BUOYANT FLOW INVESTIGATIONS ON GRAVITY CURRENTS FOR OPENINGS IN VARIOUS CONFIGURATIONS IN AN ENCLOSURE

Pavan K. Sharma, B. Gera, R.K. Singh
Reactor Safety Division, Bhabha Atomic Research Centre, Mumbai, INDIA-400085
E-mail of corresponding author: pa1.sharma@gmail.com

ABSTRACT

In the complex thermal hydraulics codes developed for fire, reactor and containment safety the junctions in the multi-compartment geometries are often modeled as uni-directional junctions and some construct of flow coefficient. However, certain large vertical size junctions are known to depict bi-directional flow behaviour under specific circumstances due to presence of buoyancy induced gravity current. Similarly a horizontal kind of opening depicts oscillatory and unstable kind of behavior. A CFD based computer code FDS was used to extend an earlier reported lattice Boltzmann study[1] in laminar restriction of simulation of gravity current in various opening configuration to a turbulent one. Numerical simulation is directed to monitor the gravity current from outside to the compartment through opening and resulting flow structures are quantified in terms of the dimensionless time for the gravity current to reach the rear wall under different opening geometries versus Rayleigh number. The author also proposes to use the available effective buoyant height for calculation of Rayleigh No. The paper presents the details of the analysis, the results obtained, and comparison with the earlier reported work. Detailed investigations have been carried out to understand the bi-directionality and oscillations of a junction by analyzing studying the outgoing hot air flow (flow coefficient) and incoming cold air (flow coefficient). These studies are very useful so that the findings can be applied to the fire/containment thermal hydraulics analysis codes for these separate effect improvements.

INTRODUCTION

The presence opening plays an important role which serves as both an inlet and outlet. Transport of gaseous mixture across rooms through openings of various sizes and shapes is a common phenomenon encountered in normal ventilation of buildings (single side ventilation) or in case of fire in buildings, a large opening in case of solar cavity receiver, tunnels etc. Even in the context of NPPs, following LOCA the steam-air mixture spreads in the containment building as a result of gas-mixture movement through intercompartmental openings. The issue have more relevance affecting the buoyancy induced mixing in transport of hydrogen in multicompartment enclosure. The complex multi-component geometry of the containment building is often modeled as a network of volumes (to represent compartments) interconnected by junctions (to represent opening across compartments) in the containment thermal hydraulic analysis following LOCA. Almost similar approach is also followed in application of zone model based fire analysis codes to study the consequences and spread of fire and smoke in the buildings. In most of the codes developed so far, for both the above mentioned applications, the junctions are primarily modeled as simple orifice type obstructions wherein application of simple Bernoulli equation [2] or one dimensional momentum equation [3] and [4] is employed to obtain the flow through them as a function of pressure drop. Such treatment of junctions, irrespective of their sizes and orientations presumes them as acting uni-directionally at any time. The flow equations assumed a well-mixed fire environment with uniform temperature inside a compartment with a small vent opening (lumped parameter formulation) [5]. For a large opening the outflow generally predicted by single equation is less. The determination of flow energy loss coefficients in few other situation for a nuclear reactor is of major interest to the nuclear power industry for safe and reliable operation [6]. Existing experimental flow factors are empirical and sometimes are inconsistent. Off late, there are enough experimental evidences [7] to show that most of the large size junctions depict simultaneously a bi-directional flow. At openings the streamlines cannot instantaneously change direction and shift the section of minimum area (vena contracta ) [8] and the discharge coefficient accounts for this decrease in cross-section [9]. Earlier experiments by [10] and [11] suggested the discharge coefficient of 0.68 (inflow and outflow). Steckler et al [12] from their compartment fire experiments (Presently simulated using CFD) reported inflow coefficient of 0.68 and the outflow coefficient as 0.73. Full scales testing for masonry veneers have shown a flow coefficient as 0.65 by [13]. Yii et al. [14] gave a model based on turbulent entrainment, rather than pressure difference. A new formula for fire-induced wall vent flow rate is developed based upon a theoretical derivation and mathematical fit to data by Quintiere and Wang [15]. A different vent flow equation has been suggested by Guigay et al. [16] in which the flow coefficient effect is replace by use of
two flow correction coefficient, one taking care of uneven distribution of velocity and the other one effect of mixing and entrainment. With the advancement in understanding of behavior of such flow junctions and their modeling methodology, it may be possible to improve the accuracy of code predictions further.

With the above objective in mind, numerical investigations have been carried out to predict the behaviour due to various opening configuration of an enclosure. Phenomenologically, an assumed fire in the enclosure causes generation of combustion-products, which along with entrained air try to escape through the side opening due to strong convective and buoyant flows. The opening being large in size, outgoing hot gases do not occupy the entire cross section, thereby permitting fresh air to enter the enclosure through the remaining area, thus establishing the complicated flow behaviour in the junction. The position of neutral axis may keep shifting up or down or even vanish depending upon the conditions in the enclosure. An attempt is made to understand the basic buoyancy induced gravity current flow associated with bi-directional/unstable behaviour of the junction. A CFD code Fire Dynamics Simulator (FDS) has been used for carrying out the analysis. The not only monitor the temperatures in the enclosure, but also detailed flow distribution transients through the junction. These were studied carefully to observe the uni/bi-directional or unstable behaviour as a function of time. The calculations carried out for four different initial temperature of the enclosure. Attention was focused on the dimensionless time for the gravity current to reach the rear wall under different opening geometries versus Rayleigh number.

The present paper describes in detail the method of analysis and results of the case study of a fire in an enclosure, which helped to understand the bi-directional behaviour of a typical large junction. The studies performed and reported in the paper are a prelude to the effort to develop a bi-directional model for flow through large openings. All the zone model software uses the flow coefficient values based on earlier experiments. The findings of the present work may be used for improving the containment analysis code CONTRAN as well as the multi-compartment fire analysis code in terms of providing the realistic values of flow.

<table>
<thead>
<tr>
<th>Case</th>
<th>Size</th>
<th>No. of grids</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30*30</td>
<td>200<em>1</em>200</td>
</tr>
<tr>
<td>2</td>
<td>9*9</td>
<td>60<em>1</em>60</td>
</tr>
<tr>
<td>3</td>
<td>3*3</td>
<td>30<em>1</em>30</td>
</tr>
</tbody>
</table>

**DESCRIPTION OF THE GEOMETRY ANALYSED**

For meeting the objective of studying, a two dimensional domain was selected based on earlier reported based on a numerical study conducted by Steckler et al. [12]. A compartment, shown in Figure 1, with the end and ceiling opening geometries is considered for simulation. The rectangular box with real lines denotes the closed compartment, whereas the rest of the domain is the outdoor space. The direction of the gravity is plumb to the bottom line. The grid origin is set at the lower left corner. The symbols u and v are the x and y components of the velocity, respectively, and the initial value of u and v is set to be 0. The opening geometries, shown in Figure 2,
include the full end opening, the upside-slot end opening, the middle-slot end opening, the downside-slot end opening, and the slot ceiling opening. The three different sizes of the enclosure were simulated as depicted in table 1 for all types of opening.

**FIRE DYNAMICS SIMULATOR**

Fire Dynamics Simulator (FDS) is a field model based software developed by NIST (USA) and has built-in dedicated computational fluid dynamics (CFD) model to describe and analyse fire in multiple compartment geometries. The code solves numerically the set of governing equations of mass, momentum (Navier-Stroke) and energy conservation appropriate for low-speed, thermally-driven flows of multispecies gas mixture to describe the smoke and heat transport arising from fires. The details of formulation of the equations and the choice of numerical algorithm available are contained in a companion document, called Fire Dynamics Simulator – User’s Guide (McGrattan et al. [17]). Fire Dynamics Simulator has two modules, the first, called simply FDS, is a Fortran 90 computer program that solves the governing equations described in elsewhere (McGrattan et al. [18]) and second, called SMOKEVIEW which is an OpenGL graphics program that allows one to visualize the results. All of the input parameters required by FDS to describe the particular scenario of interest are inferred via one or two input text files created by the user, in the format described in the input manual. A detailed systematic validation (McGrattan et al. [19]) and verification (McGrattan et al. [20]) exercise for the FDS hydrodynamic model can be referred to.

**METHOD OF ANALYSIS**

Two-dimensional analysis has been carried out using code FDS for the geometry under consideration (Fig. 2) which was discretised in a fine, structured mesh comprising of about 1.5 lakhs cells. While all the walls of the enclosure are considered adiabatic in the analysis, the atmospheric pressure boundary conditions are deployed just outside the large opening of the enclosure. The buoyancy source was modeled as a initial high temperature in the enclosure. A total of 72 simulations (Four buoyancy strength X five opening configuration X three types of enclosure sizes) have been carried out to qualitatively and quantitatively understand the complex gravity current structures.

For each simulation the dimensional less time is calculated with help of characteristics time

\[ t^* = t \left( \frac{L}{\sqrt{\beta g \Delta T H}} \right) \]

\( t \): the time taken by the gravity wave to reach at the rear wall.

**RESULTS AND DISCUSSION**

In this work only fine mesh simulations are presented after the mesh sensitivity study. In the initial phase sometimes referred to as the ‘slumping phase’ the current develops and forms a head. Somewhat later, the gravity current reaches a second, self-similar phase, where there is a buoyancy-inertia balance. After sufficient time the flow will enter a final phase, in which it is governed by a buoyancy-viscous balance. Figure 3 shows the slumping phase (upper figure) and later phase for the gravity current to reach the rear wall (lower figure) under different opening geometries. Vortex shedding occurs at the end of the gravity front and induces vortex formation in the tail and generation of rotating vortices in first four cases. The transient results were post processed for instantaneous two-dimensional divergence of flow and it was found that in side upper and middle slot opening case the enclosure left and right have the counter rotating (clockwise and anticlockwise) vortex where as in other three cases the flow rotation was in clockwise direction only (clearly indicates that dense fluid is supplied from the centre of the head to the leading edge by two main counter-rotating rolls). The dimensionless transit time of the ceiling opening is longer than that of the slot end opening, though their opening sizes are the same. Among the slot end openings, the dimensionless transit time of the down slot opening is much longer than the others, while this value is similar in the last two openings. The \( t^* \) in the full end opening is the shortest because of largest opening size. The fig. 4 depicts the summarised graph of non dimensional time for a range of Rayleigh number. Here we have calculated the Rayleigh number based on the completer height of the room which means the complete height is available for buoyancy forces. This was in line with the earlier study.
However in reality completer height is not available for the buoyant flow development in all the configuration due to flow stratification and due to configuration of opening. This aspect is very evident if we post process the flow field for various cases. Authors have thus proposed to use the Rayleigh number based on the available height for buoyancy which is determined by reducing the stratified height from the total height. The figure 5-8 depicts the non dimensional time for Rayleigh number on the basis of actual volume (height) available for the buoyancy. However this approach can not be used for ceiling slot opening configuration due to lack of available stratified height.

Fig. 3: Initial slumping phase and secondary buoyancy/viscous-inertial phase

Fig. 4: The non dimensional time as a function of Rayleigh Number where the Rayleigh number is calculated based on the complete height of the room

In a representative numerical simulation for all type of opening the transients of integrated mass flow rate of air leaving and entering the opening were monitored. The flow coefficient is defined as the ratio of mass flow rate (incoming or outgoing) to the total mass flow rate (sum of incoming and outgoing mass flow rate). The flow coefficient transients of incoming and outgoing air were thus established and are depicted in Fig. 9-13 for one temperature difference.
Fig. 5: NDT as a function of Rayleigh No. based on available buoyant height for full end opening

Fig. 6: NDT as a function of Rayleigh No. based on available buoyant height for down-slot end opening

Fig. 7: NDT as a function of Rayleigh No. based on available buoyant height for middle-slot opening

Fig. 8: NDT as a function of Rayleigh No. based on available buoyant height for up-slot opening

Fig. 9: Flow coefficient (incoming, outgoing) in case of full end opening

Fig. 10: Flow coefficient (incoming, outgoing) in case of down-slot end opening
From figure 9 to 12 the flow coefficients is oscillatory and widely of different magnitude. In case of the full end opening the variation in the flow coefficient reduces to the oscillatory mean value much earlier then other opening configuration. In case of the down slot end opening and middle slot end opening the variation in the flow coefficient remains for more time. But in case of the up-slot opening and slot ceiling opening mass coefficient remains for the higher time that means there is a bidirectional flow in these cases disturbs the flow dynamics of compartment. The reason for the up slot ceiling opening configuration is attributed to Kelvin-Helmholtz instabilities due to top and bottom density difference. This configuration have highest value of flow coefficient due to sustained unstable bidirectional flow (up/down). However the oscillations in the configuration where the density difference exist in the top and bottom part of the enclosure but the connectivity to these rooms are in sidewise(left , right) manner . This configuration results in generation of a complex, dynamic region where most of the mixing is induced by Kelvin-Helmholtz instabilities and vortex shedding (Garcia and Parsons[21]; Parsons and Garcia [22]). This process regulates the dynamics of the flow since it modifies the driving force by entraining fresh water into the current, and thus, diminishing its negative buoyancy.

**CONCLUSIONS**

The dimensionless time under the slot ceiling opening is the longest. Among the slot end openings, the dimensionless time for upside-slot end opening is highest and for downside slot opening is lowest for middle slot opening is in between. The restriction to flow is highest in upside slot opening, the lighter fluid has to overcome
more resistance as compared to downward slot opening thatswhy the nondimensional time is highest for upslot opening among the slot end openings. The flow is more oscillatory at upslot opening location as compared to other configuration of slot end openings, while there is smooth flow i.e. less oscillations at down site slot opening location. The flow through slot ceiling opening is always oscillatory for the entire duration. The full end opening shows the shortest dimensionless time in all opening geometries. In shows that the phenomena of Back draft will take place earlier in full end opening case.

REFERENCES