Preliminary CFD Study of Flow Oscillations in Parallel Channels Using the Volume Of Fluid (VOF) Method

Davide Papini, Giovanni Pastore, Antonio Cammi
Politecnico di Milano – Department of Energy
CeSNEF – Nuclear Engineering Division
Via La Masa, 34 – 20156 Milano, Italy
davide.papini@mail.polimi.it
giovanni1.pastore@mail.polimi.it
antonio.cammi@polimi.it

ABSTRACT

In this work, a preliminary CFD study is presented, aimed at investigating the transient behaviour of a two-component system experiencing flow oscillations. The addressed system comprises two parallel channels, and “parallel channel” proper boundary condition – involving the same pressure variations across the channels – is provided by the connection with common upper and lower headers. A closed 2-D domain is considered, filled with saturated water and saturated steam treated as two separated fluid components. Calculations are performed using the FLUENT code, and the Volume Of Fluid (VOF) method is selected as multiphase flow model. Liquid phase and vapour phase are perfectly separated, heat generation and heat transfer in the system are neglected, and the onset of oscillations is provided by imposing unbalanced initial conditions in terms of gravitational pressure drops.

The system behaves as a damped non-linear oscillator, and the performed simulations allow evaluating the oscillation frequency. Moreover, a picture of the fluid-dynamics involved is obtained, showing a close relationship between the oscillatory behaviour of the system and the time evolution of fluid vortices, forming near the channel ends. Finally, a sensitivity study is carried out in order to assess the consistence of the results, as well as the influence of the operating conditions (pressure) on the system behaviour.

1 INTRODUCTION

Computational Fluid-Dynamics (CFD) codes are increasingly adopted for the analysis of critical issues concerning the operation and safety of various industrial applications, since they provide a detailed insight into the physical mechanisms involved. For instance, prediction of flow oscillations in two-phase systems is of interest to the design of steam generators and boiling water nuclear reactor cores.

In this framework, a simple application of the FLUENT code [1] to the transient analysis of flow oscillations in parallel boiling channels is presented in this work. In particular, the Volume Of Fluid (VOF) method is applied to track the time evolution of a flow rate oscillation conveniently induced, and the effectiveness of the simulation is investigated. To this end, a suitable setup of the code (reference setting) is established, and a sensitivity study is carried out, which involves the discretization method for momentum and density, the initial conditions and the operating pressure.

A description of the considered case, as well as of the adopted numerical model, is provided in Section 2. The results are presented in Section 3, in terms of water-steam interface time evolution and qualitative fluid-dynamic behaviour of the system.
2 MODELLING

2.1 Case description

Two parallel boiling channels with typical Boiling Water Reactor (BWR) dimensions are considered, and equal-pressure-drop boundary condition is provided by the connection with common upper and lower headers, forming a closed system (Figure 1). A two-dimensional (2-D) domain is considered for calculations, and the fluid-dynamics of the system is analysed through the following hypotheses:

- The liquid and vapour components (saturated water and saturated steam) are perfectly separated (two-phase region is absent). A sharp water-steam interface is therefore assumed to represent the oscillating boiling boundary [2, 3], given the impossibility of reproducing phase transition phenomena by means of the FLUENT code.
- Fluid flow is supposed to be laminar.
- The oscillatory behaviour is induced through unbalanced initial conditions, i.e. the initial position of the water-steam interface is imposed to be different in the two channels at the beginning of the transient, so that gravity effects lead to oscillations due to density difference between the two phases. This practical approach allows studying a well defined flow oscillation under “parallel channel” boundary conditions (i.e., the same pressure variations – $\Delta P(t)$ – across the two channels).
- Heat generation and heat transfer in the system are neglected (adiabatic case).

![Figure 1: System geometry (not in scale, dimensions expressed in cm)](image)

2.2 Numerical setting

In the present work, simulations are carried out by means of the 6.3 version of the FLUENT code [1]. A structured mesh composed of 80556 quadrilateral elements (81978 nodes) is used (Figure 2). In the following, the reference setting is described, while variations introduced for the purpose of sensitivity study are discussed in Section 3.

The Volume Of Fluid (VOF) multiphase model [1] is adopted, which is a surface-tracking technique designed for two (or more) immiscible fluids (or phases) where the position of the interface between them is of interest. In the VOF model, a single set of momentum equations is shared by the phases (same velocity field), and the volume fraction of each phase in each computational cell is tracked throughout the domain.

The VOF formulation relies on the fact that the different phases are not interpenetrating. For each phase added to the model, a further variable is introduced, which is the volume fraction...
Figure 2: (a) Mesh view and (b) mesh detail in the zone around the junction between Channel 1 and Upper Header of the phase in the computational cell. The tracking of the interface is accomplished by the solution of a continuity equation for the volume fraction; if the volume fraction of the $q$th phase in the cell is denoted as $\alpha_q$, this equation has the following form:

$$\frac{1}{\rho_q} \left[ \frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q) \right] = \frac{1}{\rho_q} \left[ S_{\alpha_q} + \sum_{p=1}^{n} \left( \dot{m}_{pq} - \dot{m}_{qp} \right) \right]$$

(1)

where $\rho_q$ is the density of the $q$th phase, $t$ is the time, $\vec{v}_q$ is the fluid velocity vector, $S_{\alpha_q}$ is the source term (mass source for the $q$th phase) by default set to zero, $\dot{m}_{pq}$ is the mass transfer from phase $p$ to phase $q$, and $\dot{m}_{qp}$ is the mass transfer from phase $q$ to phase $p$.

The volume fraction equation – Eq. (1) – is not solved for the primary phase (which is steam in the present work); the primary phase volume fraction is computed basing on the following constraint:

$$\sum_{q=1}^{n} \alpha_q = 1$$

(2)

The pressure-based solver is used, which is required when using the VOF model, and a first-order implicit time-stepping formulation is chosen for time discretization. As concerns the solution parameters, the following selection is made [1]:

- Body force weighted discretization method for pressure.
- Second order upwind discretization method for momentum and density.
- Geometric reconstruction scheme for interface representation and face fluxes computation.
- PISO (Pressure-Implicit with Splitting of Operators) algorithm for pressure-velocity coupling.

The typical BWR reactor operating pressure (7 MPa) is considered, and operating temperature is set to 286 °C, which is the saturation temperature of water at the considered pressure. The operating density is specified to be the density of the lightest phase (a reference value of 36.525 kg/m$^3$ is considered, that is the density of steam at 7 MPa). The properties of saturated water and saturated steam in the aforementioned operating conditions are obtained from NIST REFPROP Database [4]. In accordance with literature applications of VOF techniques [3], liquid phase is assumed incompressible while vapour phase is assumed with varying density. Steam is here designated as the primary phase, and its compressibility is...
accounted for by means of the compressible ideal gas approximation. The surface tension coefficient is set to 0.017633 N/m [4].

3 DISCUSSION OF THE RESULTS

The unbalanced initial conditions and the presence of gravity force lead to oscillations in time due to the different gravitational heads. In the reference setting, the initial position of the water-steam interface (measured from the bottom of the Lower Header) is 4.70 m for Channel 1 and 1.50 m for Channel 2, corresponding to a gravitational driving force of about 22 kPa (Figure 3). The simulation is run for 7 s overall.

With a good degree of approximation, the condition of equal-pressure-drop across the channels is caught (Figure 4). Pressure variation along the channels is around 15 kPa for the reference setting, and differences between channels are below 2050 Pa (about 15%), which can be ascribed to local effects occurring near the channel ends, as well as to compressibility effects of steam component within the closed domain.

The oscillatory trend of the water-steam interface position is shown in Figure 5, both for Channel 1 and Channel 2. The unstable channel behaves as a non-linear oscillator with a frequency of 0.33 Hz (period of about 3 s), and the oscillations in the two channels occur in counterphase. Damped oscillations are recognized in consequence of energy dissipation.

Figure 3: Filled contours of water (in blue) and steam (in yellow) volume fractions at the starting instant (reference setting)

Figure 4: Pressure variation across Channel 1 and Channel 2 as a function of time (reference setting)
Flow velocity vectors after 0.8 s (i.e., near instant B indicated in Figure 5) are shown in Figure 6. At this time (about the end of the first quarter-period of oscillation), flow velocity inside the channels reaches the maximum value, approximately. It can be noticed that the CFD simulation associates the oscillatory behaviour of the system to the formation of localized high-velocity structures (vortices) close to the junctions between channels and headers. Vortex formation is interpreted as a consequence of the abrupt change in section in correspondence of the header-channel junction. In Figure 7, vortex structure near the lower end of Channel 2 is depicted at different times. The local increase of fluid velocity and the formation of a dead zone downstream the junction can be easily appreciated; the development of a vortex characterized by a dominant upward velocity component during the first quarter-period of oscillation is shown.

Figure 8 is representative of the state of fluid motion after 1.5 s (about after the first semi-period of oscillation, i.e., near instant C indicated in Figure 5), approaching the first inversion of flow direction in the channels. At this time, the downward velocity component is becoming dominant in the vortex near the lower end of Channel 2 (Figure 8-b), reflecting the inversion of flow direction in the whole channel, and a specular behaviour is observed in Channel 1. Hence, a close relationship is found between global and local behaviour of the system. Analogous fluid-dynamic details are observed during the subsequent oscillation periods.
Recent numerical studies [5, 6] indicate that the peculiarities of some instability phenomena occurring in boiling channels can be simply inferred in terms of the oscillation of the boiling boundary (basic assumption in the present work), suggesting that the present CFD approach may be addressed in the future to the study of boiling channel instabilities. Moreover, Ambrosini and Ferreri [5] confirmed by means of the RELAP5 code that flow oscillations in a boiling channel have the main effect to increase and decrease local pressure drops at inlet/outlet, as the present simulation shows and explains as a result of energy losses associated to the formation of fluid vortices in the edge zones.

3.1 Sensitivity study

To assess the consistence of the results above discussed, as well as the role played by the operating conditions, a sensitivity study is carried out. In particular, four modifications with respect to the reference setting are tested:

- First order upwind discretization method for momentum and density (first order upwind case).
- Different initial conditions, to assess the insensitiveness of the fluid-dynamic behaviour of the system to the unbalance between the positions of the water-steam interface in the two channels. An initial driving force of about 7 kPa is considered (lower unbalancing case).
- Different operating pressures (2 cases), to quantify the effect of pressure on the system behaviour. Operating pressures of 1 MPa (low pressure case) and 15.5 MPa (high pressure case) are considered.
Flow velocity vectors after 0.8 s (about the end of first quarter-period of flow oscillation, i.e. near point B of Figure 5) are reported in Figure 9 for the first order upwind case. It can be noticed (Figure 9-b) that the computed solution of the velocity field is affected by high diffusivity. The inaccuracy of the calculation is confirmed by the monitoring of the residuals (not shown here). Despite the higher computational efforts, second order upwind discretization schemes are therefore mandatory in view of a proper calculation of the fluid velocity field for the considered problem.

The simulation of the lower unbalancing case is consistent with that of the reference setting, both in terms of fluid-dynamic behaviour of the system and oscillation frequency, giving confirmation to the insensitiveness of the results to the chosen initial conditions.

The operating pressure plays a role in determining the operating density of the two phases, hence it affects the oscillation frequency through different masses entrapped in the closed domain and different pressure drops. The influence of the different pressures on the oscillation frequency of the water-steam interface position is shown in Figure 10 for Channel 1 (the behaviour of Channel 2 is symmetric with respect to Channel 1). In the low pressure case (red curve), pressure variation ΔP(t) across the channels (not shown here) is increased with respect to the reference setting, making the system stiffer and thus its oscillation frequency higher (in similarity with a spring-mass system). In the high pressure case (green...
\[ \Delta P(t) \text{ is lower, and the system is more relaxed (lower frequency).} \]
The comprehensive effects are however fairly small, being the most evident result the increase of oscillation period from 3.0 s to 3.2 s at 15.5 MPa (about 6% variation). Hence, the oscillating system behaves as a non-linear oscillator in which the mass enclosed in the domain and the pressure drop across the channels contribute in defining the frequency of oscillations [5].

4 CONCLUSIONS

In this work, a preliminary CFD study has been presented, aimed at investigating the transient behaviour of a two-component system experiencing flow oscillations. The addressed system comprises two parallel channels, and specific “parallel channel” boundary condition involving the same pressure variations across the channels is provided by the connection with common upper and lower headers. Calculations have been performed using the FLUENT code, and the Volume Of Fluid (VOF) method has been selected as multiphase flow model.

The system behaves as a damped non-linear oscillator, and the performed simulations have allowed demonstrating the capability of the VOF method in tracking the water-steam interface motion. Moreover, a picture of the fluid-dynamics involved has been obtained, showing a close relationship between the oscillatory behaviour of the system and the time evolution of fluid vortices, forming near the channel ends.

A sensitivity study has been carried out, which has allowed drawing the following conclusions:

- The first order upwind discretization method for momentum and density proves to be highly diffusive, hence not suitable for an accurate calculation of the velocity field.
- The fluid-dynamic behaviour of the system and the oscillation frequency do not depend on the initial conditions.
- By influencing the operating density of the two phases, the operating pressure affects the oscillation frequency. In this respect, the two oscillating channels have been characterized as non-linear oscillators in which the mass entrapped in the closed domain and the pressure drop across the channels (both functions of operating pressure) contribute in defining the frequency of oscillations, in similarity with the roles played by the mass and the elastic modulus in a spring-mass mechanical system.

This work represents a preliminary attempt to address the transient dynamic behaviour of a system containing a mixture of liquid and vapour by CFD, and in particular by VOF methods to track the motion of water-steam interface. The obtained results on the fluid-dynamic details of the simulated system could inspire further work on the subject. Within this respect, applications of VOF techniques are not foreseen in the near term to cases with heat generation and heat transfer. Indeed, the effectiveness and accuracy of this multiphase flow model have been preliminarily proven for simulations of adiabatic flows of water-steam mixtures. This very basic work can provide a useful background for applications to more complex systems, aimed e.g. at the prediction of pressure losses (which directly affect flow oscillation dynamics) in a channel with liquid-vapour inlet at the bottom. Moreover, simple test-cases could be built on this basis, and represent useful comparison tools to validate analytical treatments and empirical correlations for the main system parameters.

ACKNOWLEDGMENTS

The Authors wish to thank Prof. Fabio Inzoli (Politecnico di Milano) for the fruitful discussions and the very helpful technical support.
REFERENCES


