11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

MODELLILNG OF INTERACTION BETWEEN SUSPENSION AND STRUCTURE IN A TUMBLING MILL

S. Hammarberg*, **S. Larsson*** **and P. Jonsén**†

* Department of Engineering Sciences and Mathematics, Luleå tekniska universitet, 971 87 Luleå
† Department of Engineering Sciences and Mathematics, Luleå tekniska universitet, 971 87 Luleå, Sweden, Par.Jonsen@ltu.se and http://www.ltu.se/staff/p/parj-1.12049

Key words: Fluid-structure interaction, Free surface, ICFD, Tumbling mill.

Abstract. Fluid-structure interaction (FSI) and free-surface flow problems occur in many engineering applications. FSI problems includes interaction of deformable and or moveable structures with surrounding or internal fluid flows. Development of accurate methods to simulate FSI would have many benefits in numerous industrial applications e.g. reducing the need for experimental testing and recent efforts have been made in this field. Hydro power turbines, aerodynamics of wind power turbines and lubrication of mechanical components are examples of applications where FSI play an important part. This work investigates the possibility to use the new Incompressible Computational Fluid Dynamics (ICFD) solver implemented in the R7.1.0 version of LS-Dyna. The studied case is a tumbling mill, used in the mining industry, partly filled with different fluids. The interaction between the rigid cylinder casing and the fluid inside the mill, but also the behaviour of the free surface are studied topics. Modelling of wet milling is a complex multi-physics problem and usually a combination of different numerical methods are used.

One of the main purposes of this work was to investigate how well the ICFD solver in LS-Dyna could handle free surfaces and reproduce the behaviour of two different fluids. Different rotational velocities for the grinding mill and pulp viscosity were evaluated. From performed simulations a comparison between experimentally measured torque on the mill casing and torque calculated with the ICFD-solver was done. For lower rotational velocities the results show closer agreement, with increasing velocity and dynamic viscosity the error also increase.

The ICFD-solver shows good potential in handling FSI and free-surface problems when it comes to results and calculation time. Compared to other previously used methods for solving complex FSI problems, such as smoothed particle hydrodynamics (SPH), the ICFD-solver can be beneficial with shorter computational time.

1 INTRODUCTION

In numerical modelling of physical systems many aspects affects the accuracy of a mechanical response computation, for example: the smoothness and stability of the response, the inadequacies and uncertainties of the constitutive equations, the initial and boundary conditions and the uncertainties in the load. Such analyses of the computability of nonlinear problems in solid mechanics were investigated by Belytschko and Mish [1]. Validation of the models is, therefore, important for building confidence in numerical results. Including the influence of fluids in grinding mill models are important for future understanding and optimizations of grinding circuits. In the current work, the behavior of fluids in a tumbling mill is studied and validated. Validation is done by comparing numerical results with experimental measurements from grinding in an instrumented small-scale batch mill equipped with an accurate torque meter.

For a long time, when using computers for simulations, the equations of fluid mechanics have been based on the Eulerian formulation for continuous domains. This formulation, however, does not produce good enough results when analysing problems where the shape of the interface changes continuously or when analysing interaction problems between fluids and solid structures including free-surfaces involving complicated contact problems. Over the last ten years methods describing the domain by an arbitrary set of particles have raised a lot of interest due to the use of a Lagrangian formulation.

As fluid structure interaction (FSI) problems are of great interest, this work will investigate the possibility to use a new solver for these types of problems, namely the LS-DYNA R7 ICFD-solver. To evaluate the solver, a tumbling mill process used in the mining industry, partly filled with fluids of varying viscosity is used. For the tumbling mill the interaction between the rigid cylinder casing and the fluid inside the mill, together with the behaviour of the free surface are studied. In particular, the torque as a function of the angular velocity is of interest. Finally, the obtained results from the simulations will be compared to those obtained via experiments.

2 Theory

The results of this article is based on simulations using LS-DYNA'S Incompressible fluid solver also known as R7.1.0 [2], a solver which aims to solve the Navier-Stokes equations for incompressible flows.

As incompressible flows are studied in a steady state process, the mass continuity equation simplifies to the following volume equation:

$$\nabla \bullet (\overrightarrow{u}) = 0 \tag{1}$$

The governing equations are the Navier-Stokes equations, where the conservation of mass and energy are taken into account, combined with the continuity equations. After some simplifications the following set of equations are obtained, for details concerning the simplifications, see [2].

$$\rho(\frac{du_i}{dt} + u_j\frac{\partial u_i}{\partial x_j}) = \frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j \partial x_j} + \rho f_i \quad \text{in} \quad \Omega \tag{2}$$

$$\frac{\partial x_i}{\partial x_i} = 0 \quad \text{in} \quad \Omega \tag{3}$$

These differential equations are incomplete without proper boundary conditions and initial conditions. Discussions concerning proper boundary and initial conditions can be found in the literature and also in [2].

In standard CFD simulations the mesh is based on an Eulerian formulations, saying that the mesh is fixed in space and the fluid flows through the mesh. For more complex problem, such as an FSI-problem, the boundary between the fluid and the structure may be described with a Lagrangian description which deforms with the structure. To handle the combination of the Eulerian and the Lagrangian formulation y an arbitrary Lagrange-Eulerian (ALE) formulation is used. The benefit is that a strong and exact imposition of the solid boundary condition on the fluid is possible. The equations of motion are rewritten for the ALE formulation and can be found in [2].

The time integration of the Navier-Stokes equation is done by applying the Fractional Step Method. This method will not be discussed here and the interested reader is referred to [2] and the references there within.

As FSI was studied which involves moving interfaces where the structure penetrates the fluid. Due to this, a method must be used to keep track of the interplay between the interfaces. A reliable way to solve this problem is to use a level set method. The idea is to introduce an implicit function ϕ whose zero isocontour, ϕ =0 represents the interface. For instance, the domain where Navier-Stokes equations will be solved is defined by ϕ >0 and the vacuum is defined by ϕ <0. The function ϕ will be described by a distance function resulting in the level set function [2],[3].

An example of the level set method is illustrated in Figure 1. For further reading concerning the level set function and its applications in LS-DYNA's ICFD-solver the reader is referred to [2].

For controlling the the mesh an automatic volume mesher is used for the fluid domains, which simplifies the pre-processing stage. However, a good quality body fitted mesh has to be provided. With the use of the ALE formulation the mesh can be recalculated for large deformations when FSI-problems are analysed, keeping an acceptable quality of the mesh. For further reading concerning meshing and the re-meshing strategies used in the ICFD-solver, see [2].

3 Experimental Setup

The experiments were made using a laboratory scale mill instrumented and developed for accurate torque and rotational speed measurements [4]. This mill has a stainless steel drum measuring ∅ 300x450 mm with four equally spaced lifters. The rotational speed is

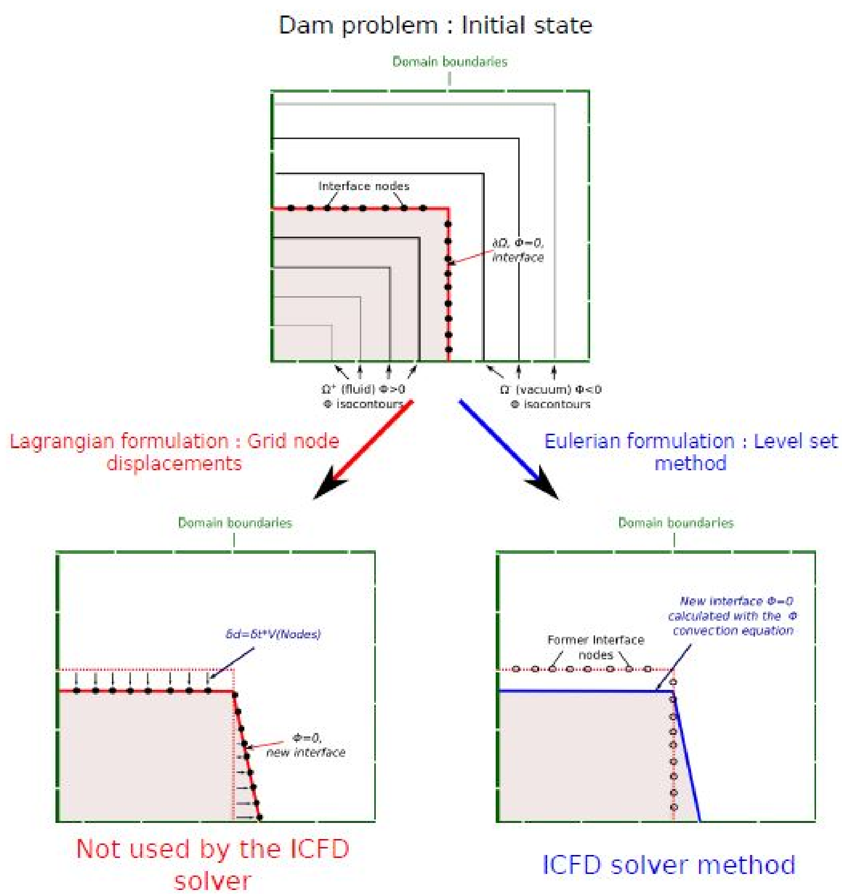


Figure 1: Free surface dam breaking with interface tracking methods [2].

Table 1: Mesh statistics

Problem	Element type (Structure)	Nr. of elements (Structure)	Nr. of surface elements (ICFD/Fluid)
Mill	Beam	960	143 178

maintained by a closed loop regulator and can be set between 10 and 100 rpm. Critical rotational speed (Nc) for the mill is 77 rpm. The torque applied to the mill charge is measured as a reaction force applied to a load cell a distance from the centre of rotation. The uncertainty in the average torque measurements is estimated to be less than ± 2 % for typical load cases. An inductive proximity sensor is measuring the rotational speed of the mill. During the experiments data is sampled at 100 Hz and average torque was calculated both as a function of time, and also as a function of mill angular position. When the torque is averaged on angular position the mill revolution is divided into 180 bins, each representing 2° of the mill rotation.

4 Modelling

As mentioned above a tumbling mill, used in the mining industry, was to be simulated using LS-DYNA's ICFD-solver. The mill has a diameter of 300 mm and is filled with a fluid to 30 % of its volume, corresponding to approximately a third of its total height, as Figure 2a illustrates. The required boundary conditions can be applied to the solid, in this case a rigid body angular velocity. The rigid body is modelled to fit in the empty space in the volume mesh discussed in the previous section. Solid and fluid are independent and the software uses a partitioned approach for FSI coupling [5]. Initial data and mesh properties for the cylinder problem are listed in Table 1.

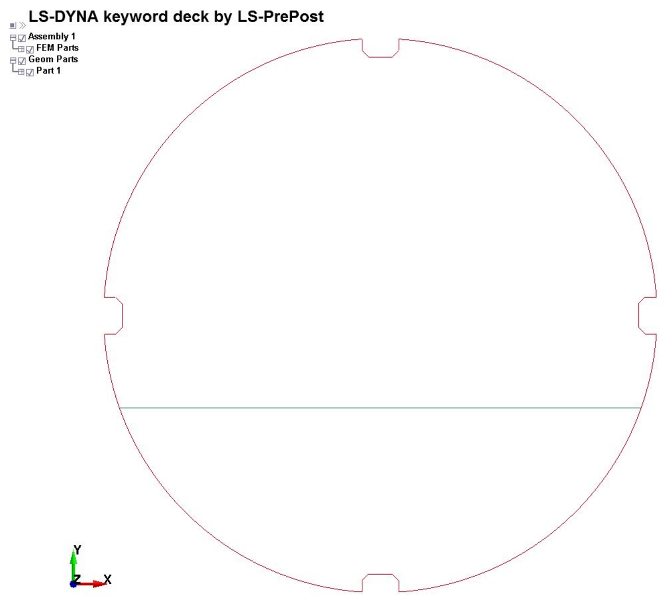
The mill casing was modelled as a rigid body and with beam elements. The mill has a diameter of $300mm$ and is filled with fluid to 30 % of its total volume, corresponding to approximately a third of the total height when viewed from the side as in the 2D model, see Figure 2a.

Initial data for volume mesh generation and mesh statistics for the mill problem are listed in Table 1. The volume meshes generated by the ICFD solver are displayed in Figure 2b.

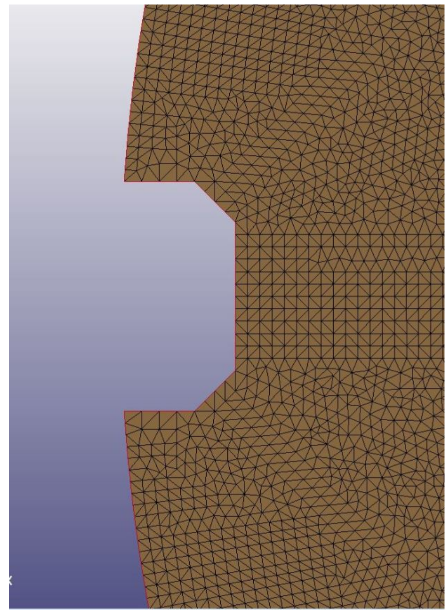
5 Results and Discussion

For the rotating mill the average torque was calculated for two different cases, the first using a dynamic viscosity of $\mu = 267$ [mPas] for the magnitite pulp and the second using water. The same rotational velocities were used as the ones used in the experiments. In the numerical modes it was possible to extract the moment around the z-axis, here interpreted as the driving torque.

In the numerical model a simulation time of 5 seconds was used throughout the calculations, after this time the solution was considered to be close to steady state.



(a) Model of mill.

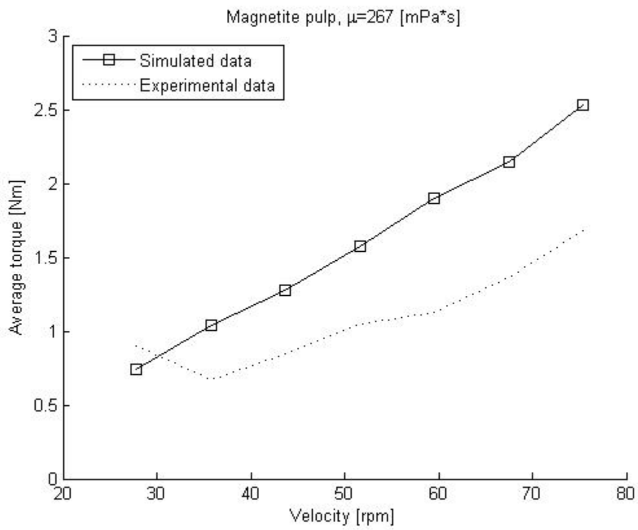


(b) Mill volume mesh.

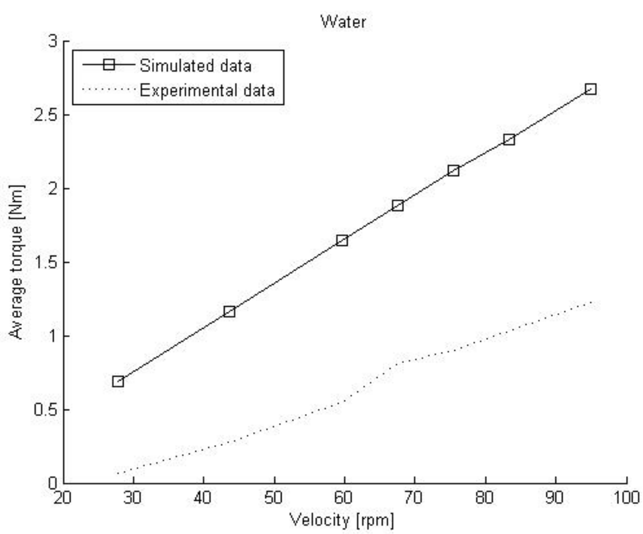
Figure 2: Geometry and volume mesh for the mill.

For a number of different rotational velocities a comparison between experimentally obtained values and simulated torque for the water and magnetite pulp was done, see Figure 3. In the case with magnetite pulp and in the case with water as fluid it is clear that the calculated results are greatly deviating from the experimentally obtained results. However, the slope of the experimental plots are roughly the same as the slope of the numerical plots, implying a consistent error. Perhaps originating from the FSI-contact.

It is noted that for the lowest rpm, in the magnetite pulp case, the calculated and experimentally measured torque are rather close in comparison, this single point is not in any way enough to draw any conclusions though. From the rather long simulation time, 5s, it is reasonable to assume a solution in steady state. Worth to mention is that the experiments were carried out over a much longer time span, 180 s, and the results were then averaged. Simulation data is not of acceptable quality at the present, there is a strong deviation from the experimentally measured values. Some of the deviation between experiments and the numerical model may occur due to the fact that a 2D numerical model is used to approximate the 3D-case. As this work is an ongoing process, further efforts will be made to optimize the numerical model, to a point where the sought after variables are obtained with satisfactory accuracy.

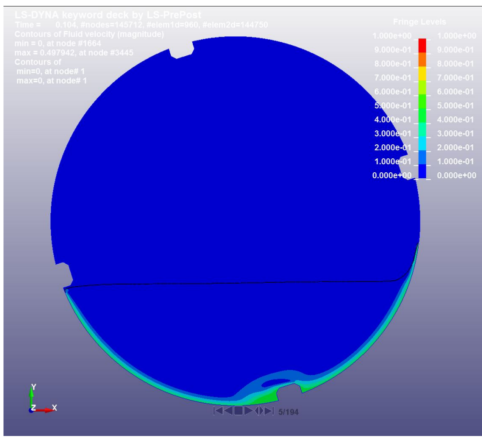


(a) Magnetitepulp with μ=267 [mPas]

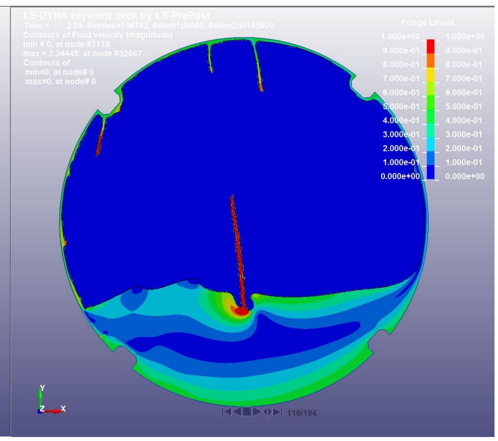


(b) Water dynamic viscosity, μ=0.001 [mPas].

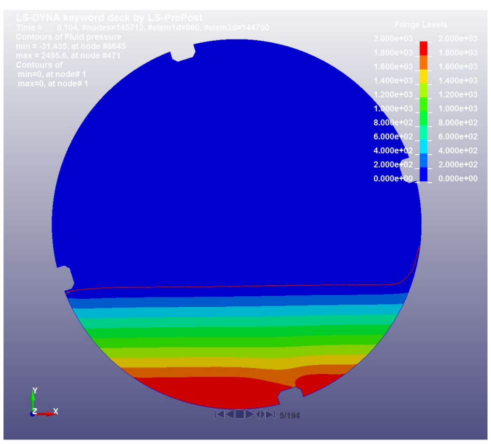
Figure 3: Torque as function of velocity for magnetitepulp and for water.



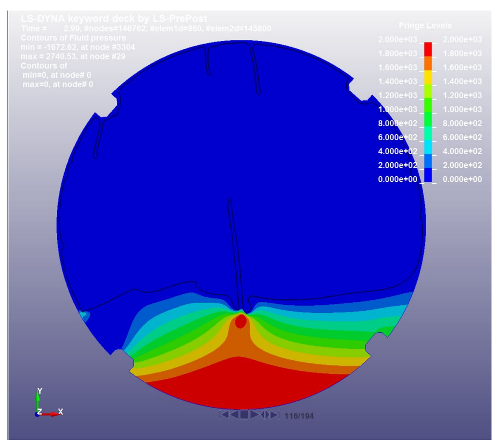
(a) Fluid velocity, magnetite, t=0.1s



(b) Fluid velocity, magnetite, t=3s

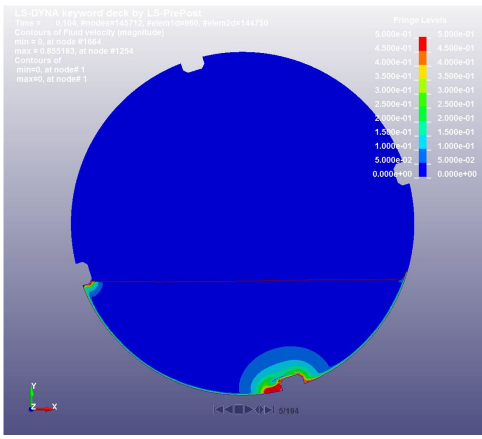


(c) Fluid pressure, magnetite, t=0.1s

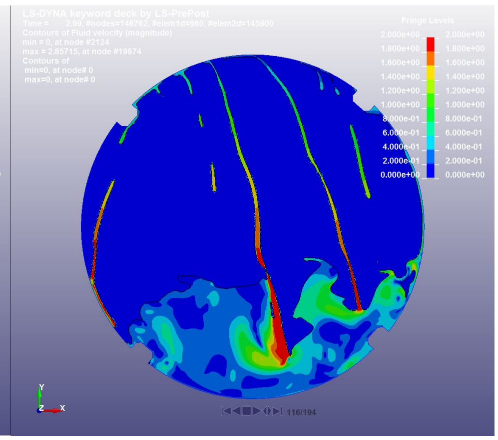


(d) Fluid pressure, magnetite, t=3s

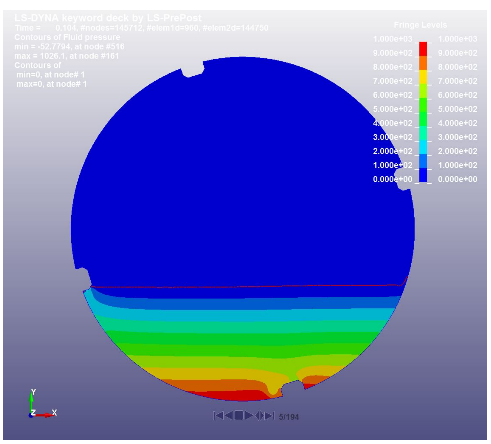
Figure 4: Fluid velocity and pressure during the 28 rpm, counterclockwise, run.



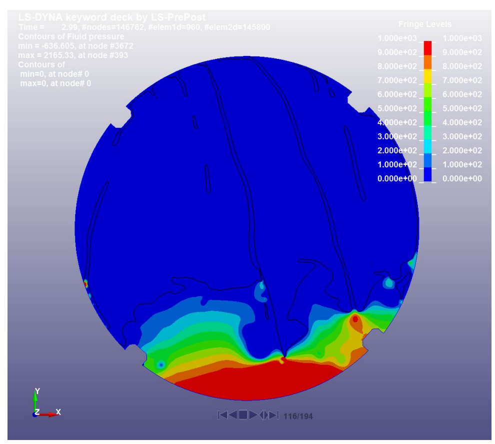
(a) Fluid velocity, water, t=0.1s



(b) Fluid velocity, water, t=3s



(c) Fluid pressure, water, t=0.1s



(d) Fluid pressure, water, t=3s

Figure 5: Fluid velocity and pressure during the 28 rpm, counterclockwise, run.

5.1 Fluid velocity and pressure

Even though no experiments were carried out in measuring the fluid velocity or pressure inside the mill it is interesting to have a brief look to these results just to show how the ICFD solver handles free surfaces and FSI. As the mill rotates the development of fluid velocity and pressure is presented in Figures 4a-4d, in these figures the fluid is the magnetite suspension with dynamic viscosity of $\mu = 267$ [mPas]. Note how the free surface is captured and also note that the solver is able to handle fluid that separates from the initial fluid domain, as droplets in Figure 4b. The pressure, Figures 4c and 4d, seems to be rather stable and physically correct during the simulation. In Figure 5a-5d the free surface handling is presented with water as fluid. From a comparison between the fluid velocity contours it is clear that with a higher dynamic viscosity the free surface is not as violent and disturbed as in the case with lower dynamic viscosity. The magnetite suspension is more viscid and slow-flowing than water, which is captured well by the ICFD-solver.

6 Conclusions

The ICFD-solver shows promise in solving FSI-problems, when it comes to the results and the time it takes to run the simulations. As it is a newly implemented code it is not surprising that it suffers from some diseases, as most new software do.

The results obtained for the mill are fairly similar to those obtained by more established solvers. When comparing the tumbling mill simulations for different fluids it is clear that a heavier fluid gives a larger pressure field in the fluid. When comparing fluid velocity it is noted that a high viscous fluid, such as the magnetite suspension, will indeed flow slower than a less viscous fluid, such as water.

A fluid structure interaction problem with a rotating geometry is a difficult problem to model, and the results shows that with increasing rotational velocity the problems with mesh deformation tends to increase. The tumbling mill problem shows promising results with lower rotational velocities and the torque calculated from simulations is fairly similar to the experimentally obtained results.

The purpose of this work was to evaluate LS-DYNA's ICFD-solver R7.1.0, to see if it was suited for solving FSI-problems and free surface flow of a tumbling mill. The success of using the ICFD-solver greatly depends on what type of FSI-problem one is trying to solve. The more complex a problem, the easier it seems to be to encounter numerically difficulties when using the ICFD-solver.

The simulation is able to complete for every angular velocity tried in the report, and the simulated results can be compared to those of the experiment. The conclusion is then that the ICFD-solver is a contender for solving problems of this kind.

7 Acknowledgement

For financial support of the project ValidMill, the Hjalmar Lundbohm Research Center, LKAB and Boliden Mineral AB are gratefully acknowledged. Luleå University of

Technology is also gratefully acknowledged for financial support.

REFERENCES

[1] T, Belytschko, T., and K. Mish, 2001. *Computability in non-linear solid mechanics*, Int. J. Numer. Meth. Engng, 53, 3–21.

[2] Livermore Software Technology Corporation (LSTC), California, Livermore. *ICFD THEORY MANUAL Incompressible fluid solver in LS-DYNA, 2013.*

[3] S. Osher, R. Fedkiw. *Level Set Methods and Dynamics Implicit Surfaces.* Springer, 2003.

[4] P. Jonsén, J.F. Stener, B.I. Pålsson and H.-Å. Häggblad, 2013. *Validation of tumbling mill charge induced torque as predicted by simulations.* Minerals and Metallurgical Processing, 30, No. 4, 220-225.

[5] F. Del Pin, I Caldichoury, *Incompressible CFD solver (ICFD) and FSI in LS-DYNA: Introduction and application.* LS-DYNA -Class Notes. Livermore Software Technology Corporation, 2013.