European Journal of Mechanics B/Fluids 61 (2017) 271–278

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

journal homepage: www.elsevier.com/locate/ejmflu

Prediction of primary atomization using Smoothed Particle Hydrodynamics

Rainer Koch *, Samuel Braun, Lars Wieth, Geoffroy Chaussonnet, Thilo Dauch, Hans-Jörg Bauer

Institut für Thermische Strömungsmaschinen, Karlsruhe Institute of Technology (KIT), Karlsruhe, Germany

ARTICLE INFO

Article history:
Available online 18 October 2016

Keywords:
Fuel injection
Atomization
Numerical prediction
Smoothed Particle Hydrodynamics

ISROMAC 2016

International Symposium on Transport Phenomena and Dynamics of Rotating Machinery

Hawaii, Honolulu

April 10-15, 2016

ABSTRACT

In modern jet engines mostly air blast atomizers are used for the liquid fuel injection. The prediction of the spray generated by such atomizers was for a long time not feasible because of restricted computing resources. However, with modern super-computers the prediction of the atomization has come into reach.

In the present paper a new approach for the numerical prediction of the primary atomization is presented. The methodology is based on the Smoothed Particle Hydrodynamics (SPH), which has originally been developed in the context of astrophysics.

The numerical predictions to be presented were performed for a planar model atomizer, for which a vast amount of experimental data was collected by us previously. The major objectives of the numerical predictions are to elaborate the mechanism governing the effect of thickness of the trailing edge of the prefilmer on the size of the droplets and the temporal droplet formation rate.

0. Introduction

In civil aviation, fuel injection is one of the most crucial processes affecting engine emissions, in particular soot and NOx emissions. First experimental investigations into air blast atomization date back to the 70s [1]. But even today, the basic mechanism governing air blast atomization is not understood, and reliable modeling approaches are not available. Present state of the art is to empirically optimize the fuel injectors. Typically, expensive measurements of the injector performance in terms of droplet size and velocity are performed at elevated pressures. Afterwards correlations like those proposed by e.g. Lefebvre [2,3] are used to extrapolate the experimental results to other operating conditions.

The prediction of the atomization process based on first principles was for a long time not feasible because of restricted computing resources. However, with modern super-computers providing 10.000 or even more CPUs, such predictions have come into reach. But the prediction of the atomization process is still a major challenge, as a wide range of scales in space and time must be captured. The spatial range extends from 1 μm which resembles the typical dimensions of the thickness of the bubbles formed during the disintegration process up to the distance required for finalizing secondary break up of larger droplets or ligaments, which is in the range of approximately 0.1 m. Time scales reach from 1 μs for the typical interaction between the air flow and the liquid at the trailing edge up to the residence time inside the combustor of typically 10 ms.

The Institut für Thermische Strömungsmaschinen (ITS) has started 7 years ago to develop a numerical method for predicting the primary atomization of air blast nozzles, which are typically used in jet engines. The numerical approach is based on the mesh free Smoothed Particle Hydrodynamics (SPH) method [4], which has originally been developed in the context of astrophysics.

The SPH method was chosen because it has some inherent advantages over the readily available and commonly used grid based methods such as the Volume of Fluid (VoF) method.

* Corresponding author.
E-mail address: rainer.koch@kit.edu (R. Koch).

http://dx.doi.org/10.1016/j.euromechflu.2016.10.007

Recent developments of applying the VoF method for predicting atomization are discussed by Herrmann [5].

During primary atomization the liquid surface is exposed to large deformations, and surface tension plays a crucial role for the liquid disintegration. Because of its Lagrangian nature, the Smoothed Particle Hydrodynamics (SPH) method adapts itself naturally to large deformations of the gas liquid interface. Furthermore, the curvature of the interface, which is required for computing the surface tension, can directly be calculated and does not require a cumbersome surface reconstruction technique like in all grid based methods.

In the present paper the application of the SPH method for predicting primary atomization will be presented and discussed. All predictions are reflecting an experiment which was performed at a generic planar prefilming airblast atomizer. The experimental setup and results have been published at several occasions [6–8].

The paper is organized as follows: First the fundamentals of the SPH method will be introduced. Then, a summary of the experimental study of the planar air blast atomizer will be given. The experimental results are predicted using the SPH method. Special emphasis is put on the effect of the thickness of the trailing edge of the prefilmer on the atomization. These predicted results are compared in terms of droplet size distribution, ligament lengths and breakup frequencies to experimental data.

1. Smoothed Particle Hydrodynamics method

The particle based SPH method has been developed in the late 1970's in the context of astrophysics [4]. Recently, it gained popularity in general computational fluid dynamics, where the main scope is on the simulation of free surface flows, e.g. [9]. In contrast to commonly used grid based Eulerian methods, the spatial discretization of a computational domain is performed via so called particles, which represent a small volume of the fluid. Due to the Lagrangian nature of the method, the particles move within the computational domain with the actual fluid velocity. The governing equations describing the flow physics are the Navier Stokes equations.

The main idea behind the SPH formalism is to evaluate the physical property of a particle or its spatial derivative by interpolating over neighbor particles within a certain radius of influence. Eq. (1) reflects the basic interpolation formalism for a particle with index i.

$$\langle \Phi \rangle_i = \sum_j V_j \Phi_j W(\vec{x}_i - \vec{x}_j, h). \tag{1}$$

In Eq. (1), Φ is a physical quantity which may be e.g. the density or the velocity etc. V_j is the volume of an adjacent particle j and $W(\vec{x}_i - \vec{x}_j, h)$ is a weighting function, which depends on the positions $\vec{x}_{i/j}$ and the smoothing length h. The summation is truncated once the distance to the adjacent particle exceeds the predefined limit Δr, which depends on the smoothing length. For a more detailed description of the SPH basics, please refer to e.g. [10]. The interpolation procedure is illustrated in Fig. 1.

Until now, the application of SPH to technical problems was limited by the lack of suitable boundary conditions. Recently, flexible and robust inflow and outflow boundaries have been introduced and been demonstrated to be successfully incorporated into a massively parallel framework [11]. In contrast to our previous work [10], realistic boundary conditions and fluid properties are now available for predictions of technically relevant flow situations.

The modeling of surface tension plays a crucial role when predicting the atomization process. This is due to the fact, that the liquid disintegration is determined by the balance of two forces acting at the liquid–air interface: The surface tension forces and the shear forces induced by the air flow. In our SPH code the surface tension is represented by the Continuum Surface Force (CSF) model, which was originally introduced by Brackbill et al. [12] in the context of the VoF method. The CSF model as adopted in our approach was proposed by Adami et al. [13]. The key feature of the CSF model is the representation of the surface tension force as continuous force acting on the volume adjacent to the interface instead of the surface of the interface between liquid and air.

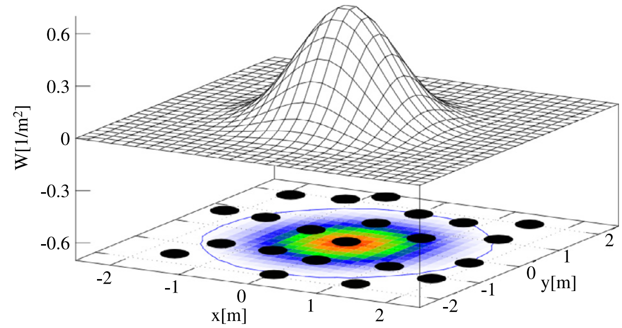

Fig. 1. Principle of the interpolation within the SPH method.

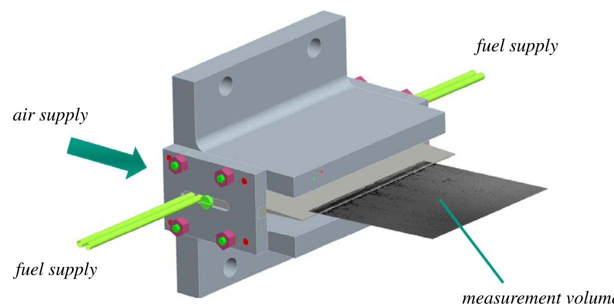

Fig. 2. Generic planar air blast atomizer.

Wetting effects, which may significantly affect primary atomization, are accounted for by the model of Wieth et al. [14]. In order to cope with the high density ratios of up to 1000 depending on the ambient air pressure, the continuity equation as proposed by Hu et al. [15] is applied

$$\langle \rho \rangle_i = m_i \sum_j W(\vec{x}_i - \vec{x}_j, h) \tag{2}$$

where m_i is the mass of the center particle.

2. Experimental study of air blast atomization

2.1. Experimental setup and diagnostics

For studying experimentally the liquid disintegration of air blast atomizers, and also for the validation of predictions, a generic experiment resembling the characteristics of a typical air blast atomizer was set up. A planar atomizer geometry was used instead of the annular design of real fuel injector nozzles [16]. This planar geometry enables better access for optical diagnostics. A sketch of the atomizer is shown in Fig. 2.

On top of the planar prefilmer plate, a kerosene surrogate is supplied. The liquid disintegration takes place downstream at the trailing edge of the prefilmer. In Fig. 3 a top view on the trailing edge of the prefilmer is depicted. The flow direction is from top to bottom. The photograph (a) was taken at low air velocity, photograph (b) at high air velocity. It is clearly visible that the liquid is accumulated at the trailing edge of the prefilmer lip. The accumulated liquid is periodically deformed into bubbles. In Fig. 4 a zoom into the region at the trailing edge is depicted, revealing the