Chemical Engineering Science 178 (2018) 12–26

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Prediction of fluid flow through and jet formation from a high pressure nozzle using Smoothed Particle Hydrodynamics

Gerald G. Pereira [a], Paul W. Cleary [a,*], Yoshihiro Serizawa [b]

[a] CSIRO Data61, Private Bag 10, Clayton South 3168, Australia
[b] Nippon Steel and Sumitomo Metals Corporation, Marunouchi 2-6-1, Chiyoda-ku, Tokyo 100-8071, Japan

HIGHLIGHTS

- Develops an SPH model for high pressure water flow through and from a nozzle.
- Predicts the break-up of the jet into a spray of fine high speed water droplets.
- The role of a nozzle insert on the flow and the resulting jet is determined.
- Very good agreement with experiments was achieved for droplet sizes.
- Generally good agreement for inlet and spray pressure distributions was obtained.

ARTICLE INFO

Article history:
Received 7 June 2017
Received in revised form 21 November 2017
Accepted 15 December 2017
Available online 16 December 2017

Keywords:
SPH
Spray
Jet
Nozzle
Fluid break-up

ABSTRACT

This paper reports on the development and evaluation of an SPH (Smoothed Particle Hydrodynamics) model for high pressure water flow through and from a nozzle and prediction of its break up into a spray of high speed water droplets. This appears to be the first application of the SPH technique to fully model a high pressure nozzle. The model predicts the internal flow and pressure distribution and enables exploration of the role of the internal geometric insert used in this design of the nozzle. It also predicts exit velocities from the nozzle as well as the pressure distribution generated by the nozzle and droplet size distribution of the resulting spray. Three different nozzle inflow rates were simulated and for all cases the numerical simulation of nozzle and spray gave generally good agreement with experiments, but complete agreement was not achieved. For better agreement, higher resolution for the SPH solution is required. The SPH simulations also show the role that the insert in the nozzle has on the flow and the resulting jet. It produces a flat inclined high velocity liquid jet within the second half of the nozzle which will generate turbulent eddies that may enhance the nucleation of the droplets in the fragmenting jet after it exits the nozzle. Overall, SPH has been shown to have a very good capacity to model high pressure nozzles and with further refinements of the technique should be able to yield accurate, quantitative data.

1. Introduction

High pressure fluid flow out of a narrow orifice is widely observed phenomenon in a range of industrial applications from food processing to ink-jet printing and spray-painting, fuel injection in automobiles and aircraft to liquid cooling of molten metal processes (to name just a few). However, it is true to say that a detailed understanding of the fundamental mechanisms of fluid flow within high pressure nozzles and the spray emanating from these devices is lacking. While experiments have been performed on these devices, they can be difficult to measure and prone to a variety of problems. For example in the region just outside the nozzle orifice optical techniques used to analyse the spray are greatly complicated by voids, non-spherical structures and high droplet density. Moreover, access to the nozzle interior is very difficult for real nozzles at the correct injection pressures (Fansler and Parrish, 2015). Thus high quality numerical modelling can help fill the gap and inform users and designers, not only in understanding the workings of the device but to also aid in optimizing the device's efficiency for specific applications.

Generally the liquid that emanates from a high pressure nozzle is in the form of a highly turbulent jet or spray. The break-up of a liquid jet has been widely studied in the past and a number of reviews give a good overview of physics of the process (Lin and Reitz, 1998; Gorokhovski and Hermann, 2008; Fansler and

* Corresponding author.
E-mail address: paul.cleary@csiro.au (P.W. Cleary).

https://doi.org/10.1016/j.ces.2017.12.033

Parrish, 2015). To characterize the nature of the different jet formations of a liquid flowing out of a nozzle it is useful to define two dimensionless numbers – the capillary number (Ca) and the Weber number (We). The capillary number is defined as $Ca = \mu v/\gamma$, where μ is the fluid viscosity, v is a typical velocity and γ is the surface tension (between the fluid being ejected from the nozzle and the external fluid which is most commonly air). The capillary number is the ratio of viscous force to surface tension force. At large Ca the viscous forces will destabilise a jet and cause it to break into small droplets. The Weber number is defined as $We = \rho R v^2/\gamma$, where ρ is the fluid density and R is the jet radius. It represents the ratio of jet kinetic energy to the surface energy. At large We inertial forces will destabilise the jet leading to break-up into many small droplets.

At very low Weber and Capillary numbers liquid drips out of the nozzle. A series of relatively large, discrete droplets slowly form from the nozzle. In this case because surface forces dominate the liquid tries to remain as contiguous as possible, but lack of fluid kinetic energy means the liquid columns eventually runs out of fluid. At higher kinetic energies of the jet (We > 4) a steady tube of liquid exits from the nozzle. This is unstable due to the classical Rayleigh-Plateau instability (Rayleigh, 1878; Plateau, 1873) and so the jet breaks up into a series of mono-sized droplets (also call jetting or Rayleigh break-up). As the velocity of the liquid coming out of the nozzle is increased further the effect of the ambient (external) air on the tube of liquid becomes important. When the Weber number of the gas becomes >0.2 (i.e., $We_{gas} = \rho_{gas} We/\rho$) the first wind-induced break-up occurs. The common example is sea-spray coming of a wave (Hoyt and Taylor, 1977). At this stage the surface tension forces are becoming insignificant compared to inertial and viscous effects. Surface tension effects can therefore be neglected and it is appropriate to next consider a third dimensionless number – the Reynolds number (which is the ratio of Weber to Capillary number), i.e. $Re = \rho R v/\mu$. At a Reynolds number of 10^5 atomization of the jet occurs. This is the formation of small droplets, much smaller than the nozzle diameter, which takes place due to short wavelength shear instability. As will be seen later, the Reynolds number for the high pressure nozzle considered in this paper is around10^5 – which places this flow in the early part of the atomization regime.

As Gorokhovski and Hermann (2008) pointed out, although there are a variety of ideas as to the physics underlying the atomization process, there is yet to be any consensus on this issue. A major reason for this is that there are a wide range of parameters which may influence this break-up. Evaluating the importance of each of these parameters over a wide variety of operating conditions is an extremely difficult task for experiments. Thus numerical modelling of the process may be able to offer significant insights into a complex physical process.

To date, there are four main numerical methods which have been applied to high pressure sprays (which undergo atomization). These are Direct Numerical Simulation (DNS), Reynolds-averaged Navier Stokes (RANS), Large Eddy Simulation (LES) all of which are based on solving the Navier-Stokes equations in an Eulerian framework. The fourth method is a probabilistic approach which models the statistical dynamics of the systems and consists of simulating the joint probability density function (PDF) of a number of fluid particles (Minier and Peirano, 2001). Jiang et al. (2010) present a review of both modelling and experimental studies of this phenomenon.

DNS methods identify the gas-liquid interface (and track it with a variety of techniques, see below) at every time step so as to provide greater numerical accuracy close to the orifice, where the liquid phase enters the gas phase. However (at the large Weber numbers which produce atomization and which are the ones of interest here) it can be extremely (computationally) expensive, since atomization is associated with a large range of length and time scales, with instabilities from the Kolmogorov scale to macro-scale. Reviews of DNS methods applied to high pressure sprays (Gorokhovski and Hermann, 2008; Jenny et al., 2012; Kolakaluri et al., 2014) give details of the variety of DNS techniques used including Volume of Fluid (VOF) method (Strom et al., 2016; Bianchi et al., 2007; de Villiers et al., 2004; Zhu et al., 2013; Srinivasan et al., 2010; Reddy et al., 2015), immersed boundary method (Uhlmann, 2005), multigrid method (Burton and Eaton, 2005), distributed Lagrangian multiplier based method (Pan et al., 2002) and conservative level-set methods (Desjardins et al., 2007). RANS approaches (which are grid-based) to the atomization process have been extensively investigated based on work by Vallet et al. (1998, 2001) and applied to diesel sprays (Beau et al., 2005; Blokkeel et al., 2004), air-blast atomizers (Jay et al., 2005; Meyers et al., 2005) and combustion (Taghavifar et al., 2016). While DNS simulations tend to focus on the droplets and RANS focus on a much larger scale encompassing many, many droplets (tending to the device-scale) the LES method often slots in between these two length-scales. It has been demonstrated for gas-solid flows (Carrara and DesJardin, 2006; Capecelatro and DesJardin, 2013), in dilute spray flows (Okong'o and Bellan, 2004) for cavitation (Oerley et al., 2015) and for diesel sprays (Jagus and Jiang, 2011; Valentino et al., 2007). Jenny et al. (2012) review the PDF approach to sprays where a joint PDF transport equation for both phase velocities, compositions, droplet diameter and turbulence frequency is developed. PDF methods have been applied to micro-mixing (Naud, 2003; Beishuizen, 2008) and combustion modelling (Beishuisen et al., 2011; Ge et al., 2008).

The numerical technique used here for modelling high pressure sprays is Smoothed Particle Hydrodynamics (SPH) which is a meshless method (explained below). Although the SPH method directly solves the Navier-Stokes equations the formulation of SPH is performed by a spatial filtering that is similar to that used to construct an LES and so SPH is able to capture structure down to the resolution used in a similar way. The primary difference is that SPH is solved in a Lagrangian manner and the version used here does not include an explicit sub-grid scale model (Cleary et al., 2007a), although this can also be included (Ting et al., 2006). Unlike the other classical approaches (mentioned above) which implement some sort of mesh on which to solve their partial differential equations, SPH implements a meshless technique to solve the Navier-Stokes equations. This meshless method could be expected to have significant advantages in the spray region of flow, where there are significant amounts of free surface and where new interfaces are being constantly created as fluid breaks up and where droplet collisions are possible. Furthermore this meshless method also has advantages over more classical mesh methods in dealing with geometries with complex-shaped internal moving parts (e.g. mixing impellers in Robinson and Cleary (2012)), such as the interior of the nozzle. Eulerian methods should be able to well cope with the prediction of the internal flow but the large pressure gradients at the nozzle opening and the very large scale of the free surfaces created immediately after this present significant challenges. Eulerian-Lagrangian methods where the internal flow is considered Eulerian and the external flow as Lagrangian could be considered but the transition in a narrow region and in the presence of significant gradients makes this challenging. Since the SPH method can, in principle, predict both the internal and external parts of the flow and avoids any complex transitions in between them it is an attractive option for predicting this type of flow.

The SPH method is a numerical technique for predicting the motion and deformation of materials based on solving continuum governing equations but in a Lagrangian frame work using discretised particles. This is most commonly used for fluid flow by solving the Navier-Stokes equations but it can also be used to solve for