Simulation of Fluid Structure Interactions (FSI) in Hydraulic Machinery

1. Introduction

Though, usually kept as a secret there are known damage cases in hydraulic machines, mostly blade cracks, caused by fluid structure interaction phenomena, see [5]. Either overloading the structure or periodic excitations of eigen frequencies of the turbine blades or other parts can result in material failure or at least noise development [3], [5] and [7]. The former is mastered quite reliably today, whereas, the latter turns out to be still a challenge. One typical problem of periodic excitation arising here is the von Karman vortex street, which, under certain conditions, emerges behind almost every body in flowing fluid. Regarding turbo machinery this body is the turbine or the stator blades, see figure 1.

![Figure 1: Pressure field behind a wing profile (NACA0012) with thick trailing edge](image)

Other periodic fluctuations in the pressure field causing dynamic loads on the structure may result from rotating stall in pump impellers, rotor stator interactions or pressure surges due to draft tube vortex ropes. Another reason is the interaction between the blades of wind or sea current turbines and the pressure field next to the pylon, which is different from the surrounding due to stagnation point effects.

All applications listed above have one thing in common; the dynamics of flow and structure are changed due to interactions between these two physical fields. But the frequencies and the amplitudes arising are very different. Due to the high frequencies of the vortex shedding von Karman streets usually excite the higher order eigen modes with high stresses, but the amplitudes are small. Rotor stator interactions exhibit higher amplitudes but lower frequencies exciting lower order modes. This means there are different strategies for the coupling and solution procedure required.

2. Theoretical background

From the physical point of view fluid structure interactions are a two field problem; one flow and one structural field. The fields are coupled via the so-called wet surface where the pressure and friction forces caused by the fluid are acting on the structure. Due to these loads the material is deformed and the deformation changes the boundaries of the flow domain.

Numerically seen, these two fields are coupled via a third field, the numerical meshes. The structural deformations are transferred to the fluid via adapting the mesh representing the flow domain, see figure 2 and [6].

There are two basic methods for the coupling procedure of the two physical domains. One is the monolithic (simultaneous) the second is the partitioned approach. The former results in one single system matrix including all fields and couplings between the fields. This matrix might be huge and ill-conditioned, [6], which leads to difficulties in the most equation solvers. Furthermore, it requires one single model for both structure and fluid. This can be a
challenging task especially for the treatment of complex problems. One the other hand the coupling between the fields is strong and accordingly the solution is robust. The latter approach allows the usage of proven and established simulation codes for flow and structure simulation. Furthermore, these codes can be replaced easily and the models can be set-up separately from each other. The solution of the smaller system matrices results in a faster but also less stable solution. Hence, for strongly coupled problems stabilisation techniques or the simultaneous approach have to be applied.

Additionally, both approaches require a formulation of the Navier-Stokes equations considering the movement of the mesh. The derivation of this ALE-formulation (Arbitrary Lagrange Euler formulation) can be found in [6].

\[
\frac{\partial \mathbf{u}}{\partial t} + c \cdot \nabla \mathbf{u} + \frac{1}{\rho} \nabla p - \nu \nabla^2 \mathbf{u} + \mathbf{f} = 0 \quad \text{momentum with } c = \mathbf{u} - \mathbf{u}_{\text{Grid}}
\]

\[
\nabla \cdot \mathbf{u} = 0 \quad \text{mass conservation, incompressible}
\]

Regarding the application there are different levels of complexity. The easiest case is the steady state one-way coupling, i.e. transferring the static pressure loads from the fluid to the structure for one analysis, see [1] and [2]. To find the equilibrium state a two-way coupling approach with transfer of deflections is necessary. This requires an iteration between structure and fluid solution. Analogously unsteady simulations can be performed.

3. FSI projects at IHS

The projects until today at IHS concerning FSI include only one-way couplings with and without equilibrium iteration. One example is the 2D channel flow with an elastic obstacle, see figure 3.

The computational grid consists of 31200 elements and is adapted manually after each flow computation. The obstacle is modelled as an Euler beam. A steady state solution is found after four iterations.
Another example is the 3D Simulation of the loads on an axial runner turbine blade. The static pressure loads come from a 3D flow simulation using the IHS simulation code FENFLOSS. Via a matching surface grid, the loads are transferred to the structure. The structural analysis is performed with the commercial code ABAQUS using linear elements. Additional loads come from the centrifugal forces. In order to reduce the computational effort, the geometries are modelled as periodic, figure 4.

Figure 4: Computed pressure on blade surface (left) and structural domain (right).

The computations with the loads described above yield maximum stresses next to the trailing edge in the hub region. But the magnitude of the stresses, see figure 5, is not high enough to produce severe material damage.

Figure 5: Stresses on axial runner turbine blade under pressure and centrifugal loads.
The results above imply that the reason for the material failures found in the past come from periodic loads and vibrations. The latest project deals with this issue. As example a NACA0012 wing profile is excited by von Karman vortices at the thick trailing edge, figure 6.

![Wing profile with thick trailing edge (left), computational domain (right).](image)

It is found that the periodic pressure fluctuations excite the first eigenmode (21.8 Hz) of the extruded wing geometry, which leads to an increasing amplitude. Due to the one-way coupling the fluid damping is not considered yet.

4. Current projects and outlook

In ongoing projects the procedures from above are extended to simulate the unsteady excitation and vibration of a real axial runner turbine blade. The challenge is the simulation of the high frequency vortex shedding (approx. 900 Hz) in 2D and 3D and the conservative transfer to the structural domain. For the eigen analysis and the transient simulation the fluid is modelled to consider the added mass effects changing the eigen frequencies of the structure.

For future two-way calculations the FENFLOSS code is enhanced with an API (application program interface), that allows coupling with structural codes via the commercial MpCCI software. Furthermore a grid deformation algorithm has to be implemented. One application will be in the field of axial fans and sea current turbines.

5. References