PIV quantification of the flow induced by an ultrasonic horn and numerical modeling of the flow and related processing times

M.C. Schenker\textsuperscript{a},*, M.J.B.M. Pourquié\textsuperscript{a}, D.G. Eskin\textsuperscript{b}, B.J. Boersma\textsuperscript{a}

\textsuperscript{a} Delft University of Technology, Laboratory for Aero & Hydrodynamics, Leeghwaterstraat 21, 2628 CD, Delft, The Netherlands
\textsuperscript{b} TA208, Brunel University, UB8 3PH Uxbridge, United Kingdom

\begin{abstract}
The flow in a confined container induced by an ultrasonic horn is measured by Particle Image Velocimetry (PIV). This flow is caused by acoustic streaming and highly influenced by the presence of cavitation. The jet-like experimentally observed flow is compared with the available theoretical solution for a turbulent free round jet. The similarity between both flows enables a simplified numerical model to be made, whilst the phenomenon is very difficult to simulate otherwise. The numerical model requires only two parameters, i.e. the flow momentum and turbulent kinetic energy at the position of the horn tip. The simulated flow is used as a basis for the calculation of the time required for the entire liquid volume to pass through the active cavitation region.
\end{abstract}

1. Introduction

High power acoustic waves can affect the structure of cast metal. This effect is known since the early 1930s (see summary in [1–3]). Due to the acoustic processing of liquid and semi-liquid metals the grain structure of a casting becomes finer and more isotropic, leading to often desired changes in the material properties such as a higher yield strength and ductility. Implementations of this technique in metallurgy on an industrial scale are, however, still limited due to the lack of fundamental understanding of the underlying phenomena and scale-up rules. Besides metal casting, acoustic treatment is also applied in other technological areas, such as the chemical [4,5] and environmental [6–9] industries.

The effective influence of high power acoustic waves on the solidification of liquid metal is in general attributed to the occurrence of mechanically violent cavitation bubbles, which activate and multiply the crystallization nuclei in the melt [1]. Besides cavitation, a second process occurring under the influence of acoustic waves is the so-called acoustic streaming. Acoustic streaming acts as a stirring mechanism, causing an increased part of the liquid volume to be treated by the local cavitation zone. The acoustic streaming is affected by the presence of cavitation.

For the implementation of ultrasonic treatment in the metal industry it is of major importance to be able to estimate the treatment time of arbitrary volumes a priori. Up to now treatment times have often been determined experimentally for certain specific geometries, configurations and materials, but to our knowledge no general applicable model for the prediction of the treatment time has been developed yet. This is the topic we want to address in the present paper.

To be able to predict the treatment time of a volume it is first required to know the flow in the entire volume. In the past, research on the induced flow patterns under cavitating conditions has been reported by a limited number of experimental [10–13] and numerical [11,14] studies. Of this last category only the recent paper by Trujillo et al. [14] aimed to predict the induced flow in an arbitrary volume. Some numerical studies make use of many boundary conditions, which have to be extracted from experiments for each specific case, limiting the general applicability. Others include the computation of the acoustic field and the action of this field on the fluid and vice versa, and are based on many assumptions and simplifications regarding the acoustic field and its boundary conditions. Also the computation of the full acoustic field requires a large computational effort and is difficult to simplify for the use in arbitrary geometries.

Acoustic streaming below the cavitation threshold, but at a sufficient driving power, applied through an acoustic point source is known to form a flow pattern equal to that of a turbulent free jet [15]. Here the minimum driving power, \( P_{\text{min}} \), is related to the minimum squared Reynolds number as: \( \rho c^{-1} P_{\text{min}} \approx Re^2 > 4 \times 10^3 \).
with \( \rho \) the density, \( c \) the speed of sound and \( \mu \) the dynamic viscosity of the liquid.

Although the situation studied in this paper is different, it is still interesting to compare the measured velocity field with the theoretical velocity profile for a turbulent free jet. When the measured jet can be described by equations similar to those of a turbulent free jet, simplified numerical approaches might be used to reconstruct the flow.

In general the cavitation zone is a very small region with respect to the total volume. In the current research only the flow induced by the cavitation in the liquid is investigated. From the experimental results the momentum induced by the cavitation zone is calculated, and this is used as input for our numerical computations of the treatment time. It is assumed that the size and location of the cavitation zone are constant, and assumed to be known and of little influence to the nature of the problem. Furthermore it is adopted that the treatment effectivity of the cavitation zone is perfect, that is, once a fluid volume passes through the cavitation zone it is taken to be treated. This assumption can easily be expanded towards partial treatment on passing the cavitation zone.

To give an overview of the paper: In section two we describe the experimental setup. Subsequently we analyze the results in section three and extract the appropriate momentum source from the measurements. In this section this momentum source is used to estimate treatment times. Conclusions and recommendations are presented in Section 4.

2. Experimental setup and measurement techniques

Although this research can be helpful for ultrasonic processing in liquids in general, the real application of our interest is the processing of molten metals and in particular liquid aluminum. In an experimental environment, apart from problems with the high temperatures, the opaque molten metal is not accessible with optical techniques for fluid flow measurements. Therefore, in the current study, we used water instead of liquid metal. Many liquid properties relevant to this type of flow are of comparable magnitude for water and aluminum, and the same physical mechanisms are taking place. For example the kinematic viscosity \( \nu \) of water at room temperature and liquid aluminum are of comparable magnitude \( (\nu_{\text{H}_2\text{O}} \approx 0.5 \nu_{\text{Al}}) [16,17] \), and the minimum squared Reynolds number as defined in Section 1 scales with a factor \( \approx 1 \). Water is often used as a modeling material for liquid metals (i.e. [18–21]), because of practical reasons, similarity of liquid properties and, very important, because liquid metals behave as a Newtonian fluid, behavior also displayed by water at room temperature.

Multiple ways to apply an acoustic field to a liquid exist. In this investigation we used an ultrasonic horn (sonotrode) slightly penetrating through the liquid surface. The strong acoustic waves generated by the horn resulted in the formation of cavitation bubbles in the liquid. They also produced a mean flow through the mechanism of acoustic streaming. The fluid motion was measured using Particle Image Velocimetry (PIV). The investigation focused on the jet-flow region and not on the recirculating flow induced by the jet. Therefore we could assume that the flow was axi-symmetric although a rectangular flow container was used. A two dimensional PIV experiment was performed, rather than measuring the full 3D flow pattern using more sophisticated but complicated and expensive PIV methods.

Applying PIV in a two-phase flow (liquid and cavitation bubbles) with a locally very dense concentration of the second phase leads to specific problems. The bubbles could not be observed as separate objects due to their dense distribution and small size, and caused most of the seeding particles to be invisible within this region. In less dense bubble regions, the bubbles might be mistaken for seeding particles, disturbing the measured displacement field. The dense bubble zone was very small in comparison with the macroscopic flow patterns of the jet. Therefore the cavitation region was excluded from the PIV analysis. A combination of the use of fluorescent seeding particles and an ensemble correlation algorithm for the obtained images was used to make a distinction between seeding particles and isolated bubbles.

All measurements were performed in a rectangular volume of water (193 \( \times \) 293 \( \times \) 180 mm-sized container, 8 l fluid). A Ti conical sonotrode with a tip diameter of 25 mm connected to a magnetostrictive transducer, \(^1\) driven by an ultrasonic generator \(^2\) running at an electrical power of 4 \( \pm \) 0.15 kW and a frequency of 17.6 kHz was used. These settings corresponded to a vibrational amplitude of 35–40 \( \mu \)m at the horn tip. The input power was estimated to be 90–100 W/cm\(^2\) [1]. The sonotrode penetrated 40 mm into the water, along the centerline of the volume. A sketch of the setup is presented in Fig. 1.

---

Footnotes:
\(^1\) Magnetostrictive Transducer, MSC-5-18 with a Ti booster, Reltec.
\(^2\) Ultrasound Generator USG-5-22, Reltec.
An important parameter in cavitation processes is the vapor pressure which is strongly related to the temperature. To obtain results which correspond to real cavitation dynamics, without interfering with effects related to boiling, the liquid temperature should remain far below the boiling point. During the measurements the water temperature was monitored and maintained below 24 °C.

The PIV experiments were executed using a pulsed laser with a flash time interval of 0.4 ms. The laser beam was transformed to a flat light sheet using a cylindrical lens. The images were recorded using a double shutter camera aligned orthogonal to the laser sheet. The liquid was seeded with fluorescent particles with a diameter of 13 μm, absorbing the wavelength of the laser (532 nm) and emitting orange light (~640 nm). The camera was equipped with a corresponding orange optical filter to remove all undesired reflections from the images.

The obtained images were filtered using a median filter and the image contrast was normalized over the whole image to minimize the effect of intensity differences over the area of the light sheet. We performed the PIV processing with a PIV algorithm written in MatLab. Interrogation windows of 32 x 32 pixels with an overlap of 50% were used. Ensemble correlation, also called correlation averaging [22,23] was applied to average over a series of images to obtain the time-averaged velocity field, while minimizing the influence of isolated bubbles on the obtained velocity field.

3. Results and discussion

3.1. Experimental results and analytical solution

Fig. 2 demonstrates the time averaged velocity-field obtained from the measurements. The jet region can be clearly observed and also the horizontal flow due to the impingement of the jet on the volume floor is clearly visible. Due to optical limitations we cannot show the entire volume; the flow extends further in horizontal direction beyond the field of view. For clarity the color scale has been truncated. It should be noted that the velocity-magnitudes extend beyond this scale. On the images a small empty or zero-velocity region near the jet origin (0,0) is present. This region represents the position of the sonotrode and the intense cavitation region surrounding it. No valid velocity-vectors could be obtained here due to the dense bubble density. Also the region near the bottom side of the image (z ≈ 104) is empty. This region corresponds to the bottom of the aquarium where no flow is present. Both empty regions were excluded from the following analysis.

When observing the obtained flow pattern, the resemblance to that of a turbulent free jet is indeed striking, and it was a logical continuation to compare the measured velocity field with that of such a jet. The analytical solution for the axial velocity profile of a turbulent free jet is known from literature, see for instance [24] or more recently [25]. In the derivation of these analytical solutions some well accepted approximations have been used. The main assumptions leading to an analytical solution for such a jet are that the streamwise evolution is much smaller than that in the radial direction and that derivatives in streamwise direction can be neglected with respect to those in radial direction. Also the Prandtl–Boussinesq hypothesis has been applied which introduces a new parameter, the turbulent viscosity ($\nu_t$), and relates this to the mean velocity field, assuming no dependence on the radial position within the jet. Comparisons with earlier experiments have shown this to be a justified assumption [24]. As a last condition it is assumed that the axial momentum is conserved within the jet.

These assumptions allow a solution in the form of a self-similar velocity profile, where the axial velocity ($u_z$) is only a function of the non-dimensional radial position ($\eta$) and the local centerline velocity (see Eq. 1). The jet width (here defined as $\delta = r$ for which $u(r) = u_0(z)/2$) and the centerline velocity evolution $u_0(z)$ along the axial direction follow from the obtained profile (see Eq. 2 and Fig. 3). The constant $a$ is related to the turbulent viscosity, $\nu_t$, and the spreading angle, $\alpha$ (see Eq. 3).

$$u_z(r, z) = u_0(z)f(\eta) = \frac{u_0(z)}{(1 + a\eta^2)^{\frac{1}{2}}}$$

(1)
with
\[ \eta = \frac{r}{\delta(z)}, \]
\[ a = \frac{2u_0(z)\delta(z)}{8v_t} = \frac{a}{8b}, \]
and
\[ \frac{\partial \delta}{\partial z} = \alpha = \text{constant}. \]

The turbulent viscosity \( v_t \) and the spreading angle of the jet are coupled through the turbulent kinetic energy (see for further details [25]). Once the spreading angle and axial momentum of the jet are determined from the experiment, the jet can be modeled using the derived formulas, without the use of additional parameters.

### 3.2. Analysis in terms of the self-similar solution of the turbulent free jet

As mentioned before (see Section 1) the main purpose of the present experiments was to develop a model to predict the treatment time of an arbitrary volume. To achieve this it was required to know the axial flow momentum of the jet. This was obtained using a fit of the analytical solution presented above (Eq. 1) to the measured data. The fitting procedure used the measured centreline velocity \( u_0 \) and the local jet diameter \( \delta \). The turbulent viscosity \( v_t \) was chosen such that the calculated profiles coincide optimally to the measured flow profiles at five axial distances and thus contain kinematic axial momentum similar to that of the measured flow. In Eq. 1 this resulted in \( \alpha = 0.12 \) and \( b = 0.034 \) (see Eqs. 3 and 4). In Figs. 4 and 5 we show the axial velocity profile at the positions corresponding to the two thin dashed lines in Fig. 2. The dashed lines represent the measured time-averaged velocity, while the thick solid lines display the fitted self-similar solution. The thin solid lines in Figs. 4 and 5 correspond to the numerical solution which will be discussed in Section 3.3.

The region were the free jet solution is valid can be visualized by displaying the centreline velocity times the jet-width as a func-

---

**Fig. 3.** The jet-geometry with a potential core and a fully developed region. The definitions for the parameter \( \delta \) and \( u_0 \) are shown.

**Fig. 4.** The axial flow velocity at \( z = 27 \) mm (see also Figs. 2 and 6). The measured velocity, the self-similarity solution and the numerical solution are shown.

**Fig. 5.** The axial flow velocity at \( z = 63 \) mm (see also Figs. 2 and 6). The measured velocity, the self-similarity solution and the numerical solution are shown.

**Fig. 6.** The centerline-velocity multiplied with the jet width \( \delta \), as a function of axial position \( z \). The vertical dashed lines correspond to the location were the velocity profiles of Figs. 4 and 5 are taken. See also Fig. 2.
tion of axial distance. For the free jet solution to be fully applicable this value should be a constant over the whole jet length. Fig. 6 di-

... displays this value, and indeed, in a region from approximately z = 50 to 80 mm $u_0 \times \delta \approx \text{constant}$. However, the range in which the self-

... similar profile scaled with the measured velocity fits the measured velocity profile extends beyond this region. This is the case in the region from approximately z = 30 to 90 mm, i.e. the conservation of axial momentum is not fulfilled in all flow regions which is probably a result of the confined space which influenced the jet behavior. The theoretical jet is defined in an infinite medium and the recirculating motion in the outer regions of the container influences the liquid entrainment by the jet.

The obtained jet-similarity supports an important assumption made in the beginning of the experiments; it indicates that the jet can be assumed to be axi-symmetric. This means that the obtained measurements can be translated to a three dimensional flow pattern resembling ordinary free turbulent jets, which have been investigated intensively throughout the years (for instance by [26]).

In well designed experiments a self-similar velocity profile for turbulent jets comes into existence after approximately 5–10 nozzle diameters [27]. In the current situation this would correspond to the horn tip diameter (25 mm). The first part of the jet, the region of flow establishment, is defined as the part of the flow where the potential core exists (see Fig. 3). The potential core is the central region of the jet which is not yet affected by entrainment of the surrounding fluid and does not show the self-similar velocity profile, but rather the velocity profile as it was present at the inflow.

This region of the flow narrows with increasing distance from the source as a result of mixing, however, for flows with a high turbulent intensity the profile is known to develop much faster [28]. This is due to an increased level of turbulent mixing. The fact that for the current measurements the self-similar profile develops much closer to the source (at approximately 10 mm from the sonotrode the profile is shaped according to the self-similar jet-profile) is therefore attributed to a high entrance turbulent intensity. The source for this high level of turbulence is most probably the cavitation zone: The violent behavior of the cavitation bubbles introduces additional turbulent kinetic energy to the jet.

### 3.3. Numerical modeling

One of the assumptions of the present study was that the jet can be modeled based on a single momentum source, rather than a fully resolved and interacting acoustic field as the driving force for the flow. The proper value for the required jet momentum was extracted from the analytical fit to the measured flow field. Subsequently, the flow was reconstructed using the computational fluid dynamics package Fluent. It was previously assumed that the jet-flow is axi-symmetric. We could thus model a cylindrical volume rather than a rectangular volume such as used for the experiments. For the diameter of the computational volume the shortest side of the rectangular volume, 193 mm, was chosen. A detailed description of the numerical parameters can be found in A.

The axial momentum source was chosen such that the axial momentum of the numerically obtained jet at a certain distance of the source coincided with that of the measured jet. Although theoretically the axial jet-momentum should not vary with increasing distance from the source, both the measured and the modeled jet (based on the measured centerline velocity) showed a slight decrease in axial direction, which was probably an effect of the limited container size. Because of this effect it was important to determine the momentum at the same axial position for both the model and the experimental results. The momentum of the numerical solution was computed using the definition for axial momentum ($M = \int_0^\infty p u^2 2\pi dr$) with the radial range truncated at the same position as the limits of the experimental field of view to avoid influence of the recirculating flow.

The Reynolds numbers reflecting the current experimental conditions ($Re = u_0 d / \nu \approx 3000$, based on the local jet-diameter and centerline velocity) fall within the range of Reynolds numbers in which a standard round jet would display transitional behavior. This means that the first part of the jet (several nozzle diameters length) should behave more or less as a laminar jet, with a corresponding small spreading angle [29]. The clearly turbulent behavior of the measured jet over the whole jet length is thus related to the way this jet is generated. This effect was probably due to the augmented mixing related to the increased turbulence intensity, which was caused by the presence of cavitation.

To mimic this effect of cavitation in the numerical model an extra turbulent kinetic energy source was added, defined in the same region as the momentum source. There was no physically based derivation available linking the acoustic intensity to the increase of turbulent kinetic energy since too many unknown parameters are related to the cavitation intensity or the corresponding turbulent intensity. In this study, the amount of turbulent kinetic energy was chosen such that the jet half width of the numerical solution resembles that of the measurements.

The obtained axial velocity profile is displayed in the thin solid lines in Figs. 4 and 5. Obviously the inner parts of the jet correspond very well with both the experimental and the analytical velocity profiles. Further away from the centreline the numerical solution resembles the measured profile better than the analytical one. This could be expected because the derivation of the theoretical analytical velocity profile is based on an infinite medium while both the measurement and the numerical solution were obtained in a confined space.

The recently published paper by Trujillo et al. [14] uses a similar approach to model the jet induced by an ultrasonic device. Their approach is similar, using the observations of Lighthill [15] that an ultrasonic horn at a sufficient driving power leads to a turbulent jet, which enables ways of simplified numerical modeling of the flow. However, their approach has some distinct differences with that described below. Besides the fact that Trujillo et al. [14] aimed at applications within the food industry, and therefore focused on heat generation and distribution within the volume, the main difference between the approaches lies in the manner the jet has been generated.

Trujillo et al. [14] used an inlet velocity at the location of the horn tip, where the inlet profile was either that of an analytical Gaussian jet, or a analytical turbulent circular jet, both satisfying momentum conservation. They justified this choice of the inlet profile on [15] who had derived such a profile to be valid very close to the origin of sound, provided that the attenuation coefficient is high. This profile has been derived for an acoustic source located at a point, rather than on a surface [15]. Since the whole tip-surface of the horn oscillates, the use of a virtual point inflow within the horn does not seem justified and does not reflect the physical behavior close to the origin of the jet.

Although it is not possible to observe the degree of self-similarity for the computed jet from the results shown in Trujillo et al. [14], it is clear that their axial velocity profile deviates from the measurements [30], especially close to the jet origin. Overall the match between simulation and experiment seems to be rather poor, due to an optimization of the fitting parameter ($S$ or $x$ for the Gaussian and analytical turbulent jet respectively) with respect to the axial over the jet length, including the initial jet region.

Using an inflow in practice also requires an outflow to be located in the computational domain. This outflow was in the...
The second significant difference between the method by Trujillo et al. [14] and that presented in this paper is the extra degree of freedom, resulting in an extra modeling variable for the latter. Trujillo et al. [14] used the power density as reported by Kumar et al. [30], who used the calorimetric method to measure the power absorbed in the system, but no such input is used for the present model. Further the liquid density, \( \rho \), and velocity of sound, \( c \), were included in the computation for the kinematic momentum and with that, the turbulent viscosity. The calorimetric method is based on the assumption that all significant acoustic energy is eventually transformed to heat and thus the temperature increase of the liquid is a measure for the absorbed power. The drawback of this method is that the acoustic energy transmitted to the liquid will be measured with the assumption that the cavitation is the only source of the incoming heat and that there are no heat sinks. When the acoustic signal is applied by using a horn partially submerged into the liquid the horn itself (especially if attached to a cooled transducer) influences the temperature of the bath, being both the heat source and sink. Trujillo et al. [14] do not report on the temperature of the horn, but in our investigation the temperature balance of the horn contributed significantly to the temperature of the liquid. For both the density and velocity of sound Trujillo et al. [14] used the values for pure water for the determination of the kinematic momentum, while the acoustic wave was largely absorbed in the cavitating region of the volume where these values loose validity. Altogether the derivation of kinematic momentum through the calorimetric measurements does not provide a reliable input parameter for the simulation.

The third point to note is the low spatial resolution of the measurements [30] used by Trujillo et al. [14] for validation of their model. Indeed Laser Doppler Anemometry (LDA) measurements do provide turbulent statistical quantities which cannot be derived using Particle Image Velocimetry, but the small amount of measurement locations along the width of the jet do not facilitate a good comparison between the LDA-experiment and the simulation in terms of jet radial profile and spreading angle.

### 3.4. Estimation of the treatment time

The numerically calculated velocity field was used as a basis to determine the treatment time for the entire volume. Since, apart from standard wall and free-surface boundary condition, the jet is the only specific model-input, the whole flow field can be computed based on the momentum and turbulent kinetic energy input near the sonotrode. Ideal numerical tracers were released in the flow and their position as a function of time was monitored. Once a tracer passed through the cavitation zone it was regarded as “treated”. No additional experiments have been performed to define the size and shape of the cavitation zone as was mentioned in Section 3.3. Therefore, approximations for the cavitation zone location and volume were made based on the available experimental observations (see Fig. 2). The obtained images showed bubbles in a region until approximately 5–10 mm distance from the horn tip, over the entire radius of the horn tip surface. Therefore a cylindrical volume, with a diameter equal to that of the sonotrode tip and a depth of 10 mm was chosen to represent the cavitation zone. Note that this treatment region is significantly larger than the volume where the momentum and kinetic energy source is located (see appendix A). The zone where treatment occurs and the streamlines of the time-averaged flow are displayed in Fig. 7.

In reality the turbulent motions would cause the particles to deviate from the streamlines of the calculated time averaged flow. This effect was included in the particle tracking mode. The turbulent kinetic energy is a measure for the local velocity of the fluid with respect to the average flow. Therefore the turbulent kinetic energy present in the CFD solution was used to give the tracer particles a deviation. This deviation is a stochastic property of the turbulent kinetic energy and thus multiple realizations for each initial position had to be made.

In the numerical solution, at each of five different positions 40 particles were injected (see B). The five injection points were located at 80 mm from the centerline and 10, 30, 50, 70 and 90 mm from the domain bottom. Each tracer followed a different path through the volume because of the fluctuations depending on the turbulent intensity. Every particle was tracked over 150,000 time steps, which was assumed to be sufficient for them to have passed through the cavitation zone. The CFD-package adapted the time-step size according to the local level of turbulent kinetic energy and velocity gradients. Details on the numerical implementation can be found in B.

The time it took for a particle to reach the cavitation area for the first time was monitored for each tracer. Of course, a particle may pass through the zone multiple times, but only the first pass is of interest for the current model. Fig. 8 shows the percentage of particles that traveled through the cavitation zone at least once as a function of time.
function of time elapsed since their injection. Of this total amount of particles (5 × 40), only two did not cross the cavitation zone within the given number of time steps. A high degree of spreading through the full domain was observed, from which it is concluded that the initial position of the particles is of little influence for the obtained traveling time.

To translate the pathline information as generated by Fluent to traveling times the data was imported to Matlab. This poses a limit on the number of particles that could be introduced. Without these limitations, a larger amount of particles could have been introduced to obtain better statistics and also the maximum number of steps could have been increased to some extent. Nevertheless, the performed simulation allowed us to reach a practically important conclusion that approximately 98% of a liquid volume of 4138 cm³ (~4 l) can be treated by cavitation in about 500 s, with about 70% being treated already in the first 130 s.

4. Conclusions and recommendations

The velocity field obtained experimentally under the ultrasonic sonotrode operating in the cavitation regime shows a jet motion similar to a turbulent free jet as would be the case for acoustic streaming without the presence of cavitation based on an acoustic point source. Although no generic input parameters have been derived for a specific sonotrode, it is shown that only two parameters, namely the input momentum and turbulent kinetic energy, have to be defined to model the treatment time of any arbitrary volume. For the specific volume used in this model (~4 l), it can be concluded that the fluid will be fully “treated” in about 500 s by the cavitation and acoustic streaming induced by the ultrasonic horn working at a frequency of 17.6 kHz and an vibrational amplitude of 35–40 μm.

Despite the usefulness of the PIV experiments, the obtained data are far from complete. Similar measurements at different powers and frequencies of the ultrasonic horn would increase the insight in the relation between the jet momentum and shape, and the applied acoustic wave. For the reconstruction of the jet velocity profile, besides input momentum directly related to the acoustic amplitude, also the turbulent kinetic energy as created in the cavitation zone is of importance. The current measurements do not provide information on the cavitation intensity within the zone as would be required for a full parametric model of the flow.

As mentioned in Section 2 the velocity field is obtained using ensemble correlation. This means that no information on the fluctuation intensity, or Root-Mean-Square velocities are available to be used in the post processing. Obtaining images of a higher quality would make the computation of vector averaged velocity fields possible, and thus provide information on the RMS velocities in a larger part of the flow. The RMS can then be compared with the numerical turbulent intensity and can be used to improve the similarity between the numerical and measured results. It would also be useful to derive a clear relation between the applied acoustic signal, the fluid properties and the cavitation induced turbulent intensity. For the computation of more realistic treatment times it would be interesting to include a full 3D flow volume as well, because the corners in a non-axi-symmetric volume might have a significant influence on the generation of stable recirculation areas.

For the same device and applied acoustic power, liquid aluminum will behave similarly, but the results for this water model cannot directly be translated to other liquids. However, once the induced momentum and additional cavitation-induced turbulent kinetic energy can be parametrized and predicted, the model can be used for arbitrary liquids, provided that the minimum squared Reynolds number and the cavitation threshold are reached. For applications in casting models, where thermal effects and solidification of the liquid metal are important the model can be easily extended. The obtained treatment times might be sensitive to the parameters in the numerical model, such as discretization schemes and the particle tracking mode adapted. This was beyond the scope of this research.

Appendix A. Numerical parameters: modeling the flow

The velocity field obtained with the measurements was reconstructed using a numerical simulation. This appendix gives a summary of all the numerical parameters involved. The computational grid was created using Gambit. Because the flow was assumed to be axi-symmetric only one half plane was modeled.

The model was 142 × 96.5 mm with a cut out at the upper left hand corner of 6.25 × 40 mm, representing the horn inserted in the liquid. The grid had 285 × 150 cells. A separate zone directly near the sonotrode tip was modeled as a separate region and was used to apply a momentum source. We used a Reynolds stress model to take the turbulent behavior of the flow into account. For the model parameters all default values as present in Fluent were adopted. All flow variables were discretized using a 2nd order upwind discretization. The pressure was treated with the discretization “standard” in Fluent.

The floor and side of the volume were modeled as walls with a no-slip boundary condition. The liquid surface was modeled as a wall without shear stress. The sonotrode was modeled as a static object and therefore also implemented as walls with a no-slip condition. As discussed in Section 3.3 the flow is driven by a combination of a momentum source and a turbulent kinetic energy source. Both were defined in a region of 2 × 6.25 mm located underneath the tip of the sonotrode, in 3D thus corresponding to a volume of 2.45 × 10⁻³ m³. The applied axial momentum was 300 000 N/m³ and the turbulent kinetic energy 2000 W/m³. The computation was performed with a time-independent solver.

Appendix B. Numerical parameters: modeling the tracer particles

To determine path lines of “fluid particles” tracer particles were inserted in the flow. To ensure that the tracers followed the flow like ideal tracers they had been given a diameter of only 0.1 μm. Turbulent dispersion of the particles was done with stochastic tracking by means of a discrete random walk model and a time constant of 0.15, which was used in the description of the integral time scale; the time a particle spends in a turbulent motion. Details on the discrete random walk model can be found in the Fluent Manual [31].

Although the model was axi-symmetric the particle motion was exerted by all components of the turbulent kinetic energy, thus deviating from the streamlines in azimuthal direction as well. There was thus no need to include a full 3D model as long as the mean flow could be described by axi-symmetry. The chosen number of time steps (150,000) specified the number of steps for which the particle was tracked. However these time steps did not have a constant length and the total traveling time therefore differed per particle.

References


Gambit, version 2.4.6.


