COUNTER CURRENT FLOW LIMITATION PREDICTION USING CFD

Thomas Keheley[1], Mathieu Martin[1]
Anca Hatman[2], Celine Lascar[3], Norbert Alleborn[3], Alexandre Chatelain[4]
[1] Areva NP Inc, 2101 Horn Rapids Road, Richland WA 99354, USA
Thomas.Keheley@areva.com, Mathieu.Martin@areva.com
[2]Areva NP Inc, 3315 Old Forest Road, Lynchburg VA 24501, USA
[3]Areva NP GmbH, Wetterkreuz 15 91058 Erlangen, Germany
[4]Areva NP SAS, 10 Rue Juliette Récamier 69456 Lyon, France

Abstract

Counter Current Flow-Limiting (CCFL) phenomena can occur during the re-flood stage of a Boiling Water Reactor (BWR) core after a loss-of-coolant accident (LOCA). Spray nozzles above the core inject water onto the upper tie plate region. Steam rising through the core due to the flashing of any liquid in the regions below the core or penetrating from above impedes the downward flow of the spray liquid into the core. Computational Fluid Dynamics (CFD) was used to investigate the CCFL phenomenon. Results for different air flow rates were compared with available test data and with traditional correlations. Good agreement between CFD predictions and the test results was achieved.

1. Introduction

Counter Current Flow-Limiting (CCFL) phenomena can occur during the re-flood stage of a Boiling Water Reactor (BWR) core after a loss-of-coolant accident (LOCA). Spray nozzles above the core inject water onto the upper tie plate region. Steam rising through the core due to the flashing of any liquid in the regions below the core or penetrating from above impedes the downward flow of the spray liquid into the core. In the case of a BWR, each fuel assembly is contained in a channel forcing the steam flow and the liquid to compete.

CCFL typically takes place in reduced flow area regions. Locally in these regions, the force of the rising steam flow becomes sufficient to hold a blanket of water. In reality, the extreme turbulence makes this phenomenon very unsteady and chaotic. In a fuel assembly the limiting regions are any flow area change, such as the upper tie plate spacer grids, or even the change from the full fuel rod diameter to the rod end caps. However, this work focuses on the upper tie plate; i.e., on the ability of the upper tie plate to allow water to penetrate the fuel assembly to assist in the cooling of the rods.

A literature survey showed that the majority of the work performed on CCFL is applied to steam generators and Pressurized Water Reactors (PWR) hot legs [3]. Reported work for BWR fuel assemblies deals with tests and correlations based on Wallis' work [2].

The commercial CFD code STAR-CD version 4.08 [1] was used to investigate the CCFL phenomenon and compared to the Kutateladze correlation. CCFL presents numerous challenges for CFD due to the extreme turbulence and the importance of properly capturing the interface between the two fluids. In this work, the focus is on the penetration rate of water against various air flows.

A methodology was developed based initially on a four-hole plate and then applied to standard upper tie plates. Because the early CCFL tests were performed using air and water, the methodology developed also uses these fluids. This work represents a first attempt at observing the CCFL phenomenon using CFD. The experience of this phenomena obtained with testing will be used to investigate the CFD capabilities in modeling this phenomenon.

The following will describe the correlation used as a reference for CCFL performance evaluation, then the test case using a 4-hole plate will be shown and finally the results for a BWR upper tie plate will be shown.

2. Correlation description

Wallis [2] developed a general flooding correlation for tubes of the form:

$$j_g^{*1/2} + m j_f^{*1/2} = C$$

Where $j_i^* = \left[\dfrac{\rho_i j_i^2}{gD(\rho_f - \rho_g)} \right]^{1/2}$

ρ = density

j_i = volumetric flux which is defined as mass flow rate divided by density and flow area

D = tube diameter

Wallis found that for turbulent flow *m* is equal to 1 and the value of *C* depended on the design of the tube ends and the way which the liquid and gas were added.

Sun and Fernandez [4] took the Wallis correlation and revised it to apply to BWR bundle geometries. Because the choice of *D*, the characteristic diameter, for fuel assemblies is uncertain, they proposed a characteristic length of:

$$\left[\frac{\sigma}{g(\rho_f - \rho_g)} \right]^{1/2}$$

Where, σ is the water surface tension and the subscript *f* is fluid and *g* is gas. Then the correlation becomes the Kutateladze correlation:

$$\left(K_g\right)^{1/2} + m\left(K_f\right)^{1/2} = C$$

Where,

$$K_g = \left\{ \frac{\rho_g j_g^2}{\left[g\sigma\left(\rho_f - \rho_g\right)\right]^{1/2}} \right\}^{1/2} \quad \text{And,}$$

$$K_f = \left\{ \frac{\rho_f j_f^2}{\left[g\sigma\left(\rho_f - \rho_g\right)\right]^{1/2}} \right\}^{1/2}$$

They then report that

$$\left(K_g\right)^{1/2} + \left(K_f\right)^{1/2} = (3.2)^{1/2}$$

The Kutateladze correlation is

$$K_g^{\frac{1}{2}} + mK_f^{\frac{1}{2}} = c$$

Where,

$$K_i = \left[\frac{\rho_i \left(\frac{m_i}{\rho_i A} \right)^2}{\left(g\sigma\left(\rho_f - \rho_g\right)\right)^{\frac{1}{2}}} \right]$$

K_i	*= K for the particular state (liquid or gas)*
ρ_i	*= density for the state specified in* K_i
A	*=flow area (case specific)*
g	*= acceleration due to gravity*
σ	*= surface tension of water*
ρ_f	*= density of water*
ρ_g	*= density of the vapor (in this case air)*
m	*= 1*
c	$=(3.2)^{1/2}$

3. CFD models

The design of the CFD models is based on the tests usually performed. The assessment of the different geometries performance will be based on the liquid penetration rate under different conditions (liquid flow rate, air flow rate). The amount of water penetrating through the upper tie plate for a certain amount of time is then measured to derive a penetration rate. Because of the abilities and limitations offered by CFD few changes will be made compared to the actual test set up.

The CFD model was designed to compare the flow penetration through different geometries based on the physics described in the correlation development. CCFL simulation requires a two-phase approach. Eulerian two-phase and multi-fluid (Volume of Fluid) models in STAR-CD were considered.

Eulerian two-phase modeling is widely used for modeling two-phase flows involving water and steam. In this case both species (liquid and gas) are modeled as continuous phases interacting with each other. Therefore, for each phase, the flow fields are defined and solved. Exchanges are allowed between the phases which, for instance, can allow transfer of mass between the phases (evaporation and condensation). Since both phases are treated similarly the grid does not required any specific refinement.

Volume of Fluid is an advection scheme that tracks the interface between different species. In this particular model only the interface between different non-miscible species is modeled. By computing the forces acting at the interface between two fluids, equilibrium is achieved and the interface is resolved. The grid needs to be sufficiently fine to resolve the majority of the eddies observed at the interface. Typically a finer grid needs to be achieved where the interface is expected in order to properly resolve the interface. This technique is quite suitable for flows where surface effects are important like surface tension.

Because of the presence of the surface tension, σ, in the Kutateladze correlation and the importance of the interaction between the fluids at the interface, the multi-fluid model in STAR-CD was chosen. It explicitly incorporates a surface tension model which is an important parameter to properly capture the CCFL phenomenon and it accurately captures the interactions between the phases at their interface. A contact angle model is also available to deal with the interaction between the interface and the walls.

Early CCFL correlations were developed based on air/water tests. AREVA has benchmarked its full scale steam/water tests against partial scale air/water tests for 9x9 BWR bundle geometries. Therefore, the analysis will be performed using air/water for the test fluids.

Due to the turbulence at the interface between the fluids, the calculations will be 3-dimensional. Because of use of the volume of fluid model of STAR-CD, a transient calculation is run for each case. Due to the extremely low time step applied, the domain is shortened to achieve the time-steps in a reasonable amount of time. The Courant Number is maintained fairly low (around 0.5) because of the high turbulence observed around the interface.

In order to be consistent with the test, the CCFL performance will be measured by calculating the amount of water penetrating the geometry over a certain amount of time. A penetration rate will then be deduced. For this first work, local phenomena are not studied as they would require limiting more the size of the model.

3.1 Flow conditions

For each run the mass flows of air and water need to be specified. One needs to specify the conditions so that the CCFL can be observed. CCFL only occurs at particular conditions. For instance, if the water is injected at a too slow rate then it will be carried away by a given air flow.

To specify the flow conditions of each run, the air flow was chosen and the water flow was calculated using the Kutateladze correlation. This gives the minimum amount of water required for the CCFL to happen according to the correlation. In order to speed up the run and guarantee that the CCFL will occur, the water flow rate is increased.

3.2 4-hole plate

A simple 4-hole plate is used to develop a CFD methodology for predicting CCFL and comparing it to the Kutateladze correlation. This very simple 4-hole geometry is shown in Figure 3-1. This plate was used to perform mesh and turbulence model sensitivity studies for methodology development. The CFD domain is shown in Figure 3-2. As for the test 2 inlets and 2 outlets are necessary, one set of inlet/outlet for each fluid.

A trim mesh is used over the domain, and a refined patch is performed around the region of the holes as this is where the CCFL is expected to occur since this is the smallest hydraulic area in the geometry.

The Kutateladze correlation is applied to this particular geometry at several flow (air and water) conditions. CFD calculations were performed at the same conditions and the penetration rates were compared with the correlation predictions.

For the CFD calculation, the flow crossing the mid-plane (shown in red in Figure 3-2) of the geometry (is monitored during the transient calculation. Using the trapezoidal rule, the amount of water penetrating through the domain can be evaluated using the following equation:

$$penetration = \left(\left(\frac{y_0 + y_n}{2} \right) + \sum_{i=1}^{n-1} y_1 \right) \Delta x$$

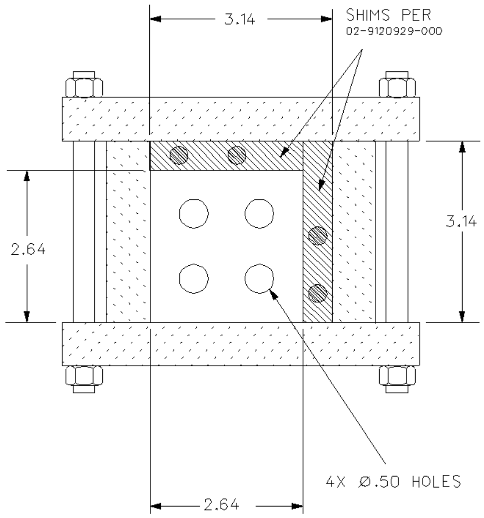


Figure 3-1: 4-hole plate geometry

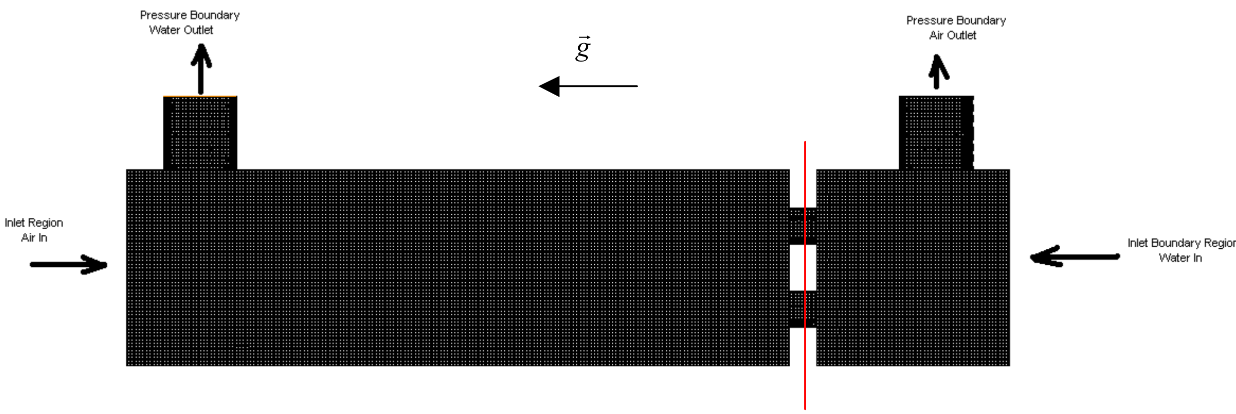


Figure 3-2: 4-hole plate CFD domain and boundaries

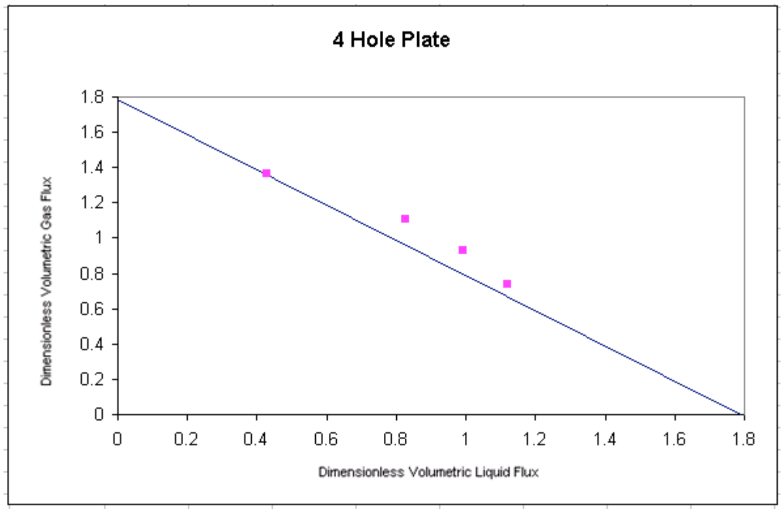


Figure 3-3: Comparison between the Kutateladze correlation (in blue) and the CFD results (in pink)

Figure 3-3 shows the comparison between the correlation and the calculated flow penetration from CFD for different flow conditions. Considering the extreme turbulence of the flow and the difficulty to resolve it the agreement achieved on this simple case was deemed fairly good.

The 4-hole plate was used to develop the methodology to calculate CCFL. Sensitivity studies were performed for grid refinement at the CCFL locations, turbulence model, and position of the boundaries. A k-ε two-equations turbulence model was used. Note that for the Multi-fluid volume, only parameters between the interface and the wall need to be defined. In STAR-CD this is defined by a contact angle at the wall that defines the transition between wetting and non-wetting contacts. This value is typically listed as a material property.

The 4-hole plate model was used because it is a simple model and can be run relatively quickly compared to a real upper tie plate. However hydraulic dimensions are consistent with the real geometry. This first test gave us some familiarity with the challenges and the parameters to be applied on realistic cases.

3.3 Upper tie plate

The upper tie plate for modern BWR designs are much more complicated than those of the early 7x7 array designs (used to develop the correlation). A typical tie plate for AREVA's ATRIUM^{TM}-10 design is shown in Figure 3-4. The methodology developed using the 4-hole plate geometry is then used on an ATRIUM^{TM}-10 upper tie plate model and compared to available ATRIUM^{TM}-10 test data and correlation predictions.

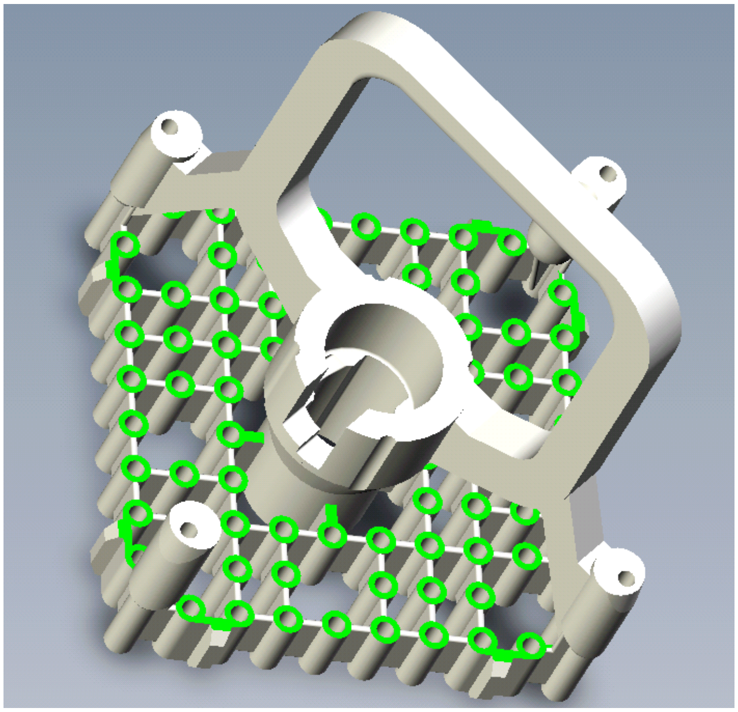
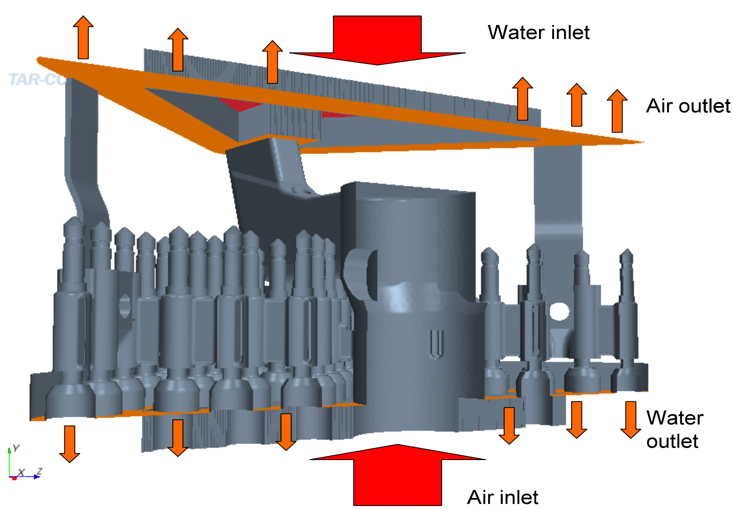


Figure 3-4: Upper Tie Plate for an ATRIUM™ type 10x10 BWR Fuel Assembly

Due to the increased complexity of the geometry compared to the 4-hole plate the level of turbulence to capture at the interface between the fluids is even more challenging. To maintain a decent amount of time for each time step the domain is shortened and cut in half. Figure 3-5 shows the domain used.



The result is shown in Figure 3-5. This graph represents the water penetrating the upper tie plate along the time. By plotting the penetration at each time one can see a transient and then a steadier regime developing. It should be noted that these observations were also made during actual testing. The initial lapse corresponds to the time for the water to hit the upper tie plate. Then the amount of water increases to reach a maximum before dropping again. A cycling phenomenon is then observed for the water penetration. The reader is reminded that this is an integrated result over a plane crossing the middle of the upper tie plate. However one can observe the net balance of the water penetrating, showing a build-up and release of the water through the upper tie plate.

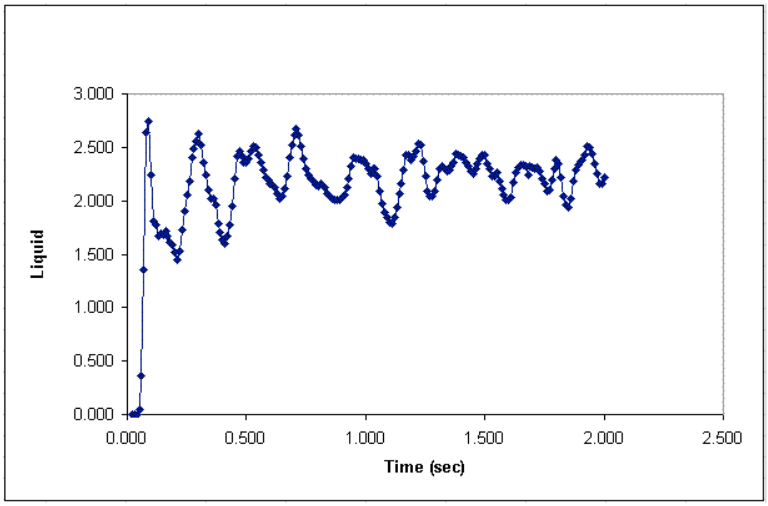


Figure 3-5: Liquid Penetration Rate (kg/s) for Air Flow at 0.005 kg/s

Figure 3-6 shows the results of the CFD calculations and the ATRIUM™-10 test results along with a plot of the Kutateladze correlation based on the ATRIUM™-10 upper tie plate flow area. As for the 4-hole plate the agreement is fairly good with both the test and the correlation. Note that CFD allowed running cases at more meaningful flow conditions than the actual test. This is why the simulated points are lower on the curve.

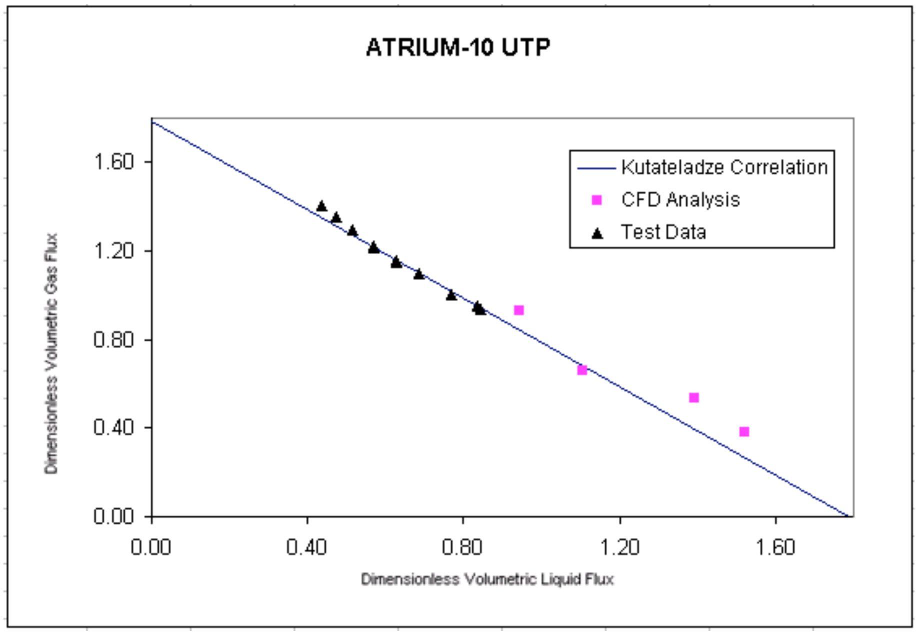


Figure 3-6: Comparison of CFD Analysis to Test Data

Figure 3-7 illustrated the turbulence generated by the CCFL phenomenon. It shows in blue the interface between air and water and it is defined by an iso-surface of the liquid fraction.

The resolution of the turbulence at the interface is between the two fluid is critical and required refined mesh where the interface was expected. The simulation shows that the falling water seems to hit the air flow and bounces back due to the resistance of the air. As explained before the air resistance is increased due to the reduction of the flow area caused by to the upper tie plate.

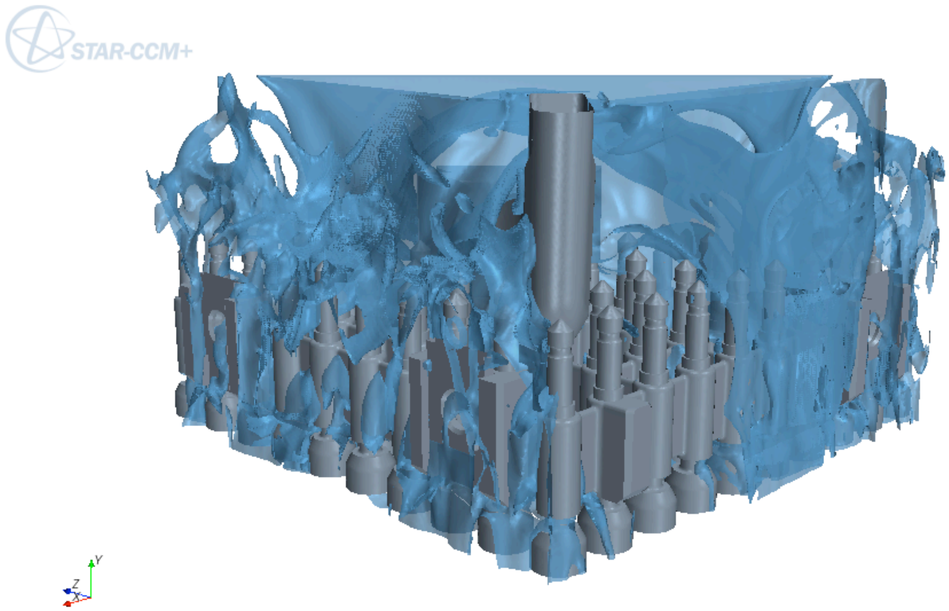


Figure 3-7: Illustration of CCFL turbulence

4. Conclusion

An investigation of the CCFL phenomenon in a BWR fuel assembly upper tie plate was performed using CFD. These first analyses showed the challenges of the simulation and were used to develop a first methodology to be applied to fuel assembly components. Good agreement to both test data and correlations were achieved when applied to BWR upper tie plate geometries.

The ability to predict the CCFL performance of a fuel assembly component is one more step towards being able to use simulation in the design and analysis of fuel assembly components. The next step will be to apply CCFL predictions tool to other geometries and designs in order to develop a methodology and a comprehensive understanding of the phenomenon. Additional sensitivity studies will need to be performed in order to better understand the importance of each parameter.

5. References

1. CCM Users Guide, STAR-CD Version 4.08, CD-adapco, Melville, NY, 2008
2. Wallis, Graham B., One-dimensional Two-phase Flow, New York, 1969
3. Deendarlianto, Christophe Vallée, Dirk Lucas, Matthias Beyer, Heiko Pietruske and Helmar Carl, Experimental study on the air/water counter-current flow limitation in a model of the hot leg of a pressurized water reactor, 2008
4. Sun, K.H., R.T. Fernandez, "Countercurrent Flow Limitation Correlation for BWR Bundles During LOCA, TRS-3: LOCA/ECCS Experiments and Analysis, pp 605-606