Progress in Nuclear Energy 85 (2015) 140–155

Contents lists available at ScienceDirect

Progress in Nuclear Energy

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

Large eddy simulation of rough and smooth liquid plunging jet processes

Lyes Khezzar [a], [*], Nabil Kharoua [a], Kenneth T. Kiger [b]

[a] *Mechanical Engineering Department, The Petroleum Institute, Abu Dhabi, United Arab Emirates*
[b] *Mechanical Engineering Department, University of Maryland, USA*

A R T I C L E I N F O

Article history:
Received 18 September 2014
Received in revised form
8 June 2015
Accepted 9 June 2015
Available online xxx

Keywords:
Plunging jet
Air entrainment
Bubble plume
LES
VOF

A B S T R A C T

Large eddy simulation based on the Smagorinsky dynamic sub-grid scale model in combination with the multiphase volume of fluid (VOF) model, was used to simulate the flow of two turbulent plunging water jets. The jets were intended to simulate a weakly disturbed jet with low turbulence content and a highly disturbed jet with a profile almost fully-developed at the exit Reynolds number of 9000. The simulation captured successfully the previous experimentally-observed topological phenomena taking place during the transient impact and continuous-entrainment regime. It showed that surface instabilities in the free jet, due to its turbulence content have remarkable effect on submerged interfacial area, air volume, and air-entrainment rate. The simulation predicted an air entrainment rate within the range of semi-empirical correlations. The calculated mean velocity field exhibited almost identical trends for both. The combination of the LES-VOF models achieved a reasonably good level of agreement with experimental and empirical results.

1. Introduction

The process of a plunging jet occurs when a liquid jet impacts a quiescent pool of the same liquid. For a low-viscosity cylindrical liquid jet, the transient impact event initiates the creation of a relatively large air cavity that grows to form an annular sheath of air from the bottom, subsequently pinching off and undergoing further breakup into smaller bubbles as it moves downwards. Meanwhile, the top conical part of the air cavity retracts towards the free liquid surface, occasionally producing droplets as the free surface ejects past the mean water line. From the time of impact and until the disintegration of the large air cavities the process is called initial transient impact regime of the jet and is subsequently followed by what is termed a continuous air entrainment regime.

Plunging jets are intimately linked to the process of air or gas entrainment into liquid pools. Examples of industrial applications include chemical reactors, waste water treatment, poring motion of liquid metal or glass and plunging entry flows in deep downshafts of large generating stations. In the nuclear industry, the activation of emergency core cooling (ECC) of a nuclear reactor, in a loss of coolant accident, includes a situation very akin to the liquid plunging jet, where detailed knowledge of steam entrainment is necessary for correct prediction of heat transfer rates (Bousbia Saleh et al., 2007). In natural environmental flows, such as in breaking waves usually found on ocean surface, plunging jets can play a significant role in gas absorption mechanisms by oceans and in scouring of solid surfaces downstream of hydraulic structures (Bollaert and Schleiss, 2003; Ma et al., 2011).

Plunging jets have been reviewed comprehensively by Bin (1993) and recently by Kiger and Duncan (2012). Most of the initial work on plunging jets in the continuous entrainment regime, as documented in Cummings and Chanson (1997a and b), Evans et al. (1996), McKeogh and Ervine (1981), Schmidtke et al. (2009), Sene (1988) and Van de Sande and Smith (1976), was experimental and semi-empirical in nature, aimed at understanding the conditions of inception, the various mechanisms of air entrainment and the development of correlations to predict the penetration depth of the bubbly plume and the rate of gas entrainment. Works aimed at understanding the topological development of the flow during the initial phase of impacting jets by way of air cavity sizes, shapes, air entrainment rates and their scaling laws include notably those of Oguz et al. (1995), Ohl et al. (2000), Soh et al. (2005) and Zhu et al. (2000). Subsequent to the initial phase of impact, Kiger and Duncan (2012) and McKeogh and Ervine (1981), in their reviews,

---

[*] Corresponding author.
E-mail addresses: lkhezzar@pi.ac.ae (L. Khezzar), nkharoua@pi.ac.ae (N. Kharoua), kkiger@umd.edu (K.T. Kiger).

http://dx.doi.org/10.1016/j.pnucene.2015.06.011
0149-1970/

discuss two main air entrainment mechanisms- (i) air entrainment for weakly disturbed plunging jets characterized by a smooth surface and (ii) air entrainment for highly disturbed jets where the jet stream is non-uniform and its surface possesses regular or irregular ripples of varying sizes depending on the jet developing length. In view of the complexity of the flow below the free surface, local velocity measurements in the bubbling area remain rare and limited to regions of low bubble density. Indeed, despite advances in instrumentation in the last several decades, it remains a challenge to obtain detailed information on the flow using experimental means as shown in Bonetto and Lahey (1993) and McKeogh and Ervine (1981) who used the Phase Doppler Anemometer. It should, however, be acknowledged that progress is being made with improved PIV for planar velocity measurements (Hamad, 2010).

Plunging water jet flows are characterized as inherently multiphase and highly transient processes with a complex temporal and spatial variation of the interface between the liquid and gaseous phases. The complexity is typically further compounded with the incoming stream and subsurface jet often being themselves within a turbulent regime. Numerical methods based on potential flow theory using the boundary integral method have been quite successfully used to simulate the phase from initial impact to pinch-off with insightful contributions by Oguz et al. (1995) and Zhu et al. (2000). Such simulations are limited to the initial pinch-off phase, and hence cannot readily answer questions related to the interaction of the deforming interface with the subsurface flow. In this region, viscous effects likely have relevance, possibly through concentrated vorticity sources (jet shear layer roll-up) and the subsurface turbulent jet. Methods based on the URANS (unsteady Reynolds averaged Navier–Stokes) approach and utilizing different multiphase models have been employed. Qu et al. (2011) performed 2D URANS transient simulations for the continuous regime using a level-set and a multiphase mixture model, concluding that methods based on interface tracking technology are better suited at capturing surface instabilities and bubbling flows when computing resources are available. Qu et al. (2012) performed 3D URANS simulation focusing on the initial impact phase of a plunging jet and showed that the URANS-VOF (volume of fluid) combination is able to capture the geometrical features of initial impact cavity development and jet penetration depth with relatively good engineering accuracy. Jet surface instabilities could not however be observed. Kendil et al. (2012) performed 2D and 3D URANS using the Euler–Euler multiphase approach within the submerged domain during the continuous entrainment regime. Their results reveal the existence of a small vortex close to the free surface in the outer region of the submerged jet. Deshpande et al. (2012) and Deshpande and Trujillo (2013) conducted experiments and computational studies of shallow angle plunging jets. Despite the fact that their numerical approach ignores turbulence modeling, the results obtained are strikingly impressive during initial impact with this viscous limit approach.

In general, classical models based on the RANS approach cannot capture certain flow field details related to structures in view of the embedded inherent averaging, and higher fidelity methods must be utilized. Direct numerical simulation (DNS) was used by Galimov et al. (2010) and Lahey (2005) to analyze the air ingestion process by a plunging liquid jet. A solver based on parallelized finite-element method with a grid size of the order of 40 million cells was used by Galimov et al. (2010). Large eddy simulation (LES) provides the ability to compute the large scales that carry most of the Reynolds stress while modeling the subgrid-scale eddies that are typically more amenable to modeling due to their nearly isotropic and universal nature. Previous approaches to LES modeling of multiphase flows included modeling of turbulence using the models developed for single phase flows such as the standard Smagorinsky subgrid scale model and to capture the interface the VOF technique was used, the LES/VOF approach. Nevertheless, the resolution of the detailed flow topology, especially in the continuous entrainment regime, presents significant challenges such as the accurate modeling of the surface tension force through its unresolved or sub-grid scale component (Liovic and Lakehal, 2012). To our knowledge there are no previous LES studies on plunging jets. The purpose of the present work is to use LES in combination with the VOF-interface capturing algorithm to simulate plunging jet flows including the development phase of the free jet (which has been hitherto neglected by most numerical studies based on viscous flow solvers), the initial impact phase and the continuous entrainment phase. Being mindful of the current limitations and absence of subgrid scale multiphase coupling models in the ANSYS FLUENT code version used, the current work limits itself to the LES/VOF methodology at moderate Reynolds and Weber numbers, where the interface is resolved using the VOF method and the dynamic Smagorinsky model is used as a sub-grid model. We focus on what happens in the transient impact phase followed by the continuous entrainment phase for weakly and highly disturbed jets, extending, thus, in a virtual environment the phenomenology that was described in the previous experimental studies mentioned above. In particular, the study aims at investigating the ability of LES-VOF combination to capture liquid surface instabilities, shed more light into its ability to predict numerically the rates of air entrainment and the details of its mechanisms and the particular influence of jet exit velocity profile and its turbulence content. The numerical results are compared with the experimental work of Qu et al. (2013).

2. Mathematical description and numerical modeling

The one-fluid multiphase formulation uses a single set of conservation equations with different material properties for the separate phases in the computational domain accounted for by solving a convection equation for the volume fraction of the secondary phase. The surface tension is represented as a volume force distributed near the interface, with the interface reconstructed from a VOF tracking method.

The flow is assumed isothermal, unsteady and turbulent. It consists of two phases: a primary liquid (water) and a secondary gas (air). The turbulent flow assumption for the weakly and highly disturbed jets, considered in this study, is justified by the fact that the liquid jet in both cases emerges from the nozzle pipe as a turbulent pipe flow and will progress as a turbulent free jet towards the liquid pool under the continuous entrainment regime. When it impacts the initially quiescent liquid pool, it is assumed that the flow in the vicinity of the jet, below the free surface, will also be turbulent. From experimental evidence (Bonetto and Lahey, 1993; Igushi et al., 1998), turbulence in this region will be present in the continuous entrainment regime and has two sources – shear, and bubble-induced due to slip between the liquid and bubbles.

The Large Eddy Simulation approach, as implemented in ANSYS FLUENT 14.0 (ANSYS FLUENT Inc., Nov., 2011), is used in this study and is based on the dynamic Smagorinsky-Lilly model under Favre-averaging of the Navier–Stokes equations (Germano et al., 1991; Lilly, 1992). The details of the model equations for conservation of mass and momentum are given below.

First the flow variables are decomposed into large-scale components (denoted here by the overbar) and small subgrid scale components by employing a top-hat filtering operation. For any variable Ø its filtered form is:

$$\overline{\phi}(\vec{x}) = \int_{fluid\ domain} \phi(\vec{y})G(\vec{x},\vec{y})dy \qquad (1)$$