**APPLICATION OF THE DIRECT QUADRATURE METHOD OF MOMENTS TO Y-JET WATER SPRAYS**

Jesper Madsen¹, Bjørn H. Hjertager², Tron Solberg³, Preben Nørskov⁴ and Jan Rusås⁵

¹Aalborg University Esbjerg, Denmark, jema@aaue.dk
²Aalborg University Esbjerg, Denmark, bhh@aaue.dk
³Aalborg University Esbjerg, Denmark, tron@aaue.dk
⁴Danfoss A/S, Denmark, px@danfoss.dk
⁵Aalborg Industries A/S, Denmark, jr@aalborg-industries.com

**ABSTRACT**

The recently formulated direct quadrature method of moments (DQMOM) is implemented in an Eulerian multi-fluid CFD code (Fluent 6.2) to simulate the droplet size distribution (DSD) in Y-jet water sprays. The essential idea behind the modeling approach is to solve continuum Eulerian equations for the gas phase and N droplet phases. Each droplet phase is characterized by a volume fraction and a diameter. The rate of breakup and coalescence determines the source terms of the transport equations of the N volume fractions and N diameters. These source terms are defined through a linear system involving the first 2N integer moments of the DSD. In this paper, two-dimensional axisymmetric computations of the atomization process of Y-jet water sprays are presented. For verification of the computations, phase Doppler anemometer (PDA) measurements have been conducted for several atomizing air ratios. The model is qualitatively assessed by examining the predicted spray characteristics such as droplet velocities and droplet sizes and their distribution, and the dependence on parametric changes.

**Keywords:** CFD, Multi-fluid model, DQMOM, Droplet size distribution, Breakup, Coalescence, PDA, Y-jet atomizer

1. INTRODUCTION

Y-jet atomizers are extensively used in oil fired utility boilers. Together with the burner aerodynamics, the atomizers represent the core of the combustion system and their good performance plays a major role to keep a low level of pollutant formation and to maintain the combustion efficiency as high as possible.

To predict the performance of a Y-jet atomizer, both the gas-liquid flow within the nozzle and the atomization process of the liquid sheet shed from it have to be modeled. According to an experimental investigation of Mullinger and Chigier [1], the annular flow regime can be assumed for the two-phase flow in the mixing chamber of the nozzle.

This paper aims to provide a complete description of the spray from a Y-jet atomizer, including the modeling of the liquid sheet atomization and secondary breakup and coalescence of droplets. The analysis of the atomization process of a Y-jet nozzle is a highly complex problem due to the gas-liquid flow interactions, which cause violent mixing of the liquid and the gas.

Sprays are usually modeled based on the Lagrangian treatment of representative parcels of droplets tracked in the surrounding gas field [2], which is also known as Discrete Droplet Model (DDM). However, the DDM approach has several limitations. The gas volume fraction in the computational cell has to be large, i.e. greater than 99%, for the assumptions of the approach to be valid. To enforce this criterion the smallest grid size that may be employed in the near-nozzle region has to be much greater than the sheet thickness. This grid size is insufficient to resolve the sharp gradients in the near-nozzle region. Furthermore the DDM approach cannot be employed to represent the liquid sheet.

Several alternate approaches to the DDM have been reported in the literature. In this work, an Eulerian multi-fluid model is employed to simulate the atomization process of Y-jet water sprays. The recently formulated direct quadrature method of moments (DQMOM) [3,4] is implemented to simulate the evolution of the droplet size distribution (DSD). The numerical limitations highlighted above are minimized with the DQMOM-multi-fluid approach.

For verification of the computations, phase Doppler anemometer (PDA) measurements have been conducted for several atomizing air ratios.

2. THE DQMOM-MULTI-FLUID MODEL

The multi-fluid model that is employed here for Y-jet sprays has been described elsewhere [4]. The essential idea behind the modeling approach is to solve continuum Eulerian equations for the gas phase and N droplet (liquid) phases. For each phase mass and momentum balances are solved as well as corresponding equations for turbulent kinetic energy and its dissipation rate. The DQMOM representation of the DSD involves the solution of an equation for the diameter for each droplet phase. The rate of droplet breakup and coalescence determines the source terms of the transport equations of the N volume fractions and N diameters. These source terms are defined through a linear system involving the first 2N integer moments of the DSD. Unlike the standard moment methods, with DQMOM the volume fractions and diameters are tracked directly, rather than the moments themselves.

Submodels are employed for the following physics:

- The interfacial momentum transfer between gas and droplet
phases includes drag and turbulent dispersion forces. The drag force is modeled through the Schiller-Naumann drag correlation. Balance equations for both turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \) are solved for each phase where corresponding interfacial transfer terms are included. The WAVE atomization model is employed to simulate liquid atomization and droplet breakup. Finally collision-coalescence and collision-breakup models are employed in which the relative velocity is assumed to be proportional to the local droplet turbulence properties.

2.1 Direct Quadrature Method of Moments (DQMOM)

The modeling approaches presented above allow one to simulate the velocity fields of the gas-droplet flow. When the aim of the study is to predict the evolution of the size distribution of droplets undergoing breakup and coalescence, the population balance has to be solved. The population balance is obtained as [4]

\[
\frac{\partial}{\partial t} n(d) + \frac{\partial}{\partial d} \left[U_r(d) n(d)\right] = S(d)
\]

where \( n(d) \) is the droplet number distribution, \( d \) is the droplet diameter, \( U_r(d) \) is the average velocity conditioned on \( d \), and \( S(d) \) is the source term, which contains breakup and coalescence. The main idea behind the approach is the solution of the closure problem, related to the evolution of the moments of the DSD, by using a quadrature approximation of order \( N \). This corresponds to the approximation of the DSD as follows:

\[
n(d) = \sum_{q=1}^{N} \omega_q \delta[d - d_q]
\]

where \( \omega_q \) is the weights (droplet number density) and \( d_q \) is the abscissas (droplet diameter) of the quadrature approximation, and \( \delta \) indicates the Dirac delta function. The \( k \)th moment of the distribution can be written as

\[
m_k = \int_0^\infty d^k n(d) \, d\,d = \sum_{q=1}^{N} \omega_q d_q^k
\]

It can be shown that, given \( 2N \) moments of the distribution, the values of \( N \) weights and \( N \) abscissas of the quadrature approximation can be calculated, forcing them to yield known values of the moments, either resorting to the product-difference (PD) algorithm and solving an eigenvalue-eigenvector problem [5], or directly solving the transport equations of weights and abscissas [3]. The DQMOM is based on the latter approach. In this work, the DQMOM, combined with the multi-fluid CFD model, consists in the solution of the following equations for the \( q \)th droplet phase volume fraction \( \omega_q \) and diameter \( d_q \):.

\[
\frac{\partial}{\partial t} \left( \alpha_q \rho \right) + \frac{\partial}{\partial d} \left( \alpha_q \rho U_{i,q} \right) = \frac{2\pi}{3} \alpha_q \rho d_q^3 S_{\delta_q} - \frac{\pi}{2} \alpha_q \rho d_q^3 S_{\omega_q}
\]

and

\[
\frac{\partial}{\partial t} \left( \alpha_q \right) + \frac{\partial}{\partial d} \left( \alpha_q U_{i,q} \right) = \frac{2\pi}{3} \alpha_q \rho d_q^3 S_{\delta_q} - \frac{\pi}{2} \alpha_q \rho d_q^3 S_{\omega_q}
\]

where \( \rho_l \) is the liquid-phase density and \( U_{i,q} = U_i(d_q) \) is the \( q \)th droplet phase velocity. \( S_{\delta_q} \) is the source term for the droplet number density \( \omega_q \), which is related to the droplet volume fraction by

\[
\omega_q = \frac{\delta_q \alpha_q}{\pi d_q^3}
\]

and \( S_{\omega_q} \) is the source term for the weighted diameter \( \delta_q = \omega_q d_q \). It is possible to show that the source terms can be evaluated solving a linear algebraic system, obtained from the population balance equation after application of the quadrature approximation and forcing the moments to be tracked with high accuracy [3]. This linear system can be written in matrix form as:

\[
Ax = b
\]

where the \( 2N \times 2N \) coefficient matrix \( A = [A_1, A_2] \) is defined by

\[
A_1 = \begin{bmatrix}
1 & \cdots & 1 \\
0 & \cdots & 0 \\
-d_1^2 & \cdots & -d_N^2 \\
\vdots & \ddots & \vdots \\
(2(1-N)d_1^{2N-1}) & \cdots & (2(1-N)d_N^{2N-1})
\end{bmatrix}
\]

and

\[
A_2 = \begin{bmatrix}
0 & \cdots & 0 \\
1 & \cdots & 1 \\
2d_1 & \cdots & 2d_N \\
\vdots & \ddots & \vdots \\
(2(N-1)d_1^{2N-2}) & \cdots & (2(N-1)d_N^{2N-2})
\end{bmatrix}
\]

The 2N vector of unknowns \( x \) is defined by

\[
x = [S_{m_1}, \cdots, S_{m_{2N}}, S_{\delta_1}, \cdots, S_{\delta_N}]^T
\]

and the known right-hand side is

\[
b = [\bar{S}_{m_1}, \cdots, \bar{S}_{m_{2N-1}}]^T
\]

where \( \bar{S}_{m_q} \) is the source term for the \( k \)th moment defined by

\[
\bar{S}_{m_q} = \int_0^\infty d^k S(d) \, d\,d
\]

These source terms for the moments, resulting from the summation of several contributions, each one corresponding to a specific process, need to be evaluated in order to solve the linear system in Eq. (7). Applying the quadrature approximation, Eq. (2), the source terms for the moments of the distribution due droplet breakup,
coalescence of two droplets, and collision induced breakup (fragmentation) of two droplets are evaluated as

\[
S_{n_s} = \sum_{q=1}^{N} \sum_{p=1}^{N} \frac{1}{2} \left( d_q^{(1)} + d_q^{(2)} \right)^{1/3} c_{pq} \omega_p \omega_q - \sum_{p=1}^{N} \sum_{q=1}^{N} d_q^{(2)} c_{pq} \omega_p \omega_q
\]

where \( a_q \) is the breakup kernel, which is the frequency of breakup of a droplet of diameter \( d_q \), \( c_{pq} \) is the coalescence kernel, and \( e_{pq} \) is the collision induced breakup kernel. \( \bar{B}_q^{(i)} \) and \( \bar{F}_{pq}^{(i)} \) are the moment transforms of the breakup daughter distribution functions \( b(d|d_q) \) and \( f(d|d_q,d_p) \), which contain information about the daughter diameter of the droplets during the breakup of a droplet of diameter \( d_q \) and the collision-induced breakup of two droplets of diameters \( d_p \) and \( d_q \), respectively:

\[
\bar{B}_q^{(i)} = \frac{\gamma}{\omega_q} \int_0^\infty d^4 b \left( d|d_q \right) dd
\]

\[
\bar{F}_{pq}^{(i)} = \frac{\gamma}{\omega_p \omega_q} \int_0^\infty d^4 f \left( d|d_p,d_q \right) dd
\]

2.2 Droplet Breakup

To model atomization, the blob injection model is employed, in which liquid blobs are injected with a diameter equal to the sheet thickness. The primary breakup of these blobs and secondary breakup of droplets is modeled using the standard WAVE breakup model [6]. In this work symmetric fragmentation is considered, i.e. the breakup of an unstable droplet with diameter \( d_q \) produces two droplets of diameter \( \frac{2^{-1/3}}{d_q} \). The daughter distribution function corresponding to this breakup mechanism is:

\[
b \left( d|d_q \right) = \begin{cases} 2 & \text{if } d = 2^{-1/3} d_q \\ 0 & \text{otherwise} \end{cases}
\]

The moment transform of the daughter distribution function becomes:

\[
\bar{B}_q^{(i)} = 2 \left( 3^{-1/3} \right) d_q^{i}
\]

The original WAVE model is formulated via a rate approach for Sauter mean diameter (SMD) reduction of the parent drops within the DDM model [6]. This has been transformed to the breakup kernel by:

\[
a_q = \frac{1 - d_{st}/d_q}{(2^{1/3} - 1) \tau_{eq}} = \frac{1 - d_{st}/d_q}{0.26 \tau_{eq}} \quad d_{st} < d_q
\]

where \( \tau_{eq} \) is the breakup time and \( d_{st} \) is the stable diameter [6]. This ensures that the SMD is consistent with the original model. However, other daughter distribution functions [4] or other models using the rate approach can easily be implemented.

2.3 Droplet-Droplet Collision

The droplet-droplet collision model is semi-empirical and has two stages. The first stage is to determine the collision rate between droplets. Collision rates are expressed by a collision coefficient \( \beta_{pq} \) defined such that \( \beta_{pq} n(d) n(d) \) is the number of collisions per unit volume and time between two sizes of droplets having number densities \( n(d) \) and \( n(d) \). According to the kinetic theory [7], the collision coefficient is given by:

\[
\beta_{pq} = \pi d_{pq}^{2} U_{rel}
\]

where \( d_{pq} = (d_p + d_q)/2 \) and \( U_{rel} \) is the relative velocity between the two droplet classes. The following form for the average relative velocity \( U_{rel} = |\hat{U}_p - \hat{u}_q| \) between colliding droplets is assumed:

\[
U_{rel} = \sqrt{U_p - U_q} + 2(1 - \rho_{pq})(k_p + k_q)
\]

where \( \hat{U} \) is the mean droplet phase velocity and \( k \) is the droplet phase turbulent kinetic energy. \( \rho_{pq} \) is a correlation coefficient \( (0 \leq \rho_{pq} \leq 1) \), describing the correlation of the fluctuating velocity. Considering two droplets separated by \( d_{pq} \) (for two droplets in contact) the correlation coefficient is assumed to be:

\[
\rho_{pq} = 1 - \frac{3.6}{k_p + k_q} \left( \frac{\varepsilon_{pq} d_{pq}}{2} \right)^{2/3}
\]

where \( \varepsilon \) is the droplet turbulent kinetic energy dissipation rate.

The second stage of the model determines the outcome of the collisions. The model takes coalescence and bounce into account, as well as collision-induced breakup (fragmentation). With the collision coefficient, the expression for the coalescence kernel becomes:

\[
c_{pq} = \min \left( E_{beam}, E_{coll} \right) \beta_{pq}
\]

where the probability of coalescence at low \( We_{coll} \) is [8]

\[
E_{beam} = \min \left[ 1, \left( \frac{\text{We}_{coll}}{4.8 \rho(f(y))^2} \right)^{0.3} \right]
\]

and the probability of coalescence at higher \( We_{coll} \) is [9]

\[
E_{coll} = \min \left[ 1, \left( \frac{4.8 f(y)}{\text{We}_{coll}} \right) \right]
\]

\( \text{We}_{coll} \) is the collisional Weber number defined as

\[
\text{We}_{coll} = \frac{\rho U_{rel}^2 d_{coll} \sigma}{\varepsilon}
\]
where \( d_{\text{small}} \) is the diameter of the smaller of the interacting droplets and \( \sigma \) is the surface tension. The function \( f(\gamma) \) is defined as [8]

\[
f(\gamma) = \gamma^3 - 2.4\gamma^2 + 2.7\gamma
\]  

(25)

where \( \gamma = \frac{d_{\text{large}}}{d_{\text{small}}} \) is the droplet size ratio.

One possible outcome of droplet-droplet collisions is the breakup of the original droplets into smaller droplets. The collision-induced breakup kernel is expressed as

\[
e_{pq} = (1 - E_{\text{cool}}) \beta_{pq}
\]  

(26)

Georjon and Reitz [10] proposed a model for "shattering" collisions between droplets. They assumed that after the droplets collide they form a ligament, which stretches due to the inertia of the collision. An estimation for the diameter of the droplet fragments \( d_{\text{frag}} \) is given by a correlation of Post and Abraham [11]

\[
d_{\text{frag}} = \frac{1.89\left(d_p^3 + d_l^3\right)\beta}{\sqrt{2.01Wk_c^2\left(1 + \gamma^2\right)^{1/2} + 1}}
\]  

(27)

The moment transform of the daughter distribution function corresponding to this collision induced breakup mechanism is

\[
\mathcal{F}_{pq}^{(t)} = \left(d_p^3 + d_l^3\right)\beta
\]  

(28)

3. EXPERIMENTAL SETUP

To validate the CFD model, an experimental test rig was established. Compressed air and tap water were used to supply the gas-liquid spray system, and measurements were carried out with a phase Doppler anemometer (PDA).

According to the requirement of oil fired boilers, several single-hole Y-jet nozzles were designed, one of which was chosen to be the reference and used for comparison in this work. A schematic of the Y-type atomizer used in this work is shown in Figure 1. The most important dimensions of the Y-jet nozzle tested are the diameters of the air-injection orifice \( D_a = 2.6 \) mm, the liquid-injection orifice \( D_l = 1.7 \) mm, the mixing chamber \( D_m = 3.7 \) mm, and the length of the mixing chamber \( L_m = 14.8 \) mm.

Experiments were performed with a liquid mass flow rate of 190 kg/h, and a ratio between the atomization air to the liquid flow rate (mass loading ratio \( \text{MLR} = m_l / m_a \)) of 3.75%, 5.00%, and 6.25%. For characterization with the PDA system, the atomizer was positioned in the vertical direction with its tip pointed downward. The droplet size and axial velocity distributions were measured at up to 25 different locations along the radial direction \( r \) at the cross sections located at the spray axial locations of \( x = 110, 150 \) and 190 mm, respectively. For all the radial profiles presented, the data represents the laser entrance where the laser beam blockage by the spray was low.

Figure 1: Y-jet atomizer.

4. COMPUTATION DETAILS

The computations are performed on two two-dimensional axisymmetric computational grids using the CFD code Fluent 6.2. Figure 2 shows the grids (A and B) for the Y-jet atomizer. The nozzle and spray regions are represented within a single calculation domain. The spray domain is the same for both grids and 422.6 mm long by 100 mm in radius with 170\times63 cells. In Grid A, the nozzle mixing chamber is 14.8 mm (= \( L_m \)) long by 1.85 mm in radius with 15\times9 cells. In Grid B, the nozzle mixing chamber is reduced to 1 cell in the axial direction with a length of 1 mm.

The liquid injection is comprised of the wall-adjacent cell and the air injection is comprised of the central 8 cells. The liquid film injection is modeled by specifying a uniform droplet size equal to the film thickness \( d_{\text{inj}} = 206 \mu m \) and a velocity of \( U_{\text{inj}} = 23.4 \) m/s at the boundary. The core airstream has a flat velocity profile with velocities of \( U_{\text{inj},g} = 190, 254, \) and 317 m/s for the three mass loading ratios, respectively.

Figure 2: Computational grids. Spray domain together with a close-up view near the nozzle of Grid A and Grid B.

The flow in the mixing chamber of the nozzle and in the near-field is sensitive to the injection boundary values of \( k \) and \( \varepsilon \). To assess the effect of the assumed boundary
conditions for turbulence intensities, $I_l$ and $I_g$, and turbulent length scales, $\ell_l$ and $\ell_g$, computations are performed for Grid B with different sets of turbulence boundary values. Table 1 shows a summary of the runs made.

Table 1: Description of the simulations made. $D_j = 1.7$ mm, $D_a = 2.6$ mm, $d_{ij} = 206$ μm.

<table>
<thead>
<tr>
<th>Run #</th>
<th>Grid</th>
<th>$B_1$</th>
<th>Turbulence boundary values</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A</td>
<td>10</td>
<td>4 6 0.07D_j 0.07D_a</td>
</tr>
<tr>
<td>2</td>
<td>B</td>
<td>10</td>
<td>4 6 0.07D_j 0.07D_a</td>
</tr>
<tr>
<td>3</td>
<td>B</td>
<td>1.73</td>
<td>4 6 0.07D_j 0.07D_a</td>
</tr>
<tr>
<td>4</td>
<td>B</td>
<td>1.73</td>
<td>10 10 0.07D_j 0.07D_a</td>
</tr>
<tr>
<td>5</td>
<td>B</td>
<td>1.73</td>
<td>10 10 0.14d_{ij} 1.12d_{ij}</td>
</tr>
<tr>
<td>6</td>
<td>B</td>
<td>1.73</td>
<td>5 4 0.14d_{ij} 1.12d_{ij}</td>
</tr>
</tbody>
</table>

The WAVE model constant $B_1$ [6] has been given a variety of values between 1.73 and 80. In this study, the computations are performed with two values of $B_1$, Reitz [6] used $B_1 = 10$ and based on comparisons to the TAB model $B_1$ has been set as $B_1 = \sqrt{3} \approx 1.73$.

It is well known that the standard k-ε model does not fit the data for single phase axisymmetric jets. A common practice is to modify one of the constants in the model, $C_{\mu}$, form 1.92 to 1.87. This correction is applied in the present simulations.

The nozzle and spray regions are treated with the same DQ MOM-multi-fluid model where the droplet breakup and coalescence models are applied in both regions. The model is tested with $N = 3$ droplet phases.

The measurements were made in steady sprays whereas the computational model solves the time-dependant governing equations. The time step used for all Runs is $2.5 \times 10^{-7}$ s, corresponding to a maximum Courant number of 0.1. The subsequent discussion in the next section focuses on the steady axial velocities and SMD values.

### 5. RESULTS AND DISCUSSION

#### 5.1 Comparison with Experiment

The radial profiles of the axial droplet velocity calculated with the DQ MOM-multi-fluid approach (Run #4) are compared with the measured ones (Exp.) in Figure 3, and the radial distribution of the droplet SMD is shown in Figure 4. The profiles are determined at three axial locations for $MLR = 5.00\%$.

It is seen that the measured mean velocity reaches a maximum value at the spray centerline, and decreases toward the spray edges as the spray spreads out. The air entrainment from the surroundings to the spray region is responsible for this behavior of the mean velocity distribution. The variation of the velocity profile along the spray axis $x$ in the downstream direction indicates the expansion of the spray region caused by the effect of spray-air interaction and air entrainment. Along the downstream direction, the variation of the droplet velocity near the central region is just opposite to that near the edge. Near the center, the axial mean velocity decreases as the distance from the nozzle increases, whereas close to the edge it increases downstream. At the nozzle exit, the initial large velocity difference between the liquid and the airflow results in high velocities of the droplets because of the strong momentum transfer and the small sizes of the droplets. After a certain distance downstream of the nozzle exit, the droplets are accelerated to the velocity of the air, which slows down as a result of the loss of its momentum to the droplets and the entrained surrounding air. Then as the droplets and the co-flowing air are transported further downstream and spread out, they are decelerated because of the air entrainment from the surroundings. Thus, the velocity profile flattens as the spray region expands downstream along the spray axis.

![Figure 3: Comparison of the calculated and measured radial profiles of the axial droplet velocity for $MLR = 5.00\%$.](image3.png)

![Figure 4: Comparison of the calculated and measured radial profiles of the SMD for $MLR = 5.00\%$.](image4.png)
velocity and the high fluctuating component of the droplet velocities in the spray central region results in small droplets there because of aerodynamic breakup and collision-induced breakup. The small droplets are entrained by the high-velocity core airstream and are accelerated to higher velocities in the central region of the spray. The larger droplets, having larger inertia, tend to follow their own trajectories, and are less affected by the air entrainment motion near the spray edges. It is also seen that some increase in the SMD with the axial distance occurs near the central region, whereas close to the edge the SMD decreases downstream, similar to that observed by Li and Shen [12]. This is mainly because of relatively large droplets migrating toward the central region.

The predicted radial distribution of axial velocity is shown for Run #4 in Figure 3. The centerline axial velocity is predicted to low, and the droplet velocities at larger radial distances are over-predicted. This is a phenomenon that is seen in all runs made on Grid B, suggesting that this may be an injection condition problem.

Radial distributions of predicted droplet SMD are shown in Figure 4. The trend of increasing droplet size with radial distance is correct, but the rate of change is too large, with SMD of the small droplets on the centerline under-predicted and the SMD of the large droplets toward the periphery over-predicted. There are two main reasons for this. The first is that the sheet breakup in the computation results in large droplets from the injected blobs with many surrounding smaller droplets, especially between the "sheet" and the spray axis. This results in small values of the axis SMD and large values of the periphery SMD. The other reason is that the collision rate of droplets in the spray may be over-predicted. The rate of change of droplet size with axial distance is approximately correct.

5.2 Parametric Tests

The influence of grid, droplet breakup, and turbulence boundary conditions on the spray will be discussed next. Computations for all mass loading ratios show similar trends.

The results obtained at the axial location of $x = 190$ mm from running the simulation on two different grids are shown in Figure 5 and Figure 6 for $MLR = 6.25\%$. The long nozzle mixing chamber in Grid A cause the droplets to be accelerated to the velocity of the air, hence the droplet velocity farther downstream of the nozzle exit is over-predicted. At the nozzle exit in Grid B the lower droplet velocities result in a faster slow down of the air velocity and lower droplet velocities downstream. The smallest downstream SMD values are predicted when using Grid A. This is due to the large region within the nozzle with high velocity difference between the phases, resulting in small droplets there because of droplet breakup and thus smaller downstream SMD values.

In Figure 7 the spray droplet size is seen to be sensitive to the value of the WAVE model breakup constant $B_1$. Smaller droplet sizes result as $B_1$ is lowered because of faster droplet breakup. The droplet axial velocity is not sensitive to the reduced droplet breakup time, so this figure is not presented here.
effect to that seen in single phase jets and two-phase particle laden jets in which the jets are spread more as the turbulence intensity is increased. This demonstrates that the effect is due to the breakup and coalescence of the droplets. In the case of Run #3 and #6, the larger droplets, having larger inertia, tend to follow their own trajectories, and are less affected by the air entrainment motion near the spray edges. As a result the spray spreads more.

Runs #4 and #5 show that the values of the SMD and the axial velocity are not very sensitive to the injection boundary values of turbulent length scales \( \ell_1 \) and \( \ell_g \).

The parametric tests suggest that the error is in the injection boundary values, and that the conditions computed in the near-nozzle region are inaccurate. Unfortunately, it is not possible to perform rigorous tests since the experimental values in the near-nozzle region are not known.

5.3 Droplet Size Distribution (DSD)

The DQMOM does not directly give information about the DSD since it tracks only its moments. Nevertheless, the droplet diameters \( d_q \) and weights \( \omega_q \) can be seen as an approximation of the real DSD. The calculated droplet number distribution in comparison with the experimental data for MLR = 5.00% is given in Figure 10. Four measured DSDs at different positions are shown and the differences between them are small, only a higher amount of small droplets can be observed for the distributions nearest to the spray centerline. Towards the spray edges existence of larger droplets can be observed. Additionally, the results of droplet size predictions of Run #4 are shown. A reasonable correspondence between droplet diameters and number densities and the measured DSD is evident. The same shape of the DSD is predicted. Only the existence of larger droplets and the SMD is somewhat over-predicted.

Figure 8: Effect of different injection turbulence boundary values (Table 1) on predicted SMD for MLR = 3.75%.

Figure 9: Effect of different turbulence boundary values (Table 1) on predicted axial velocity for MLR = 3.75%.

Figure 10: Normalized DSDs (number density function) at four different positions for MLR = 5.00%. The gray bars represent the experimental data. The DQMOM prediction of Run #4 is represented by the vertical black peaks centered on the respective droplet phase diameter \( d_p \), whose height is the corresponding number density \( \omega_q \) normalized by the total number density \( m_0 \).
The performance of the DQMOM-multi-fluid model for Y-jet water sprays has been discussed. The DQMOM removes the need to discretize the DSD into fixed size classes by means of tracking the moments of the DSD. One of the advantages of the DQMOM-multi-fluid model is that it can be used on different kinds of nozzles.

In this paper the model has been applied to the simulation of Y-jet water sprays and compared with experimental data. It has been shown that the model is able to reproduce measured trends in steady droplet SMD and axial velocities. The parametric explorations have served to demonstrate that the model reacts in the correct qualitative manner to changes in input parameters. However, the measured values are not reproduced qualitatively with adequate accuracy.

The accuracy of the calculation results strongly depends on the consistent modeling of the nozzle flow characteristics and the models adopted for breakup and collision processes. Hence, careful validation of the individual models and the availability of proper boundary conditions at the nozzle exit serve as major prerequisites for the successful analysis and optimization of Y-jet sprays.

7. ACKNOWLEDGEMENTS

This work is financially supported by Danfoss and Aalborg Industries.

8. NOMENCLATURE

\( a_q \) breakup kernel \([s^{-1}]\)
\( B_l \) WAVE breakup constant \([-]\)
\( b(q)d_q \) breakup daughter distribution function \([\text{m}^3]\)
\( B_q \) moment transform of \( b(q)d_q \) \([\text{m}^3]\)
\( c_{pq} \) coalescence kernel \([\text{m}/\text{s}]\)
\( d \) droplet diameter \([\text{m}]\)
\( E \) coalescence efficiency \([-\%]\)
\( e_{pq} \) collision induced breakup kernel \([\text{m}/\text{s}]\)
\( f_{pq} \) moment transform of \( f(q,d_q,d_q) \) \([\text{m}^3]\)
\( I \) turbulent intensity \([-\%]\)
\( k \) turbulent kinetic energy \([\text{m}^2/\text{s}^3]\)
\( \ell \) turbulent length scale \([\text{m}]\)
\( \text{MLR} \) mass loading ratio \([-\%]\)
\( m_k \) \(k^{th}\) moment of the DSD \([\text{m}^3/\text{m}^3]\)
\( N \) number of droplet phases \([-\%]\)
\( n(d) \) droplet number distribution \([\text{m}^3]\)
\( r \) radial coordinate \([\text{m}]\)
\( S \) source term \([-\%]\)
\( U \) velocity \([\text{m/s}]\)
\( We \) Weber number \([-\%]\)
\( x \) axial coordinate \([\text{m}]\)

Greek letters
\( \alpha \) volume fraction \([-\%]\)
\( \rho_{pq} \) collision coefficient \([\text{m}^3/\text{s}]\)
\( \delta_q \) weighted diameter \([\text{m}^3]\)
\( \varepsilon \) turbulent kinetic energy dissipation rate \([\text{m}^3/\text{s}^2]\)
\( \gamma \) droplet size ratio \([-\%]\)
\( \omega_q \) weight (droplet number density) \([\text{m}^3]\)

\( \rho \) density \([\text{kg/m}^3]\)
\( \rho_{pq} \) correlation coefficient \([-\%]\)
\( \tau_{bu} \) breakup time \([\text{s}]\)

Subscripts
\( boun \) bounce
\( coal \) coalescence
\( coll \) collision
\( frag \) fragments
\( g \) gas
\( inj \) injection
\( l \) liquid
\( p \) \( p^{th}\) droplet phase
\( q \) \( q^{th}\) droplet phase
\( rel \) relative
\( st \) stable

9. REFERENCES