequencies of an immersed structure in a fluid. Indeed, the calculations conducted on the low-density fluid gave an error of 5.51% compared to the analytical value with a tetrahedral mesh for the fluid domain. But this result was improved by considering a hexahedral mesh for the fluid domain surrounding the solid structure. Such a mesh gave an error of 1.85%. Besides, even if the hexahedral mesh was shown to be more accurate than the tetrahedral mesh, we were unable to get converged simulations using the most refined meshes, M4 and M5. Indeed, the diminution of the mesh size forced us to decrease the time step, which led to the apparition of added-mass instabilities.

With a higher density fluid, stability issues still occur, even with the tetrahedral mesh. The relaxation is not sufficient to overcome these issues and the instability limits the possible time step size. Even if our discretisation scheme is implicit, and so is expected to be stable, the coupling between the fluid and the solid and the increase of the added-mass effect due to the fluid density cause instabilities to appear. Thus, the simulations with the hexahedral fluid mesh and the M4 and M5 simulations with the tetrahedral mesh were unable to converge due to these instabilities.

With the increase of the fluid density, we must take particular care in choosing the simulation parameters. The time discretisation order can be decreased and the time step size can be increased to stabilise the calculations. But in return, these stabilisations have a cost on the accuracy. Hence, the challenge is to find the optimal configuration to conciliate both stability and accuracy criteria.

References

[1] Saeed R, Galybin A and Popov V 2010 Adv. Eng. Soft. 41 1245–55
[2] Coutu A, Monette C and Velagandula O 2007 Hydro 2007
[3] Coutu A, Proulx D, Coulson S and Demers A 2009 Int. J. Num. Methods in Fluids 60 515–38
[4] Seeley C, Coutu A, Monette C, Nennenmann B and Marmont H 2012 Smart Mat. Str. 21 035027
[5] Liaghat T, Allenbach L, Guibault F, Nennenmann B and Avellan F 2013 5th Int. Workshop Cav. Dyn. Prob. Hydr. Mach. 40
[6] Ha S T, Ngo L C, Saeed M, Jeon B J and Choi H 2017 J. Mech. Sc. Tech. 31 281–7
[7] Naseri A, Lehmkuhl O, Gonzales I and Oliva A 2016 J. Phy. 745 5
[8] Naudascher E and Rockwell D 2012 Flow-induced vibrations: an engineering guide Courier Corp. 28
[9] NPARC Alliance CFD Verification and Validation 2008 Examining spatial (grid) convergence NASA [Online] https://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html
[10] Glick M, Breuer M, Durst F, Hallmann A and Rank E 2001 J. Wind Eng. Indus. Aerodyn. 89 1351–68
[11] Namkoong K, Choi H and Yoo J 2005 J. fluids str. 20, 51-69
[12] Liaghat T 2014 Master Thesis, École Polytechnique de Montréal
[13] Förster C, Wall W A et Ramm E 2007 Comput. methods appl. mech. eng. 196 1278-93