CFD modeling of annular flow for prediction of the liquid film behavior

MARIA CAMACHO
CFD modeling of annular flow for prediction of the liquid film behavior

MARIA CAMACHO

KTH Supervisor: Henryk Anglart
Westinghouse Supervisor: Tobias Strömgren
Examiner: Jan Dufek

TRITA-FYS 2016:52
ISSN 0280-316X
ISRN KTH/FYS/--16:52-SE
ABBREVIATIONS

CFD – Computational Fluid Dynamics
BWR – Boiling Water Reactor
CHF – Critical Heat flux
LPT – Lagrangian Particle Tracking
DPM – Discrete Phase Model
DRW – Discrete Random Walk
EWF – Eulerian Wall Film
FVM – Finite Volume Method

APPENDICES

APPENDIX 1. Boundary and/or initial conditions
APPENDIX 2. User-Defined Functions
APPENDIX 3. Deliverables

NOMENCLATURE

\[
\begin{align*}
\rho & \quad \text{density, kg/m}^3 \\
u & \quad \text{velocity, m/s} \\
S_m & \quad \text{mass source term, kg/s·m}^2 \text{ or kg/s·m}^3 \\
S_{mom} & \quad \text{momentum source term, N/m}^2 \text{ or N/m}^3 \\
p & \quad \text{pressure, Pa} \\
\tau & \quad \text{shear stress, N/m}^2 \\
\tau^{eff} & \quad \text{effective stress tensor, N/m}^3 \\
g & \quad \text{gravitational acceleration, m/s}^2 \\
q^\prime & \quad \text{heat flux, W/m}^2 \\
H & \quad \text{latent heat, J/kg} \\
F & \quad \text{force, N} \\
x & \quad \text{position vector, components in m} \\
m & \quad \text{mass, kg}
\end{align*}
\]
$V$ volume, $m^3$

diameter, m

$\gamma$ relaxation time, s

$\mu$ dynamic viscosity, Pa s

$k$ coefficient, drag: dimensionless, entrainment: m/s

$Re$ Reynolds number, dimensionless

$h$ liquid film thickness, m

$J$ superficial velocity, m/s

$\dot{m}$ mass flux, kg/s m$^2$

$\sigma$ surface tension, N/m

$e$ exponent, dimensionless

$D$ pipe diameter, m

$f$ friction factor, dimensionless

$y$ distance from the wall, m

**SUBSCRIPTS**

$v$ vapor

$f$ liquid film

$d$ droplet

$vap$ vaporization

$ent$ entrainment

$dep$ deposition

$w$ wall

$i$ interfacial

$c$ cell
ABSTRACT

Annular flow is the characteristic flow pattern at the upper part of Boiling Water Reactor (BWR) fuel assemblies. Consequently, at this part of the core, cooling occurs mainly due to the presence of a liquid film flowing on the fuel rods external surface. If the liquid film dries out, Critical Heat Flux (CHF) phenomena could be reached, causing deterioration of the heat exchange capability and compromising the safety of the operation.

A model of annular flow in a pipe was developed in ANSYS Fluent to study the liquid film behavior at BWR operational conditions. The simulation includes a two-dimensional Eulerian liquid film model coupled to a three-dimensional core model (including vapor flow and droplets) based on the Eulerian – Lagrangian approach. The simulations were performed using two different droplet diameters and validated with experimental liquid film mass flow rates. The model was also compared to a similar OpenFOAM model developed in a related project for one of the droplet diameters.

The liquid film mass flow rates obtained with the largest droplet diameter are in good agreement with the experimental ones but slightly underpredicted. On the other hand, the results obtained with the OpenFOAM model differ significantly due to a different approach used in the calculation of the interfacial shear stress.
1 INTRODUCTION

Since the vapor content in the water coolant increases while flowing from inlet to outlet of the core of a BWR, different flows regimes or patterns are developed. Consequently, the removal of heat from the fuel rods occurs due to different heat exchange mechanisms. For instance, annular flow is the characteristic flow pattern at the upper part of a BWR fuel assembly and heat is removed mainly by evaporation of the liquid film.

Annular flow is characterized by gas flowing as a continuous phase in the middle of a channel while liquid is present as dispersed droplets in the gas and as a liquid film at the walls of the surface. The liquid film mass is constantly changing due to three main interactions: deposition of liquid droplets onto the liquid film surface, liquid film evaporation on the liquid-gas interface and liquid film entrainment as droplets into the vapor flow.

When in a BWR the liquid film disappears or achieves a critical film thickness or flow rate, the dryout condition is reached. If dryout remains for a long period of time, then it could lead to CHF phenomena causing deterioration of the heat exchange capability. Owing to this, the fuel rods temperature could increase drastically and fuel damage may occur.

In the mindset of nuclear energy safety, many efforts have been made to be able to predict the dryout condition. According to Anglart’s (2013) thermal-hydraulics textbook: the overall experimental effort in obtaining CHF data is enormous. It is estimated that several hundred thousand CHF data points have been obtained in different labs around the world. More than 200 correlations have been developed in order to correlate the data. (p.153)

However, even though most of the empirical correlations are highly reliable and accurate, they are accompanied by two main drawbacks. In the first place, each one is only valid in the specific conditions of the experiments such as geometry,
pressure, heat flux, mass flows; and they can be hardly extended. And secondly, they require a considerable number of experiments which can be extremely expensive and time consuming.

Eventually, attention has turned to the development of models which can capture most of the phenomena happening and can easily be adapted to different geometries and operational conditions. Computational Fluid Dynamics (CFD) tools allow us to solve conservation equations of fluids simultaneously, coupling of different models, and many other features. The development of a complete model can be very time consuming but once set up will enable to study different scenarios.

Westinghouse Electric Sweden and the Nuclear Reactor Technology department at KTH Royal Institute of Technology have been working together in the NORTHNET project to build a tool able to predict dryout in fuel bundle geometries. The first step to achieve this goal was the development of an annular flow model in a simpler geometry, a pipe, using the open source CFD code OpenFOAM.

The annular flow is simulated using a three-field approach: vapor, droplets and liquid film; and three main interactions: droplets deposition, film evaporation and film entrainment. The liquid film is simulated as a two-dimensional Eulerian model while the vapor-droplets model is simulated as a three-dimensional Eulerian – Lagrangian model, both models are simultaneously coupled.

This master thesis project was born as a new proposal aiming at building a model of annular flow in a pipe, able to predict the liquid film behavior and eventually dryout condition by using a commercial CFD tool. The tool used for the simulation is ANSYS Fluent and the approach is the same as the one used in OpenFOAM. The model is set up to simulate only the annular flow part, it means, the development of the flow from sub-cooled water to annular flow is not considered. Furthermore post-dryout condition is not included.
This thesis report is organized in seven sections. First, the current introduction provided the background, justification and objective of the project. Section 2 contains the most relevant theory needed to understand the project and the approach implemented. Section 3 describes the case used for the validation of the model. Section 4 explains how the computational set up was developed, the ANSYS Fluent models, adjustments and boundary conditions. The results are presented and discussed in section 5 while conclusions and recommendations are stated in section 6. Finally, all the references used during the development of the report are listed in section 7.


2 THEORETICAL BACKGROUND

This part of the report describes the most relevant theory required for understanding the development of the model.

2.1 FLOW PATTERNS IN TWO-PHASE MIXTURES

Flow patterns in heated and unheated channels have been widely studied for vertical and horizontal pipes. In general, it has been observed that for a given fluid, channel geometry, pressure and flow rate; as the vapor content of the mixture increases (measured as vapor mass rate fraction and commonly known as flow quality), the liquid – vapor spatial distribution changes and different flow patterns are developed. Collier and Thome (1994) present a resume of the flow patterns experimentally observed in vertical channels with upwards flow. (p.10)

- **Bubbly Flow**: vapor phase flows as dispersed small bubbles across the liquid continuous phase.

- **Slug Flow**: vapor phase is present as larger bubbles and separated one to another by liquid slugs.

- **Churn Flow**: the large bubbles formed in slug flow are breakdown, forming a chaotic vapor flow at the center of the channel and displacing the liquid to the walls. Not clearly observed in heated channels where flow transition goes from slug to annular pattern.

- **Annular Flow**: a thin liquid film flows at the channel wall while the gas phase flows as a continuous phase at the center of the channel; liquid phase also appears as discrete droplets into the vapor continuous phase.

- **Mist flow**: liquid phase is present only as dispersed droplets across the continuous vapor phase. Only present in heated channels.
Figure 1 shows a graphical representation of the flow patterns described above.

Figure 1. Flow patterns in vertical channels with upwards flow. a, b: bubbly flow, c: slug flow, d: churn flow, e: annular flow, f: mist flow. Source: Buongiorno (2010). (p.4)

2.2 HEAT TRANSFER MECHANISMS IN BWRs

The core of a BWR can be considered as a heated channel; therefore, different flow patterns and heat transfer mechanisms for removing heat from the fuel rods are present along the height of the core. Figure 2 displays the different flow patterns and heat transfer regimes present in a vertical heated channel assuming an axially uniform heat flux.
Figure 2. Flow and heat transfer regimes in a heated vertical channel.
Source: Modified from Anglart (2013). (p.147)
- **Section 1:** The fluid enters the channel at an inlet temperature below the saturation temperature of the fluid. The fluid is present as single phase liquid flow and the heat transfer mechanism is convective heat transfer.

- **Section 2:** When the wall temperature exceeds the saturation temperature, small bubbles are formed on the heated surface (initial stages of bubbly flow pattern). Sub-cooled boiling is the heat transfer mechanism removing heat from the surface.

- **Section 3:** The point where the fluid temperature reaches the saturation temperature is called the Onset of Nucleate Boiling (ONB) identified as $x=0$ in figure 2. Up here, more bubbles are formed on the heated surface, growing and detaching developing a fully-bubbly flow pattern. As the flow quality increases, the amount of bubbles increases, bubbles start colliding and the slug flow regime is developed. The characteristic heat transfer mechanism in this section is saturated nucleate boiling.

- **Section 4:** As the flow quality increases, vapor concentrates at the center of the channel as a continuous phase and liquid is displaced to the walls forming a thin liquid film. Droplets are present in the vapor core coming from entrainment of the liquid film. The characteristic flow pattern is annular flow and it is accompanied by a change in heat transfer from nucleate boiling to forced convective heat transfer.

The transition mentioned above occurs because the film thickness decreases reducing its thermal resistance at a point in which the wall superheat (difference between the wall temperature and the saturation temperature) is so low that the wall temperature is not enough to allow the formation of bubbles on the surface and nucleate boiling is suppressed. As a result, the convective heat transfer to the liquid film surface and consequent evaporation is enhanced.
Section 5: The liquid film decreases due to evaporation and entrainment and when it dries out; post-dryout heat transfer becomes the leading mechanism. Thus, the heat transfer occurs through the evaporation of the dispersed droplets present in the vapor phase and through convective heat transfer to vapor.

The heat transfer coefficient decreases abruptly due the low heat conductivity of the vapor and the wall temperature increases drastically according to Newton’s law of cooling \( q^* = k_{\text{heat}} (T_{\text{wall}} - T_{\text{fluid}}) \). Where \( q^* \) is the heat flux, \( k_{\text{heat}} \) is the heat transfer coefficient, \( T_{\text{wall}} \) is the temperature of the wall and \( T_{\text{fluid}} \) is the temperature of the fluid.

2.3 THEORETICAL APPROACH OF ANNULAR FLOW

In this section, the theory behind the approach of annular flow as implemented in ANSYS Fluent will be described. Differences and comparisons with the model developed in OpenFOAM will be discussed in section 5. All the equations shown in this section are already implemented inside ANSYS Fluent solver unless something else is specified.

2.3.1 General

The three-field approach of annular flow treats the system as three different components: vapor, droplets and liquid film. This approach appears to be the most physically accurate representation of annular flow due to the coexistence of liquid film and liquid droplets with significant differences in velocity and flow direction. Taking into consideration that droplets are dispersed into the vapor flow, these two fields can be represented together by a multiphase model (called from this point the gas core model). The gas core model is simultaneously coupled to a liquid film model by introducing sources terms into the conservation equations of the gas core and the liquid film. Figure 3 illustrates the annular flow three-field approach.
Three main interactions between the liquid film and the gas core are taken into account in the present model, see figure 4.

- The evaporation of the liquid film because of the heat released by the fuel rods.

- The entrainment of the liquid film as additional droplets into the vapor flow.

- The deposition of droplets present in the vapor flow on the liquid film surface.

Figure 3. Illustration of annular flow three-field approach.
Source: Li and Anglart (2015a). (p.5)

Figure 4. Representation of the main interactions between the liquid film and gas core.
Source: Li and Anglart (2015a). (p.5)
There are other possible interactions when droplets impact the liquid film surface. One example is bouncing or rebound; the droplet hits the liquid film and leaves the surface relative intact but with a change in velocity. Another possible outcome is splashing, when the droplet leaves the surface in the form of several smaller droplets. According to ANSYS Fluent Theory Manual 17.1 (p.429), based on Stanton and Rutland (1996, p.783), rebound occurs after the droplets reached a critical temperature above the saturation temperature and splashing prevails when the impact energy achieves a critical value.

However, it is generally assumed in the literature when modelling annular flow that the fluid exists as saturated liquid and vapor and all heat supplied is used for the evaporation of the liquid film. Consequently, there is no super heat of the fluid and the saturation temperature would not be exceeded. Additionally, the impact energy of the droplets is expected to be small taking into account that the wall normal velocities of the droplets are much lower than the axial velocities in the conditions of annular flow. In conclusion, it is reasonable to neglect those interactions and to consider that only full absorption occurs when droplets reach the liquid film surface.

Another consequence of the fluid saturation condition is that the energy conservation equation can be neglected. As mentioned in the previous paragraph, all heat added to the liquid film is considered to participate only to its evaporation. Consequently, there is no temperature distribution and no need to solve the energy equation. Hence, only mass and momentum conservation equations are solved by the model.

Before presenting the theoretical approach of the gas core and liquid film models, it is important to understand how the equations to be presented in sections 2.3.2 and 2.3.3 are solved. For this reason, it is necessary to clarify how ANSYS Fluent treats each of the fields. ANSYS Fluent uses the Finite Volume Method (FVM) to solve the conservation equations of the fluid flow which in this project corresponds to the vapor flow with the dispersed droplets. This
method consists in dividing the computational domain in smaller control volumes called cells and solving the conservation equations of the fluid flow in each cell. Regarding the liquid film, it cannot be simulated by ANSYS Fluent as a fluid flow; therefore, a liquid film model is applied on the wall boundary surface of the computational domain as a boundary condition. Consequently, the wall becomes a film wall and the equations for the liquid film model are solved in each of its faces. More details about the computational domain are shown in section 4.1.

2.3.2 Gas core model

The gas core model is solved as a two-phase flow three-dimensional system. Due to its multiphase flow nature, it can be formulated in two different ways: Eulerian – Eulerian or Eulerian – Lagrangian. The eulerian frame of reference describes the fluid flow from the point of view of a stationary observer. As a result, the Eulerian – Eulerian approach treats the vapor flow and the droplets as two different continuous phases.

On the other hand, the lagrangian frame of reference describes the fluid flow from the point of view of an observer moving with the particles. Therefore, the Eulerian – Lagrangian approach considers the vapor flow as a single, continuous phase while the droplets are treated as dispersed particles and solved by tracking them through the calculated flow field. This tracking method is usually called Lagrangian Particle Tracking (LPT).

Both approaches are considered reliable to model multiphase flows and have their own advantages and disadvantages. Yet, the Eulerian – Lagrangian approach was chosen for the development of this model for two main reasons. First, it represents the physical phenomena in a more realistic way. Second, it results in a straightforward formulation of the interaction with the liquid film.
Furthermore, ANSYS Fluent Theory Manual 17.1 (p.490) suggests this approach when the volume fraction of the dispersed phase is lower than 10%. Hence, the Eulerian – Lagrangian approach is the right choice since the void fractions in the upper part of a BWR are expected to be in the range of 80% – 100%, leaving a liquid volume fraction between 0% – 20% distributed between the liquid film and the droplets.

A two-way flow coupling is applied to consider mutual interaction between the vapor flow and the droplets. In other words, the vapor flow impacts the droplets behavior and the droplets affect the vapor flow solution. On the other hand, droplet – droplet interactions are not included in this model due to the complexity of the phenomena and lack of reliable information. Additionally, many references mention that droplet – droplet interactions can be neglected when the dispersed second phase occupies low volume fractions.

The mass and momentum conservation equations for the vapor flow correspond to those developed in the literature for single-phase flow plus the addition of the source terms due to the interactions with the droplets and the liquid film.

**Mass conservation:**

\[
\frac{\partial \rho_v}{\partial t} + \nabla \cdot (\rho_v u_v) = S_{m,\text{vap}} \tag{1}
\]

**Momentum conservation:**

\[
\frac{\partial (\rho_v u_v)}{\partial t} + \nabla \cdot (\rho_v u_v u_v) = -\nabla p_v + \nabla \cdot \tau_v^{\text{eff}} + \rho_v g + S_{\text{mom,\text{vap}}} + S_{\text{mom,d}} \tag{2}
\]

Where \( \rho_v \) is the vapor density, \( u_v \) is the vapor mean velocity, \( p_v \) is the vapor pressure, \( \tau_v^{\text{eff}} \) is the effective stress tensor of the vapor, \( g \) is the gravity vector, \( S_{m,\text{vap}} \) and \( S_{\text{mom,\text{vap}}} \) are the mass and momentum source terms due to evaporation of the liquid film respectively, and \( S_{\text{mom,d}} \) is the momentum source...
due to interaction with the droplets. $S_{m,vap}$ and $S_{mom,vap}$ are added by the user as described in section 4 while $S_{mom,d}$ is automatically calculated by ANSYS Fluent with the established computational setup.

The effective stress tensor, $\tau^e_v$ is calculated according to equation 3.

$$\tau^e_v = \mu_v \left[ (\nabla u_v + \nabla u_v^T) - \frac{2}{3} \nabla \cdot u_v I \right]$$  \hspace{1cm} (3)

Where $\mu_v$ is the viscosity of the vapor and $I$ is the unit tensor. The effective stress tensor is solved by applying the Boussinesq hypothesis, therefore, a new parameter called the turbulence viscosity should be computed to solve equation 3. The turbulence viscosity is found by solving separated transport equations according to the turbulence model chosen for the calculations.

The mass source terms due to the evaporation of the liquid film, $S_{m,vap}$, is found by an energy balance at saturated conditions as shown in equation 4.

$$S_{m,vap} = \frac{q^*}{H_{vap}}$$  \hspace{1cm} (4)

Where $q^*$ is the heat flux applied on the film wall and $H_{vap}$ is the latent heat of vaporization.

The momentum source term due to evaporation of the liquid film, $S_{mom,vap}$, is calculated according to equation 5.

$$S_{mom,vap} = S_{m,vap} u_f$$  \hspace{1cm} (5)

Where $u_f$ is the liquid film velocity.
The values calculated in equations 4 and 5 are added only to the cells adjacent to the film wall where evaporation occurs. Given that those values are computed in terms of film wall area, to be added into the vapor flow conservation equations, it is necessary to multiply them by the area of the film wall face and divide by the cell volume.

Because of the two-way coupling between the vapor and the droplets, an additional momentum source term, $S_{mom,d}$, is added to the vapor flow momentum equation to account for the vapor-droplets interaction. This source term is computed as stated by equation 6.

$$S_{mom,d} \text{ (cell } c) = \frac{1}{V_{cdt}} \sum a m_d (u_{d,out} - u_{d,in}) \quad (6)$$

Where $m_d$ is the droplet mass, $u_{d,out}$ is the droplet velocity at the outlet of the cell and $u_{d,in}$ is the droplet velocity at the inlet of the cell.

The equation of motion for the droplets is deduced in the literature by applying a force balance over each droplet.

$$m_d \frac{du_d}{dt} = \sum F \quad (7)$$

The location of the droplet is calculated by:

$$\frac{dx_d}{dt} = u_d \quad (8)$$

Assuming spherical droplets:

$$m_d = \rho_d d_d^3 \frac{\pi}{6} \quad (9)$$

Where $m_d$ is the droplet mass, $d_d$ is the droplet diameter, $\rho_d$ is the droplet density, $u_d$ is the droplet velocity, $x_d$ is the droplet position vector and $F$
represents the forces acting on the droplet. The most representative forces acting on the droplets are the gravitational and drag forces.

Other forces such as “virtual mass” and pressure gradient can be neglected due to the large difference between the droplet density and the vapor flow density in this case. Moreover, the Saffman lift force is not included because according to Wang et al. (1997) it overpredicts the optimum force for depositing particles. Additionally, it was also demonstrated that when the optimum lift force is omitted altogether from the equation of motion, the overall effect is small since only a slight reduction in deposition rates is perceived.

The gravitational and drag forces are calculated according to equations 10 and 11 respectively.

\[ F_{\text{gravitational}} = m_d g \]  
\[ F_{\text{drag}} = m_d \left( \frac{u_v - u_d}{\gamma_d} \right) \]

Where \( \gamma_d \) is the droplet relaxation time.

\[ \gamma_d = \frac{4}{3} \frac{\rho_d d_d^2}{\mu_v k_{\text{drag}} Re_d} \]  

Where \( k_{\text{drag}} \) is the drag coefficient and it is a function of the Reynolds number of the droplet which can be found by:

\[ Re_d = \frac{\rho_v d_d}{\mu_v} |u_v - u_d| \]

There are many correlations reported in the literature to compute the drag coefficient. The one used in this project is the Morsi-Alexander drag model based on an experimental drag curve for single, rigid, spherical particles. This model is an improvement of the well-known Schiller – Neumann model, by adjusting the drag coefficient for a wider range of \( Re_d \).
\[ k_{\text{drag}} = a_1 + \frac{a_2}{Re_d} + \frac{a_3}{Re_d^2} \]  \hspace{1cm} (14)

Where the constants \( a_1, a_2 \) and \( a_3 \) are defined according to different Reynolds ranges (Ansys Fluent Theory Guide 17.1, 2016, p.545).

### 2.3.3 Liquid film model

The conservation equations for the liquid film are deduced by applying the equations developed in the literature for single-phase flow. However, for solving the liquid film, the thin film assumption is made. Owing to this, the spatial gradients of the properties of the liquid film are considered negligible in the tangential direction compared with those in the wall normal direction. Additionally, the liquid is assumed to flow only parallel to the wall, in other words, flow in the wall normal direction is considered negligible. These assumptions imply that the conservation equations of the liquid film can be integrated in the wall normal direction to obtain a two-dimensional system.

**Mass conservation:**

\[
\frac{\partial (\rho_f h)}{\partial t} + \nabla_s \cdot (\rho_f h u_f) = S_{m,\text{dep}} - S_{m,\text{ent}} - S_{m,\text{vap}} \quad (15)
\]

**Momentum conservation:**

\[
\frac{\partial (\rho_f h u_f)}{\partial t} + \nabla_s \cdot (\rho_f h u_f u_f) = -h \nabla_s p_f + \rho_f h g_t + S_{mom,w} + S_{mom,t} + S_{mom,\text{dep}} - S_{mom,\text{ent}} - S_{mom,\text{vap}} \quad (16)
\]

Where \( \rho_f \) is the density of the liquid film, \( u_f \) is the mean film velocity, \( h \) is the liquid film thickness, \( \nabla_s \) is the surface gradient operator, \( S_{m,\text{dep}} \) is the mass source term due to deposition of droplets on the liquid film surface, \( S_{m,\text{ent}} \) is the
mass source term due to entrainment of the liquid film, $S_{m,vap}$ is the mass source term due to evaporation of the liquid film, $p_f$ is the total pressure acting on the liquid film surface, $g_t$ is the gravity in the tangential direction (parallel to the film), $S_{mom,w}$ is the shear force on the film-wall interface, $S_{mom,i}$ is the shear force on the gas-film interface, $S_{mom,dep}$ is the momentum source due to deposition of droplets on the liquid film surface, $S_{mom,ent}$ is the momentum source due to entrainment of the liquid film, and $S_{mom,vap}$ is the momentum source due to evaporation of the liquid film.

The LPT method considers that each droplet located on the liquid film surface according to equation 8 is immediately deposited or absorbed into the liquid film. Hence, the source terms due to deposition of droplets on the liquid film surface $S_{m,dep}$ and $S_{mom,dep}$ are calculated automatically by ANSYS Fluent with the established computational setup. $S_{mom,w}$ and $S_{mom,i}$ are calculated directly by ANSYS Fluent as it will be explained later in this section. The other mass and momentum sources are added by the user as described in section 4.

The mass source term due to evaporation of the liquid film, $S_{m,vap}$, is calculated by equation 4. In contrast, the computation of the mass source term due to entrainment of the liquid film, $S_{m,ent}$, requires more efforts. Entrainment of liquid film has been deeply studied in many industries. As a result, it is possible to find plenty of correlations in order to predict entrainment rates. Secondi, Adamsson and Le-Corre (2009) performed an assessment of the performance of several published entrainment correlations using available measurements of droplet entrainment rates. Their conclusions state that Okawa, et al. (2003) demonstrated good predictive capability and seems preferable compared to the other correlations.

Okawa, et al. (2003) developed an entrainment correlation based on the assumption that the dominant mechanism of droplets entrainment from liquid film is the breakup of roll waves due to the interfacial shear force. Consequently,
it is assumed that the entrainment rate is directly proportional to the interfacial shear force which causes the entrainment and inversely proportional to the surface tension which restrains the entrainment. Additionally, it is considered that the entrainment rate is affected also by the droplets present in the gas core so the effects of inertia of the gas core flow due to the entrained droplets is taken into account by using the density ratio of gas and liquid phases. Finally, Okawa combined these assumptions and proposed the following correlation.

\[
\dot{m}_{\text{ent}} = k_{\text{ent}} \rho_f \frac{f_i \rho_f h^2}{\sigma} \left(\frac{\rho_f}{\rho_v}\right)^e = S_{m,\text{ent}}
\]  

(17)

Where the entrainment mass transfer coefficient \( k_{\text{ent}} \) and \( e \) exponent are recommended from experimental data to be equal to 4.79 \times 10^{-4} \text{ m/s} \) and 0.111 respectively; \( f_i \) is the interfacial friction factor, \( J_v \) is the vapor superficial velocity and \( \sigma \) is the surface tension. From experimental data it was also found that roll waves are not formed if a critical film Reynolds number is not reached, so the entrainment only occurs when \( Re_f \geq 320 \).

\[
Re_f = \frac{\rho_f J_f D}{\mu_f}
\]  

(18)

Where \( J_f \) is the liquid superficial velocity, \( \mu_f \) is the liquid film viscosity and \( D \) is the hydraulic diameter, in case of a circular pipe, the pipe diameter.

According to Okawa, et al. (2001), since the liquid film is thin, the superficial velocities can be simplified by the following expressions:

\[
J_v \approx u_v
\]

\[
J_f \approx \frac{4u_f h}{D}
\]
The interfacial friction factor $f_i$ is estimated by the correlation proposed by Wallis (1969).

$$f_i = 0.005 \left(1 + 300 \frac{h}{D} \right) \quad (19)$$

Next, coming back to the liquid film momentum equation, eq. 16, the total pressure acting on the liquid film surface, $p_f$, is the sum of the vapor flow pressure and the hydrostatic pressure. The first value is taken directly from the vapor flow results and the second one is calculated according to equation 20.

$$p_{hydrostatic} = -\rho_f h (n \cdot g) \quad (20)$$

Where $n$ is the surface normal vector and $g$ is the gravity vector.

It is also assumed that the film velocity follows a parabolic profile. The velocity equation is developed by a force balance between the gravitational force and the wall shear force and applying the following boundary conditions: non-slip at the wall and zero-gradient at the interface with the vapor flow. Then the liquid film velocity in the wall normal direction can be calculated using equation 21.

$$u(y) = \frac{3u_f}{h} \left( y - \frac{1}{2h} y^2 \right) \quad (21)$$

Where $y$ is the distance from the wall.

Using this velocity profile, the shear force on the film-wall interface, $S_{mom,w}$, is computed by equation 22.

$$S_{mom,w} = -\mu_f \left( \frac{du}{dy} \right)_{y=0} = -\mu_f \frac{3u_f}{h} \quad (22)$$

On the other hand, ANSYS Fluent applies the following boundary condition at the vapor-film interface.
\[ S_{mom,i} = \tau_{f,i} = \tau_{v,i} \]  \hspace{1cm} (23)

Where \( \tau_{f,i} \) is the shear force of the liquid film and \( \tau_{v,i} \) is the shear force of the vapor flow both at the vapor-film interface usually called the interfacial shear stress. The vapor flowing next to the liquid film is solved in a moving reference, it means, relative velocities are used to solve the conservation equations for the cells adjacent to the film wall. Besides, \( \tau_{v,i} \) is computed by using the vapor flow relative velocity gradients at the vapor-film interface. Therefore, the interfacial shear stress is a function of the liquid film surface velocity. At the same time, the liquid film surface velocity is compound of two velocity vectors: the inertial and shear force velocities. The first one is calculated by equation 21 when the distance from the wall is equal to the film thickness \( (u_{fs} = 3u_f/2) \) and the second one is a function of the interfacial shear stress. Consequently, the interfacial shear stress and the liquid film surface velocity are found by an iterative process.

The momentum source term due to evaporation of the liquid film, \( S_{mom,vap} \), is calculated by equation 5. Finally, the momentum source terms due to deposition and entrainment are evaluated according to equations 24 and 25 respectively.

\[ S_{mom,dep} = S_{m,dep}u_{d,dep} \]  \hspace{1cm} (24)

\[ S_{mom,ent} = S_{m,ent}u_{d,ent} \]  \hspace{1cm} (25)

Where \( u_{d,dep} \) and \( u_{d,ent} \) are the deposited droplet velocity and the entrained droplet velocity respectively. The first value is calculated automatically by ANSYS Fluent when solving equation 6 for the droplets located on the liquid film surface. The second one is found as explained in section 4.3.
3 VALIDATION CASE

The case used to validate the model is described in this section.

3.1 GENERAL DESCRIPTION

Adamsson and Anglart (2006) performed measurements of film flow rates in diabatic annular flow in a pipe at BWRs operating conditions. Their experiments were performed at 7 MPa in a circular pipe with 14 mm of inner diameter ($D$), total heated length 3.65 m ($L_{total}$) and 10 K of inlet sub-cooling ($\Delta T$). Three different cases were run with various axial power distributions. For the purpose of validation of the model developed during this project, case 2 was selected with a uniform axial heat flux distribution.

Case 2 corresponds to a total inlet mass flux of 1250 kg/s*m$^2$ ($\dot{M}_{in, tot}$) and an annular flow length of 1.06 m ($L_{annular}$). The outlet vapor qualities were measured during the experiments and for this case 51 % ($x_{out}$) was reported. Nomenclature in brackets is used in Appendix 1. The results of the measurements reported for case 2 with uniform axial heat flux distribution are shown in table 1. Figure 5 illustrates the test section of the experiments.

![Figure 5. Representation of the test section](Source: Modified from Li and Anglart (2015b). (p.5)
Table 1. Results of film measurements for case 2 with uniform axial power distribution.

<table>
<thead>
<tr>
<th>z position (m)</th>
<th>Film flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.59</td>
<td>0.0258</td>
</tr>
<tr>
<td>2.72</td>
<td>0.0250</td>
</tr>
<tr>
<td>2.86</td>
<td>0.0208</td>
</tr>
<tr>
<td>2.99</td>
<td>0.0187</td>
</tr>
<tr>
<td>3.12</td>
<td>0.0148</td>
</tr>
<tr>
<td>3.25</td>
<td>0.0125</td>
</tr>
<tr>
<td>3.39</td>
<td>0.0087</td>
</tr>
<tr>
<td>3.52</td>
<td>0.0075</td>
</tr>
<tr>
<td>3.65</td>
<td>0.0053</td>
</tr>
</tbody>
</table>

3.2 PHYSICAL PROPERTIES

As mentioned before, in annular flow it is assumed that the fluid exits as saturated liquid and vapor. Hence, the fluid properties used during the simulation correspond to the saturated values at 7 MPa. Both, the liquid film and the droplets use the data for saturated water.

Table 2. Physical properties used for the simulation.

<table>
<thead>
<tr>
<th>Property</th>
<th>Water</th>
<th>Vapor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>739.72</td>
<td>36.525</td>
</tr>
<tr>
<td>Viscosity (kg/m³s)</td>
<td>9.1249E-05</td>
<td>1.8960E-05</td>
</tr>
<tr>
<td>Surface tension (N/m)</td>
<td>0.017633</td>
<td>-</td>
</tr>
<tr>
<td>Heat of vaporization (J/kg)</td>
<td></td>
<td>1504900</td>
</tr>
<tr>
<td>Gravitational acceleration* (m/s²)</td>
<td></td>
<td>-9.81</td>
</tr>
</tbody>
</table>

* Value of axial component
3.3 BOUNDARY CONDITIONS

The following boundary conditions are required to solve the model: uniform axial distributed heat flux supplied over the external wall of the pipe, vapor velocity, droplets flow rate and liquid film flow rate at the inlet of the annular flow length (this length is shown in figure 5 for the experiment and in figure 6 for the computational mesh). All of the boundary values can easily be found from the data provided in the validation case description and the physical properties reported in table 2.

For instance, the heat flux is obtained by performing a general heat balance over the total heated length (shown in figure 5). The vapor velocity is found by applying a vapor mass balance over the whole annular flow length. Similarly, the droplets flow rate is calculated by a liquid mass balance at the inlet of the annular flow length. Finally, the liquid film flow rate is given by the experiment (see table 1). Details of the calculations can be found in Appendix 1. Table 3 presents a summary of the boundary, operational and geometrical conditions used in the simulation.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat flux</td>
<td>983443 W/m²</td>
</tr>
<tr>
<td>Inlet vapor velocity</td>
<td>12.0352 m/s</td>
</tr>
<tr>
<td>Inlet droplets mass flow rate</td>
<td>0.0989204 kg/s</td>
</tr>
<tr>
<td>Inlet liquid film mass flow rate</td>
<td>0.0258 kg/s</td>
</tr>
<tr>
<td>Pressure</td>
<td>7 MPa</td>
</tr>
<tr>
<td>Annular flow length</td>
<td>1.06 m</td>
</tr>
<tr>
<td>Pipe diameter</td>
<td>0.014 m</td>
</tr>
</tbody>
</table>
4 COMPUTATIONAL SETUP

The model was developed in ANSYS Fluent version 17.1 using a pressure-based solver. All variables are handled in SI units.

A transient simulation is applied for two reasons. First, according to other CFD users and the experience during the development of this model, the use of a dispersed second phase in the continuous fluid flow makes it hard to achieve a converged solution using a steady state simulation. Second, according to Wolf, Jayanti and Hewitt (2001), air – water annular flow experiments (with evaporation, entrainment and deposition) have shown that film flow parameters only reached a quasi-steady state after advancing 100 – 300 diameters from the inlet. Taking into account that at least a minimum of 1.40 m is required in this case to reach a steady state solution and that the annular flow length is only 1.06 m, the unsteady solution seemed to be preferable for this model.

4.1 GEOMETRY AND MESH

The same mesh used in the OpenFOAM model was uploaded to the ANSYS Fluent model developed in this project. It consists of a cylinder with a diameter of 14 mm and a length of 1.06 m with a multi-block hexahedral mesh. The mesh contains approximately a total number of cells of 120000, a non-dimensional wall spacing between 30 and 150 and a distribution around 100 * 35 * 40 cells in the axial, radial and tangential direction respectively. Li and Anglart (2015) performed a mesh dependency study of the case presented in section 3 in OpenFOAM and concluded that the results using this mesh and refined ones are basically the same.

During the development of the project, some issues came out when building the liquid film model. In short, the initialization of the liquid film model requires that a liquid mass flux is applied at the wall faces close to the inlet of the pipe; as a result, the liquid film requires a long length to stabilize. For that reason, the
geometry was doubled in the axial direction allowing the injection and stabilization of the liquid film in half of the pipe length. That is done to achieve the required liquid film mass flow rate at the inlet of the annular flow length (see table 3). The mesh for the annular flow length remains exactly the same as the one described above. The mesh as used in ANSYS Fluent is shown in figure 6. The *inlet-annular* internal surface becomes the inlet to the annular flow length.

![Figure 6. Illustration of the mesh used in the ANSYS Fluent model.](image)

The *injection wall* is where the liquid mass flux is applied. The *stabilization wall* is where the liquid stabilizes until reaching quasi-constant parameters (without interactions between the fields). And finally, the *annular wall* is where the interactions between the three fields take place; evaporation, entrainment and deposition. Furthermore, the fluid zone is divided in two zones. The first part corresponds to the cells between the inlet of the pipe and the inlet of the annular flow length (*inlet-annular* internal surface) and it is called inside the solver as *pre-fluid zone*. The second part which goes from that point to the outlet of the pipe is the *annular-fluid zone*. 
4.2 FLUENT MODELS

Each model must be enabled in the Setup/Models task page. Models options are set inside their own dialog box. In this section, the models used for the simulation and the most relevant options selected for each model will be described.

4.2.1 Turbulence Model

The use of a turbulence model is necessary to take into account the effects of turbulence in the flow solution. According to Damsohn (2011) “the modeled film is hardly influenced by the turbulence model” (p.153); therefore, the wall treatment can use a high Reynolds number approach in order to save computational cost. The Sheer-Stress Transport (SST) $k$-$\omega$ model ($k$ is the turbulent kinetic energy and $\omega$ is the specific dissipation rate) was selected for this simulation. This turbulence model blends the robust and accurate formulation of the $k$-$\omega$ model in the near-wall region with the accurate prediction of the freestream of other turbulence models.

4.2.2 Discrete Phase Model (DPM)

As mentioned before, the gas core model is simulated using the Eulerian – Lagrangian approach for numerical calculations of multiphase flows. The ANSYS Fluent model which follows this approach is the Discrete Phase Model (DPM). In addition to solving the conservation equations for the continuous phase, DPM enables to simulate a discrete second phase in a Lagrangian frame of reference. This second phase corresponds to particles dispersed in the continuous phase which trajectories are computed by ANSYS Fluent.

It is important to clarify that even when in reality droplets are individual physical particles, ANSYS Fluent does not track each physical particle but groups of particles with the same properties such as diameter, mass flow rate and velocity.
This group of particles is called a parcel. In other words, the model tracks a number of parcels, and each parcel is representative of a fraction of the total mass flow released in a time step. The number of particles in a parcel is automatically set in such a way that it satisfies the mass flow rate.

When enabling *Interaction with Continuous Phase*, the two-way flow coupling is included. Furthermore, *Unsteady tracking* of particles is enabled; therefore, particles and the flow develop in time together concurrently. Some other selected features inside this model include the injection of particles with the continuous phase flow time step and the *Number of Continuous Phase Iterations per DPM Iteration* set to 1.

The latter was chosen according to figure 24.31 of the ANSYS Fluent User Guide 17.1, p.1313. It shows the minimum number of DPM updates required for the DPM sources to reach their final values. Therefore, in this model, for a DPM under-relaxed factor of 0.1 and 50 continuous phase iterations per time step, 50 DPM iterations are necessary after any change to the DPM sources (for example a new injection) to ensure that the change has taken effect.

The droplets entering at the inlet of the annular flow length are created by setting a surface injection from *inlet-annular*. By selecting a surface injection, one parcel is released from each one of the injection surface faces every time step. Droplets are injected with the vapor velocity at the inlet of the annular flow length and a constant mass flow rate. Velocity and mass flow rate are reported in table 3. The start time of the injection is set to 1 s to reach a quasi-steady state solution before droplets enter into the system.

Because the droplets size is an unknown parameter, two different values were chosen to compare how the results are affected by them. The values are 0.1 mm and 0.7 mm, which are the Sauter mean diameters reported by Xie, Koshizuka and Oka (2004) for calculated droplet size in BWR conditions with a steam quality of 0.35. They calculated the first value using Kataoka, Ishii,
Mishima´s correlation and the second one using Ambrosini, Andreussi and Azopardi´s correlation. The last value is in agreement with the Sauter mean diameter reported by Le Corre et al. (2015) from their size droplet distributions measurements under BWR operating conditions. Therefore, it will be expected that a droplet diameter of 0.7 mm will give a better correlation with the experimental case used to validate the model.

Finally, the spherical drag law and Discrete Random Walk (DRW) model are enabled. The first one activates the Morsi-Alexander drag model (see section 2.3.1) and the second one is used to predict the dispersion of particles due to turbulence in the fluid phase using stochastic tracking. In the stochastic tracking approach, ANSYS Fluent predicts the turbulent dispersion of particles by integrating the trajectory equations for individual particles, using the instantaneous fluid velocities.

### 4.2.3 Eulerian Wall Film (EWF) model

ANSYS Fluent includes the Eulerian Wall Film (EWF) model which is used to predict the behavior of thin liquid films on the surface of the walls. When enabling the EWF model, the mass conservation equation for the liquid film is automatically solved. However, *Solve Momentum* should be enabled for solving the momentum conservation equation.

In addition, *DPM Collection* must also be selected for the model to automatically account for the interaction of the droplets with the liquid film due to droplets deposition onto the liquid film surface. Lastly, for including the first five terms on the right hand side of the liquid film momentum equation (eq. 16), the following options should be selected: *Gravity Force, Surface Shear Force, Pressure Gradient and Spreading Term*.

The EWF model does not include a feature for the evaporation of the liquid film. Nevertheless, it incorporates a pre-defined stripping model based on the
research developed by Mayer (1961) for liquid film entrainment at very high gas velocities. This stripping model was not used in this simulation because of the following reasons. First, this is a model for liquid atomization and probably unsuitable for large particle size. Additionally, it is an entrainment model more applicable to gas jet erosion applications and not so appropriate for the gas velocities handled in this case. Finally, according to Kolev (2007), p.285, it requires more research to be considered reliable. Kolev (2007) based his comment on the fact that the values of the constants to be applied in the formulas vary from researcher to researcher. Section 4.3 describes how evaporation and entrainment where implemented in the model.

There is one important issue about the EWF model worthy to be mentioned. This model does not allow setting boundary conditions for the liquid film at the inlet of the liquid film flow but only at the wall. This occurs because the EWF model is a boundary condition which can be applied over wall surfaces, but it is not a model to be applied over a fluid flow. When assigning the EWF model to a wall, the ANSYS Fluent solver designates the wall as a film wall where liquid film conservation equations are solved.

The above situation became an issue because a specific liquid film mass flow rate is required at the inlet of the annular flow length (see section 3). Therefore, it was necessary to build a wall area close to the inlet of the pipe where a liquid mass flux is injected and leave the liquid film to build in time. By doing so, the liquid film takes a long axial distance to stabilize and the pipe axial length had to be increased two times. Consequently, the running time increases due to the enlargement of the computational domain and many attempts (using different liquid mass fluxes at the injection wall) were needed to set the required liquid film mass flow rate at the inlet of the annular flow length.
4.3 MODELING EVAPORATION AND ENTRAINMENT

The evaporation rate of the liquid film is a constant value as suggested by equation 4. Therefore, the same evaporation rate is applied at each face of the film wall. Mass and momentum sources terms due to evaporation are added to the conservation equations of the vapor flow and the liquid film. Evaporation starts at 1s running time to allow the vapor-liquid film system to reach quasi steady state solution before applying a new change.

The entrainment rate, as expressed in equation 17, is not a constant value and it will vary from one film wall face to another. The entrainment rate is calculated by applying a loop over each face of the film wall, reading the required properties and computing the entrainment flow rate value. Then, mass and momentum sources terms due to entrainment are added to the conservation equations of the liquid film. Entrainment starts at 1.5 s running time to allow the vapor-liquid film system to reach a quasi-stable solution before applying a new interaction.

Additionally, it is also required to create the new entrained droplets. Consequently, a new surface injection is created but this time from the film wall. The injection positions and initial velocities of the droplets were selected taking into account the comparisons made by Li and Anglart (2015b). They ran the case presented in section 3 in OpenFOAM with different combinations between two injection positions and two initial velocities. The initial positions were the average film thickness and 5 times that value; the initial velocities were the velocity of the local droplets and the liquid film surface velocity. The best match with the experimental results was obtained when setting the injection positions at 5 times the average film thicknesses and the initial velocities as the velocities of the local droplets.

The result obtained for the injection positions of the entrained droplets is better explained when considering what was mentioned in section 2.3.3, the dominant mechanism of droplets entrainment is the breakup of roll waves. Therefore, it
could be expected that the waves have a higher height than the average film thickness. This situation was appreciated in the development of the model, because when the cells adjacent to the film wall were selected as the injection positions, more than 80% of the entrained droplets where being reabsorbed immediately after entrainment.

Finally, the entrained droplets are injected in a distance 5 times the liquid film thickness from the wall, they are assumed to have the same diameter as the droplets entering at the inlet of the annular flow length and their initial velocities were expected to be defined as the local velocities of the vapor flow. However, under the author’s knowledge of ANSYS Fluent solver and frame time, it was not possible to find the cells of the new positions (where entrained droplets are injected) to be able to read their local velocities. Therefore, the velocities of the entrained droplets were set to the local velocities of the cells adjacent to the respective film wall faces.

Evaporation and entrainment of the liquid film could not be implemented by using the existent features of the EWF model. Thus, User-Defined Functions (UDFs) were developed to be able to recreate these processes. Appendix 2 contains the file with the seven UDFs built for this purpose. The first UDF represent the mass sources applied over the liquid film, including evaporation and entrainment term. Inside this UDF, other values are calculated to pass later to another UDFs through User-Defined Memories.

The second UDF corresponds to the initialization of the entrained droplets injection. As mentioned before, the entrainment rate is different for each face of the film wall; therefore, a simple surface injection cannot be used because entrainment flow rates, velocities and positions vary according to the properties calculated in each film wall face. Then, to recreate the entrained droplets, a dummy injection is created over the film wall surface and then the UDF makes a loop over each cell adjacent to the film wall (where a surface injection releases the particles) and modifies the flow rates, initial positions and velocities.
The third and fourth UDFs calculate the mass and momentum source terms of the vapor flow due to evaporation of the liquid film. Since evaporation occurs in the liquid film surface, these source terms are only added to the cells adjacent to the film wall. Finally, the fifth, sixth and seventh UDFs calculate the momentum source terms to be applied over the liquid film in the z,y,x-direction respectively.

4.4 BOUNDARY CONDITIONS

Next, the boundary conditions applied over the fluid zones and boundary surfaces are described.

4.4.1 Pre-fluid zone

In this zone, the initial vapor flow velocity is fixed so the conservation equations are not solved. This section only exists because the injection wall and stabilization wall are required to build the liquid film.

4.4.2 Annular-fluid zone

Two UDFs are hooked in this zone, the mass and momentum source terms to be applied in the vapor flow conservation equations due to evaporation of the liquid film.

4.4.3 Inlet

It is defined as a velocity-inlet boundary condition. The conditions at the inlet of the pipe are set here and remain constant until the inlet at the annular flow length (inlet-annular internal surface). These inlet boundary conditions correspond to the operational pressure set to 7 MPa and the velocity of the vapor flow calculated in section 3.3 as 12.0352 m/s in the axial direction.
Additionally, the initial values of $k$ and $\omega$ are required to initialize the turbulence calculations; $k$ is set to 0.233025 m$^2$/s$^2$ and $\omega$ to 899.321 s$^{-1}$. These initial values are calculated according to the specifications in section 6.3.2.1 in the ANSYS Fluent User Guide 17.1, p.287. For details of the calculations, please refer to Appendix 1. The DPM boundary conditions at the inlet and outlet of the pipe are set as escape, it means, droplets leave the calculation computational domain.

### 4.4.4 Outlet

It is defined as a pressure-outlet boundary condition. The pressure is set at 7 MPa and the values of $k$ and $\omega$ in case of back flow are equal to the ones at the inlet.

### 4.4.5 Injection wall

EWF model is enabled. A wall mass flux boundary condition is set to satisfy the liquid film mass flow rate at the inlet of the annular flow length (0.0258 kg/s, see table 3). However, the set mass flux boundary condition, kg/s·m$^2$, does not correspond to the liquid mass flow rate divided by the injection wall area but to a higher value, probably because by injecting from the wall some liquid flows downwards while the liquid film stabilizes and also because pressure losses. At the end, a trial an error process was required to find the necessary liquid mass flux rate boundary condition and a value of 6.15 kg/s·m$^2$ is set (it corresponds to a 0.0298 kg/s liquid mass flow rate). During the development of the model, it was found that applying also a small momentum flux boundary condition helped to achieve faster stabilization of the liquid film without affecting the conservation equations. The set value is 60 N/m$^2$. 
4.4.6 Stabilization wall

EWF model is enabled. All initial conditions, film height and velocities, are specified as 0 to allow the liquid film to build up from the liquid mass flux applied in the injection wall.

4.4.7 Annular wall

EWF model is enable. All initial conditions, film height and velocities, are specified as 0 to allow the liquid film to build up from the liquid mass flux applied in the injection wall. Additionally, User Source Terms is enabled to include the UDFs with the mass and momentum source terms due to evaporation and entrainment of the liquid film.

4.5 SOLUTION METHODS

Initialization values are set equal to the inlet boundary condition values. The coupled algorithm was selected to speed up the solution convergence and second-order solutions to improve accuracy. A time step of 1E-04 is assigned and a total run time of 2 seconds. 50 iterations per time step is applied and a DPM under-relaxation factor of 0.1 is chosen to increase the stability of the coupled calculation procedure by letting the impact of the discrete phase change only gradually.
5 RESULTS AND DISCUSSION

In this part of the report the final results obtained with the model are presented. The results are organized in 3 different sections. First, a comparison between the results obtained when using droplet diameters of 0.1 mm and 0.7 mm in the ANSYS Fluent model. Second, a comparison between the results obtained using a droplet diameter of 0.1 mm when modeling in the ANSYS Fluent model developed in this project and the OpenFOAM model developed for the NORTHNET project. Finally, some results related to the building up of the liquid film. Local values along the circumference of the pipe are averaged at each axial position, the average values at the end of the simulation (2 s) are plotted unless something else is specified.

5.1 RESULTS WITH TWO DIFFERENT DROPLET DIAMETERS

The liquid film mass flow rate as a function of the axial direction of the pipe is presented in figure 7. Three different curves are plotted: the experimental data presented in table 1, and the results obtained with the model when using droplet diameters of 0.1 mm and 0.7 mm.

![Figure 7. Liquid film mass flow rate for two different droplet diameters.](image-url)
The results shown a high overprediction of the liquid film flow rate when using a droplet diameter of 0.1 mm while a slight underprediction when using a droplet diameter of 0.7 mm. Additionally, when using a droplet diameter of 0.7 mm, the underprediction is significant from the inlet of the annular flow length at 6.24 m to around a position of 7 m, however from that point until the outlet of the pipe at 7.3 m, the experimental and predicted liquid film mass flow rate values are close to each other but still underprediction is evident.

To understand why there is such a huge difference between both simulation results when using two different droplet diameters, figure 8 illustrates the mass fluxes due to the deposition of the droplets on the liquid film surface, the entrainment and the evaporation of the liquid film. The evaporation rate is the same for both cases because as mentioned in section 4.3 it is a constant value depending only on the heat flux applied on the wall of the pipe.

Figure 8. Deposition, entrainment and evaporation mass fluxes for two different droplet diameters.
As observed in figure 8, fewer droplets are deposited on the liquid film surface when the droplet diameter increases. This behavior has been observed and explained by plenty of researches, some of them are summarize by Damsohn (2011), p.105. Basically, large droplets are hardly affected by the vapor flow, it means that they are more driven by their initial velocities than by the drag force and turbulence diffusion acting on them due to the vapor velocity and turbulence respectively. Considering that the eddies of the turbulent vapor flow are the ones inducing the lateral velocities that transport the droplets to the liquid film surface, fewer large droplets would be expected to reach the liquid film surface.

Additionally, there are other interesting aspects of the results shown in figure 8. Firstly, the mass flux due to the evaporation of the liquid film is around 6 times the mass flux due to the entrainment of the liquid film, therefore, it seems that evaporation is the main mechanism of removal of liquid film mass in annular flow under BWR conditions. Secondly, the deposition mass fluxes fluctuate around the evaporation mass flux when droplet diameter is 0.1 mm, and around values lower than the evaporation mass flux when droplet diameter is 0.7 mm. Thirdly, the mass entrainment fluxes are fairly constant when droplet diameter is 0.1 mm while when droplet diameter is 0.7 mm, they slightly decrease in the axial direction and with respect to the values for 0.1 mm.

All the observations mentioned above are related to the way the liquid film mass flow rate is behaving in figure 7 but to understand the connection, it is important to have a look at the liquid film thickness behavior through the whole axial direction shown in figure 9.
The liquid film thickness is a direct measurement of the liquid film mass. From figure 8, it can be deduced that the overall mass losses in the liquid film are higher when the droplet diameter is 0.7 mm than 0.1 mm. When the droplet diameter is 0.1 mm, the mass of the deposited droplets manages to compensate part of the mass loss due to evaporation of the liquid film while when the droplet diameter is 0.7 mm the difference between the liquid film mass being removed and added is larger. This explains the results obtained in figure 9, where the liquid film thickness decreases faster in the axial direction when the droplet diameter is 0.7 mm than 0.1 mm. However, the liquid film flow rate is not only a function of the liquid film thickness but also of the liquid film velocity illustrated in figure 10.
In general, the liquid film velocity increases in the axial direction when the droplet diameter is 0.1 mm but decreases when the droplet diameter is 0.7 mm. This explains the behavior of the curves shown in figure 7. When the droplet diameter is 0.1 mm, the reduction in the liquid film thickness is more or less compensated by the increment of the liquid film velocity, keeping a fairly constant mass flow rate along the axial direction. On the other hand, when the droplet diameter is 0.7 mm, both film thickness and velocity decrease, reducing abruptly the liquid film flow rate.

Additionally, looking back to figure 8, when the droplet diameter is 0.1 mm, the reduction of the liquid film thickness in the axial direction decreases the entrainment mass flux but it remains more or less constant along the axial direction due to the increment of the liquid film velocities which causes that entrainment occurs in more locations according to the entrainment criterion stated in section 2.3.3. On the other hand, when the droplet diameter is 0.7 mm, the entrainment mass flux decreases in the axial direction due to the reduction of both the liquid film thickness and velocity; and they are also lower compared
to a droplet diameter of 0.1 mm due to the faster decrease of the liquid film thickness and less entrainment due to lower velocities.

But, why does the liquid film velocity behave so different for different droplet diameters? The reason of this should be related to the liquid film momentum equation (eq. 16). From that equation and the results illustrated in figure 10, it could be said that when the droplet diameter decreases at a specific limit, then the significant reduction of the deposition mass flux creates a condition where the momentum flux due to the deposition of droplets does not provide enough help to keep the liquid film acceleration upwards. Consequently, the liquid film velocities start decreasing but the liquid film is still flowing upwards thanks to the shear force acting on the liquid-vapor interface due to the vapor flow velocity.

Most of the experimental measurements in annular flow have focused on getting values for liquid film thicknesses or liquid film flow rates. Therefore, there is not enough information about the behavior of the liquid film velocity in different operational conditions. The experiments developed by Wolf et al. (2001) measured the wave velocity which can be correlated to the liquid film surface velocity. Their experiments were developed in air-water with an inlet pressure of 0.24 MPa and without heating so evaporation did not take place. Their results showed that the liquid film velocity increases in distance until becoming fairly constant. This behavior was also observed in the measurement of Würtz (1978) for steam-water at 7 MPa in adiabatic conditions, unfortunately wave velocities where not reported for the diabatic experiments. From those experiments everything that could be implied is that without evaporation, the liquid film is accelerated in the upward direction due to the interfacial shear stress and the momentum provided by the deposition of droplets. However, it is not possible to deduce any information about how the mass of deposited droplets could influence the liquid film velocity when evaporation is taking place.
Next, coming back to figure 8, it seems that the deposition mass fluxes are low close to the inlet of the annular flow length where the droplets are injected. This is more evident when the droplet diameter is 0.7 mm and it can be observed that deposition mass fluxes start increasing after half of the annular flow length. This could explain why the liquid film flow rate shown in figure 7 is less underpredicted close to the outlet. Figure 11 illustrates for both droplet diameters the relation between the deposition mass flux and the concentration of the droplets in the cells adjacent to the film wall.

![Graph](image1.png)

*Figure 11. Relation between deposition mass flux and concentration of droplets for two different droplet diameters.*
The concentration of droplets is higher close to the inlet of the annular flow length as expected because that is the surface of the injection of droplets. However, the deposition of droplets in the liquid film surface seems to be inhibited by the large concentration of droplets. This behavior has been mentioned by other researchers and it was first observed by Namie and Ueda (1972). It is stated that the deposition mass transfer coefficient decreases when droplets concentration increases possibly because droplet-droplet interaction becomes a dominant effect.

Finally, table 4 presents the liquid film mass flow rate at the outlet of the pipe and other values of interest.

Table 4. Experimental and predicted values for two different droplet diameters.

<table>
<thead>
<tr>
<th></th>
<th>Experiment</th>
<th>d = 0.1 mm</th>
<th>d = 0.7 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet liquid film mass flow</td>
<td>5.333E-03</td>
<td>2.2561E-02</td>
<td>4.6152E-03</td>
</tr>
<tr>
<td>rate (kg/s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Droplets average residence</td>
<td>-</td>
<td>1.66</td>
<td>1.49</td>
</tr>
<tr>
<td>time (s)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Average Weber number</td>
<td>-</td>
<td>5.4757E-05</td>
<td>3.3045E-05</td>
</tr>
<tr>
<td>droplets next to film</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The liquid film mass flow rate at the outlet of the pipe are overpredicted for the droplet diameter of 0.1 mm and underpredicted for the droplet diameter of 0.7 mm as mentioned at the beginning of this section. However, the values reported for 0.7 mm are much closer to the experimental values and coincide with the Sauter droplet diameter reported by Le Corre et al. (2015) under BWR operational conditions. Therefore, it could be said that the droplet diameter in the experimental case are closer to 0.7 mm than 0.1 mm and that the other liquid film parameters such as thickness and velocity behave more accordingly to the results presented for 0.7 mm.
The deviations from the experimental results can come from deviations in the liquid film thickness calculations, in the liquid film velocities calculations or both. The film thickness calculations are mainly dependent on the evaporation, deposition and entrainment mass fluxes. The film velocities calculations depend also on the mass fluxes but besides, on the way the wall and interfacial shear forces are being handle. Having experimental data where additionally to the liquid film flow rate at least one of the other parameters were known would allow to understand better where the deviation is coming from and how to adjust the model.

However, some possible explanations for the underprediction of the liquid film mass flow rate when using a 0.7 mm droplet diameter could still be discussed. The first one could be related to the empirical correlation being used to predict the mass flux due to entrainment of the liquid film. Even though, Okawa´s entrainment correlation has been demonstrated to provide the best predictive capability compared to other correlations, it is also recognized by Secondi et al. (2009) that it tends to overpredict the entrainment and some improvements could be done. Another possible reason for the underprediction of the liquid film mass flow rate is that not all the heat flux supplied on the wall of the pipe is used to evaporate the liquid film but part of it is transferred to the vapor flow.

Other possible explanations for the underprediction of the liquid film mass flow rate with a droplet diameter of 0.7 mm could be related to the treatment given to the droplets in the model. For example, as mentioned before, the Saffman lift force was not included because according to Wang et al. (1997) it overpredicts the shear-induce lift force acting on the droplets. Wang et al. (1997) developed an optimum lift force based on the shear and wall-induced components. The wall-induce lift force could be an important factor for the deposition of droplets on the liquid film surface and increase the deposition mass flux in the model. Even though, Wang et al. (1997) demonstrated that when any lift force is omitted altogether from the equation of motion, the overall effect is small since only a slight reduction in deposition rates is perceived, the effect in the annular flow
under BWR conditions could be significant. Using a distribution of droplet diameters could also improve the deposition of droplets on the liquid film surface as well as the adjustment of the drag force coefficient due to possible deformation of the droplets.

Previously, it was mentioned that there is less droplet deposition when the droplet diameter increases. This could also be also observed in table 4 where the droplets average residence time is shorter for droplet diameter of 0.7 mm than 0.1 mm, therefore, implying that more droplets are leaving the domain and faster when their diameter is 0.7 mm than 0.1 mm. Finally, the average Weber numbers of the droplets next to the film wall is also presented in table 4. The Weber number is calculated as \( We = \frac{\rho_d u_{a,n}^2 d_d}{\sigma} \) where \( u_{a,n} \) is the velocity of the droplet in the normal direction of the wall. As expected, droplets with 0.7 mm of diameter have lower Weber number because they tend to travel more towards the axial direction than the normal one. Additionally, both Weber numbers are very low and therefore the assumption made that the dominant interaction when droplets reach the liquid film is absorption is supported. No splashing should be expected at those Weber numbers values.

5.2 RESULTS USING A DROPLET DIAMETER OF 0.1 mm IN ANSYS FLUENT AND OPENFOAM MODELS

Both simulations were performed using a droplet diameter of 0.1 mm. The results in both models were taken from a straight line in the film wall from the inlet of the annular flow to the outlet, therefore they are local values. The results are presented in this way due to difficulties to get average values in the OpenFOAM model, it is expected that the behavior in the straight line represents the general behavior in the circumference of the pipe along the axial direction. The results of the liquid film flow rates through the axial direction in the ANSYS Fluent and OpenFOAM models are presented in figure 12.
Surprisingly, the OpenFOAM and ANSYS Fluent models are presenting totally different results for the same droplet diameter. In fact, the results of the OpenFOAM model are in better agreement with the results in ANSYS Fluent for a droplet diameter of 0.7 mm. Definitely, for a droplet diameter of 0.1 mm, the results in the OpenFOAM model are closer to the experimental values with some underprediction. Figure 13 presents a comparison of the mass fluxes due to evaporation, entrainment and deposition for both models.
It seems that the entrainment mass fluxes are more or less the same while depositions occur in more points in the ANSYS Fluent model. This is a peculiar behavior considering that same forces and equations are solved in both models for the lagrangian approach of the droplets. The only difference is that to compute the drag coefficient, the ANSYS Fluent model uses the Morsi – Alexander correlation while the OpenFOAM model uses the Schiller – Neumann correlation. However, the first one is the same to the second one but adjusted to more Reynolds Numbers ranges and it should not have such an impact in the results. Figures 14 and 15 illustrates the liquid film thickness and velocity respectively.

Figure 14. Liquid film thickness for two different models.

Figure 15. Liquid film velocity for two different models.
In general, the liquid film thicknesses computed in the OpenFOAM model are lower than those calculated in the ANSYS Fluent model, the exception is close to the inlet of the annular flow length but the behavior in that part of the pipe could be more related to the way the inlet boundary condition was implemented in both models. This behavior was expected considering that more deposition is observed in the ANSYS Fluent model in figure 13. On the other hand, the liquid film velocities are increasing in the axial direction for the ANSYS Fluent model while for the OpenFOAM model they are decreasing with distance. This is the same behavior observed in section 5.1, but this time additionally to be related to less momentum source due to fewer deposited droplets, it could also be related to the differences in the computational set up used in the models. As mentioned before, the liquid film velocity is influenced from one side for the mass fluxes and also for the other forces acting on the liquid film such as the wall and interfacial shear force. The major differences in the calculation of the momentum sources terms used in the momentum conservation of the liquid film (eq. 16) is the approach used to compute the interfacial shear force.

The OpenFOAM model calculates the interfacial shear stress by applying the general definition of interfacial friction factor which is the ratio between interfacial shear force and the kinetic energy of the vapor phase. The interfacial shear force on the vapor-film interface is calculated as $\tau_i = f_i \rho_v \left( u_v - u_{sf} \right)^2 / 2$ where the interfacial friction factor, $f_i$, was calculated according to Wallis (1969) correlation as stated in equation 19, $u_v$ is the bulk velocity of the vapor flow adjacent to the liquid film and $u_{sf}$ is the liquid film surface velocity calculated by equation 21 when the distance from the wall is equal to the film thickness ($u_{sf} = 3u_f/2$). Consequently, the OpenFOAM model considers that the liquid film behaves always as a laminar film. On the other hand, the interfacial shear stress is calculated by ANSYS Fluent considering that the liquid film is composed of a layer where the film is laminar and the waves crests where the film is turbulent. Therefore, the liquid film surface velocity is compound of two velocity vectors: the inertial and shear force velocities. The first one is calculated in exactly the same way as in OpenFOAM and the second one is a function of the interfacial
shear stress. The interfacial shear stress is found by solving the effective stress tensor of the cells adjacent to the film and using the relative velocity \( u_v - u_{sf} \). In this case, the interfacial shear stress is a function of the liquid film surface velocity and this one is a function of the interfacial shear stress and to solve both of them an iterative process is done.

5.3 LIQUID FILM EVOLUTION IN THE AXIAL DIRECTION

Figure 16 shows how the liquid film evolves in distance before evaporation, deposition and entrainment start (all local values are plotted at 1 s).

![Graph showing liquid film evolution with distance](image)

Figure 16. Liquid film evolution with distance.

The liquid film local values do not fluctuate distinctively in the stabilization wall; they look pretty much uniform in each axial location but decrease with the distance. On the other hand, in the annular wall the average film thickness is constant along the whole length but the local values are clearly fluctuating. This behavior seems to agree with the observations in the experiments developed by Wolf et al. (2001), where the formation of waves is evident after the liquid film has advanced some distance from the inlet of the pipe. Therefore, the maximum local film thicknesses observed in figure 16 would correspond to the waves crests and the minimum film thicknesses to the waves troughs.
6 CONCLUSIONS AND RECOMMENDATIONS

The results in ANSYS Fluent when the droplet diameter is 0.7 mm are in good agreement with the experimental liquid film mass flow rate but slightly underpredicted. On the other hand, the results with a droplet diameter of 0.1 mm are extremely overpredicting the experimental data. These results were expected taking into account that 0.7 mm is the droplet size reported in the only known experiment where droplet measurements were done under BWR operational conditions.

To improve the model and prediction it is important to review some aspects such as the empirical model used for calculating the entrainment mass flux, the use of a wall-induced lift force, using a droplet distribution instead of a uniform diameter, study the possibility of deformation of the droplets and the impact in the drag force coefficient, study the possibility that not all the heat flux is used for the evaporation of the liquid film.

The results in the ANSYS Fluent model suggest that there is a minimum deposited mass flux necessary to keep the liquid film being accelerated upwards. Above that mass flux, a fairly constant liquid film mass flow rate does not mean that the liquid film thickness is not decreasing, however it decreases slowly. Below that mass flux, the liquid film thickness, velocities and liquid film flow rate will decrease very fast.

The results in the ANSYS Fluent model and the OpenFOAM model with the same droplet diameter of 0.1 mm suggest that the different approaches used for the computation of the interfacial shear stress could cause that the liquid film is accelerated upwards or not. Additional experiments under BWR operational conditions are necessary to validate the behavior present in the ANSYS Fluent model when there are different deposited mass fluxes and the approach used for the calculation of the interfacial shear stress. Those experiments require that more variables are measured.
For future work on this model, the validation using more experimental cases and an extension to complex geometries, it is necessary to be able to apply an inlet boundary condition in the liquid film. Therefore, the Eulerian Wall Film model used in ANSYS Fluent to simulate and predict the behavior of liquid films will require some modifications which can only be done by its designers. The approach used in this project to simulate a liquid film from a wall boundary condition demands too much computational cost and efforts to be able to set the right conditions at the inlet of the annular flow length.

It would be also interesting to find the best way to calculate the vapor quality at least at the outlet of the pipe to compare it with the result reported by the experiment. Even though some options were checked at the end of the project, the author did not find an accurate method for the Eulerian - Lagrangian approach used in the model. Also, it would be interesting to study the difference in deposited droplets between the ANSYS Fluent model and the OpenFOAM model because according to the way both models were set they should not be showing such significant differences.
7 REFERENCES


APPENDIX 1. BOUNDARY AND/OR INITIAL CONDITIONS

Heat flux supplied over the pipe

\[
q \left[ \frac{W}{m^2} \right] = \frac{m_{\text{in,tot}} \bar{C}_p \Delta T + \dot{m}_{\text{v, out}} \Delta H_{\text{vap}}}{A_{\text{wall total}}}
\]

\[
\dot{m}_{\text{in,tot}} = \dot{M}_{\text{in,tot}} A_{cs}
\]

\[
A_{cs} = \frac{\pi D^2}{4}
\]

\[
\bar{C}_p = \frac{Cp_{\text{sat}} + Cp_{\text{inlet}}}{2}
\]

\[
\dot{m}_{\text{v, out}} = \dot{m}_{\text{in,tot}} x_{\text{out}}
\]

\[
A_{\text{wall total}} = \pi DL_{\text{total}}
\]

Where \( m_{\text{in,tot}} \) is the total mass flow rate at the inlet of the pipe, \( \bar{C}_p \) is the average heat capacity between the inlet and saturation conditions, \( \Delta T \) is the inlet subcooling temperature, \( \dot{m}_{\text{v, out}} \) is the vapor flow rate at the outlet of the pipe, \( \Delta H_{\text{vap}} \) is the heat of vaporization, \( A_{\text{wall total}} \) is the total heated wall area, \( \dot{M}_{\text{in,tot}} \) is the total mass flux at the inlet of the pipe, \( A_{cs} \) is the cross-sectional area of the pipe, \( D \) is the pipe diameter, \( x_{\text{out}} \) is the vapor quality at the outlet of the pipe and \( L_{\text{total}} \) is the total heated length.

Vapor velocity at the inlet of the annular flow length

\[
\frac{u_{\text{v, in}}}{S} = \frac{\dot{m}_{\text{v, in}}}{\rho_v A_{cs}}
\]

\[
\dot{m}_{\text{v, in}} = \dot{m}_{\text{v, out}} - \dot{m}_{\text{v, vap}}
\]
\[ m_{v,\text{vap}} = \frac{q^* A_{\text{wall annular}}}{\Delta H_{\text{vap}}} \]

\[ A_{\text{wall annular}} = \pi D L_{\text{annular}} \]

Where \( m_{v,in} \) is the vapor mass flow rate at the inlet of the annular flow length, \( \rho_v \) is the vapor density, \( m_{v,\text{vap}} \) is the mass flow rate of the evaporated liquid film, \( A_{\text{wall annular}} \) is the annular flow wall area and \( L_{\text{annular}} \) is annular flow length. \( A_{cs} \), \( m_{v,\text{out}} \) and \( q^* \) were previously computed in this appendix.

**Droplets flow rate at the inlet of the annular flow length**

\[ m_{d,\text{in}} \left[ \frac{kg}{s} \right] = m_{l,\text{in}} - m_{f,\text{in}} \]

\[ m_{l,\text{in}} = m_{\text{in, tot}} - m_{v,\text{in}} \]

Where \( m_{l,\text{in}} \) is total liquid mass flow rate at the inlet of the annular flow length, including liquid film and droplets; \( m_{f,\text{in}} \) is the liquid film mass flow rate at the inlet of the annular flow length, given in table 1; \( m_{\text{in, tot}} \) and \( m_{v,\text{in}} \) were previously computed in this appendix.

**Turbulence kinetic energy**

\[ k \left[ \frac{m^2}{s^2} \right] = \frac{3}{2} (u_{v,\text{in}} L)^2 \]

\[ I = 0,16 (Re_v)^{-1/8} \]

\[ Re_v = \frac{\rho_v u_{v,\text{in}} D}{\mu_v} \]
Where $I$ is the turbulence intensity, $Re_v$ is the vapor flow Reynolds number and $\mu_v$ is the vapor viscosity. $u_{v,in}$ was previously computed in this appendix.

**Specific dissipation rate**

$$\omega [s^{-1}] = \frac{k^{1/2}}{C_\mu l}$$

$$l = \frac{0.07D}{C_\mu^{3/4}}$$

Where $C_\mu$ is an empirical constant equal to 0.09 and $l$ is the turbulence length scale. $k$ was previously computed in this appendix.
APPENDIX 2. USER-DEFINED FUNCTIONS

#include "udf.h"
#include "sg_film.h"

real hvap = 1504900;
real heat_flux = 983443;
real n = 0.111;
real ke = 0.000479;
real rhovap = 36.525;
real rholiq = 739.72;
real dh = 0.014;
real r = 0.007;
real muliq = 0.000091249;
real Ref_crit = 320;
real sigma = 0.017633;
real p_diam = 0.0003;

DEFINE_PROFILE(film_mass_source,tf,i)
{
    real film_thickness, film_vel, vap_vel, Jf, Jg, fi, Ref, area_face, area[ND_ND],
    pos[ND_ND], r1, angle, x_inj, y_inj, z_inj, evap_mass_source,
    evap_mom_source, ent_mass_source;
    real flow_time = CURRENT_TIME;
    face_t f;
    cell_t c;
    Thread *tc;

    begin_f_loop(f,tf)
    {
        film_thickness = F_EFILM_HEIGHT(f,tf);
        film_vel = F_EFILM_W(f,tf);
        c = F_C0(f,tf);
        tc = THREAD_T0(tf);
        vap_vel = C_W(c,tc);
        Jf = 4*film_vel*film_thickness/dh;
        Jg = vap_vel;
        fi = 0.005*(1+300*(film_thickness/dh));
        Ref = (rholiq*Jf*dh)/muliq;

        if (flow_time >= 1.5 & & Ref > Ref_crit)
            ent_mass_source
                =((ke*rholiq*fi*rhovap*Jg*Jg*film_thickness)/sigma)*pow(rholiq/rhvap,n);
        else
            ent_mass_source = 0.0;
    }
}
if (flow_time >= 1)
    evap_mass_source = heat_flux/hvap;
else
    evap_mass_source = 0.0;

evap_mom_source = evap_mass_source*film_vel;
F_AREA(area,f,tf);
area_face = NV_MAG(area);
F_CENTROID(pos,f,tf);
r1 = r-5*film_thickness;
angle = acos(pos[0]/r);
x_inj = r1*cos(angle);
y_inj = r1*sin(angle);
z_inj = pos[2];

C_UDMI(c,tc,0) = evap_mass_source;
C_UDMI(c,tc,1) = evap_mass_source*area_face;
C_UDMI(c,tc,2) = evap_mom_source;
C_UDMI(c,tc,3) = evap_mom_source*area_face;
C_UDMI(c,tc,4) = ent_mass_source;
C_UDMI(c,tc,5) = ent_mass_source*area_face;
C_UDMI(c,tc,6) = x_inj;
C_UDMI(c,tc,7) = y_inj;
C_UDMI(c,tc,8) = z_inj;

F_PROFILE(f,tf,i) = -(ent_mass_source+evap_mass_source);
}
end_f_loop(f,tf)
}

DEFINE_DPM_INJECTION_INIT(entrainment_inj,I)
{
    Particle *p;
    cell_t c;
    Thread *tc;

    loop(p,I->p_init)
    {
        c = P_CELL(p);
        tc = P_CELL_THREAD(p);
        P_POS(p)[0]=C_UDMI(c,tc,6);
        P_POS(p)[1]=C_UDMI(c,tc,7);
        P_POS(p)[2]=C_UDMI(c,tc,8);
        P_FLOW_RATE(p)= C_UDMI(c,tc,5);
        P_DIAM(p)=p_diam;
        P_VEL(p)[0]=C_U(c,tc);
        P_VEL(p)[1]=C_V(c,tc);
        P_VEL(p)[2]=C_W(c,tc);
    }
}
C_UDMI(c,tc,9) = C_U(c,tc);
C_UDMI(c,tc,10) = C_V(c,tc);
C_UDMI(c,tc,11) = C_W(c,tc);
}
}
DEFINE_SOURCE(vapor_mass_source,c,tc,dS,eqn)
{
  real vol_cell, evap_mass_source;
  vol_cell = C_VOLUME(c,tc);
  evap_mass_source = C_UDMI(c,tc,1)/vol_cell;
  return evap_mass_source;
}
DEFINE_SOURCE(vapor_mom_source,c,tc,dS,eqn)
{
  real vol_cell, evap_mom_source;
  vol_cell = C_VOLUME(c,tc);
  evap_mom_source = C_UDMI(c,tc,3)/vol_cell;
  return evap_mom_source;
}
DEFINE_PROFILE(z_film_mom_source,tf,i)
{
  face_t f;
  cell_t c;
  Thread *tc;
  real drop_vel, ent_flux;

  begin_f_loop(f,tf)
  {
    c = F_C0(f,tf);
    tc = THREAD_T0(tf);
    drop_vel = C_UDMI(c,tc,11);
    ent_flux = C_UDMI(c,tc,4);
    F_PROFILE(f,tf,i) = -(C_UDMI(c,tc,2)+ent_flux*drop_vel);
  }
  end_f_loop(f,tf)
}

DEFINE_PROFILE(y_film_mom_source,tf,i)
{
  face_t f;
  cell_t c;
  Thread *tc;
  real drop_vel, ent_flux;
begin_f_loop(f,tf)
{
    c = F_C0(f,tf);
    tc = THREAD_T0(tf);
    drop_vel = C_UDMI(c,tc,10);
    ent_flux = C_UDMI(c,tc,4);
    F_PROFILE(f,tf,i) = -ent_flux*drop_vel;
}
end_f_loop(f,tf)

DEFINE_PROFILE(x_film_mom_source,tf,i)
{
    face_t f;
    cell_t c;
    Thread *tc;
    real drop_vel, ent_flux;

    begin_f_loop(f,tf)
    {
        c = F_C0(f,tf);
        tc = THREAD_T0(tf);
        drop_vel = C_UDMI(c,tc,9);
        ent_flux = C_UDMI(c,tc,4);
        F_PROFILE(f,tf,i) = -ent_flux*drop_vel;
    }
    end_f_loop(f,tf)
}
APPENDIX 3. DELIVERABLES

- **Case files:**

  Before running again any of the cases, the respective UDFs file must be compiled first. Please consider that the case file and the UDFs file must be saved in the same folder.

  - Droplet diameter of 0.1 mm  
    - annular_flow_diam01.cas
  - Droplet diameter of 0.7 mm  
    - annular_flow_diam07.cas

- **Data files:**

  - From 0 s to 0.9 s  
    - saved in folder *pre*
  - From 1 s to 2 s  
    - saved in folder *d_0.1mm*
  - Droplet diameter 0.1 mm
  - From 1 s to 2 s  
    - saved in folder *d_0.7mm*
  - Droplet diameter 0.7 mm

- **UDFs files:**

  - Droplet diameter of 0.1 mm  
    - source_terms_annular_diam01.c
  - Droplet diameter of 0.7 mm  
    - source_terms_annular_diam07.c