STATE-OF-THE-ART AND NEEDS FOR JET INSTABILITY AND DIRECT CONTACT CONDENSATION MODEL IMPROVEMENTS

by

Anis BOUSBIA-SALAH, Fabio MORETTI, and Francesco D'AURIA

Received on December 4, 2006; accepted in revised form on May 22, 2007

There is a common understanding among thermal-hydraulic experts that the system analysis codes have currently reached an acceptable degree of maturity. Reliable application, however, is still limited to the validated domain. There is a growing need for qualified codes in assessing the safety of the existing reactors and for developing advanced reactor systems. Under conditions involving multi-phase flow simulations, the use of classical methods, mainly based upon the one dimensional approach, is not appropriate at all. The use of new computational models, such as the direct numerical simulation, large-eddy simulation or other advanced computational fluid dynamics methods, seems to be more suitable for more complex events. For this purpose, the European Commission financed NURESIM Integrated Project (as a part of the FP6 programme), was adopted to provide the initial step towards a Common European Standard Software Platform for modelling, recording and recovering computer data for nuclear reactor simulations. Some of the studies carried out at the University of Pisa within the framework of the NURESIM project are presented in this paper. They mainly concern the investigation of two critical phenomena connected with jet instabilities and direct contact condensation that occur during emergency core cooling. Through these examples, the state-of-the-art and the need for model improvements and validation against new experimental data for the sake of getting a better understanding and more accurate predictions are discussed.

Key words: advanced computational tools, multi-fluid flow, jet instabilities, direct contact condensation

INTRODUCTION

In the near past, most of the phenomena that occur in two-phase flows were simulated using best estimate (BE) thermal-hydraulic codes. The latter have been initially developed for industrial applications of large geometrical scales and are generally based on the one-dimensional approach. This prevents the existing codes from performing realistic simulations or evaluating the safety and performance of advanced reactor systems with sufficient reliability. For instance, in such "large-scale" codes, the structures of dispersed phases such as bubbles or droplets cannot be recognized individually, although additional field describing of the additional phase (*e. g.* the droplets, separately from the liquid film on the wall) has been implemented in certain oil industry codes [1, 2]. Furthermore, advanced reactor systems often adopt new design features which are based upon multi-dimensional-multi-fluid thermal-hydraulic characteristics. Consequently, multi-dimensional modelling of multi-phase flows has become more prevalent as computer capabilities have significantly expanded. Such analyses become necessary for deriving accurate estimates of the complex phenomena that occur during normal or out-of-the ordinary operating conditions. It is, therefore, quite natural to try to understand the multi-dimensional phenomena either on the basis of sophisticated experiments or more reliable computer codes. Developing and testing these tools requires intensive work on complex physical modelling and numerical schemes. For this purpose, the European Commission NUclear REactor SIMulations (NURESIM) project was financed with the aim of setting up an initial step towards a Common European Standard Software Platform for model-

Technical paper
UDC: 532.51/.52:519.876.5
BIBLID: 1451-3994, *22* (2007), 1, pp. 58-66
DOI: 10.2298/NTRP0701058B

Authors' address:
Department of Mechanical and Nuclear Engineering,
University of Pisa
2, Via Diotisalvi, 56126 Pisa, Italy

E-mail address of corresponding author:
b.salah@ing.unipi.it (A. Bousbia-Salah)

ling, recording and recovering computer data for nuclear reactor simulations.

Advanced computational methods such as computational fluid dynamics (CFD) have now been recognized as an important tool, as discussed in [3]. This method was initially developed for one-phase flows, but it is also increasingly been applied for the calculation of two-phase ones. Velocity, pressure and temperature fields can be determined with a high degree of accuracy and therefore higher degree of universal validity is to be expected. However, a prerequisite for these model improvements is the provision of adequate experimental data.

The validation tests for these calculation tools include separate effect tests (SET) and some real size industry-like experiments; they are used to validate the physical models implemented in the codes [4]. Experimental data are also needed for the quantification of code uncertainties, for checking the scale-up capability to fill the gap between the real plant and test facility scales and for the development of new generation codes or advanced reactor systems.

The pressurized thermal shock (PTS) constitutes one of the most important safety issues in the design of nuclear power plants (NPP) [5]. Nowadays, this topic is rigorously investigated through the use of BE computational tools, more powerful numerical methods that take into account the multi-dimensional, multi-phase flow modelling. The most severe PTS scenario limiting the reactor pressure vessel (RPV) lifetime is a cold water emergency core cooling (ECC) injection into the cold leg during a LOCA. The injected water mixes with the hot fluid present in the cold leg and the mixture flows towards the downcomer of the RPV. Under such a scenario, extreme thermal gradients in the structural components may occur and last for a long period of time leading, consequently, to very high thermal stresses. Therefore, the loads upon the RPV must be assessed reliably. To achieve this goal, relevant physical phenomena as ECC jet instabilities and direct contact condensation before mixing should be accurately modelled when a two-phase PTS scenario occurs. These two issues are discussed hereafter in more detail.

JET INSTABILITY PHENOMENON

Overview

The stability analysis of a liquid jet injected into a gaseous medium is a crucial matter for several technical and industrial applications. In particular, a good understanding of the jet break-up mechanism is fundamental for the development of enhanced injection systems ranging from ink-jet printing, fueling of Formula 1 race cars, direct fuel injection for diesel engines, to such applications as ECC in PWR nuclear plants. Due to these technological applications, the problem of jet stability has been widely investigated, both experimentally and theoretically.

The pioneering work on the topic was performed in the 19th century by Rayleigh who developed a theory for surface waves on a liquid jet caused by small perturbations in velocity and pressure. He made a linearized analysis of the capillary stability of an incompressible liquid. The linear stability theory allows predicting the break-up length (*i. e.* the distance from the nozzle exit where the break-up occurs) and the drop's dimensions. These quantities were shown to depend upon the Weber number ($W \quad \rho V^2 a/\sigma$) which represents the "ratio" between the inertial and capillary forces.

A liquid jet issuing from a nozzle may breakup into small drops when it is subjected to even minute disturbances. The fundamental mechanism responsible for the break-up of a liquid jet is that of surface tension induced instability. Other mechanisms, however, can modify the break-up process and alter both the continuous length of the jet and the size distribution of the drops. These mechanisms include other effects that influence jet stability, such as:

the velocity profile relaxation at the nozzle exit: a laminar jet, for instance, is characterized by a parabolic velocity profile; after the jet exits from the nozzle, however, no more wall shear stresses are present and the profile tends to become uniform,

the interaction between the liquid jet and the surrounding gas: the relative motion between the jet and the surrounding air gives rise to pressure forces on the jet, thus enhancing instability. Moreover, at high jet velocities, the shear forces may strip away ligaments of fluid from the jet's surface, and

the heat and mass transfer at the jet surface: the transfer may produce thermal or concentration gradients which in turn determine a surface tension-driven flow within the jet; jet stability may thus be enhanced or reduced.

Therefore, the main objectives of the studies on liquid jet instability have been to obtain the growth rates of the initial disturbances (as a function of the disturbance wave number), the cut-off wave number, the drop sizes after the break-up, the break-up length, and the break-up time; as well as to determine drop behavior after the break-up (*e. g.* satellite merging) and the effects of the initial disturbance amplitude, disturbance-type (such as surface, pressure or velocity disturbances), the initial velocity profile of the jet, and fluid properties.

Jet modelling

A liquid jet is naturally unstable and breaks up into droplets. The mode of disintegration is strongly related to the character of the jet and velocity difference between the liquid and the surrounding gas. This difference controls the level of aerodynamic forces. In the case of a low liquid velocity jet, in an ambient gas at rest, the aerodynamic forces are negligible and the jet breaks up into drops with a diameter comparable to the jet diameter under the capillary pinching phenomenon. This is the Rayleigh mode regime. For high values of liquid or gas velocity, the instabilities lead the jet to break up into drops with diameters much smaller than the jet diameter.

Liquid jet stability has been studied extensively in the past and there is considerable literature on the characteristics of "submerged" jets (*i. e.,* water into water or air into air). However, much less is known about the flow of "free" jets (*i. e.*, water into air). Although the former, after an initial region of the flow development, is completely described by the local momentum conditions and thus amenable to simple mathematical modeling, the latter displays a total absence of self-similarity and is governed by parameters such as pressure, nozzle size and configuration, density, viscosity and surface tension of the jet fluid. The breaking up of free jets has, nevertheless, attracted a good deal of attention (see fig. 1).

In 1879, Rayleigh carried out the first important study on jet stability. He studied the temporal stability of inviscid jets by means of the linear stability theory and proved that only axis-symmetric disturbances with a wavelength larger than the circumference of the undisturbed nozzle grow in time. Neglecting the effect of ambient fluid, the viscosity of the jet liquid and gravity, Rayleigh demonstrated that a cylindrical liquid jet is unstable with respect to disturbances characterized by wavelengths larger than the jet's circumference [7].

The general model considered for a water jet is, as shown in fig. 2, issuing at time (t) from a nozzle of a radius R with a velocity $u(0, t)$ that is uniform across

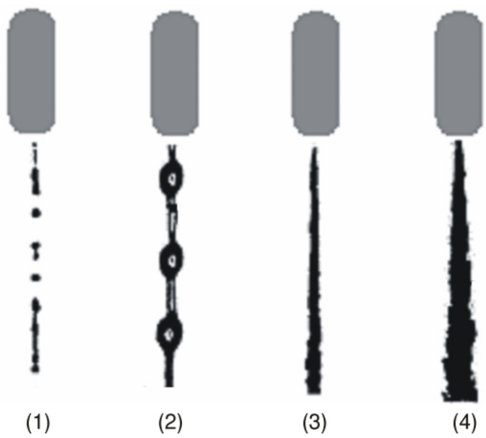

Figure 1. Different regimes of the break-up of fluid jets *(1) Rayleigh's regime, (2) first wind induced regime, (3) second wind induced regime, (4) atomization regime [6]*

the exit area, but periodic around a mean value of U and expressed as:

$$u(0,t) \quad U \qquad U \sin 2\pi f(t) \tag{1}$$

where U is the average exit velocity and U and f are the amplitude and frequency of the modulation, respectively.

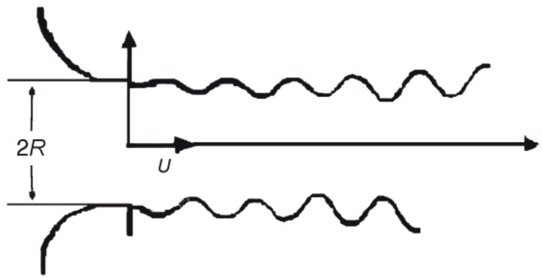

Figure 2. Infinite axis-symmetric jet

Rayleigh's model consists of imposing, at $t = 0$, an interfacial disturbance with Fourier wave numbers k and n in the z and θ directions, respectively. He determined the evolution of this initial disturbance to an inviscid jet using normal modes $e^{i(kz + n) + s(k, n)t}$, where z is along the thread axis and $s(k, n)$ is complex.

Rayleigh established that only axis-symmetric ($n = 0$) disturbances with wavelengths (2 /k) larger than the undisturbed thread circumference (2 a) grow in time [$s(k, n = 0) > 0$], and that there is a maximum in the growth rate at a wave number (k_{max}) equal to 0.696/a for which the disturbances grow without traveling, [$s(k, n = 0) = 0$]. This temporal stability of the static thread was reconsidered in the case when the thread fluid is viscous and surrounded by a second immiscible viscous liquid.

Since capillarity drives instability, the range of unstable wavelengths remains the same when viscous effects are included, but growth rates are reduced and the maximally growing waves are shifted to longer wavelengths. The disturbances were generally modeled as growing in time. In most applications, the jet issues from an orifice and breaks downstream, *i. e*., disturbances continually imposed at the nozzle tip (either intentionally, through periodic oscillations, or through tip imperfections) grow by the destabilizing action of capillarity as they are diffused downstream, until they cause the jet to break up into drops.

Keller [8] first noted this fact. It suggested a base state in which the jet is modelled as a circular infinite jet moving with uniform velocity upon which interfacial disturbances with Fourier frequency modes ω are imposed, locally, in space and, periodically, in time. For an inviscid jet and axis-symmetric disturbances, Keller [8] determined the spatial growth for the harmonic response ($e^{ik(\omega)z + i\omega t}$; $k(\omega)$ complex, $k_i < 0$ for spatial growth) as a function of the Weber number ($W = \rho V^2 a/ \quad \sigma$ is the fluid density). This demonstrated that for frequencies in the unstable range, the drop size and distance to the break-up point can be predicted, since the drops which are formed at this point in time have a size which scales as $1/k_r(\omega)$ and the distance downstream from the break-up is of an order of $1/k_i(\omega)$.

Numerical and experimental studies

A large body of experiments studying the break-up of jets into drops has been undertaken, usually by applying disturbances generated with different techniques:

acoustic-induced vibrations,
electromagnetic-induced vibrations,
mechanically-induced vibrations, and
thermal modulation of the surface tension.

Generally, a sinusoidal periodic disturbance of frequency ω applied at the nozzle tip using acoustic, electromagnetic or piezo-electric-induced pressure vibrations or a vibrating impinging needle. In all of the above mentioned studies, the distance between two successive peaks determines the wavelength of the disturbances. The (spatial) growth rate was measured by measuring the break-up length or by measuring the amplitude of the disturbance at two successive peak/troughs and by assuming exponential spatial growth. In addition, the waves were assumed to diffuse with jet velocity, so that the temporal theory applies in the moving frame.

Drazin and Reid [9] performed stability analysis for plane jets similar to Rayleigh's analysis of a cylindrical jet. They found that a non-viscous infinite plane jet is stable, *i. e.* not subject to any disturbances, because surface tension always has a damping effect and forces the surface back to its initial shape when perturbed.

Sou and Tomiyama [10] studied the effect of a turbulent flow within an atomization nozzle and liquid jet discharged from the nozzle. Their numerical simulation was based upon a hybrid numerical method, a combination of a two-way bubble tracking method and an interface tracking method. This hybrid method enabled them to examine the effects of bubbles on liquid jet deformation. As a result of bubble tracking and hybrid simulations, the following conclusions were reached:

the method yields good predictions for the distribution of pressure and bubbles within the nozzle, as well as for the relation between injection pressure and liquid flow rate,

the collapse of bubble clouds within liquid jet enhances the deformation of the liquid jet, and

the vortices generated in a nozzle play a more important role in jet deformation than interfacial forces acting on the liquid jet.

On the other hand, Cinnella [11] made a preliminary step towards the understanding of non axis-symmetric liquid jet break-up mechanisms. Specifically, the objective was to investigate whether, and how, the use of non-circular nozzles influences jet break-up properties. A numerical approach was used, *i. e.* a finite volume code. The volume of fluid (VOF) front capturing methodology was considered to represent the liquid/gas interface.

Numerical results concerning jets in Rayleigh's regime emanating from either circular or square-edged nozzles were emphasized. In this model, the jets are subject to a sinuous velocity perturbation at the nozzle inlet and characterized by growing values of the Weber number. For square jets, at least in the range of the Weber number considered, a significant reduction of the break-up length with respect to the axis-symmetric case, as well as a reduced sensitivity of the break-up length to the jet Weber number, have been found. These phenomena seem related to the appearance of secondary flows whose intensity grows with jet speed, promoting jet.

A numerical study was carried out by Ibrahim [12] on the evolution of asymmetrical disturbances on a viscous liquid jet in an inviscid gas medium. The asymmetrical disturbances were shown to become relevant at high Weber numbers.

Alternatively, Furlani [13] studied the effect of surface tension on the break-up and drop formation. The surface tension was locally modulated by heating the surface of the jet as it leaves its orifice. An analytical closed-form expression for time dependence of the jet radius was derived. This expression enables rapid parametric analysis of jet instability (time to drop formation) as a function of jet radius, velocity, viscosity, density, and surface tension.

Sami and Ansari [14] studied the effects of velocity profile relaxation on the instability of laminar liquid jets cast from long tubes. There is substantial evidence to indicate that the results of these effects are similar to those of the relative motion between the jet and its ambient gas, *i. e.* as the velocity is increased, the jet length reaches a maximum and then decreases as the velocity increases further. The effect of velocity profile relaxation on the drop size distribution, however, has not yet been measured.

The break-up of the jet can additionally be dependent on nozzle geometry. Karasawa [15] clearly demonstrated the effect of nozzle length on the development of instabilities on the liquid jet's surface. With the increase in nozzle length, the jet surface shape changes from smooth to wavy. The initial velocity profile at the nozzle exit can, thus, have a substantial influence on liquid jet stability and its sensitivity to the velocity profile is particularly true for low mean velocity (laminar) flows. The effect of nozzle turbulence on drop size distribution was experimentally investigated by Sterling, and Abbott [16].

Pan and Suga [17] used direct numerical simulations (DNS) to simulate water jets into air. The level-set method is adopted in present simulations as a front-capture methodology to track the dynamically evolving liquid/gas interface. Reynold's number at jet exit ranges from 480 to 15,000, Weber's number from 3 to 10,000. The liquid/gas density ratio is 816. The dynamic features of liquid/gas jet flows and the primary break-up are reasonably captured by their simulations. Numerical simulations confirm the effects of the gravity and surface tension forces on the break-up process at relatively low Reynold's numbers. It has also been observed that the radial velocity, induced during the relaxation of the axial liquid velocity profile, leads to the disintegration of high speed jets (see fig.3). The study supports the experimental observation that high-speed laminar jets are more prone to instability and break-up in an extremely violent pattern, much sooner than fully developed turbulent jets. Nevertheless, velocity profile relaxation with turbulence effects needs to be further investigated.

Demuren and Wilson [18] investigated complex streamwise vorticity fields in the evolution of non-circular jets. Generation mechanisms are investigated via Reynolds-averaged (RANS), large-eddy (LES), and direct numerical (DNS) simulations of laminar and turbulent rectangular jets. Complex vortex interactions are found in the DNS of laminar jets, but axis-switching is observed only when a single instability mode is present in the incoming mixing layer. With several modes present the structures are not coherent and no axis-switching occurs. RANS computations also produce no axis-switching. On the other hand, LES of high Reynolds number turbulent jets produce axis-switching even in cases with several instability modes in the mixing layer. Three-dimensional simulations of laminar and turbulent jets with rectangular cross-sections were performed. At low Reynold's numbers, the DNS were performed, while at higher Reynold's numbers, LES or RANS computations were carried out. Figure 4 shows the contours of the instantaneous total vorticity for rectangular jets at low and high Reynold's numbers.

Figure 3. Simulation of high speed water jets into air

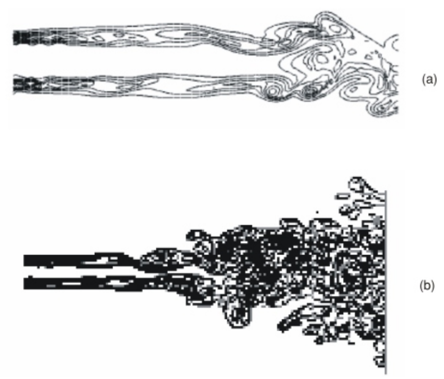

Figure 4. Instantaneous broad mode instabilities total vorticity of rectangular jet at Re = 750, with DNS (a) and at Re = 75 000 with LES (b) [18]

DIRECT CONTACT CONDENSATION

Direct contact condensation (DCC) constitutes a critical issues for LWR safety-related situations. However, there are no adequate models to simulate accurately such phenomenon as the condensation of a large bubble containing steam and non-condensable gases in water [19].

DCC generally occurs around a jet injection during an ECC transient in pressurized water reactors. In large break LOCAs, very large injection flow rates may occur particularly during the accumulator discharge. These situations are characterized by a strong thermo-mechanical coupling and strong instabilities may be created by high instantaneous condensation rates. This leads to plug formation in cold legs which strongly affects the flow regime and the delivery of water to the vessel (intermittent water delivery to the downcomer). These phenomena were observed experimentally in the scale-1 upper plenum test facility (UPTF) by Weiss [20]. Therefore, the prediction of thermal-hydraulic interactions by contact condensation of steam on the water surface is of interest for an efficient ECC injection.

The experimental results performed in the COSI test facility [21] show that a large fraction of the total condensation occurs in the zone of the jet and, more particularly, around the injection. Since the condensation increases in the vicinity of the injection, this suggests that the turbulence induced by the jet is the main controlling mechanism.

In ideal conditions, when the jet is continuous without a breakup, Janicot and Bestion [21] suggested the use of the Iciek correlation to predict the condensation on the jet, notwithstanding the fact that no experimental evidence exists to confirm this model. The correlation is expressed in terms of the Stanton number:

$$\mathrm{St} = 0.00835 \left(\frac{l}{d}\right)^{0.28} \mathrm{Fr}^{\,0.1} \tag{2}$$

where l is the jet length, d – the injection diameter, and Fr – the Froude number.

Various models describing heat and mass transfers at an interface have been proposed either in terms of the surface renewal theory or Reynolds averaged turbulent diffusion. However, most of the developed correlations are valid only under the test facility conditions under which they are derived, but not necessarily for different geometries and scales.

In addition, an accurate modeling of such mechanisms requires the consideration of liquid re-circulations and the calculation of temperature profiles in the liquid layer. Therefore, both experimental and numerical limitations exist.

Simulations of contact condensation have shown that standard interfacial mass transfer models are not sufficient to predict this phenomenon accurately. These limitations could be overcome by considering models entirely based on 3-D flow regime approaches. The numerical limitations are nowadays being surmounted. In recent works, as those presented by [22-24], several solution methods are investigated.

As a continuation of the work of Janicot and Bestion [21], the work of Coste [24] represents a significant attempt to calculate the direct contact condensation occurring at the ECC injection while taking into account the multi-D features of the two-phase flow. A 2-D computational approach is adopted in order to describe the injection of a sub-cooled water jet into a horizontal pipe filled with flowing steam close to saturation. The geometry of the calculation domain, as well as the boundary conditions for the calculations, are derived from a set of experiments performed at the COSI facility. The model is based upon a set of simple models, including sub-grid scale effects of turbulence and a liquid/gas heat transfer model for condensation based on the surface renewal concept. The results were validated against eighteen COSI tests.

The calculated condensation rate is compared against the measured one and the obtained errors for different runs were generally less than 20%, despite the limitations related to the use of a 2-D approach. The flow topology identified in ref. [21] (distinction between the recirculation zone, high turbulent mixing zone, and the downstream stratified flow) is also reproduced by Coste's calculations, although only qualitatively. Both measurements and calculations indicate that a great part of the condensation takes place in the jet zone and that it is mainly governed by the turbulence induced by the jet itself. Downstream from the jet region, the condensation rates are lower than in the jet region. The presence of non-condensable gases reduces the condensation rate, since those gases tend to form something like an insulating layer between the vapour and the liquid, thus hindering phase interaction. In such conditions, heat transfer is controlled by the vapour molecular diffusion through the air layer. However, this last effect was not included in the calculations.

Generally, much better tendencies of phenomena and quantifications of some of the effects were provided by these predictions. However, the calculations under-predicted thermal stratification, probably due to the limits imposed by not taking into account the 3-D effects, as well as those of overestimating turbulent liquid conductivities and viscosities.

Khoo [22] used the DNS method to simulate vapor condensation at a turbulent liquid interface. On the other hand, Maksic and Mewes [23] proposed a numerical simulation for the condensing flow of subcooled water injected into a pipe filled with saturated steam. For this purpose, a 2-D model was developed for the description of the flow with direct contact condensation. The mass flow rate of condensation is calculated as a function of the interfacial area and turbulent energy dissipation of water in the vicinity of the interface. The solution is obtained using the commercial CFD-code (fig. 5), validated against UPTF tests.

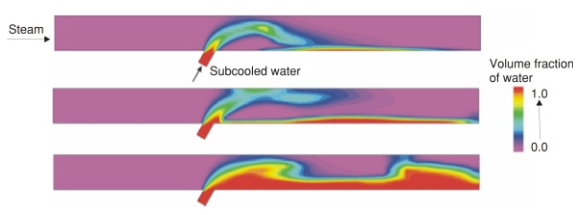

Figure 5. Calculated results for phase distribution in time [23]

MODEL IMPROVEMENTS

Current numerical tools are now capable of modeling even structures with complex geometry by using a 3-D numerical model and solving complex physical aspects of respective flows [25]. However, two-phase

CFD is much less mature than single phase CFD. The flows are much more complex and many basic phenomena may take place at various scales. Thus, it is clear that the physical modeling will have to be improved over a long period of time. Fundamental questions related to the averaging or filtering of equations are not yet as clearly recognized. This makes the separation between physics and numerics not always well defined [26]. Therefore, additional advancements are needed to include further developments of the two-phase CFD models on the basis of experiments and on the basis of new systems of equations and/or new closure laws based on theoretical and/or empirical basis using interface tracking methods (ITM).

The PTS simulations, including jet flows and DCC phenomena, must be enhanced beyond the current state-of-the-art by substantially improving the two-phase flow modeling capabilities of current CFD and system codes. Improvements are necessary, both for the physical models (heat transfer coefficient at the interface between liquid and vapor, instabilities of the interface), as for the numerical schemes (accuracy, CPU time). The improvement of current state-of-the-art CFD-codes must be based on existing experiments, as well as on new experiments equipped with novel measuring techniques (*e. g*., mean velocity, turbulent fluctuations, liquid temperatures, high-frequency wire-mesh sensors) that offer sufficient resolution in space and time for comparison with the CFD computations.

The desired model improvement could, for instance, be inspired by recent works published in the open scientific literature such as [27], which proposes a two-fluid model based on accurate mechanistic computational fluid dynamics that can predict a wide variety of steady and transient multiphase flow phenomena. A wide range of mechanistic models are also to be found in the paper dedicated to gas-oil multiphase flow, as found in [28]. However, as outlined by Yadigaroglu [29], in the current generation of codes, the mechanistic models of interfacial areas depend on flow regime; the flow regimes are typically determined using a collection of mechanistic transition criteria with a relatively simple flow regime "maps" as a result. This approach may lead to abrupt flow regime changes and, possibly, to certain non-physical situations.

For calculation techniques, Lakehal [30] proposed novel developments in the DNS-based turbulence modeling strategy for calculating jets in crossflows. The new trends in computational methods for nuclear reactor thermal-hydraulics are discussed by Yadigaroglu [31] who proposes a new computational multi-fluid dynamics (CMFD) for a better simulation of multiphase flow phenomena.

CONCLUSIONS AND RECOMMENDATIONS

Multi-dimensional modeling of multiphase flows has become more prevalent as computer capabilities have significantly expanded. Such tools are nowadays intensively investigated in order to obtain more accurate simulations of complex phenomena that occur in industrial and nuclear power plants.

Although a huge amount of investigations have been carried out to understand the mechanisms related to jet instabilities, an approach capable of describing the phenomenon which would take into account all the parameters that govern the stability of a jet (gravity, the viscosity of both the liquid and the surrounding gaseous medium, nozzle internal flows, turbulence, velocity profile relaxation, asymmetry, satellite drop formation, *etc*.) seems to be still missing. Most of the achieved results are based on analytical or semi-analytical models within the linear perturbation theory approach, with a number of simplifying assumptions that limit their applicability. The use of CFD simulations based on the volume-of-fluid method shows promising results. Nevertheless, further developments in modeling are needed and much is expected from DNS and multi-phase CFD approaches, in particular from the interface-tracking techniques, as well as experimental data for the validation of numerical techniques.

ACKNOWLEDGEMENT

The current paper was performed under the EC-FP6 NURESIM project (http://www.nuresim.com/) for the subtask D2.1.1 related to the "Identification of relevant PTS scenarios, state-of-the-art of modelling and needs for model improvements".

REFERENCES

[1] Faluomi, V., Bonuccelli, M., Bousbia-Salah, A., Simplified Transient Multiphase Model for Oil Field Development Analysis, *Proceedings*, 3rd International Symposium on Two-Phase Flow Modelling and Experimentation, Pisa, Italy, September 22-25, 2004, pp. I-351–I-358

[2] Bonuccelli, M., Faluomi, V., Ansiati, A., Bousbia-Salah, A., Blotto, P., Modelling Annular Flow at High Gas Velocities for Well Blowout Analyses, *Proceedings*, 3rd International Symposium on Two-Phase Flow Modelling and Experimentation, Pisa, Italy, September 22-25, 2004, pp. I-359–I-366

[3] ***, IAEA/NEA Workshop on the Use of CFD Codes for Safety Analysis of Reactor Systems Including Containment, Pisa, Italy, November 11-13, 2002

[4] Egorov, Y., Validation of CFD Codes with PTS-Relevant Test Cases, *Proceedings*, 5th Euratom Framework Programm ECORA project WP4, 2004

[5] Sievers, J., Schulz, H., Bass, B. R., Pugh, C. E., Final Report on the International Comparative Assessment

Study of Pressurized Thermal-Shock in Reactor Pressure Vessels (RPV PTS ICAS), NEA/CSNI/R(99) 3, GRS-152, 1999

[6] Sleuyter Franky, C. B., Stability of Flashing and Non-Flashing Liquid Jets, Master thesis, University of Eindhoven, The Netherlands, 2004

[7] Chauhan, A., Maldarelli, C., Rumschitzki, D. S., Papageorgiou, D. T., An Experimental Investigation of the Convective Instability of a Jet, *Chemical Engineering Science, 58* (2003), pp. 2421–2432

[8] Keller, J. B., Rubinow, S. I., Tu, Y. O., Spatial Instability of a Jet, *Physics of Fluids, 16* (1973), pp. 2052–2055

[9] Drazin, P. G., Reid, W. H., Hydrodynamic Stability, Cambridge University Press, Cambridge, UK, 1982

[10] Sou, A., Tomiyama, A., Numerical Simulation of Liquid Jet Deformation Based on Hybrid Combination of Interface and Bubble Tracking Methods, *Proceedings on CD-ROM*, 3rd European-Japanese Two-Phase Flow Group Meeting, Certosa di Pontignano, Italy, September 21-27, 2003

[11] Cinnella, P., Pastore, L., Laforgia, D., Comparison of the Stability Properties of Round and Square Liquid Jets, in: 59° Congresso ATI, Generazione di Energia e Conservazione dell'Ambientе, Vol. II, SG Editoriali, Padova, Italy, 2004, pp. 965-975

[12] Ibrahim, E. A., Asymmetric Instability of a Viscous Liquid Jet, *Journal of Colloid and Interface Science, 189* (1997), pp. 181-183

[13] Furlani, E. P., Temporal Instability of Viscous Liquid Microjets with Spatially Varying Surface Tension, *J. Phys. A: Math. Gen., 38* (2005), pp. 263–276

[14] Sami, S., Ansari, H., Governing Equations in a Modulated Liquid Jet, *Proceedings*, First US Water Jet Conference, Golde, Colorado, USA, April 7-9, 1981, pp. 14-24

[15] Karasawa, T., Masaki, T., Abe, K., Shiga, S., Kurabayashi, T., Effect of Nozzle Configuration on the Atomization of a Steady Spray, *Atomization Sprays, 2* (1992), pp. 411–426

[16] Sterling, A. M., Abbott, W. T., Mechanisms of Water Jet Instability, *Proceedings*, First US Water Jet Conference, Golden, Colorado, USA, April 7-9, 1981, pp. 40-46

[17] Pan, Y., Suga, K., Direct Simulation of Water Jet into Air, 5th International Conference on Multiphase Flow, ICMF'04, Yokohama, Japan, May 30 – June 4, 2004, Paper No. 377

[18] Demuren, A. O., Wilson, R. W., Streamwise Vorticity Generation in Laminar and Turbulent Jets, NASA/CR-1999-209517. ICASE Report No. 99-33, 1999

[19] Yadigaroglu, G., Andreani, M., Dreier, J., Coddington, P., Trends and Needs in Experimentation and Numerical Simulation for LWR Safety, *Nuclear Engineering and Design, 221* (2003), pp. 205–223

[20] Weiss, P., UPTF Experiment: Principal Full Scale Test Results for Enhanced Knowledge of Large Break LOCA Scenario in PWR's, *Proceedings*, 4th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-4), Karlsruhe, Germany, 1989, Vol. 1, p. 60

[21] Janicot, A., Bestion, D., Condensation Modelling for ECC Injection, *Nuclear Engineering & Design*, 145 (1993), pp. 37–45

[22] Khoo, B. C., Patera, A. T., Sonin, A. A., Direct Numerical Simulation of Pure Vapor Condensation at a Turbulent Liquid Interface, An Extracted-Subdomain Approach, *Heat Transfer Division*, ASME, 114 (1989), pp. 39-50

[23] Maksić, S., Mewes, D., Numerical Calculation of the Direct Contact Condensation by Cold Leg ECC-Injection, *Proceedings*, 39th European Two-Phase Flow Group Meeting, Aveiro, Portugal, June 17-20, 2001

[24] Coste, P., Computational Simulation of Multi-D Liquid-Vapour Thermal Shock with Condensation, 5th International Conference on Multiphase Flow (ICMF'04), Yokohama, Japan, May 30 – June 4, 2004, Paper No. 420

[25] Bousbia-Salah, A., D´Auria, F., Use of Coupled Code Technique for Best Estimate Safety Analysis of Nuclear Power Plants, *Progress in Nuclear Energy, 49* (2007), pp. 1-13

[26] Bestion, D., Recommendation on Use of CFD Codes for Nuclear Reactor Safety Analysis, 5th Euratom Framework Program ECORA Project-WP8, 2004

[27] Lahey Jr, R. T., Drew, D. A., The Analysis of Two-Phase Flow and Heat Transfer Using a Multidimensional, Four Field, Two-Fluid Model, *Nuclear Engineering and Design, 204* (2001), pp. 29-44

[28] Petalas, N., Aziz, K., A Mechanistic Model for Multiphase Flow in Pipes, *J. Can. Pet. Tech., 39* (2000), pp. 43-55

[29] Yadigaroglu, G., The Interfacial Area in Transient Computations: Review and Developments, Trends in Numerical and Physical Modeling for Industrial Multiphase Flows, Minisymposium on Interfacial Area, Institut d'Etudes Scientifiques de Cargèse (Corse), France, September 27-29, 2000

[30] Lakehal, D., DNS and LES of Turbulent Multifluid Flows, in: Keynote Lecture, *Proceedings*, 3rd Symposium of Two-Phase Flow Modelling and Experimentation, Pisa, Italy, September 22-25, 2004, pp. I-67–I-78

[31] Yadigaroglu, G., Computational Fluid Dynamics for Nuclear Applications: from CFD to Multi-Scale CMFD, *Nuclear Engineering and Design, 235* (2005), pp. 153-164

Анис БУСБИА-САЛАХ, Фабио МОРЕТИ, Франческо ДАУРИА

САВРЕМЕНИ МОДЕЛИ НЕСТАБИЛНОГ МЛАЗА И ДИРЕКТНЕ КОНТАКТНЕ КОНДЕНЗАЦИЈЕ И ПОТРЕБА ЊИХОВОГ УНАПРЕЂЕЊА

Међу стручњацима за термо-хидраулику постоји опште уверење да су програми за анализу система управо достигли прихватљив ниво зрелости. Међутим, њихова поуздана примена још увек је ограничена само на потврђени делокруг. Постоји растућа потреба за квалификованим кодовима за оцену сигурности постојећих реактора и развој напредних реакторских система. Под условима који укључују вишефазну симулацију тока, употреба класичних метода углавном заснованих на једнодимензионом приступу није уопште одговарајућа. Коришћење нових модела рачунања, као што су директна нумеричка симулација, великовртложна симулација, или друге унапређене методе динамике флуида, чини се погоднијим за сложеније догађаје. НУРЕСИМ интегрисани пројект финансиран од Европске комисије (као део ФП6 програма) био је прихваћен са циљем да учини први корак према општеевропској стандардној софтверској платформи за моделовање, бележење и обнављање рачунарских података за симулацију рада нуклеарног реактора. Нека од ових проучавања која су обављена на Универзитету у Пизи у оквиру пројекта НУРЕСИМ приказана су у овом раду. Тичу се углавном истраживања две критичне појаве повезане са нестабилношћу млаза и директном контактном кондензацијом које се догађају током ванредног хлађења језгра. Посредством ових примера, ради бољег разумевања и тачнијег предвиђања, разматрани су савремени модели и потреба за њиховим побољшањем и провером са новим експерименталним подацима.

Кључне речи: унапређени рачунарски алати, вишефлуидни ток, нестабилност млаза, директна контактна кондензација