INVESTIGATION OF NATURAL CIRCULATION TWO-PHASE FLOW BEHAVIOUR IN HEADER MANIFOLD USING CFD CODE*

P. Gulshani
NEWTECH Services
3313 Fenwick Crescent
Mississauga, Ontario
L5L 5N1

ABSTRACT

The three-dimensional (3-D), multiphase, computational fluid dynamic (CFD) code FLUENT is used to simulated two-phase flow behaviour in a CANDU header manifold under low (natural circulation) flow conditions. This behaviour was previously inferred from experimental data. The CFD simulations reported here are being used to support these inferences and to obtain a better understanding of phase distribution in the header manifold. The simulations seem to show that the vapor-water mixture models in the FLUENT code do not capture properly phase separation in the header and proper phase branching at the header-feeder connections that have been observed in experiments at low flows. The simulations using discrete-phase model in FLUENT, which tracks the pathlines of the individual vapor bubbles in the water continuum phase, show interesting, complicated and, in some cases, unexpected bubble trajectories from the point of injection of the bubbles at a feeder connection to the other parts of the header and other feeder connections. These simulations have the potential of providing needed insight into the vapor-phase behaviour in the header and may be useful in accident analyses.

1. INTRODUCTION

Low two-phase flow conditions can exist in a CANDU header manifold under certain postulated accident natural circulation conditions (i.e., for reduced loop inventory combined with loss of forced flow). Under these conditions, vapor-water mixture in some of the feeders flowing into the header may separate in the header, and only single-phase water may flow out the header in some of the feeders (i.e., reverse flow in these feeders) as shown in Figure 1. The distribution of the vapor phase in the header affects the amount of vapor that can be entrained into a feeder with reverse water flow as shown in Figure 1. Figure 1 depicts a postulated scenario where the vapor bubbles in the mixture flowing into the header from the HS7 feeder are entrained into the reverse water flow in the HS8 feeder. This scenario was used in Refs [1,2] to estimate the extent of fuel cooling in the HS8 fuel channel under two-phase thermosyphoning conditions. In Refs [1,2], the vapor bubbles entrained into the HS8 feeder was postulated to occur, as inferred from experimental data, when the flow along the header was low at the axial

* This work was partly carried out when the author was associated with the University of Ontario Institute of Technology.

location of the HS7-HS8 feeder connections. For some postulated accident scenarios, two-phase vapor-water mixture may be injected into the header through header outlet pipe (turret). This two-phase mixture impinges on the header bottom, may undergo hydraulic jump, and the phases may separate along the header. Water may then flow through some of the feeders, vapor through the other feeders, and a mixture through the remaining feeders (refer to Ref. [3]).

It is desirable to confirm the occurrence of the vapor entrainment and/or other types of vapor-phase behaviour in the header manifold using a 3-D, multiphase CFD code. It is also desirable, as has been recommended by some industry and university members, to compare the predictions of a CFD code with those obtained from simplified two-phase flow models used in Refs [1,2,3] and in other studies.

2. CFD CODE FLUENT

FLUENT is the state-of-the-art computer program (written in C language) for modeling steady and unsteady fluid flow and heat transfer in complex geometries. It includes the pre-processors code GAMBIT for generating the geometry of the facility of interest and creating a (unstructured or structured) mesh or grid on the geometry (for discretizing and solving the governing equations). The geometry-mesh combination is then imported into FLUENT, appropriate models (namely: segregated or coupled, steady or unsteady solver, single or multiphase fluids, including separated, mixture, Eulerian models, and discrete-phase injection models, energy and momentum transfers, laminar/turbulent properties, slip, drag, lift, and buoyancy models, etc) and the associated governing equations of the physical problem and the appropriate material properties are selected, and the boundary conditions are imposed on the geometry. (In the mixture multiphase model, FLUENT treats the phases as interpenetrating continua interacting at their interfaces with the possibility of slip between the phases, and solves the mixture governing equations. In the Eulerian multiphase model, FLUENT solves the governing equations of the phases separately and they are coupled through exchanges among the phases.) The governing mass, momentum, and energy equations are then solved using an appropriate solution scheme (such as explicit or implicit finite difference, Eulerian or Lagrangian methods) and convergence criteria. The results are then processed using FLUENT post-processing capabilities.

3. GAMBIT GEOMETRY AND MESH GENERATION

The geometry for the feeder-header manifold was created using the code GAMBIT. This code provides geometrical objects of various shapes that can be combined and trimmed to form any desired geometry. The code allows generating meshes of various types on the geometry. Figure 2 show the header-feeder geometry and the associated mesh generated by GAMBIT.

4. FLUENT MIXTURE AND EULERIAN MODEL SIMULATIONS

The mixture or Eulerian models were used to determine the nature of the fluid in the feeder pipe HS8 when steady or oscillating vapor-water mixture was imposed as boundary conditions at the feeder pipes HS5, HS6, HS7, and HS9 and the header-outlet pipe, i.e., the pipe leading to the boiler in Figure 1. The results indicated that, for either the mixture or Eulerian model in the FLUENT, a two-phase mixture was predicted in the HS8 feeder at high and low flows in feeders HS5, HS6, HS7, and HS9 and the header-outlet pipe. This result disagrees with the experimental observation (Ref [2]) that a vapor-water mixture exists in HS8 only when the flow in the other feeders becomes low. Therefore, it appears that neither the FLUENT mixture nor Eulerian model properly accounts for the separation of the vapor and water phases in the header.

5. FLUENT DISCRETE-PHASE MODEL SIMULATION OF FEEDER VAPOR ENTRAINMENT

In this simulation, the discrete-phase model in the FLUENT was used to inject vapor from a feeder pipe into the header where a steady flow of water had been established, and to observe the resulting nature of the fluid in feeder HS8. First, steady single-phase water model in the FLUENT was used to establish steady water flow into the header at the feeders HS5, HS6, HS7, and HS9 and outlet water flow at header-outlet pipe. Water was predicted to flow out the header into feeder HS8 as expected. This was achieved at low and high feeder water flows. Then, for either low or high feeder water flow, unsteady flow model in the FLUENT was activated and vapor was injected into the header from each of the feeders, one at a time. Figures 3 to 9 show the resulting vapor-bubble trajectories.

Figure 3 shows the vapor-bubble trajectory predicted by the discrete-phase model in FLUENT for vapor injection into the header from feeder HS5 at high feeder water flows. Figure 3 shows that some of the injected vapor bubbles flow to the closed end of the header to the right of HS5 and become trapped there (in the FLUENT animation window, these bubbles are seen to travel endlessly in closed paths in this region of the header). Some of the remaining bubbles are entrained into HS8 feeder, and the rest of the bubbles flow to the downstream part of the header. Some of these bubbles exit the header through the header-outlet pipe and the rest follow complicated paths in the region of the header between HS8 and HS9 feeder connections. These bubble paths are determined by the drag and lift forces in the water velocity fields in the header generated by the water flows in feeders HS8 and HS9.

Figure 4 shows the bubble trajectory at high feeder water flows when the vapor is injected into the header from feeder HS7, which located directly opposite to feeder HS8. Interestingly, none of the bubbles enter feeder HS8. The header axial flow sweeps the vapor bubbles to the downstream part of the header. Some of these bubbles exit the header through the header-outlet pipe, and the remaining bubbles flow back toward the middle of the header, and then return to the downstream end of the header. These trajectories are determined by the drag and lift forces in the water velocity fields in the header generated by the water flows in feeders HS8 and HS9.

Figure 5 shows the bubble trajectories at high feeder water flows when the vapor is injected into the header from feeder HS9, which is located near the downstream end of the header. Surprisingly, the vapor bubbles flow upstream and some of them exit the header through the feeder HS8, and the remaining bubbles flow back to the downstream end of the header. Some of these bubbles exit the header through the header-outlet pipe, and the remaining bubbles flow downstream and then upstream forming repeating loops of bubbles in this region of the header. This type of bubble behaviour is determined by the drag and lift forces in the water velocity fields in the header generated by the water flows in feeders HS8 and HS9. This result shows that vapor bubbles can be entrained into the reverse-flow in the HS8 feeder from an upstream feeder (i.e. HS9) in addition to the bubbles entrained from an adjacent feeder (i.e., HS7) and downstream feeder (i.e., HS5 and HS6) as assumed in Refs. [1,2].

Figure 6 shows the vapor-bubble trajectory predicted by the discrete-phase model in FLUENT for vapor injection into the header from feeder HS5 at low feeder water flows. The bubble behaviour is similar to that at high feeder water flows in Figure 3 except that all of the bubbles exit the header through HS8 feeder and none of them flow to the downstream part of the header. This behaviour is expected because the water flow downstream of HS8 is very low.

Figure 7 shows the bubble trajectory at low feeder water flows when vapor is injected into the header from feeder HS6. Most of the bubbles exit the header through feeder HS8. The remaining bubbles flow to the downstream part of the header and exit through the header-outlet pipe.

Figure 8 shows the bubble trajectory at low feeder water flows when vapor is injected into the header from feeder HS7. Unlike the trajectory at high feeder water flows in Figure 4, most of the bubbles are entrained into the feeder HS8. The remaining bubbles flow downstream and exit the header through the header-outlet pipe. This vapor behaviour is similar to that postulated in Refs. [1,2] and as depicted in Figure 1.

Figure 9 shows the bubble trajectories at low feeder water flows when vapor is injected into the header from feeder HS9. The bubble trajectories are similar to those at high feeder water flows in Figure 5 except that very few bubbles exit the header through the header-outlet pipe. This bubble behaviour is somewhat unexpected.

6. FLUENT DISCRETE-PHASE MODEL SIMULATION OF TWO-PHASE INJECTION INTO HEADER TURRET

This case was simulated similarly to those in Section 5 except that the water and vapor phases were injected into the header outlet pipe (turret) only.

Figures 10 and 11 show the FLUENT predicted vapor bubble pathlines viewed along and from the side of the header respectively. On striking the bottom of the header, the bubbles flow to the upper parts of the header on either side of the turret vertical axis. The bubbles on the left side of the axis are trapped and circulate in closed paths in this region of the header closed-end. The bubbles on the right side of the turret axis flow to the upper part of the header. Near the middle of the header, some of these bubbles flow downward and exit through feeders HS7 and HS9. The remaining bubbles continue to flow downstream of the header. Some of these bubbles exit through feeders HS5 and HS6. The remaining bubbles become trapped and circulate in closed paths in the region near the downstream closed-end of the header. Therefore, the simulation predicts that every feeder receives a mixture of steam and water. These predicted trajectories are somewhat similar to those inferred from experiments in Ref. [3], but no feeder is predicted to receive steam only (i.e., stratified flow is not predicted) although the upper-located feeders receives more steam than those located at lower elevation along the header. It is noted that the header model used in this simulation is shorter and with fewer feeders than that used in Ref [3], and this difference contributes to differences between the simulation results presented in this paper and those in Ref [3].

7. CONCLUDING REMARKS

The FLUENT-GAMBIT simulations of the vapor-water phase distributions in the CANDU header manifold reported in this paper indicated some interesting vapor bubbles trajectories in the header, some of which may be unexpected. These trajectories are determined by the nature of the drag and lift forces in the water velocity fields in the header generated by the feeder water flows. Additional simulations and parametric surveys are needed to obtain further physical insight into the header phase distribution and determine their usefulness in accident analyses.

REFERENCES

1. P. Gulshani and H.M. Huynh, “Assessment of fuel cooling under natural circulation conditions in intact loop LOCA scenarios in gentilly 2”, 17th Annual Conference of Canadian Nuclear Society, Frederiction, New Brunswick, Canada, 1996 June 9-12.
2. P. Gulshani and H.M. Huynh, “Modeling inlet or outlet feeder voiding and fuel simulator cooling under natural circulation conditions in the RD-14M loop”, 5th CNS International Conference on Simulation Methods in Nuclear Engineering, Montreal, Quebec, Canada, 1996 September 8-11.
3. P. Gulshani, “MISSFINCH: Models for steady stratified flow in CANDU header”, 14th Annual Nuclear Simulation Symposium of Canadian Nuclear Society, Pinawa, Manitoba, Canada, 1988 April 25-26.

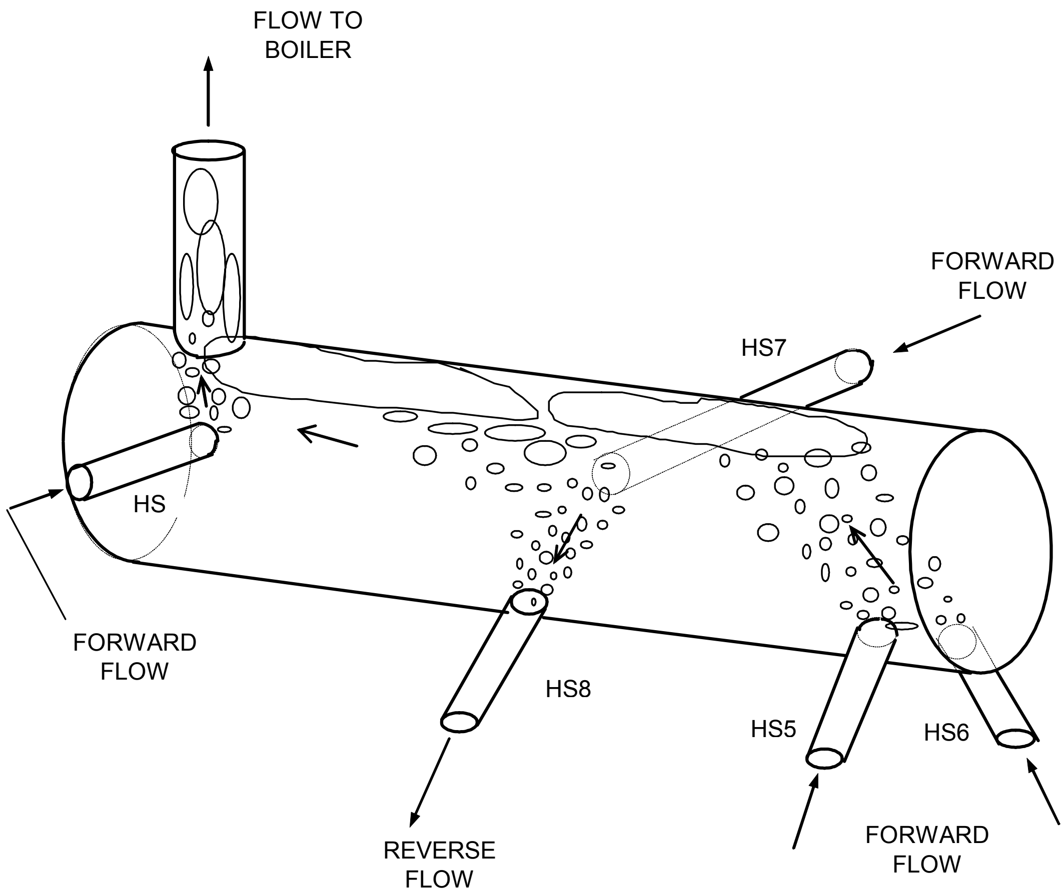

FIGURE 1: CONCEPTUAL TWO-PHASE FLOW PATTERN IN RD-14M OUTLET HEADER SHOWING BUBBLE ENTRAINMENT WITH REVERSE OUTLET FEEDER FLOW

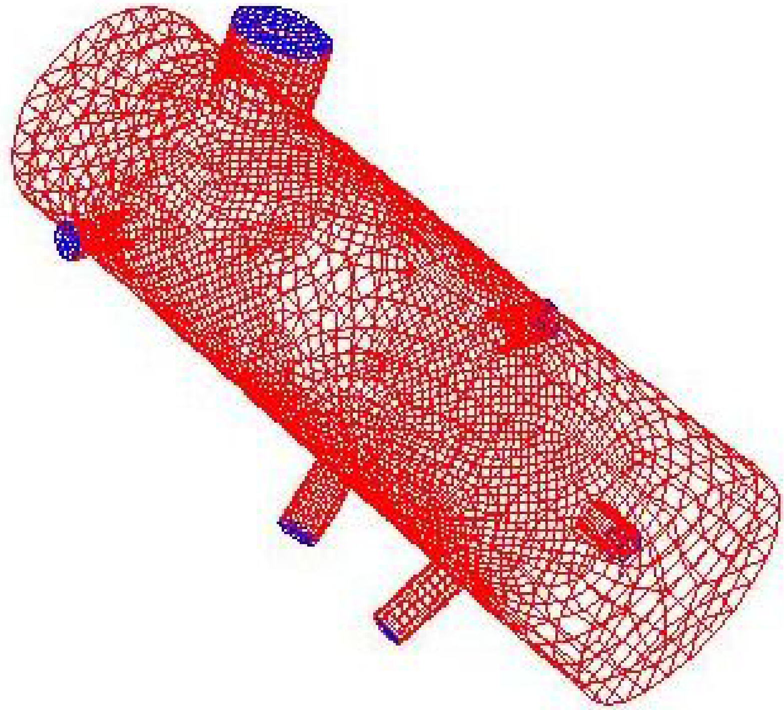

Figure 2: GAMBIT-generated header-feeder geometry and mesh

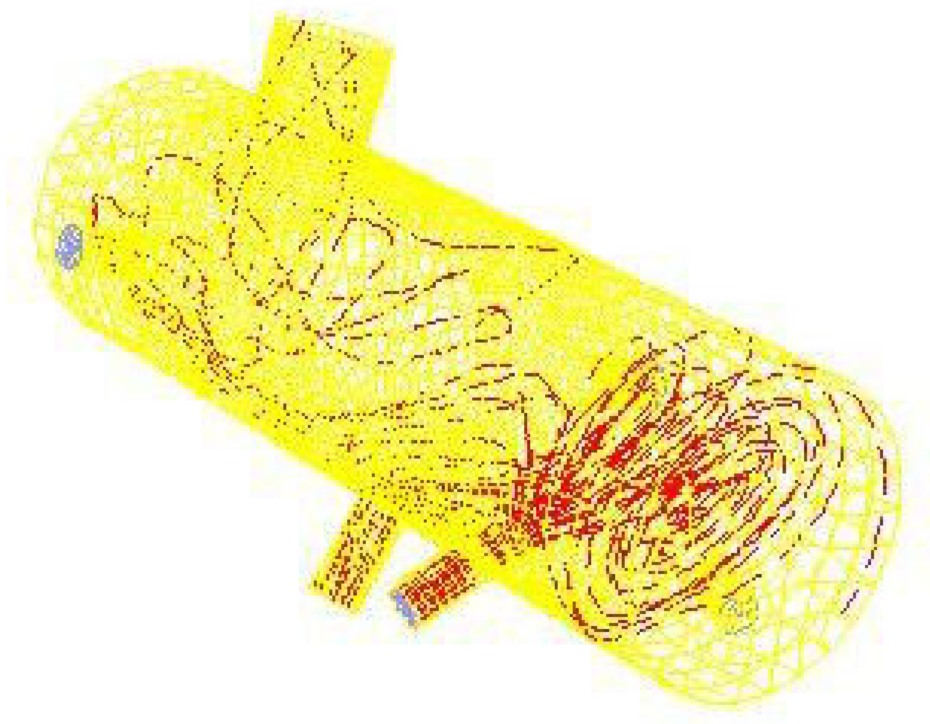

Figure 3: FLUENT-generated vapor trajectories for vapor injection at feeder HS5 and high feeder water flows

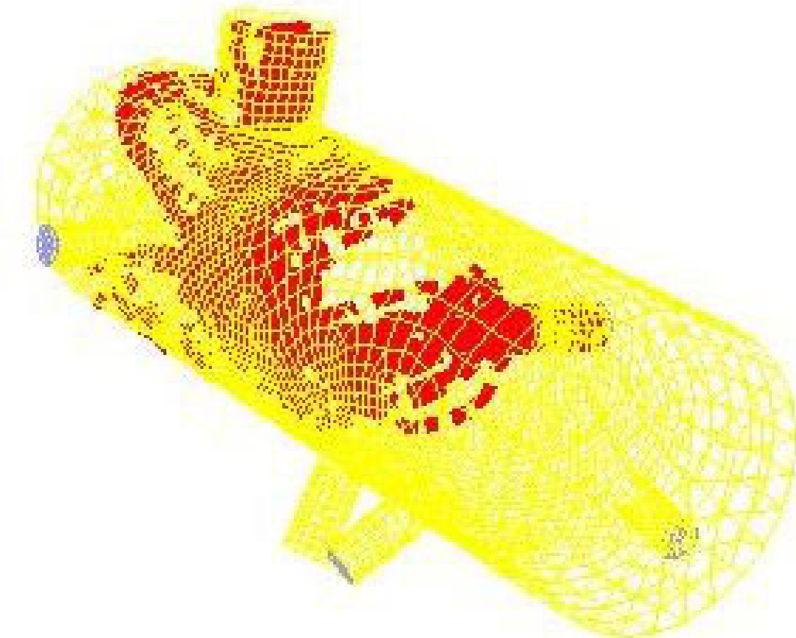

Figure 4: FLUENT-generated vapor trajectories for vapor injection at feeder HS7 and high feeder water flows

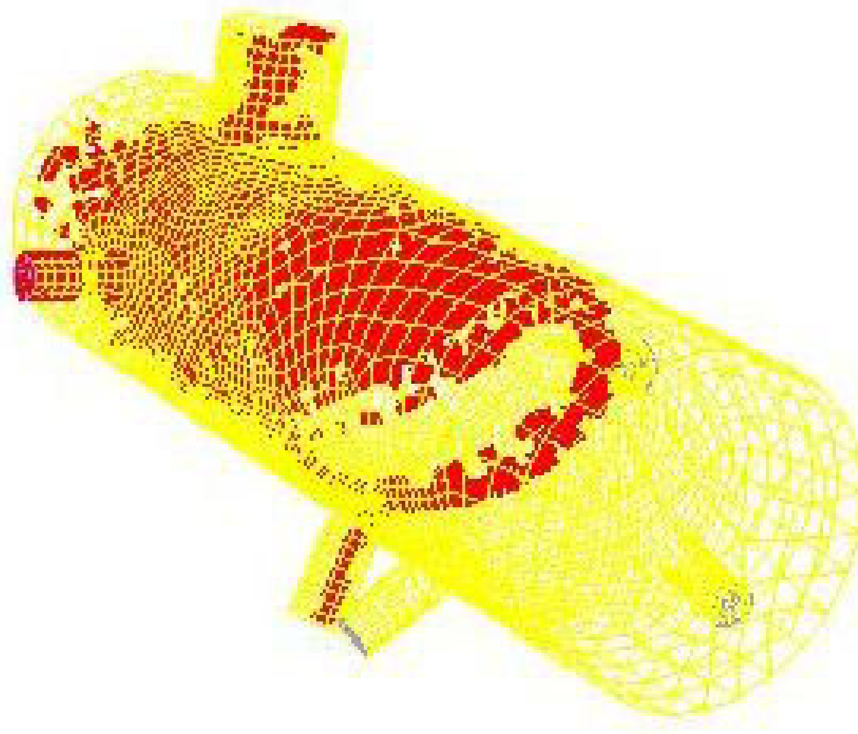

Figure 5: FLUENT-generated vapor trajectories for vapor injection at feeder HS9 and high feeder water flows

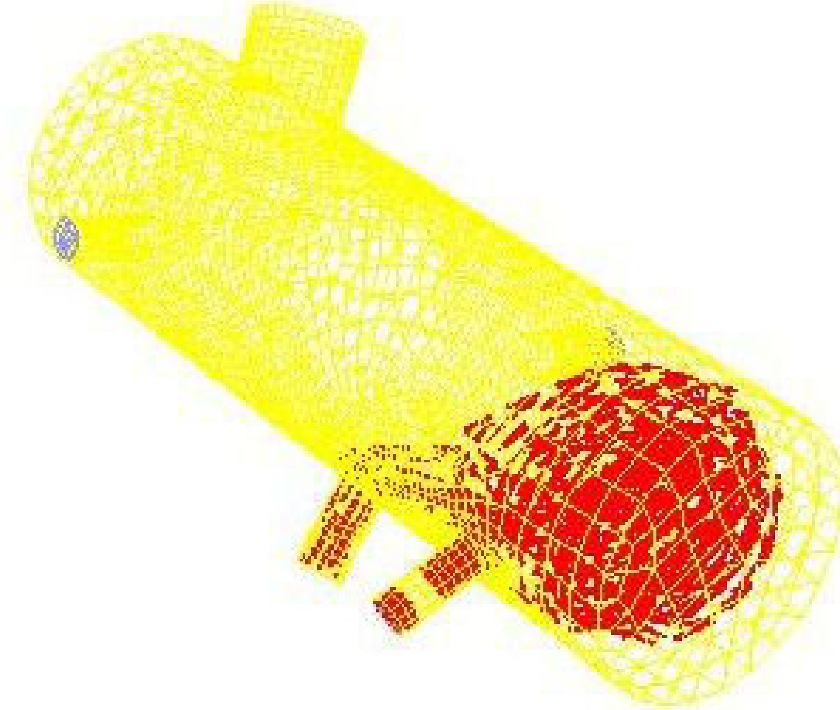

Figure 6: FLUENT-generated vapor trajectories for vapor injection at feeder HS5 and low feeder water flows

Figure 7: FLUENT-generated vapor trajectories for vapor injection at feeder HS6 and low feeder water flows

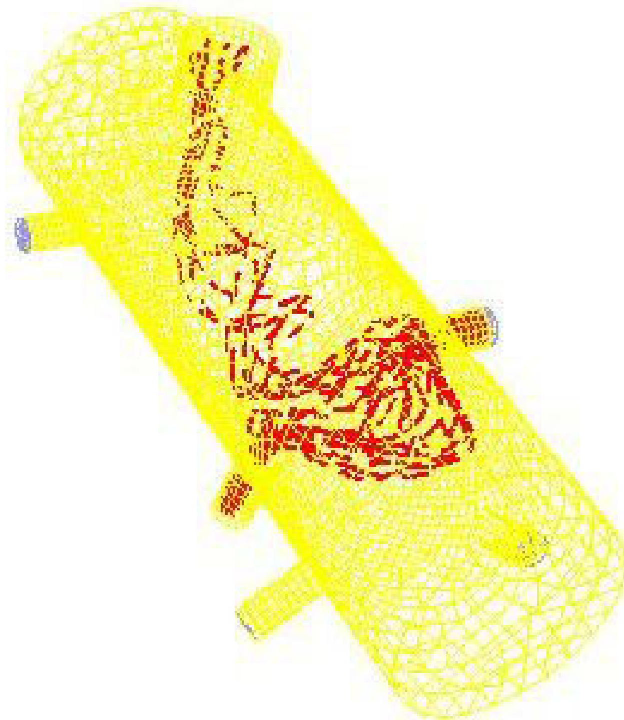

Figure 8: FLUENT-generated vapor trajectories for vapor injection at feeder HS7 and low feeder water flows

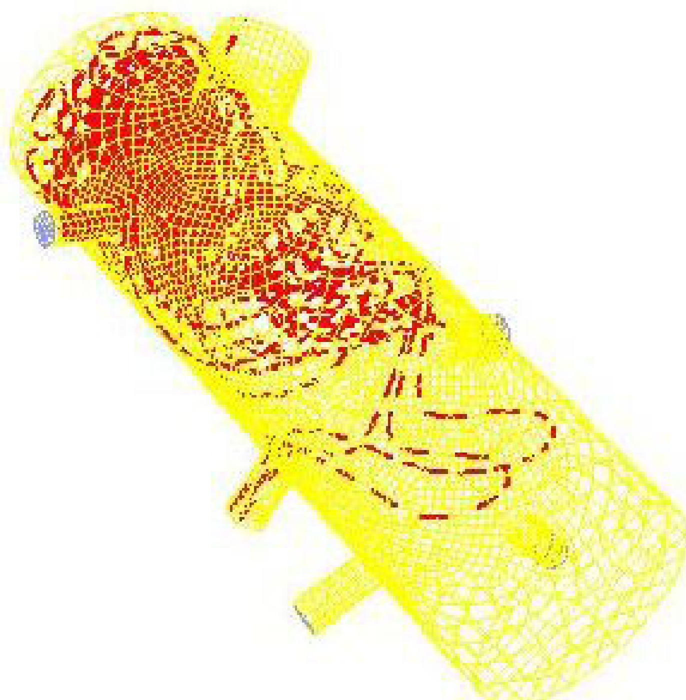

Figure 9: FLUENT-generated vapor trajectories for vapor injection at feeder HS9 and low feeder water flows

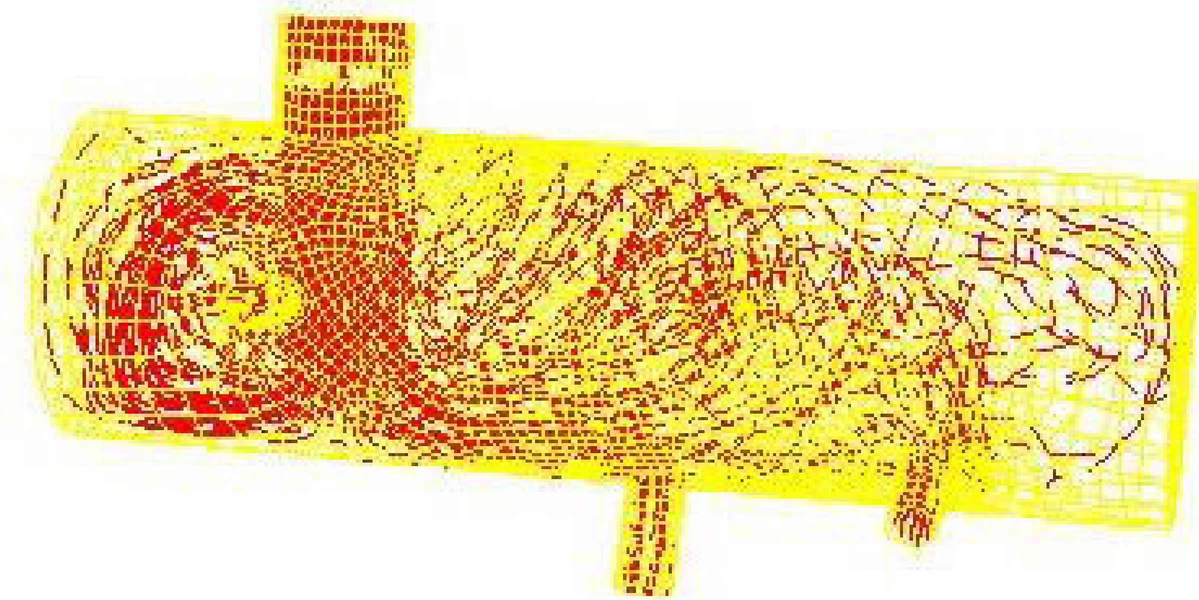

Figure 10: Vapor bubble flow pattern along header for two-phase vapor-water injection into header turret – axial view

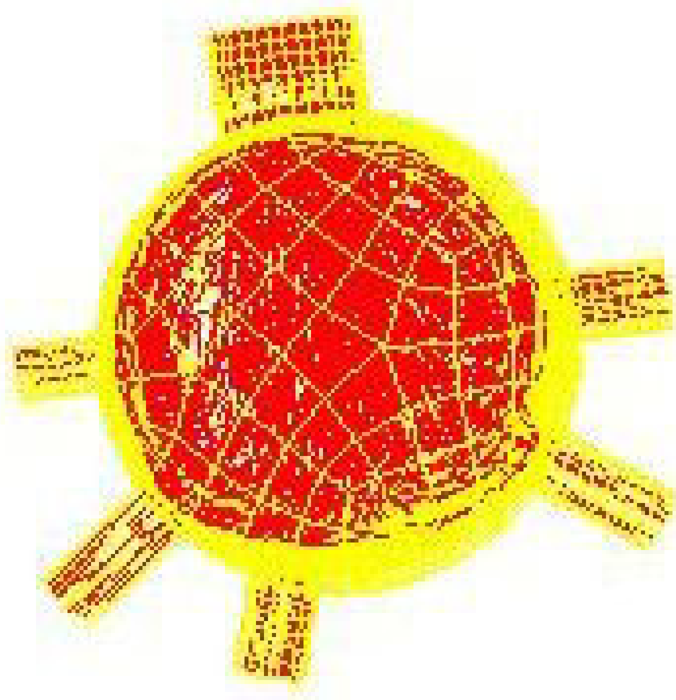

Figure 11: Vapor bubble flow pattern along header for two-phase vapor-water injection into header turret – end view