Numerical Modeling and Validation Concept for Acoustic Streaming Induced by Ultrasonic Treatment

Donato Rubinetti[1] Daniel A. Weiss[1] Jonas Müller[2] Arne Wahlen[2]

***Abstract*—Acoustic streaming (AS) denotes the effect where a flow in a fluid is driven by the absorption of a sound wave. In order to have a significant effect in practice, the sound wave requires a high frequency (typically order of ultrasound) and high amplitude. In metal processing industry this treatment is applied to obtain grain morphology adjustments during the solidification of metal. Improvement and further development of this technique focus on numerical modeling to reduce substantial costs for test rigs and field tests.**

This study presents a numerical model which takes into account acoustics and fluid dynamics. An oscillating sonotrode is placed into the fluid, emitting high-amplitude into the fluid. The distribution of the resulting sound field induces a body force described by an additional source term in the momentum balance of the fluid.

The results have been experimentally validated with a small-scale laboratory model using water, seed oil and glycerin as sample fluids. By adjusting material properties a variety of fluids such as molten metal can be simulated. The concept presented for the numerical modeling of AS is numerically stable and appropiate. It can be adapted to related applications involving sound driven fluid motion.

***Keywords*—Acoustic Streaming, Attenuation, Pressure Acoustics, CFD, Time-Averaged Variables.**

I. INTRODUCTION

Metal processing industry applies the acoustic streaming ultrasonic treatment for grain morphology adjustments during the solidification of metal. By placing a sonotrode as sound emitter operating at high frequencies (typically ∼20kHz) into a liquid material a steady fluid motion is achieved. Research and further development of the AS treatment focus on numerical modeling which poses a multiphysics problem. It includes the challenging coupling of high-frequency acoustics and fluid dynamics with timescales significantly below the ones of acoustics. The purpose of this study is to conceive an experimentally validated numerical model for adaptation to other fluids by changing the material parameters.

II. PHYSICAL MODEL

The physical model is based on the inhomogeneous Helmholtz equation for the acoustic part and on incompressible Navier-Stokes equations for the fluid dynamics. By segregating the frequency domains into high-frequency for acoustics and low-frequency for fluid dynamics compressible and incompressible studies are applicable. This circumstance is commonly misused in literature. The acoustics is calculated within a time-harmonic study which yields complex-valued results. The governing equation for the acoustics writes

$$-\frac{\omega^2}{a^2}p + \texttt{h.o.t} = \nabla^2 p + \frac{4}{3}\frac{i\omega\mu}{a^2\rho_0}\nabla^2 p \qquad (1)$$

with ω being the angular frequency, a the speed of sound, the dependent variable p for pressure, `h.o.t` for *higher order terms*, dynamic viscosity μ and ρ_0 being the constant component of the fluid density.

The fluid dynamic part of this multiphysics application can be evaluated with a common CFD tool. To link the acoustic pressure field to the flow field an intermediate step is taken through the following force term [1], [2]

$$\boldsymbol{F} = \frac{1}{2}\mathrm{Re}\left[\rho_e^*\omega\boldsymbol{u}\right] + \frac{1}{2}\rho_e\left(\mathrm{Re}\left[u_r^*\frac{\partial\boldsymbol{u}}{\partial r}\right] + \mathrm{Re}\left[u_z^*\frac{\partial\boldsymbol{u}}{\partial z}\right]\right) \qquad (2)$$

with ρ_e^* being the complex conjugate of density perturbation, $\boldsymbol{u}$ the particle velocity and its components u_r, u_z in cylindrical coordinates. The model is simplified assuming isothermal behavior, neglecting cavitation and turbulent effects. The latter is sensitive in terms of physical accuracy due to widely spread Reynolds-numbers. Nonetheless it improves numerical stability significantly. Density and pressure perturbations are coupled in the acoustics part which requires a compressible fluid description. In the stationary flow evaluation compressibility has no relevance.

III. NUMERICAL MODEL

The following numerical model is created using COMSOL Multiphysics 5.2. For the acoustics part the *Pressure Acoustics Frequency Domain*-interface is selected. The fluid flow is defined by the *Laminar Flow*-interface.

A. General Setup

An 2D-axisymmetric test case, to match the experimental setup, is chosen according to figure 1. The simulation is structured within three studies as follows

- Study 1, Acoustics - Calculates the acoustic pressure field. The governing equation contains an additional dipole source term for attenuation;
- Study 2, PDE - A *Coefficient Form PDE* grants access to higher order derivatives from study 1;

---

[1] Institute of Thermal and Fluid Engineering
University of Applied Sciences Northwestern Switzerland
e-mail: donato.rubinetti@fhnw.ch, daniel.weiss@fhnw.ch

[2] Institute of Product and Production Engineering
University of Applied Sciences Northwestern Switzerland
e-mail: jonas.mueller@fhnw.ch, arne.wahlen@fhnw.ch

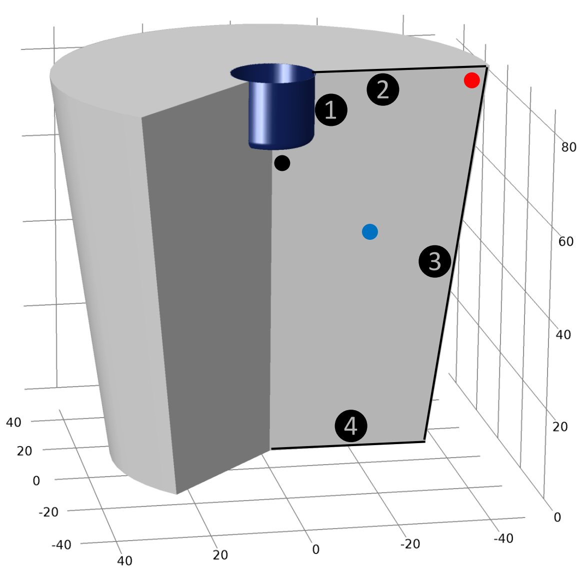

Fig. 1: Revolved representation of the sample geometry. The colored dots indicate the location of tracing massless particles. Boundaries: 1 sonotrode, 2 open interface, 3 + 4 fluid-wall interface. Units in mm.

Table I: Boundary conditions for the acoustics and fluid flow (CFD) parts of the simulation.

Boundary	Part	Acoustic BC	CFD BC
1	sonotrode	acceleration	no slip wall
2	free surface	sound soft	free slip wall
3	wall	sound hard	no slip wall
4	wall	sound hard	no slip wall

- Study 3, CFD - The force term according to (2) is implemented via volume force inducing a stationary flow.

B. Boundary Conditions

Table I lists the boundary conditions used in the simulation. Study 2 is an intermediate step to store higher order derivatives with one simplified partial differential equation. As it has no physical meaning no boundary conditions are needed.

The sonotrode acceleration a_z in axial direction is given by equation (3) [3]

$$a_z = 4\pi^2 f^2 A \tag{3}$$

with f being the frequency and A the amplitude - typically in orders of $10 - 50\mu$m. The sound soft BC physically imposes zero impedance, while sound hard BC imposes infinite impedance. The CFD boundary conditions are divided into free slip walls for the open boundary and no slip wall, where the flow velocity is constrained to zero.

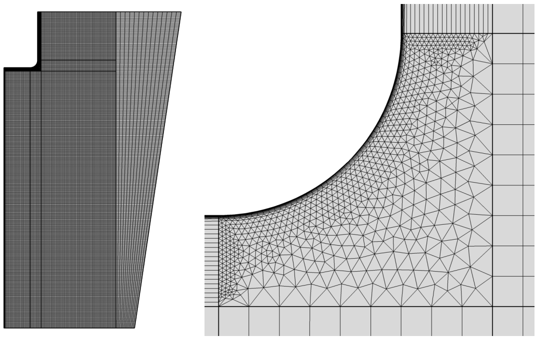

Fig. 2: Computational mesh consisting of hexa- and tetra-elements including boundary layers.

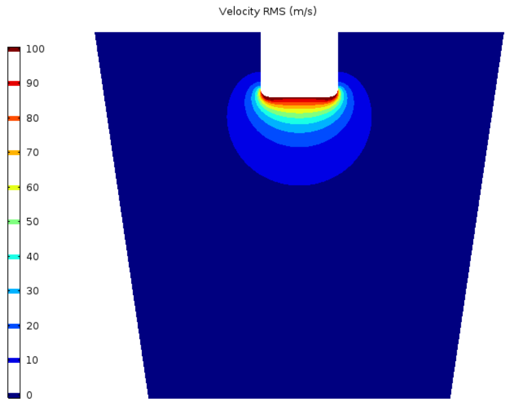

Fig. 3: Resulting acoustic velocity RMS field

C. Mesh

All studies are carried out with the same mesh as illustrated in figure 2. Due to remarkable gradients of the source terms in the sonotrode region boundary layers are required. The grid is predominantly mapped to minimize numerical diffusion.

D. Results

1) Acoustics: The frequency domain results of the acoustics are shown in figure 3. The sonotrode acceleration leads to a sharp acoustic particle velocity rise. The force term (2) is based on the acoustic velocity field. In a separate test-case the functionality of the attenuation term defined as dipole source q_d

$$q_d = \frac{4}{3}\frac{i\omega\mu}{a^2\rho_0}\nabla p \tag{4}$$

within the governing equation (1) has been analytically verified with Stokes' law of sound attenuation [4].

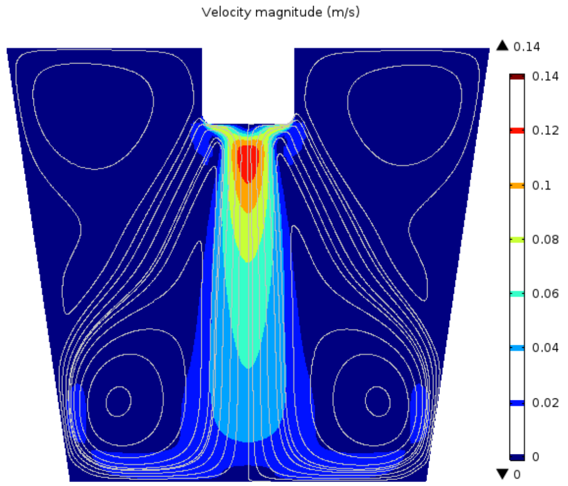

Fig. 4: Resulting stationary velocity field of the study 3.

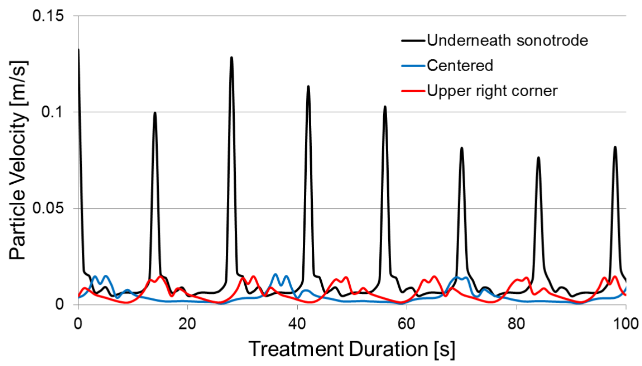

Fig. 5: Velocities of the massless particles located as shown in figure 1.

2) CFD: Figure 4 shows the resulting stationary velocity field for an aluminium melt at an amplitude of 30μm and frequency of 20kHz. The flow pattern is driven by an axial jet emanating from the tip of the actuating sonotrode. As this jet is deflected from the bottom wall it creates a vortex in the region of lower corners. Underneath the sonotrode the momentum reaches its maximum, whereas in the open interface zone the flow is close to standstill. Figure 5 compares the velocities of three tracing particles initially dispersed. Particles located under the sonotrode experience a remarkable acceleration, increasing the amount of cycles.

IV. EXPERIMENTAL VALIDATION

A. *Experimental setup and procedure*

The experimental setup is shown in figure 6. The aluminium sonotrode is dipped into the fluid-filled crucible. The experiments are carried out with a frequency of 20kHz and with an amplitude of $10, 20, 30\mu$m. Water, seed oil and glycerin are used as test fluids.

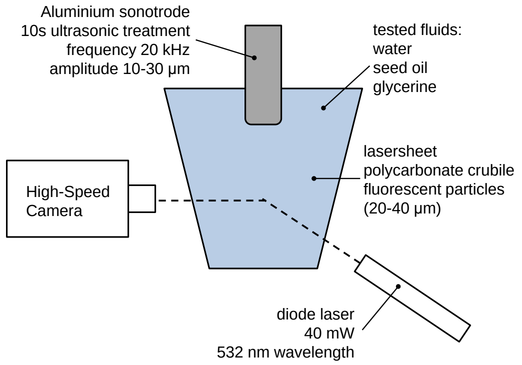

Fig. 6: Experimental setup

A high-speed camera in combination with a diode laser and lasersheet allows the tracking of the fluorescent particles. By *Particle Image Velocimetry* the corresponding velocity field is derived. The ultrasonic treatment is limited to 10 seconds for the presented setup.

B. *Results comparison*

Figure 7 compares the velocity fields of the seed oil test case. It is obvious that the jet occurs in simulation and experimental results. Even though the experimental data does not appear completely symmetric, on the right-hand side of the crucible the streaming pattern is recognizable. In this area, the direction and location of the induced flow matches the predicted behavior by the simulation. Figure 8 compares the velocities along the rotational axis. Experiment and simulation are in good agreement in proximity of the sonotrode tip. The simulation reaches its peak velocity within 10mm from the sonotrode tip which is more than twice the experiments maximum velocity. However, with increasing axial distance the gap between simulation and experiment results decreases showing the same characteristic decline behavior. The deviation can be explained primarily by the lack of accuracy of the optical measurements and by simplifications taken in the simulation.

V. CONCLUSION

The presented model is numerically stable and appropiate for testing parameter variations, geometry modifications and material adjustements. Simplifications and stabilizations methods allow the coupling of acoustics and fluid dynamics, which operate on timescales orders of magnitude from each other. Due to highly transient behavior and short treatment time experimental data proves to be delicate to obtain. To achieve stationary conditions an extended ultrasonic treatment such as industrial application is beneficial. A validated numerical model to predict behavior of the flow characteristics for different fluids and for parameter variations turns out to be a reasonable approach.

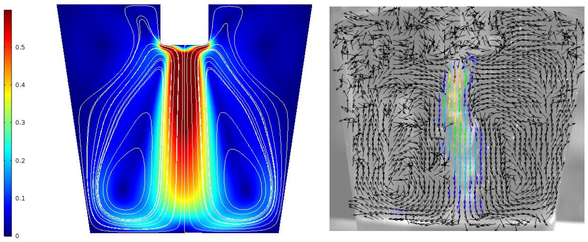

Fig. 7: Comparison of the velocity fields (m/s) of the experiment and simulation for seed oil actuated at an amplitude of 30μm and frequency of 20kHz.

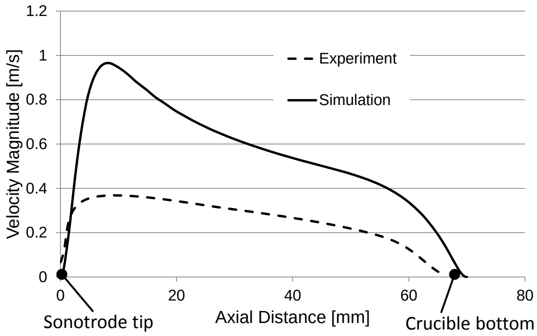

Fig. 8: Comparison of the velocity magnitude along the rotational axis.

REFERENCES

[1] Muller, P. et al. (2012): *A numerical study of microparticle acoustophoresis driven by acoustic radiation forces and streaming-induced drag-forces*. Lab Chip, The Royal Society of Chemistry, Vol. 12, p. 4617-4627.

[2] COMSOL (2016): *Acoustic Streaming in a Micro-tunnel Cross Section*. https://www.comsol.com/model/acoustic-streaming-in-a-microchannel-cross-section-17087 . Accessed on September 7, 2016.

[3] Gallego, J. et al. (2015): *Power Ultrasonics, Applications of High Intensity Ultrasound*. Woodhead Publishing, Cambridge.

[4] Rossing, T.D. (2014): *Springer handbook of acoustics*. 2nd ed. Springer, Berlin.