SIMULATING FLUID-STRUCTURE IINTERACTION FOR MIXING DEVICE DESIGN

Ralf Löffler, Fluent Deutschland GmbH, Darmstadt, Germany
Dr. Mark Pelzer, Fluent Deutschland GmbH, Darmstadt, Germany
Luke Munholand, PhD., Fluent Inc., Lebanon, NH

Introduction

The changes in chemical production processes, in particular the shift from chemical production to biotechnology, puts more emphasis on shear rate control of mixing devices. At the same time, mixing devices should be highly optimized with respect to power consumption, both for economical reasons and due to the prevention of additional heat-up of the mixing process. In addition, most biotechnological mixing processes include a two-phase fluid consisting of a liquid phase and a gas phase.

Computational fluid dynamics (CFD), has been established as a methodology that helps design mixing devices in the past years. Under the above conditions, however, the fluid flow simulation may no longer be independent of the associated structural deformation problem:

- the agglomeration of the gas phase in the wake of the impeller blades can significantly change the dynamic structural load of the impeller, even leading to premature fatigue failure of the impeller;
- for critical impeller blade designs, a moderate torsion of the blade may already significantly change the shear rate exposition near the leading edge of the impeller, thus leading to a decrease in the production rate of the microorganisms.

This paper will therefore explain the *procedure* to perform a coupled Fluid-Structure Interaction (FSI) simulation, consisting of a part including the fluid flow simulation with the CFD code FLUENT, and a structural analysis part based on Finite Element Analysis. Particular focus will be given to numerical aspects influencing the quality of the results, such as meshing, coupling methodology, physical models used etc. In addition, the computational effort required for this kind of simulation will be reported and compared to the potential gain of the coupled simulation, i.e. increased product quality or output and enhanced device reliability.

Numerical Methodology – CFD, FEA, FSI Coupling

Computational Fluid Dynamics (CFD) is a well-known numerical methodology to predict the behavior of fluid flow and related phenomena such as heat transfer and transport of chemical species. The basic idea behind CFD is pretty much the same than with the Finite Element Analysis (FEA): if physical phenomena are being described by equations which are for some reason not solvable analytically, numerical approaches can be used to solve particular problems.

Computational Fluid Dynamics

As far as CFD is concerned, the basic equation for CFD, the Navier-Stokes-Equation, can be solved analytically only for very simple types of flow. Therefore, numerical methods to predict the flow behavior for more complex types of flow have become very common, and among them the Finite Volume Method is the most frequently used one. The basic idea is that one can subdivide the total flow domain into smaller parts (the cells in the numerical grid) and just balance the inflow and outflow of the neighboring cells. In theory, generally speaking the finer the numerical grid, the closer this approach would be to the exact solution – very similar to the trapezoidal rule to calculate the area below a functional graph.

CFD results basically consist - for 3D fluid flow problems – at least of the solution quantities for pressure and the velocity vector, and eventually additional scalar quantities for turbulence, temperature, species concentrations etc..

Finite Element Analysis

As far as Finite Element Analysis is concerned, the finite element method works similar, with the exception that it is nodal based rather than cell based. In addition, the structural mechanics problem could usually potentially be solved analytically for single parts of the structure – it is then the complexity of geometry rather than the basic equation which advocates for a numerical solution.

Typical solution quantities are stresses, strains, moments etc..

Coupling

The ***physical*** coupling between CFD and FEA is first given by the loads which are introduced through the flow onto the structure – pressure acting on walls, eventually thermal stresses etc.. If flow loads are not significant, coupling is then a one-way road which ends here. However, if the flow loads are high, the structure could deform significantly and force a new flow pattern. And sometimes the movement of the walls could be the major driving force for a flow, e.g. the flight of birds and insects. This represents a two-way coupling then.

It is obvious that as long as it is unclear whether the fluid flow loads are significant, a cautious structural design procedure would ask for at least a one-way coupling to figure out the contribution of the flow-induced load to the total structural load. This is actually done quite often, but rather as a rule-of-thumb approach for the dynamic pressure (of the flow) than as a coupled simulation.

Since we recognize that the FEA method and the CFD method are similar in how they work (using a numerical grid, producing results values for all grid positions), one can imagine that coupling between the two methods is not too difficult in principle. However, for a flexible approach many "usability tasks" need to be addressed by coupling methods, like mesh interpolation of results (to allow different grid resolutions for the CFD and FEA simulation, respectively), automatic translation of unit systems (if different systems are used for FEA and CFD), and in the case of two-way coupling the staggering of CFD and FEA calculation loops, etc. etc..

These "usability tasks" are usually addressed by the coupling interfaces, either external ones like MpCCI or internal ones like the ANSYS Workbench coupling tools.

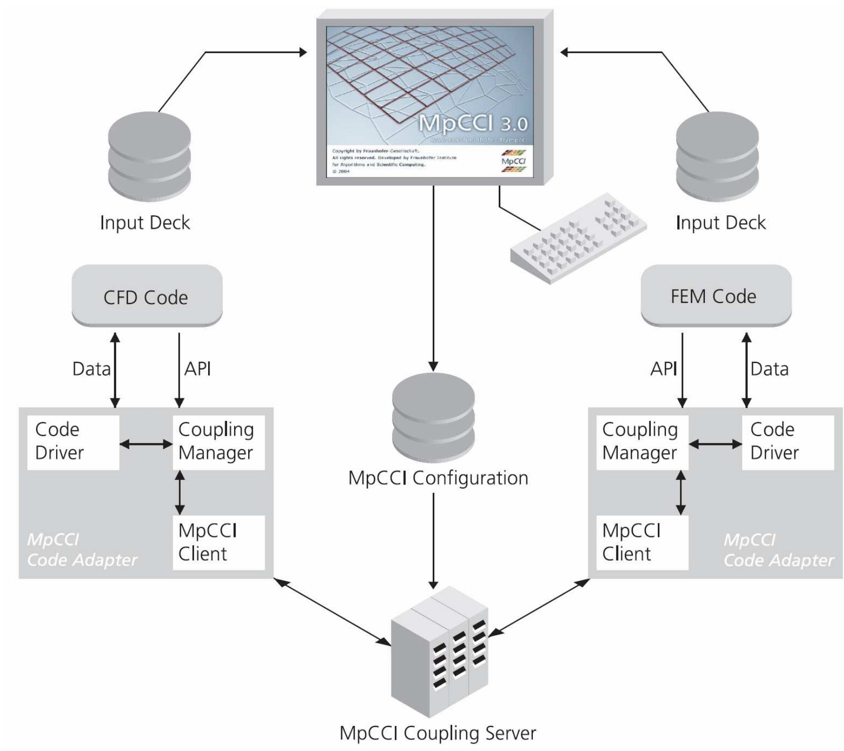

Figure 1: MpCCI Coupling software – Working schematics

Procedure for Coupled FSI Mixing Tank Simulations

CFD Simulation

The CFD simulation of a mixing tank can be regarded as a standard task – CFD codes like FLUENT and its sister code MixSim provide all functionality required to successfully perform such simulations.

Earlier papers emphasize that for accurate results care should be taken regarding the choice of turbulence models (if necessary) and the mesh resolution in particular around the blades of the mixer to properly capture the surface pressure and shear stresses, which feed into the results for momentum of the shaft etc.

Two basic approaches to simulate the movement of the mixing device within the mixing tank are possible: the stationary one with a so-called multiple reference frame approach assumes that the relative position of the mixing device with respect to e.g. baffles is not important for the flow field and can therefore be neglected. The transient one with the so-called moving mesh does in fact take the relative motion into account and fully represents it in the computational domain.

If applicable, obviously the stationary approach is much more efficient, since one does not need to perform a transient simulation. If a coupled FSI simulation is intended, this is even more advantageous in terms of simulation effort. However, one has to remind that on the structural side fatigue-related failure is usually more critical (and also more likely) than static failure, so despite of the higher effort transient FSI simulations may be much more relevant. This is particularly true for the two-phase problem including bubble agglomeration mentioned above, since periodic or transient detachment behavior of agglomerated gas, dependent on the local concentration and resulting lift forces, can be assumed.

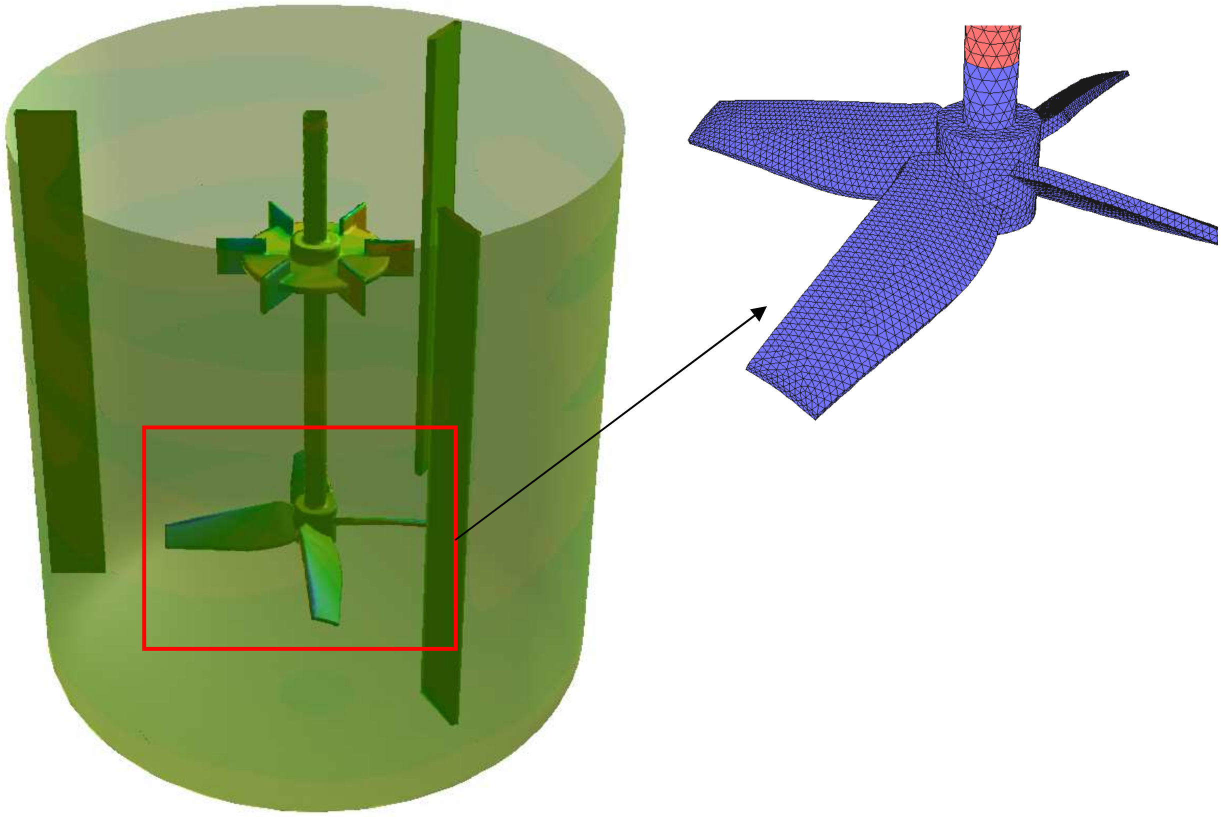

Figure 2: FLUENT CFD simulation model; pressure contours, mesh on impeller

FEA model

The FE analysis is done based on the geometry model created by MixSim, utilizing the geometry export functions Fluents preprocessing tools. Apart from the standard structural loads (gravitational forces, impeller rotational forces), particular emphasis is put on the deformation of the impeller blades due to the pressure and stress exaggerated on the impeller by the fluid.

The deformation of the blade is then returned to the CFD simulation to check whether it influences the maximum shear rate.

CFD: pressure, viscous stresses	→	FEA: Deformation?	→	CFD: *changed* (?) pressure, viscous stresses

All coupling activities are done in an automatized manner by utilizing the MpCCI interface.

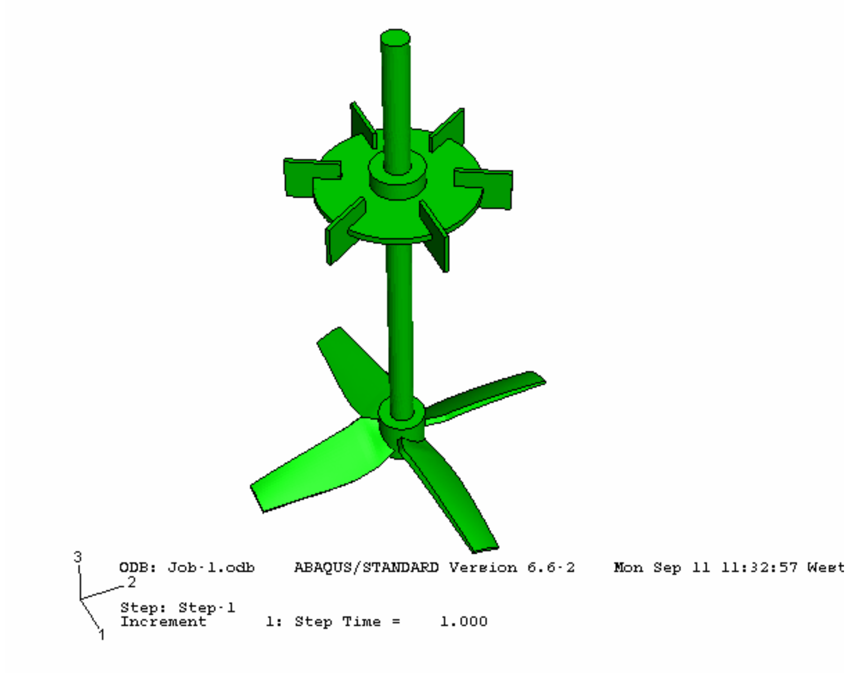

Figure 3: FEA model of the impeller configuration

Results and Discussion

Results for simple sample configurations under different loads are presented in the talk, including a description of the total computational effort for the isolated simulations (CFD and FEA) and the overhead effort of the coupling as such.

Roughly speaking, the additional effort for a coupled FSI approach is directly related to the amount of physical coupling of the problem – the bigger the interaction, the higher the effort to perform the coupled simulation. Since timescales of the coupling effects can be significantly different from the natural timescales of the isolated CFD and FEA problem – and since a coupled simulation needs to resolve all timescales involved – heavily coupled problems are challenging in terms of effort required.

However, one-way coupling of CFD and FEA as a first step can often help to estimate the influence of fluid flow phenomena on the structural response of the system, therefore allowing the gain of additional knowledge or security in design with modest effort.