Influence of Mixture Non-Uniformity on the Performance of an Effervescent Nozzle

K. Pougatch†*, M. Salcudean‡, and J. McMillan‡

†Department of Mechanical Engineering
University of the British Columbia
Vancouver, BC V6T 1Z4 Canada
‡Edmonton Research Centre
Syncrude Canada Ltd.
Edmonton, AB T6N 1H4 Canada

Abstract
This paper investigates the influence of the inlet gas-liquid mixture non-uniformity on the performance of the TEB type effervescent nozzle. The flow conditions studied are chosen to be relevant to industrial Fluid Coking operations. The investigation is limited to radial non-uniformity and conducted numerically by applying the mathematical model previously developed by the authors. The comprehensive mathematical model includes the liquid continuous flow through the variable cross-section nozzle, atomization, and spray dispersion. The variety of particle (bubble or droplet) sizes is represented by the average local diameter that can vary throughout the domain. Break-up and coalescence of bubbles (inside the nozzle) or droplets (outside of the nozzle) are accounted for through the particle number density approach. To quantify the inlet non-homogeneity, a new criterion related to the slope of the volume fraction inlet profile is proposed and utilized in the analysis. The simulation of a number of cases that cover the whole range of possible scenarios from the fully separated flow with liquid at the center to that with gas at the center demonstrates that the inlet mixture non-uniformity does not produce substantial variations in the nozzle performance in terms of the spray quality. Further analysis reveals that the first convergent section of the TEB nozzle generates enough turbulence to facilitate phase mixing as the flow moves through the following divergent section. Therefore, as the mixture reaches the second convergent section, which is responsible for the atomization and subsequent droplets distribution, the radial profile is similar for all cases regardless of the inlet conditions.

*Corresponding author: pougatch@mech.ubc.ca
Introduction

Liquid atomization is widely used in processes where a large liquid surface area is required. There is a variety of ways to atomize liquid, and, as a result, a variety of atomizer designs. Some of them utilize the secondary fluid (usually gas) to help atomize the primary one; such atomization is called twin-fluid or gas-assisted. Most methods of gas-assisted liquid atomization are characterized by liquid break-up due to the interface instability because of the velocity shear. The atomization by phase inversion, on the other hand, is achieved by a rapid bubble growth in a two-phase mixture as a result of a sudden pressure drop and the corresponding gas expansion. The growing bubbles coalesce with their neighbors, and the liquid that becomes trapped between bubbles forms droplets. The nozzles utilizing this technology are called effervescent [1]. They have the advantage of smaller droplet sizes and lower injection pressure and gas to liquid mass ratio (GLR) requirements. The effervescent nozzles have applications in a variety of industrial processes from aerated fuel injection in gas turbines to bitumen injection into a Fluid Coking reactor [2]. The latter process has an important role in the synthetic oil production from bituminous sand deposits. The bitumen atomized with the assistance of steam is injected into a fluidized bed; upon collision with hot coke particles a thin layer is formed on the particles’ surface. The Fluid Coking reactor facilitates thermal cracking of the bitumen. The quality of atomization plays an important role to ensure a high yield of valuable lighter hydrocarbons [3].

There is a substantial literature that reports experimental investigation of effervescent atomizers. Review [1] thoroughly discusses the work conducted to investigate the influence of injection pressure, atomizer geometry, GLR, gas and liquid physical properties and flow rates, and atomizer design on the resulting droplet velocity and diameter, spray expansion angle, gas entrainment, liquid distribution, and other spray parameters. More recent work is mostly related to in-depth investigations of the particular atomizer design or applications. For example, in [4] the authors applied an effervescent atomizer design to industrial burners and studied the influence of the nozzle design parameters on the spray quality. Other researchers [5] studied the variation of the atomizer discharge coefficient for a wide variety of inlet flow rates and through the different flow and atomization regimes.

With the advances in the computer power and numerical algorithms, there is an increasing number of papers on the effervescent atomization. The models range from the analytical and semi-empirical to the comprehensive multi-dimensional models based on the first principles. The work of Panchaquina and Sojka [6] in which a correlation for the spray velocity is developed is an example of the former, and the work of Xiong et al. [7] where the Navier-Stokes and Lagrangian equations are solved for the gas and liquid phases correspondingly, is an example of the latter. Broukal and Hajek [8] applied commercial CFD code to the modeling of the effervescent spraying to simulate the resulting Sauter mean droplet diameter distribution. Qian and Lin [9] presented a review of numerical modeling applications to effervescent atomization.

It can be seen from the substantial volume of the research carried out that the performance of the effervescent nozzles strongly depends on their design. While most of the studies examine the atomizer configurations generally similar to the originally proposed, e.g. [10], the specific nozzle containing two convergent and one divergent section (TEB type) that is utilized in the Fluid Coking reactor [11] received less wide attention.

Ariyapadi et al [12] experimentally studied droplet velocity and diameter distribution in a water spray obtained from a scaled down version of the TEB nozzle. Further research [13] demonstrated that the addition of surfactant to the water reduces droplet diameter in the spray. Ejim et al. [14] investigated the influence of the liquid viscosity and surface tension on the average droplet diameter for the same scaled down TEB nozzle. They discovered that while the increase of the viscosity and the decrease of the surface tension increase the droplet diameter, the dependency is rather weak. Subsequently, Gomez et al. [15] extended the work to study the effect of the bubble diameter in the mixture before it enters the nozzle and found a positive correlation. Recently, Rahman et al. [16] demonstrated that the variation of the gas molecular weight does not produce an appreciable change in the produced droplet diameters.

A numerical model for the effervescent nozzle of TEB type was developed by Pougatch et al. [17] and validated by comparison with experimental measurements of liquid mass flux and average droplet diameter distribution [2]. The model was applied to various flow conditions and utilized to investigate the influence of nozzle design, liquid properties such as viscosity and surface tension, and GLR on the resulting characteristics of the spray.

The previous model applications assumed that the bubbles were distributed uniformly at the entrance to the TEB nozzle, i.e. the incoming mixture was homogeneous. However, as it was shown in [16], insufficient mixing can influence the characteristics of the spray for a scaled down nozzle. At the same time the mixing quality of the steam-bitumen mixture that is supplied at the nozzle entrance is not well known. There might be some non-uniformity present due to the insufficient mixing or partial stratification. Therefore, it is essential to evaluate the influence of the mixing non-homogeneity on the atomization quality in order to as-
sess the importance of good mixing and provide guidance for the mixer design. As it is not feasible to carry out experiments at the nozzle operating conditions, a numerical investigation is conducted. The present study attempts to cover this topic by the application of the previously developed mathematical model to the effervescent TEB type nozzle operating at industrially relevant conditions.

**Mathematical model**

The mathematical model that has been developed by Pougatch et al. [2] specifically for the effervescent atomization process and compared with a variety of experimental data is utilized to simulate the nozzle operation. The model is based on an Eulerian-Eulerian representation of gas and liquid phases and includes compressibility effects for the gaseous phase. The continuous phase is determined based on the local values of the phase volume fractions; thus, both liquid flow with dispersed bubbles and gas flow with dispersed droplets can coexist in the computational domain. The primary atomization is treated as a catastrophic phase inversion, meaning that the locally continuous phase changes over an infinitely thin surface. The phase inversion can be visualized with the help of Figure 1. Bubble growth results in the liquid trapped in-between them; subsequent coalescence of the bubbles produces droplets. As can be seen from Figure 1, our assumption about the phase inversion mechanism means that the number of newly born droplets is the same as the number of bubbles before the phase inversion. It is also assumed that the other flow parameters, such as pressure, velocity, turbulence, etc., do not change across the phase inversion surface. According to our previous investigation [17], the value of the gas volume fraction that triggers the phase inversion is 80%.

![Phase inversion schematic](image)

**Figure 1.** Phase inversion schematic.

Ensemble averaged mass and momentum conservation equations are written in the following form for continuous and dispersed phases [18].

\[
\frac{\partial}{\partial t} \rho \alpha_i + \nabla \cdot \rho \alpha_i \mathbf{V}_i = 0 \quad i = c, d
\]

\[
\frac{\partial}{\partial t} \rho_i \mathbf{V}_i + \nabla \cdot \rho_i \mathbf{V}_i \mathbf{V}_i = \nabla \cdot \mathbf{\tau} - \alpha_i \nabla \rho_i + \sum_k \mathbf{F}_{ik} \quad i = c, d
\]

The stress tensor is calculated with the Boussinesq approximation.

\[
\mathbf{\tau}_i = \alpha_i \left( \mu_i + \mu_i' \left( \nabla \mathbf{V}_i + \nabla \mathbf{V}_i^T \right) - \frac{2}{3} \mathbf{I} \nabla \right) - \frac{2}{3} \alpha_i \rho_i k_i \mathbf{I} \quad i = c, d
\]

As steam and bitumen are assumed to have the same temperature, heat transfer between them is neglected. Instead of solving the steam energy conservation equation, it is assumed that its total enthalpy is constant.

\[
H_i = c_v T_s + \frac{V_i^2}{2} + k_i
\]

The ideal gas equation of state connects the density with pressure.

\[
p_v = \frac{\rho R_v T_s}{M_s}
\]

The bubble/droplet diameter distribution at each location is represented by the average diameter that can vary throughout the flow field accounting for the break-up and coalescence by solving a particle number density transport equation [19].

\[
\frac{\partial n}{\partial t} + \nabla \cdot \left( \mathbf{V}_n n \right) = \nabla \cdot \left( \frac{\mu'}{\rho_s \text{Sc}^f} \nabla n \right) + n \left( f_v - f_c \right)
\]

The particle number density, defined as the number of particles per unit volume, can be connected with the volume fraction and diameter.

\[
n = \frac{6 \alpha_i}{\pi d_i^3}
\]

The above conservation equations are not sufficient to completely define the problem. Several terms such as the interfacial forces in (1), the turbulent viscosity in (2), and the break-up and coalescence rates in (6) are still unknown. Therefore, a number of constitutive equations are required. These equations are obtained from a variety of sub-models that essentially translate the micro-scale physical phenomena to the macro-scale parameters. We consider the interfacial and turbulent drag forces and the virtual mass force. The interfacial drag correlations vary depending on the flow regime. The fluctuating motion of phases is accounted for through the mixture turbulence model with the turbu-
lence response coefficient. Bubbles and droplets break-up and coalescence are considered separately because there are different physical mechanisms involved. Some of the sub-models, such as break-up and coalescence, have already been published and needed adaptation for specific flow conditions, and others, such as multiphase wall functions, have been developed specifically for the effervescent nozzle modeling. For the sake of brevity, the closure equations are not presented in this paper; however, the complete model description is available in our previous publications [2] and [17].

The model equations are solved on the multi-segment structured non-uniform curvilinear grid that allows accurate representation of the nozzle geometry. Second order upwind discretizations are utilized for the convective terms in the momentum transport equations; a flux limiter approach is used for the particle number density and the volume fractions. As there is a strong coupling between the phases, the phase momentum equations are solved together for each velocity component with the interfacial drag and virtual mass terms implicitly discretized. The modified Spalding’s InterPhase Slip Algorithm (IPSA) [20] is utilized to provide the pressure-velocity coupling. To solve the linear system of equations that arises after the discretization, the Generalized Minimal RESidual (GMRES) algorithm [21] with the Incomplete Lower Upper decomposition of zero degree (ILU(0)) preconditioning is employed. The computational code has been developed by the authors at the University of British Columbia and applied to a wide range of multiphase flow problems.

**Description of cases**

A standard industrial size TEB nozzle [11], schematically shown in Figure 2, is chosen for our investigations. It contains a short convergent section (the contraction angle equals 31°), which is preceded by a relatively short strait pipe and followed by a longer divergent section (4°), and finishes with a short convergent section (25°) with a small straight orifice piece.

![Figure 2. TEB nozzle schematics (not to scale). All dimensions are in mm.](image)

We consider flow parameters that are expected in an industrial Fluid Coker operation and have already been used in our previous work [22]. Steam and bitumen heated to 350 °C are used as working fluids at GLR 0.87%. The nozzle is issuing spray in quiescent steam, and the fluidized bed is not considered. The modeling domain includes the nozzle and the downstream spraying area stretching to 1 m axially and 0.15 m radially. The gage pressure at the boundary of this downstream area equals 3.4475×10⁶ Pa (60 psi). Inlet flow rates and other necessary physical properties of the steam and bitumen are listed in Table 1. In order to simplify our calculations, we ignore gravity and assume that the flow has axial symmetry. No-slip boundary conditions are assumed at the nozzle walls.

**Table 1. Flow parameters and physical properties.**

<table>
<thead>
<tr>
<th></th>
<th>Steam</th>
<th>Bitumen</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mass flow rate, kg/s</strong></td>
<td>0.026</td>
<td>3.0</td>
</tr>
<tr>
<td><strong>Density, kg/m³</strong></td>
<td></td>
<td>851</td>
</tr>
<tr>
<td><strong>Viscosity, Pa s</strong></td>
<td>2.2×10⁻⁵</td>
<td>0.01</td>
</tr>
<tr>
<td><strong>Molar mass, kg/mol</strong></td>
<td>0.018</td>
<td></td>
</tr>
<tr>
<td><strong>Surface tension, N/m</strong></td>
<td></td>
<td>0.018</td>
</tr>
</tbody>
</table>

Next, it is important to characterize the inlet non-uniformity. While there is only one uniform inlet distribution for any given combination of flow rates, there can be many non-uniform distributions even within the confines of the axial symmetry assumption. First of all, we consider only spatial variations, leaving the transient non-homogeneity out of current consideration. Second, we assume that the phase velocity is uniform across the inlet and is the same for steam and bitumen. The consequence of this assumption is that all the difference is now contained in the volume fraction values because all other components of the flow rates (the density and velocity) are fixed. This second assumption actually brings us close to the representation of the worst case scenario because the interfacial velocity difference would promote the turbulence generation and, as a result, improve mixing. The volume fractions can change along the radial coordinate with any frequency. However, the higher frequency produces the larger surface areas available for mixing. Therefore, following our intention to investigate the worst case scenario, we choose the lowest possible frequency. That is, the maximum concentration of each component is always either at the center or at the periphery (nozzle wall) and the minimum concentration is correspondingly either at the periphery or at the center. Evidently, it can be either steam or bitumen in higher concentrations at the pipe center, and there can be variations of how the volume fractions change across the profile.

Some representative cases are schematically shown in Figure 3. They range from the worst situation, where the phases are completely separate and there is no premixing, Figure 3 (a) and (d), to a milder center to circumference gradient, Figure 3 (c) and (f). For simplicity, we assume that the volume fraction variation is linear. To facilitate the quantification of the non-uniformity, we plot volume fraction profiles for the
liquid phase along the radial coordinate for three representative cases. The difference between the profiles shown in Figure 4 is clearly seen. Note that the radial location of the completely separated flow (curve (a)) interface is uniquely determined from the equal velocity constraint. It is apparent that up to three different flow zones can be formed: the liquid flow, the gas flow, and the mixture flow with the linear variation of volume fractions. Evidently, the slope of the curve (or the middle part of the curve) can be used to characterize non-homogeneity. We denote $\alpha_1$ bitumen concentration at the wall, and $\alpha_2$ bitumen concentration at the center. $\alpha_1$ can change between zero and its homogeneous value (around 0.5), and $\alpha_2$ can change from the homogeneous value to unity. Also, we mark $R_1$ the inner radius at which the bitumen volume fraction starts to change and $R_2$ the outer radius at which the volume fraction stops changing. $R_i$ can change from zero to the completely separated flow interface value (about 8.6 cm for TEB nozzle), and $R_2$ can change from the separated flow interface value to the pipe radius $R$. Therefore, the slope of the curve can be easily obtained as $(\alpha_2 - \alpha_1)/(R_2 - R_1)$. Finally, we non-dimensionalize the slope relationship with the pipe radius to obtain the uniformity criterion $k$.

$$k = R \frac{\alpha_2 - \alpha_1}{R_2 - R_1}$$  \hspace{0.5cm} (8)

It can be seen that for the liquid-in-the-center cases, $k$ can vary from zero – for the uniform distribution – to the infinity – for the complete phase separation. As it is marked in Figure 4, for $k > 1$ all three zones are present with the middle zone decreasing in size as $k$ grows, finally disappearing when $k$ arrives at infinity. For $0 < k < 1$, there are no clear liquid or gas zones since there is a certain degree of phase mixing that already exists at the center and at the periphery.

Similar discussion also applies to the cases with the higher gas concentrations at the center of the nozzle. Figure 5 presents typical liquid volume fraction profiles for these cases. Keeping the same notations for $\alpha_i$ and $R_i^1$, it is evident that the uniformity criterion $k$ is going to be negative. Analogously to the already considered cases, the shape of the profile changes at $k = -1$ from having three zones to only one, and the complete separation of phases is achieved as $k$ reaches negative infinity.

In our numerical investigation we consider the full range of $k$ values, from $-\infty$ through zero to $\infty$, to represent various degrees of the radial non-uniformity.

\footnote{Note that the ranges of variations for variables $\alpha_1$ and $\alpha_2$ would change to the ranges of $\alpha_2$ and $\alpha_1$ in the liquid centered cases correspondingly.}

![Image](315x180 to 540x324)

**Figure 3.** Gas-liquid distribution at the nozzle entrance. Black color represents bitumen and white color represents steam; (a) $k \to \infty$; (b) $k > 1$; (c) $0 < k < 1$; (d) $k \to -\infty$; (e) $k < -1$; (f) $-1 < k < 0$.

![Image](325x590 to 365x622)

**Figure 4.** Inlet radial profiles of volume fractions; (a) $k \to \infty$; (b) $k > 1$; (c) $0 < k < 1$.

![Image](365x571 to 378x585)

**Figure 5.** Inlet radial profiles of volume fractions; (a) $k \to -\infty$; (b) $k < -1$; (c) $-1 < k < 0$.
Results and discussion

A computational grid shown in Figure 6 has been developed for the previously described domain that includes the nozzle and the surrounding area. The grid contains 10,130 cells non-uniformly distributed to provide a good resolution inside the nozzle and immediately downstream of it and at the same time keep computational costs to a minimum.

We started our simulations with a uniform case ($k = 0$) and obtained a transient solution. The time-variations, however, are rather small. Liquid volume fraction distribution in the spray shown in Figure 7 demonstrates a fairly narrow jet with peripheral maximum of droplets concentration. It is interesting that it has visibly less dispersion than a comparable air-water case [17]. In addition, the peripheral maximum is much stronger and develops almost immediately after the atomization.

Figure 8 shows the average droplet diameter distribution in the spray. For better presentation, the areas where the liquid volume fractions are too small ($\alpha_d < 10^{-4}$) are blanked out. It can be seen that the droplet size is smaller in the jet center and gradually increases towards the circumference. This structure is consistent along the length of the jet. Such behavior is quite different from the previously studied air-water cases [2, 17], where the largest droplet diameter is located at the center, and the droplet size reduces for about $\frac{3}{4}$ of the spray width. Even though those cases do have a secondary peripheral droplet diameter maximum, it is smaller and less pronounced. These observations underline the existing uncertainty with a proper scaling procedure to substitute steam-bitumen mixture at elevated pressure and temperature with other materials suitable for experiments at standard conditions in research labs.

Before proceeding further with the rest of the simulations, we ensure that the solution is not grid dependent. In addition to the original grid described above (Figure 6), we developed two more grids (medium and coarse) by coarsening the mesh distribution with 7,836 and respectively 5,544 grid cells. The solution analysis demonstrates that while the coarse grid solution deviates substantially from that obtained on the original grid, the medium grid solution is reasonably close to it. Therefore, the original grid is sufficient for the grid independent solution, and it is utilized for the investigations presented further in this paper.
Figure 9. Bitumen mass flow rate profiles for $k \geq 0$ (a) and $k \leq 0$ (b) across the spray at two different axial distances from the nozzle exit.

In order to evaluate the impact of the inlet flow uniformity on the spray, we simulated ten additional cases that cover the whole range of the uniformity criterion $k$: $\{0; -5; -0.75; -0.45; 0.45; 0.75; 2; 5; \infty\}$. As the converged solutions were obtained, we observed that the transient effects became more noticeable with increasing the absolute value of $k$, especially for negative $k$. Nevertheless, the transient spray fluctuations are still relatively minor, and the time-averaged solution is fully representative of the process. In the following, the time-averaged solutions are compared.

We start the evaluation with the cross-sectional profiles of the bitumen flow rate shown in Figure 9 for various $k$ at two different axial distances. It is evident that the inlet non-uniformity has little effect on the droplets’ flow rate distribution in the spray. Very minor variations are reported for positive $k$ (higher concentration of liquid at the nozzle center) and slightly more noticeable variations are present for negative $k$ (higher concentration of gas at the center). It is likely that the negative $k$ solutions deviations from the uniform case that results in a slightly wider spray are the consequences of the time-dependent fluctuations. This effect does not dissipate with the increase of the distance from the nozzle.

Next, we turn our attention to the average droplet diameter profiles shown at Figure 10 for 0.3 m distance from the nozzle orifice. Similar to the flow rate profiles, the curves are fairly close to each other for all values of $k$. However, it is also visible that the less uniform cases (higher absolute value of $k$) exhibit slightly more uniform cross-sectional distribution. That is, for these cases the droplet diameter is slightly larger at the jet center and slightly lower at the periphery. It is also instructive to plot the droplet diameters that are mass-averaged across the full spray cross-section at different axial distances from the nozzle exit against the uniformity criterion $k$. Such graphs are presented in Figure 11. It confirms our previous observation that the droplets produced by the nozzles with various degrees of inlet non-uniformity are rather similar. The droplet average di-
Diameter steadily increases due to coalescence as the droplets move away from the nozzle. Despite the noise present in the results, we can still see a small tendency for the liquid-centered inlet flow to produce smaller droplets than the gas-centered flow. This tendency persists with the increase of the axial distance from the nozzle orifice.

Now, as we discovered that the inlet radial non-uniformity does not play a significant role in the resulting spray properties, we examine the flow in the nozzle itself to see and recognize the reasons for such behavior. Figure 12 shows the liquid volume fraction contours inside the nozzle for all investigated inlet $k$ values. The difference in the initial conditions is clearly seen on the plots. It is apparent that the flow is determined by the initial conditions until the first convergent section as there is no noticeable change of the phase distribution along the straight inlet piece for almost all of the cases. The only exceptions are the two fully separated flow cases ($k = -\infty, k = \infty$) for which the sharp boundary between the phases becomes slightly dispersed shortly after the entrance. While some mixing starts to occur as the flow passes through the first convergent section, the mixing intensifies greatly at the beginning of the divergent section. The higher the degree of the initial non-homogeneity is, i.e. $|k|$, the longer it takes for the profile to reach uniform distribution. Nevertheless, by the end of the divergent section, all cases exhibit similar volume fraction distributions.

![Figure 11](image1.png)  
**Figure 11.** Cross-sectionally mass-averaged droplet diameter variation for various $k$ at different axial distances from the nozzle exit.

![Figure 12](image2.png)  
**Figure 12.** Liquid volume fraction contours inside the nozzle for various $k$.

![Figure 13](image3.png)  
**Figure 13.** Air velocity contours inside the nozzle for various $k$. (in m/s)

![Figure 14](image4.png)  
**Figure 14.** Mixture turbulent kinematic viscosity contours inside the nozzle for various $k$. (in m$^2$/s)
In order to understand the flow better, we plot contours of the air velocity in Figure 13 and the mixture turbulent kinematic viscosity in Figure 14. We can see that the flow structure in the first convergent section differs depending on the initial phase distribution. For the liquid-centered flows, the bitumen tends to occupy a larger portion of the cross-section squeezing the steam to the peripheral layer and, as a result, increasing the steam velocity near the wall. For the gas-centered flows, the picture is reverse as the bitumen is directed radially by the section profile towards the center from the periphery. Consequently, the steam velocity near the pipe axis is increased. Due to a large phase density difference (about 100 times), the effect of the bitumen being pushed to the center is much stronger. It is confirmed by the higher velocity values at the end of the convergent section compared with the liquid-centered cases. The higher velocity leads to the increased turbulence production by shear that can be observed on turbulent viscosity plots (Figure 14). This, in turn, facilitates mixing that is driven by the turbulent dispersion. Additionally, the negative radial momentum obtained by the bitumen promotes mixing naturally by forcing phases together. This momentum also destabilizes a steam-bitumen interface that can explain more transient fluctuations for the gas-centered cases discussed previously. All of it has the consequence of the observed length of the part of the divergent section where the phases are still separated being shorter for the cases with negative $k$ values when compared with the corresponding positive $k$ cases.

It is interesting to see that the most turbulence after the first convergent section is generated for the cases with the uniform or close to the uniform phase distribution. This happens because the one of the main mechanisms of the turbulence production is dependent on the interfacial velocity. This velocity increases as the flow passes through the convergent section because of the difference in the inertia between the phases [17]. However, the slip velocity would generate appreciable turbulence only if there is a sufficient local concentration of both phases.

As the flow distribution is mostly equalized before the second convergent section, the turbulence generated there is comparable for most cases. Still, a closer look reveals slightly higher values for more uniform cases ($|k| < 1$) and somewhat lower values for the gas-centered cases ($k < -1$). This can be explained in part by some residual turbulence that has not dissipated as the flow passes through the divergent section and remains in the flow. As the flow turbulence parameters are passed along through the phase inversion surface to the spray, the turbulence present in the flow before the phase inversion plays a major role in the secondary droplet break-up. Therefore, we can explain some key features of average droplet diameter variations with $k$ observed in Figure 11 by connecting droplet sizes to the turbulence available in the flow. Thus, the cases with the near uniform inlet distribution produce smaller droplets than all others, and the gas-centered cases produce smaller droplets than the liquid-centered cases.

Following our analysis, we can see the way the particular features of the TEB nozzle geometry contribute to its consistent performance. The first convergent section helps to generate turbulence that is necessary for good mixing. The divergent section provides a sufficient space where this mixing can take place, preparing the flow for the final step by distributing phases across the cross-section. The second convergent section produces a pressure drop required for the fast and efficient atomization by phase inversion and generates turbulence necessary to promote the secondary droplet break-up.

**Conclusions**

A comprehensive mathematical model developed for the gas-assisted premixed atomization that resolves the flow inside as well as outside of the nozzle is applied to investigate the influence of the inlet mixture radial uniformity on the performance of the TEB type of effervescent nozzles. The uniformity criterion $k$ that is related to the inlet volume fraction gradient is proposed and utilized in the investigations.

The analysis of simulation results demonstrated that the radial variations of the inlet volume fractions have only a minor influence on the resulting liquid mass flux and the average droplet diameter distributions in the spray. This relative independence from the inlet mixture radial homogeneity can be attributed to the high degree of mixing taking place inside the nozzle itself. The turbulence generated at the first convergent section of the nozzle is sufficient to facilitate mixing inside the long divergent area before reaching the second convergent section that determines the resulting spray properties.

**Acknowledgements**

This work was financed by Natural Sciences and Engineering Research Council of Canada (NSERC) and Syncrude Canada Ltd.

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>specific heat</td>
</tr>
<tr>
<td>$d$</td>
<td>diameter</td>
</tr>
<tr>
<td>$f$</td>
<td>frequency</td>
</tr>
<tr>
<td>$H$</td>
<td>total enthalpy</td>
</tr>
<tr>
<td>$I$</td>
<td>unit tensor</td>
</tr>
<tr>
<td>$F$</td>
<td>force</td>
</tr>
<tr>
<td>$k$</td>
<td>uniformity criterion</td>
</tr>
</tbody>
</table>
\( k' \)  turbulent kinetic energy  
\( M \)  molar mass  
\( n \)  particle number density  
\( p \)  pressure  
\( R \)  radial distance  
\( R_g \)  universal gas constant  
\( Sc_t \)  turbulent Schmidt number  
\( T \)  temperature  
\( t \)  time  
\( V \)  velocity  
\( \alpha \)  volume fraction  
\( \mu \)  viscosity  
\( \rho \)  density  
\( \tau \)  stress  

Subscripts  
\( br \)  break-up  
\( c \)  continuous  
\( cl \)  coalescence  
\( d \)  dispersed  
\( g \)  gas  

Superscripts  
\( t \)  turbulent  

References  