Article

Influence of Modification of the Geometry of the Wave-Plate Mist Eliminators on the Droplet Removal Efficiency—CFD Modelling

Łukasz Makowski , Jakub Łaskowski and Mariusz Tyrański

Faculty of Chemical and Process Engineering, Warsaw University of Technology, Waryńskiego 1, 00-645 Warsaw, Poland; j.laskowsky@gmail.com (J.Ł.); mariusz.tyranski.dokt@pw.edu.pl (M.T.)

* Correspondence: lukasz.makowski.ichip@pw.edu.pl

Abstract: This study is concerned with droplet separation using wave-plate mist separators. The influence of continuous phase velocity, droplet size, and geometry on droplet removal efficiency has been investigated. The following modifications were analysed: drainage channel presence, length, angle, and modification to the streamlined shape. Furthermore, the influence of the physical parameters of the separated substances on separator efficiency was investigated. Calculations were conducted using computational fluid dynamics (CFD). The results were compared with the experimental data from the literature. Based on the results obtained, a new shape of drainage channels was proposed, which is characterised by high droplet removal efficiency with relatively low pressure drop.

Keywords: wave-plate mist eliminator; computational fluid dynamics; droplet removal efficiency

Citation: Makowski, Ł.; Łaskowski, J.; Tyrański, M. Influence of Modification of the Geometry of the Wave-Plate Mist Eliminators on the Droplet Removal Efficiency—CFD Modelling. *Processes* **2021**, *9*, 1499. <https://doi.org/10.3390/pr9091499>

Academic Editor: Jacek Gebicki

Received: 19 July 2021

Accepted: 23 August 2021

Published: 25 August 2021

Publisher's Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.

Copyright: © 2021 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

The necessity of removing liquid droplets from the process gas streams is a common problem in industrial technologies. Rising raw materials prices and strict environmental protection plans have become challenging to design more innovative and efficient systems for the droplet removal from process gases and vapour streams. The presence of gas-liquid aerosols (mist) can result from spontaneous condensation or desublimation in supersaturated gas-vapour mixtures or chemical reactions. It is undesired in many gas-liquid contact devices such as absorbers, quench coolers, or condensers [1]. Eliminating droplets from the gaseous phase aims to prevent the failure of process equipment requiring the use of gases or vapours with high purity, protecting the environment against pollution and degradation, recovering valuable substances, or realising the desire to obtain a product of the highest purity. Wave-plate mist eliminators are applied when necessary to remove droplets of size ranging from a few to tens of micrometres. However, removing very fine droplets requires different solutions such as a glass fibre bed separator intended for removing droplets of a diameter smaller than 2 µm [1]. Wave-plate mist eliminators need to have appropriate geometry (in practice, the channel through which the gas flow has a zig-zag shape, which causes a sharp change in the direction of the flowing fluid stream). The gas containing liquid droplets flows across the bends, forcing it to change the flow direction a few times. Due to their high density compared to gas, liquid droplets are subject to inertial forces, which causes the deflection of the droplet trajectories from the gas motion patch and separation of droplets on the walls. Those forces depend on droplet mass and velocity as well as the geometry of mist eliminator [2]:

$$F_s = \frac{m_p u_p^2}{R} = \frac{\pi d_p^3 \rho_p u_p^2}{6R} \quad (1)$$

where F_s represents inertial force, R is a bending place curvature radius, and m_p , u_p , d_p , and ρ_p are the mass, velocity, diameter, and density of the droplets, respectively. It can be seen that inertial force is proportional to the d_p^3 . Due to this fact, wave-plate mist eliminator efficiencies decrease sharply for smaller droplets. For example, in the same conditions, for 37.5 μm droplets, the separation efficiency of the wave-plate separator was over 90%, but for droplets of diameter less than 25 μm , it dropped sharply to less than 40% [3]. Droplets that have not been re-entrained by the gas can adhere, coalesce, form a film, and flow down under the influence of gravity to be removed from the separator space. The advantage of inertial droplet separators is a low-pressure drop. They can be used to remove foam-forming liquids and high-viscosity liquids, which in the case of mesh and fibre separators is ineffective due to the robust clogging spaces between wires or fibres by liquid droplets. Furthermore, most wave-plate separators have relatively higher resistance to droplet re-entrainment than wire mesh separators, although they cannot obtain high efficiency under conditions of low gas velocities or/and fine droplets [3].

Three essential parameters affect the efficiency of the inertial droplet separator [4]:

- The linear velocity of the gas;
- Separator channels' width;
- Presence of drainage channels.

As the inertial force value is proportional to the square of the velocity, increasing the gas velocity has a complex influence on droplet removal efficiency. Below 6 m/s, the increase in velocity is beneficial for efficiency; however, for velocities above this value, the influence of secondary droplet entrainment increases, reducing droplet removal efficiency. Nevertheless, for some wave-plate separators with multiple bends, re-entrainment will not occur as long as the gas velocity is lower than 8 m/s [3]. High velocities can also cause flooding phenomenon if a mist eliminator is working in the vertical position. In that case, an upward gas flow prevents the gravitational downflow of droplets [2]. The presence of drainage channels allows maintaining high efficiency even at high linear gas velocities. The shape and type of drainage channels can significantly affect the droplet removal efficiency of the separator.

The problem of optimising the operation of an inertial mist eliminator has been widely described in the literature [5–10]; however, an interesting issue is the influence of drainage channel geometry on droplet removal efficiency. Computational fluid dynamics has a unique role in understanding this process. It allows understanding the system's conditions better and easily estimates the influence of process parameters and geometry changes on droplet removal efficiency.

Some successful CFD analyses of airflow and droplet transport wave-plate mist eliminators are described in the literature [11,12]. There is also proof that the length of drainage channels has a crucial influence on droplet removal efficiency and pressure drop [11]. The novelty of this paper is presenting CFD simulations of the influence of the different aspects of the geometry of drainage pockets such as length, angle, and shape on the droplet removal efficiency and different types of droplets and process medium velocity. The first part of the work is focused on the selection of turbulence and turbulent dispersion models. Next, the crucial part of the work presents the influence of the presence, width, and slope angle of drainage channels. The results concern not only the efficiency but also the pressure drop in the system. Based on the results obtained, a new shape of drainage channels was proposed, which is characterised by a high efficiency of droplet removal with reasonably low pressure drop. In addition, droplet flows of media used in the chemical industry such as n-octane, diesel fuel, sulphuric acid, and aqueous ammonia solution were simulated in the new system.

2. Materials and Methods

2.1. Materials

The present work assumed that air is a continuous phase and water is a discrete phase. Section 3.4 also included parameters of three other media used to simulate discrete phases.

2.2. Experimental System

In the present work, different configurations of commercial wave-plate mist eliminators have been investigated using numerical methods. In the first stage, two types of geometries described by [5] were analysed: one without drainage channels (geometry A) and another with zig-zag shaped drainage channels (geometry B_10_0). Visualisations of considered geometries are presents in Figures 1 and 2. Those results were used to validate chosen numerical methods for further calculations.

Figure 1. Wave-plate mist eliminator with inflow and outflow area (geometry A) dimensions in mm.

Figure 2. Visualisation of wave-plate mist eliminator with drainage channels (geometry B_10_0).

In the second stage of work, variants with analogical drainage channels of different lengths (a), which reduce the width of the main channels (L), were simulated. Secondly, there were modelled variants with drainage channels of 10 mm length at different angles (α). Geometrical details of every configuration are listed in Table 1 and presented in Figure 3.

Table 1. Geometrical details of analysed mist-wave eliminators.

Geometry	Presence of Drainage Channels	a [mm]	L [mm]	α [°]
A	No	-	35.5	-
B_10_0	Yes	10.5	25	0
B_10_30	Yes	10.5	25	30
B_10_60	Yes	10.5	25	60
B_10_90	Yes	10.5	25	90
B_15_0	Yes	15.5	20	0
B_20_0	Yes	20.5	15	0
B_25_0	Yes	25.5	10	0
B_30_0	Yes	30.5	5	0

Figure 3. Details of geometries of analysed wave-plate mist eliminators dimensions in mm.

2.3. Simulation Parameters and Assumptions

In order to analyse the effects of using different geometries, medium velocities, and droplet diameters, the commercial software Fluent developed by Ansys Inc. was used. Calculations were carried out using the 2D domain. Calculations were carried out for two different average velocities on the inlet: 2.0 m s^{-1} and 4.0 m s^{-1} , which are typical values in industrial practice. The upper limit of velocity was determined to avoid dragging droplets.

In order to conduct simulation, the following were assumed:

- Air is a continuous phase;
- Water is a discrete phase;
- Inlet overpressure equal to 5 MPa;
- Gravitation force on droplets is negligible;
- No coalescence between droplets;
- Droplets behave as hard spheres;
- No droplet-droplet interaction;
- Droplets do not form a film on the walls;
- Droplets do not affect velocity profiles of gas phase;
- Lift forces are negligible;
- Droplets are removed immediately after hitting a wall;
- Droplet diameters range from 1 to $40 \text{ }\mu\text{m}$;
- Mass fraction of water was lower than 0.1.

The design assumptions presented above are commonly used to determine the effect of geometry on system efficiency [3–5,11]. In particular, this refers to the value of the mass fraction of water, which below 0.1 should not lead to the formation of a liquid film on the wall, and the phenomenon of droplet coalescence should not significantly affect the system efficiency. The agreement between simulation results and experimental data presented later in this paper confirms the applicability of the model assumptions.

The viscosity and density of continuous phase (air) and discrete phase (water) were specified as constant in constant temperature equal to $15 \text{ }^{\circ}\text{C}$. Those parameters are listed below:

- $\rho_{\text{air}} = 1.225 \text{ kg m}^{-3}$;
- $\nu_{\text{air}} = 1.461 \cdot 10^{-5} \text{ m}^2 \text{ s}^{-1}$;
- $\rho_{\text{water}} = 999 \text{ kg m}^{-3}$.

2.4. Continuous Phase Modelling

The SIMPLE method was used for the pressure–velocity coupling, and second-order discretisation schemes were used for all variables to minimise numerical diffusion effects [13]. Computations were regarded as satisfactory converged when total normalised residuals were smaller than 10^{-6} . Computational meshes were generated using Ansys

Meshing software and for every variant of geometry. The size of numerical grids was approximately 850,000 quadrilateral cells. The mesh was densest in the wave-plate region, and the near-wall cell sizes were set accordingly to satisfy the $y^+ \sim 1$ condition. Mesh independence was checked at the highest tested Reynolds numbers using two quantities: average wall shear stress value at the walls and average turbulence energy dissipation rate in the system. The results of both these two quantities were constant (less than 3% difference), even with using denser meshes than those described. The obtained mesh was validated by comparing calculated results with experimental data. The final mesh for the B_10_0 variant was presented in Figure 4.

Figure 4. Final computational mesh of geometry B_10_0: (a) view on a fragment of mist eliminator; (b) detailed view on drainage channel.

In the first stage, simulation of B_10_0 geometry was conducted, and obtained results were compared with accurate experimental data with a maximum uncertainty below 5% presented in [5] to select the appropriate turbulence model for further calculations. The gas-phase velocity in a broad part of wave-plate separators is relatively high, and turbulent flow occurs, which means that the $k-\epsilon$ model should be accurate. However, wave-plate

channels are narrow, and the flow is less turbulent, making the k - ω model more precise than k - ε in this part of the device. Furthermore, the SST k - ω includes the turbulence shear stress transport, and it is accurate for simulating flow under near-wall conditions [2]. In the present work, three turbulence models based on Reynolds-averaged Navier–Stokes (RANS) were compared:

- k - ε with standard and enhanced wall treatment;
- Transition SST k - ω ;
- Reynolds Stress Model (RSM) with enhanced wall treatment.

Many works have been published in recent years presenting the possibility of using more advanced mathematical models, so-called large eddy simulation (LES), to simulate industrial processes [13–16], enabling the simulation of processes in a wide range of Reynolds numbers. However, it should be remembered that such a solution increases both the calculation time and the demand for computing power many times. It is recommended to use it only when the calculation results are significantly more accurate than those obtained using lower-order models. In a similar process, [17] carried out calculations using RANS and LES models. They observed that differences between models results are 3% and less than 7% on average for the pressure drop and the removal efficiency, respectively, and the RANS approach can be used successfully significantly enhanced with a turbulent dispersion of droplets model.

The following equations govern the continuity and momentum in RANS models:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (2)$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j}(-\rho \overline{u'_i u'_j}). \quad (3)$$

For the k - ε model, the turbulent kinetic energy, k , and the specific dissipation rate of k , ε , are obtained from the following equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (4)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon. \quad (5)$$

For the SST k - ω model, the turbulent kinetic energy, k , and the specific dissipation rate of k , ω , are obtained from the following equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k + G_b \quad (6)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega + G_{\omega b}. \quad (7)$$

The Reynolds Stress Model is more complicated than the other turbulence models mentioned above because of using anisotropic Reynolds stress tensors. It requires most computational resources, and it was used only as a reference.

2.5. Discrete Phase Modelling

Once continuous phase flow is computed, the droplet behaviour can be calculated. In order to simulate droplet motion, a Discrete Phase Model (DPM) was applied. This model

uses a Lagrangian approach for modelling droplet trajectories. Droplet motions are tracked using Newton's second law [2]. The following equation governs forces acting on droplets:

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} + F_x \quad (8)$$

where u and u_p are gas and droplet velocities, respectively; ρ and ρ_p are gas and droplet densities, respectively; g is gravitational acceleration, and F_x is an additional acceleration. In this work, one-way coupling between continuous and discrete phases was considered. It is assumed that droplet movement does not affect the continuous phase flow pattern. In Equation (8), $F_D(u - u_p)$ is a resistant force of continuous phase per droplet mass ($m s^{-2}$), and it can be calculated from the Reynolds number and droplet parameters. Coefficient F_D can be expressed as:

$$F_D = \frac{18\mu}{\rho_p d_p^2} \frac{C_D Re}{24} \quad (9)$$

where Re is the relative Reynolds number, and C_D is the drag force coefficient, which is obtained from the following expressions [18]:

$$C_D = 0.44 \text{ for } Re_d > 1000 \quad (10)$$

$$C_D = \frac{1 + 0.15 Re_d^{0.687}}{\frac{Re_d}{24}} \text{ for } Re_d \leq 1000 \quad (11)$$

where Re_d is the droplet Reynolds number defined by:

$$Re_d = \frac{\rho |u - u_p| d_p}{\mu} \quad (12)$$

For efficiency calculations, we used droplets of size within the range from 1 μm to the size when the calculated efficiency equals 100%. In all cases, droplets were injected in a perpendicular direction to the inlet plane. Efficiency is defined as a ratio of the number of droplets removed from continuous phase ($c_{in} - c_{out}$) to the total injected droplets (c_{in}):

$$\eta = \frac{c_{in} - c_{out}}{c_{in}} \cdot 100\% \quad (13)$$

The droplet motion was calculated using two variants of the DPM model:

- Without turbulent dispersion;
- With turbulent dispersion affecting droplet motion.

To include the phenomenon of turbulent dispersion in simulating droplet motion, stochastic tracking model was used, which allows calculating droplet motion trajectories by using instantaneous velocity:

$$u(t) = \bar{u} + u'(t). \quad (14)$$

Many particles are injected at one point in a stochastic model, and their motion trajectories are calculated using a fluctuation module.

The results of all RANS models described in Section 2.4 were verified using experimental data [5]. Figure 5 and Tables 2–5 show a summary of results and relative errors for geometry B_10_0, which has the same structure and size as used in the work of [5]. In this case, the turbulent dispersion model was implemented to improve turbulent models used for calculations. The influence of including this model in calculations will be presented in the next section.

Figure 5. Comparison of calculated results with experimental data for geometry B_10_0 with inlet velocity: (a) 2 m s^{-1} ; (b) 4 m s^{-1} .

Table 2. Comparison of calculated results with experimental data for geometry B_10_0 with inlet velocity 2 m s^{-1} .

Droplet Diameter [μm]	Experimental Data [5] [-]	k- ϵ , std. Wall tr. [-]	k- ϵ , enh. Wall tr. [-]	RS, enh. Wall tr. [-]	SST with turb. disp. [-]
8	0.20	0.766	0.487	0.605	0.270
10	0.48	0.807	0.619	0.717	0.468
12	0.75	0.809	0.688	0.762	0.695
13	0.93	0.819	0.732	0.802	0.798
15	0.98	0.856	0.828	0.895	0.946
18	0.99	0.901	0.913	0.954	0.996
21	1	0.944	0.968	0.992	1

Table 3. Relative errors between calculated results and experimental data for geometry B_10_0 with inlet velocity 2 m s^{-1} .

Droplet Diameter [μm]	k- ϵ , std. Wall tr. [%]	k- ϵ , enh. Wall tr. [%]	RS, enh. Wall tr. [%]	SST with turb. disp. [%]
8	282.98	143.40	202.29	34.91
10	68.21	28.86	49.41	2.58
12	7.87	8.27	1.6	7.33
13	11.88	21.27	13.76	14.18
15	12.65	15.51	8.67	3.47
18	8.99	7.79	3.64	0.61
21	5.60	3.20	0.80	0

Analysing Figure 5 and Tables 2–5, one can assume that all used models predict the trend of increase in efficiency with droplet size. However, there are significant differences in expected values. The most accurate results were observed using the SST model, especially for higher efficiencies. Furthermore, the SST model is the only one that accurately predicts almost 100% efficiency for droplets with a diameter of $18 \mu\text{m}$ (2 m s^{-1}) and $12 \mu\text{m}$ (4 m s^{-1}), which are very close results to experimental data. This conclusion is similar to [5] and [15]. As a result, the SST model was found as the most suitable for simulating the gas phase.

Table 4. Comparison of calculated results with experimental data for geometry B_10_0 with inlet velocity 4 m s^{-1} .

Droplet Diameter [μm]	Experimental Data [5] [-]	k- ϵ , std. Wall tr. [-]	k- ϵ , enh. Wall tr. [-]	RS, enh. Wall tr. [-]	SST with turb. disp. [-]
6	0.20	0.759	0.616	0.634	0.328
7	0.40	0.797	0.707	0.719	0.493
8	0.68	0.837	0.789	0.799	0.686
10	0.86	0.883	0.883	0.902	0.944
11	0.95	0.887	0.897	0.924	0.982
13	1	0.904	0.922	0.959	0.995

Table 5. Relative errors between calculated results and experimental data for geometry B_10_0 with inlet velocity 4 m s^{-1} .

Droplet Diameter [μm]	k- ϵ , std. Wall tr. [%]	k- ϵ , enh. Wall tr. [%]	RS, enh. Wall tr. [%]	SST with turb. disp. [%]
6	279.50	208.00	217.00	64.00
7	99.13	76.65	79.80	23.37
8	23.14	16.07	17.47	0.88
10	2.68	2.73	4.83	9.76
11	6.66	5.60	2.70	3.34
13	9.57	7.77	4.06	0.5

2.6. Comparing Calculations with and without Turbulent Dispersion

In Figure 6 and Tables 6–9, the results and relative errors with and without using turbulent dispersion for the SST model are presented and compared with experimental data.

Figure 6. Comparison of calculated results obtained with SST model with and without using turbulent dispersion with experimental data for geometry B_10_0 with inlet velocity: (a) 2 m s^{-1} ; (b) 4 m s^{-1} .

Table 6. Comparison of calculated results obtained with SST model with and without using turbulent dispersion with experimental data for geometry B_10_0 with inlet velocity 2 m s^{-1} .

Droplet Diameter [μm]	Experimental Data [5] [-]	SST without turb. disp. [-]	SST with turb. disp. [-]
8	0.20	0.072	0.270
10	0.48	0.138	0.468
12	0.75	0.228	0.695
13	0.93	0.485	0.798
15	0.98	1	0.946
18	0.99	1	0.996
21	1	1	1

Table 7. Relative errors between calculated results obtained with SST model with and without using turbulent dispersion with experimental data for geometry B_10_0 with inlet velocity 2 m s^{-1} .

Droplet Diameter [μm]	SST without turb. disp. [%]	SST with turb. disp. [%]
8	63.83	34.91
10	71.25	2.58
12	69.60	7.33
13	47.81	14.18
15	2.04	3.47
18	1.01	0.61
21	0	0

Table 8. Comparison of calculated results obtained with SST model with and without using turbulent dispersion with experimental data for geometry B_10_0 with inlet velocity 4 m s^{-1} .

Droplet Diameter [μm]	Experimental Data [5] [-]	SST without turb. disp. [-]	SST with turb. disp. [-]
6	0.20	0.080	0.328
7	0.40	0.276	0.493
8	0.68	0.471	0.686
10	0.86	0.778	0.944
11	0.95	0.889	0.982
13	1	1	0.995

Table 9. Relative errors between calculated results obtained with SST model with and without using turbulent dispersion with experimental data for geometry B_10_0 with inlet velocity 4 m s^{-1} .

Droplet Diameter [μm]	SST without turb. disp. [%]	SST with turb. disp. [%]
6	60.00	64.00
7	31.08	23.37
8	30.69	0.88
10	9.54	9.76
11	6.42	3.34
13	0	0.5

Analysing Figure 6 and Tables 6–9, one can observe that more accurate results, compared to the experimental results, were received using the SST model with turbulent dispersion, particularly for droplets of small diameter obtaining higher efficiency, which is following expectations, because for smaller droplets, the model predicts a higher acceleration of droplets, impacting fluctuations of the continuous phase. Based on the obtained results, all subsequent calculations will be done using the SST model with turbulent dispersion as the most suitable model describing the work of wave-plate mist eliminator.

2.7. Non-Dimensional Numbers

Calculated values of Reynolds number on the inlet, for hydraulic diameter $d_h = 0.68$ m, are presented in Table 10.

Table 10. Reynolds number values for different variants of calculation.

The Average Velocity on the Inlet [m s^{-1}]	Reynolds Number [-]
2.0	94,114
4.0	188,277

The Reynolds number is the most important non-dimensional number considered case, because it illustrates the nature of gas flow (Re) and droplet flow (Re_d). It allows one to select the appropriate turbulence model for CFD calculations and parameters used in the equations. However, the Stokes number, representing the ratio between droplet relaxation time and gas characteristic time, is also loosely related to droplet removal efficiency, even though it is not related to turbulent dispersion effects [5]. Since the characteristic length is channel width, the Stokes number can be calculated by the following equation:

$$St = \frac{\rho_p d_p^2 \left(\frac{u_{in}}{\sin \alpha_c} \right)}{18\mu(L \sin \alpha_c)} \quad (15)$$

where u_{in} is gas velocity on inlet and α_c is the bend angle between wave-plates. The relation between Stokes number and removal efficiency is presented in Figure 7. It can be noticed that the regression curve shape is similar to the droplet diameter–efficiency curve obtained from calculations.

Figure 7. Comparison of calculated Stokes numbers with results obtained with the SST model with and without using turbulent dispersion with experimental data for geometry B_{10_0} with inlet velocity: (a) 2 m s^{−1}; (b) 4 m s^{−1}.

2.8. Boundary Conditions

For the continuous phase, the following boundary conditions were used:

- “Velocity-inlet” with constant value on the inlet to the computational domain;
- “Outflow” on the outlet of the computational domain;

- “Stationary wall” without slip on the edges of the wave-plate mist eliminator and drainage channels;
- “Symmetry” on the edges of the computational domain outside the wave-plate mist eliminator excluding inlet and outlet.

For the discrete phase, the following boundary conditions were used:

- “Trap” on the edges of the wave-plate mist eliminator and drainage channels;
- “Escape” on the rest of the edges of the computational domain.

Droplets of the dispersed phase were put into the domain through the inlet by the uniform injections of spheres separately for every diameter from investigated range (1 to 40 μm , every 3 μm) with corresponding velocity and constant total mass flow rate of 0.00355 kg s^{-1} . Every injection was calculated separately.

3. Results

From a practical and industrial point of view, three parameters were crucial to simulate flow fields of analysed geometries:

- The removal efficiency of droplets;
- Pressure drop on wave-plate mist-eliminator;
- The mean force acting on the single drainage channel.

Pressure drop was calculated as a difference between pressures on the inlet and outlet of investigated mist eliminators. The mean force acting on the drainage channel was calculated as an arithmetic average from forces acting on every drainage channel. Calculation results for both parameters are listed in Tables 11 and 12.

Table 11. Pressure drops on mist eliminators and the mean force acting on drainage channel; inlet velocity 2 m s^{-1} .

Geometry	A	B_10_0	B_10_30	B_10_60	B_10_90	B_15_0	B_20_0	B_25_0	B_30_0
Pressure drop [Pa]	82	87	106	151	205	109	246	587	1977
Mean force acting on drainage channel [N]	-	0.38	0.48	0.63	0.78	1.38	4.20	11.64	43.61

Table 12. Pressure drops on mist eliminators and the mean force acting on drainage channel; inlet velocity 4 m s^{-1} .

Geometry	A	B_10_0	B_10_30	B_10_60	B_10_90	B_15_0	B_20_0	B_25_0	B_30_0
Pressure drop [Pa]	346	367	446	643	872	442	1000	2371	7960
Mean force acting on drainage channel [N]	-	1.53	1.96	2.59	3.26	5.62	17.11	47.07	175.81

Based on the above results, it can be assumed that according to Darcy–Weisbach law, the pressure drop (Δp) increases proportionally to the square of the medium velocity (u):

$$\Delta p \sim u^2. \quad (16)$$

The results of pressure drop are following the literature data [2,14]. From an economic point of view, the pressure drop is an undesirable phenomenon. As a result, geometry variant B_30_0 most likely has no practical usage. Furthermore, it can be observed that the force acting on the drainage channel is proportional to the pressure drop on the mist eliminator. In the following chapters, the influence of various parameters was considered in detail.

3.1. Influence of Gas Inlet Velocity on Droplet Removal Efficiency on Wave-Plate Mist Eliminator

The calculations were carried out for two inlet velocities, 2 m s^{-1} and 4 m s^{-1} , and two characteristic geometries were investigated:

- A;
- B_10_0.

The obtained results are shown in Figure 8.

Figure 8. Comparison of calculated results of droplet removal efficiency for geometries A and B_10_0 with inlet velocity: (a) 2 m s^{-1} ; (b) 4 m s^{-1} .

The results confirm that the increase of the inlet velocity of the gas increases the droplet separation efficiency. In B_10_0, it can be seen that for the speed of 4 m s^{-1} , almost 100% efficiency is obtained for the droplet diameter of 12 μm (18 μm for the lower velocity). Nevertheless, it should be noted that this is done at the cost of more than four times higher pressure drop. The results also confirm that drainage channels positively affect separation efficiency for larger droplet diameters, increasing it by about 20–30%. For droplet sizes smaller than 9 μm for 2 m s^{-1} and 7 μm for 4 m s^{-1} , the drop separation efficiency is higher for a system without channels, which is obviously due to the inertia forces that act on the drops. In a system with drainage channels, the cross-section near the channels decreases, which causes an increasing velocity, and smaller drops follow the gas rather than deposition on the walls.

3.2. Influence of Drainage Channels Length on Droplet Removal Efficiency on Wave-Plate Mist Eliminator

The next step was to investigate the influence of the length of drainage channels crucial for the mist eliminator. In the present work, we investigated geometry variants with drainage channels with the following lengths:

- 10.5 mm—geometry B_10_0;
- 15.5 mm—geometry B_15_0;
- 20.5 mm—geometry B_20_0;
- 25.5 mm—geometry B_25_0;
- 30.5 mm—geometry B_30_0.

The obtained results are shown in Figure 9.

Based on the results, the droplet removal efficiency increases with drainage channel length. This phenomenon is associated with the increased velocity of the continuous phase, decreased area of flow, and increased area of the drainage channel. Figures 10 and 11 show velocity fields and vectors of investigated geometries. It can be observed that the highest gas velocities are obtained in the close neighbourhood to drainage channels and that the velocity increases with the next channel, which means that self-similarity flow conditions for the subsequent cross-sections near the channel have not been achieved. These results are similar to those of [5].

Figure 9. Comparison of calculated results of droplet removal efficiency for geometries with different drainage channel lengths with inlet velocity: (a) 2 m s^{-1} ; (b) 4 m s^{-1} .

Figure 10. Cont.

Figure 10. Velocity fields (a,c) (m s^{-1}) and vectors (b,d) and vectors (m s^{-1}) of investigated geometry variants with different drainage channel lengths: (a,b) B_10_0; (c,d) B_25_0; with inlet velocity 2 m s^{-1} simulated using a transition SST model.

Figure 11. Cont.

Figure 11. Velocity fields (a,c) (m s^{-1}) and vectors (b,d) and vectors (m s^{-1}) of investigated geometry variants with different drainage channel lengths: (a,b) B_{25_0}; (c,d) B_{30_0}; with inlet velocity 4 m s^{-1} simulated using a transition SST model.

With areas with high-velocity values, stagnation areas and recirculation areas are observed in and just behind drainage channels. All observed phenomena cause the drainage channels with higher lengths to provide better results, although it also generates higher pressure drops, which are undesirable. Those variants of geometries can be used if high-pressure drops are acceptable.

3.3. Influence of Drainage Channels Angle on Droplet Removal Efficiency on Wave-Plate Mist Eliminator

Simulations were conducted for geometry variants with different angles of drainage channels. Geometry B_10_0 was modified to obtain three different angles of drainage channels:

- 30°—geometry B_10_30;
- 60°—geometry B_10_60;
- 90°—geometry B_10_90.

The results in Figures 12–14 show the velocity fields of the investigated geometries.

Figure 12. Comparison of calculated results of droplet removal efficiency for geometries with different drainage channel angles with inlet velocity: (a) 2 m s^{−1}; (b) 4 m s^{−1}.

Figure 13. Cont.

(b)

(c)

Figure 13. Velocity fields (m s^{-1}) of investigated geometry variants with different drainage channel angles: (a) B_10_30; (b) B_10_60; (c) B_10_90; for inlet velocity 2 m s^{-1} , simulated using a transition SST model.

Droplet removal efficiency increases with drainage channel angle (α). This phenomenon can result from moving drainage channels to areas with higher gas velocities, which causes higher inertial force acting on droplets in those areas and generates higher local pressure drops. The maximum velocity increases with the slope of the channels. In B_10_0, the maximum velocity is 5.5 times higher than the system inlet velocity, and in B_10_90, it is 8.5 times higher. Simultaneously, the separation efficiency for drainage channel slopes of 0 and 30 degree increases by about 40%, with the pressure drop by about 25%. Higher angles can also increase secondary droplet entrainment; however, the model does not include this effect.

Figure 14. Velocity fields (m s^{-1}) of investigated geometry variants with different drainage channel angles: (a) B_10_30; (b) B_10_60; (c) B_10_90; for inlet velocity 4 m s^{-1} , simulated using a transition SST model.

3.4. Influence of Physicochemical Properties of the Droplet on Removal Efficiency on Wave-Plate Mist Eliminator

Mist eliminators are used for removing droplets of media with different physicochemical properties. In the present work, we analysed the influence of different types of droplet materials on the work of the mist eliminator listed in Table 13.

Table 13. Materials of droplets used in calculations and their densities.

Medium	Density [kg m^{-3}]
water	999
n-octane	720
diesel fuel	830
95% sulphuric acid	1834
25% aqueous ammonia solution	865

Calculations were carried out for geometry variants A and B_10_0. The results are shown in Figure 15.

Based on the plots shown in Figure 15, it can be observed that the density of the medium has a strong influence on the droplet removal efficiency. This phenomenon is since the inertia force acting on droplets is directly proportional to droplet mass. A higher density of medium causes better droplet removal efficiency. A similar effect can be observed by analysing the change in droplet size. Higher diameters of droplets cause higher droplet mass and hence inertia force, making them easier to remove by a mist eliminator. The presented results again show a better droplet removal efficiency when using a system with drainage channels, where the efficiency increases by about 20%. However, it should be remembered that this happens at the cost of an increased pressure drop of about 6%. A more visible effect is observed for larger droplet sizes.

Figure 15. Cont.

Figure 15. Comparison of calculated results of droplet removal efficiency for different media: (a) geometry A, inlet velocity 2 m s^{-1} ; (b) geometry A, inlet velocity 4 m s^{-1} ; (c) geometry B_10_0, inlet velocity 2 m s^{-1} ; (d) geometry B_10_0, inlet velocity 4 m s^{-1} .

3.5. Modification of Wave-Plate Mist Eliminator with Streamline Drainage Channels

The results presented in the previous sections show that the computational fluid dynamics correctly predict the influence of process and geometric parameters on the removal efficiency. After analysing the results, the authors proposed a new drainage channel to achieve high process efficiency with a slight pressure drop. When analysing the pressure distribution for the cases under consideration, e.g., geometry B_15_0 shown in Figure 16, it can be seen that high-pressure drop values appear near the sharp edges of the drainage walls, predominantly perpendicular to the flow direction.

Figure 16. Pressure fields (Pa) of B_15_0 geometry for inlet velocity 2 m s^{-1} simulated using a transition SST model.

For longer edges, this effect is even more visible. Therefore, it was decided to propose drainage channels with a much more streamlined shape. The geometry of the new system is shown in Figure 17, and the main dimensions are shown in Table 14.

Figure 17. Details of geometry C_15_0 with streamlined drainage channels.

Table 14. Geometrical details of analysed mist-wave eliminator with streamlined drainage channels.

Name	Presence of Drainage Channels	a [mm]	L [mm]
C_15_0	Yes	15.5	18.5

The results of droplet removal efficiency are shown in Figures 18 and 19. The authors decided to compare the results obtained for the new geometry for the most similar previous cases: B_10_0 and B_15_0.

Figure 18. Comparison of calculated results of droplet removal efficiency for geometries B_10_0, B_15_0, and C_15_0 with inlet velocity: (a) 2 m s^{-1} ; (b) 4 m s^{-1} .

(a)

(b)

(c)

Figure 19. Cont.

Figure 19. Velocity fields (m s^{-1}) of geometry variant C_15_0 with streamline drainage channels with inlet velocities (a,b) 2 m s^{-1} and (c,d) 4 m s^{-1} , which were simulated using a transition SST model. Velocity fields (a,c) (m s^{-1}) and vectors (b,d) (m s^{-1}) of geometry variant C_15_0 with streamline drainage channels with inlet velocities (a,b) 2 m s^{-1} and (c,d) 4 m s^{-1} , which were simulated using a transition SST model.

Based on the plots presented in Figure 16, geometry variant C_15_0 provides the highest droplet removal efficiency.

Figure 19 shows velocity fields and vectors of the investigated new geometries, and Tables 15 and 16 present the pressure drop and the mean force acting on drainage channels for the three, comparing geometry. For comparison, Figure 20 shows velocity fields and vectors for the analogous case with non-streamline drainage channels.

Table 15. Pressure drops on mist eliminators and the mean force acting on the drainage channel; inlet velocity 2 m s^{-1} .

Geometry	B_10_0	B_15_0	C_15_0
Pressure drop [Pa]	87	109	107
Mean force acting on drainage channel [N]	0.38	1.38	1.40

Table 16. Pressure drops on mist eliminators and the mean force acting on drainage the channel; inlet velocity 4 m s^{-1} .

Geometry	B_10_0	B_15_0	C_15_0
Pressure drop [Pa]	367	442	406
A mean force acting on drainage channel [N]	1.53	5.62	6.20

Figure 20. Velocity fields (a) and vectors (b) (m s^{-1}) (a) and vectors (b) (m s^{-1}) of geometry variant B_15_0 with inlet velocities 2 m s^{-1} simulated using a transition SST model.

Analysing the velocity distributions of the considered cases for 2 m s^{-1} , one can observe that the new geometry gets higher maximum velocities than the corresponding variant with non-streamlined drainage channels. This phenomenon is related to the shape of the external drainage channel walls. In C_15_0, the main gas stream is flattened, which is mainly visible on the third channel flows. A critical phenomenon is the occurrence of vortexes inside the channels. In the case of simple channels, the vortex is flattened in an oval shape. Around the channel's edges, the fluid from inside comes into contact with a gas with a nearly maximum velocity. Although, in the case of the profiled channels, a different phenomenon occurs. The forming vortex is round in shape, and the channel's edge disturbs the flow to a lesser extent, which will substantially impact the entrapment of deposited droplets—this can be well observed analysing the droplet trajectories presented in Figures 21 and 22.

Figure 21. Droplet trajectories for geometries B_15_0 (a,c,e,g) and C_15_0 (b,d,f,h) for velocity 2 (m s⁻¹) and droplet diameters: 8 (μm) (a,b); 10 (μm) (c,d); 12 (μm) (e,f); 14 (μm) (g,h).

(a)

(b)

(c)

(d)

(e)

(f)

(g)

(h)

Figure 22. Droplet trajectories for geometries B_15_0 (a,c,e,g) and C_15_0 (b,d,f,h) for velocity 4 (m s⁻¹) and droplet diameters: 6 (μm) (a,b); 8 (μm) (c,d); 10 (μm) (e,f); 12 (μm) (g,h).

Based on the visible results, the droplet removal efficiency is about 100% for droplets in the size of 14 μm . This new variant also has over two times higher droplet removal efficiency than geometry variant B_10_0 for droplets of 6 μm and 80% higher droplet removal than variant B_15_0. However, the pressure drop is about 23% higher than variant B_10_0 and similar to variant B_15_0 for the inlet velocity of 2 m s^{-1} . One can observe that the results are even better for higher velocity. The streamline construction of drainage channels should also minimise the phenomenon of secondary droplet entrainment. Naturally, experiments should validate every result obtained through simulation.

4. Conclusions

The work of the inertial droplet separator was modelled using computational fluid mechanics. The influence of practical important parameters on the droplet removal efficiency was investigated: the inlet velocity of the continuous phase, the droplet size, the effect of the presence of drainage channels, and their geometry, i.e., the length and angle of the slope. The simulation results show that drainage channels increase the pressure drop, which is undoubtedly a disadvantage of using drainage channels in inertial droplet separators. On the other hand, the removal efficiency values of the separator with drainage channels are much higher than those without drainage channels. Drainage channels cause the velocity of droplets to increase, which results in increased removal efficiency. At the same time, it should be remembered that the literature data show that the effect of droplet entrainment (not included in the calculation) will be much higher in the separator without drainage channels, which will further reduce the separation efficiency.

Three turbulence models and two variants of turbulent dispersion modelling were used in the present work. The best results were obtained for the transition SST model with the turbulent dispersion model attached. Such conclusions are consistent with the literature, and an excellent agreement with the experimental literature data allows simulating other parameters such as changes in slope angle and the size of drainage channels.

The simulation results show that small changes in geometry significantly improve the separator efficiency with a relatively small increase in pressure drop. The analysis of the obtained results allowed authors to propose a new geometry of drainage channels with streamlined shapes. This solution allows achieving a significant increase in droplet removal efficiency while keeping a comparable pressure drop for lower velocities and even decreasing for higher velocities than the basic geometry of channels of similar length. These results should be confirmed by extending the theoretical model, particularly with the effects of secondary entrainment of droplets and forming a liquid film on the wall, and comparing these with the result of the experimental data.

Author Contributions: Conceptualisation, Ł.M. and J.Ł.; methodology, Ł.M.; validation, Ł.M. and J.Ł.; formal analysis, Ł.M.; investigation Ł.M., J.Ł., M.T.; writing—original draft preparation, M.T.; writing—review and editing, Ł.M. and M.T.; visualisation, M.T.; supervision, Ł.M.; project administration, Ł.M. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data available on request.

Conflicts of Interest: The authors declare no conflict of interest.

Symbols

The following symbols are used in this manuscript:

Latin symbols:

a	drainage channel length, mm
c_{out}	number of droplets removed from the system
c_{in}	number of droplets injected into the system
C_D	drag force coefficient
d_p	droplet diameter, m
d_h	hydraulic diameter of the inlet
F_D	coefficient in drag force acceleration, s^{-1}
F_s	droplet inertial force, N
g	gravitational acceleration, $m\ s^{-2}$
k	turbulent kinetic energy, $m^2\ s^{-2}$
L	main channel width, mm
m_p	droplet mass, kg
p	pressure, Pa
R	curvature radius of the bending place, m
Re	Reynolds number
Re_d	droplet Reynolds number
St	Stokes number
t	time, s
u	continuous phase velocity, $m\ s^{-1}$
$\bar{u}$	continuous phase average velocity, $m\ s^{-1}$
u'	fluctuation of continuous phase velocity, $m\ s^{-1}$
u_{in}	continuous phase velocity on inlet, $m\ s^{-1}$
u_p	droplet velocity, $m\ s^{-1}$
y^+	dimensionless wall distance

Greek symbols:

α	drainage channel angle, $^\circ$
α_c	angle between wave-plates, $^\circ$
δ	delta function
ε	turbulent dissipation rate, $m^2\ s^{-3}$
η	droplet removal efficiency
μ	continuous phase dynamic viscosity, Pa·s
μ_t	continuous phase turbulent viscosity, Pa·s
ν_{air}	continuous phase kinematic viscosity, $m^2\ s^{-1}$
$\rho; \rho_{air}$	continuous phase density, $kg\ m^{-3}$
$\rho_p; \rho_{water}$	droplet density, $kg\ m^{-3}$
σ_k	turbulent Prandtl number for k
σ_ε	turbulent Prandtl number for ε
σ_ω	turbulent Prandtl number for ω
ω	specific dissipation rate, s^{-1}

Acronyms

CFD	Computational Fluid Dynamics
DPM	Disperse Phase Model
LES	Large Eddy Simulation
RANS	Reynolds-Averaged Navier–Stokes
RSM, RS	Reynolds Stress Model
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
SST	Shear Stress Transport

References

1. Del Ministro, C.; Bosio, F.; Spreafico, S.; Brunazzi, E. The gx fiber bed—A novel mist eliminator with improved aerosol removal performances. *Chem. Eng. Trans.* **2018**, *69*, 811–816. [[CrossRef](#)]
2. Yuan, S.; Fan, Y.; Li, J.; Zhou, S.; Cao, Y. Influence of droplet coalescence and breakup on the separation process in wave-plate separators. *Can. J. Chem. Eng.* **2018**, *96*, 1627–1636. [[CrossRef](#)]
3. Liu, Y.; Yu, D.; Jiang, J.; Yu, X.; Yao, H.; Xu, M. Experimental and numerical evaluation of the performance of a novel compound demister. *Desalination* **2017**, *409*, 115–127. [[CrossRef](#)]
4. Gharib, J.; Moraveji, M.K. Determination the factors affecting the vane-plate demisters efficiency using CFD modeling. *J. Chem. Eng. Process Technol.* **2011**, *1*, 1–5. [[CrossRef](#)]

5. Galletti, C.; Brunazzi, E.; Tognotti, L. A numerical model for gas flow and droplet motion in wave-plate mist eliminators with drainage channels. *Chem. Eng. Sci.* **2008**, *63*, 5639–5652. [[CrossRef](#)]
6. Azzopardi, B.J.; Sanaullah, K.S. Re-entrainment in wave-plate mist eliminators. *Chem. Eng. Sci.* **2002**, *57*, 3557–3563. [[CrossRef](#)]
7. Houghton, H.; Radford, W. Measurement on eliminators and the development of a new type for use at high gas velocities. *Trans. Am. Inst. Chem. Eng.* **1939**, *35*, 427–433.
8. James, P.; Wang, Y.; Azzopardi, B.; Hughes, J. The role of drainage channels in the performance of wave-plate mist eliminators. *Chem. Eng. Res. Des.* **2003**, *81*, 639–648. [[CrossRef](#)]
9. Wang, Y.; James, P. The calculation of wave-plate demister efficiencies using numerical simulation of the flow field and droplet motion. *Chem. Eng. Res. Des.* **1998**, *76*, 980–985. [[CrossRef](#)]
10. Wang, Y.; James, P. Assessment of an eddy-interaction model and its refinements using predictions of droplet deposition in a wave-plate demister. *Chem. Eng. Res. Des.* **1999**, *77*, 692–698. [[CrossRef](#)]
11. Estakharsar, M.H.H.; Rafee, R. Effect of drainage channel dimensions on the performance of wave-plate mist eliminators. *Korean J. Chem. Eng.* **2013**, *30*, 1301–1311. [[CrossRef](#)]
12. Rafee, R.; Rahimzadeh, H.; Ahmadi, G. Numerical simulations of airflow and droplet transport in a wave-plate mist eliminator. *Chem. Eng. Res. Des.* **2010**, *88*, 1393–1404. [[CrossRef](#)]
13. Michalcová, V.; Kotrasová, K. The numerical diffusion effect on the CFD simulation accuracy of velocity and temperature field for the application of sustainable architecture methodology. *Sustainability* **2020**, *12*, 10173. [[CrossRef](#)]
14. Makowski, Ł.; Bałdyga, J. Large eddy simulation of mixing effects on the course of parallel chemical reactions and comparison with k- ϵ modeling. *Chem. Eng. Process. Process Intensif.* **2011**, *50*, 1035–1040. [[CrossRef](#)]
15. Makowski, Ł.; Orciuch, W.; Bałdyga, J. Large eddy simulations of mixing effects on the course of precipitation process. *Chem. Eng. Sci.* **2012**, *77*, 85–94. [[CrossRef](#)]
16. Wojtas, K.; Makowski, Ł.; Orciuch, W. Barium sulfate precipitation in jet reactors: Large eddy simulations, kinetics study and design considerations. *Chem. Eng. Res. Des.* **2020**, *158*, 64–76. [[CrossRef](#)]
17. Ruiz, J.; Kaiser, A.; Zamora, B.; Cutillas, C.; Lucas, M. CFD analysis of drift eliminators using RANS and LES turbulent models. *Appl. Therm. Eng.* **2016**, *105*, 979–987. [[CrossRef](#)]
18. Gosman, A.D.; Loannides, E. Aspects of computer simulation of liquid-fueled combustors. *J. Energy* **1983**, *7*, 482–490. [[CrossRef](#)]