STELLINGEN

behorende bij het proefschrift

PERFORMANCE OF NOVEL MIST ELIMINATORS

van

Cor Verlaan

1. Goed onderzoek kan het best gedaan worden aan slechte scheiders.

2. De ontwikkeling van fluid dynamics software ligt nagenoeg stil door een gebrek aan goed fundamenteel onderzoek.

3. Voor gasreiniging bij hoge gasdichtheid is de axiaal cycloon het beste apparaat.

4. Het feit dat in de praktijk de grootte van de druppels in de te reinigen gasstroom niet bekend is maakt de selectie van een gas-vloeistof scheider een hachelijke zaak. Het is dan ook niet voor niets dat een scheidingsvat vaak uit drie scheidingsstappen is opgebouwd.

5. Bij hoge gasdichtheid toepassing berust de werking van een vane-type scheider hoofdzakelijk op turbulente diffusie en niet op afscheiding door traagheidskrachten.

6. Wil de ontwerpersopleiding beter van de grond komen dan moet het 1e fase diploma gedevalueerd worden.

7. Voetbal zal aanzienlijk aantrekkelijker worden als er minder spelers in het veld rondlopen.
8. Het bestaan van klassieke stromingen in de wetenschap is niet bevorderlijk voor de ontwikkeling er van.


10. Wapenembargo’s worden pas ingesteld als het betreffende land een oorlog begint. Aangezien zo’n land dan al overbewapend is, is het beter wapenembargo’s af te kondigen voor alle potentiële oorlogslanden.

11. Het woord hydrocycloon geeft een beperking van zijn toepassingsgebied weer.
PERFORMANCE OF
NOVEL MIST
ELIMINATORS

COR VERLAAN
PERFORMANCE OF NOVEL MIST ELIMINATORS

Proefschrift

ter verkrijging van de graad van doctor aan de Technische Universiteit Delft,
op gezag van de rector Magnificus,
prof. drs. P.A. Schenck
in het openbaar te verdedigen
ten overstaan van een commissie
aangewezen door het College van Dekanen
op dinsdag 17 september 1991 te 16.00 uur

door

Cornelis Christoffel Johannes Verlaan,
geboren te Stein,

Scheikundig Ingenieur.
Dit proefschrift is goedgekeurd door de promotor
Prof. dr. ir. J. de Graauw

CIP-DATA KONINKLIJKE BIBLIOTHEEK, DEN HAAG

Verlaan, Cor

Subject headings: gas-liquid separation / vane-type demisters / axial flow cyclones.
PERFORMANCE OF NOVEL MIST ELIMINATORS

INDEX

SUMMARY

1 INTRODUCTION

1.1 Offshore production of natural gas 1
1.2 Liquid removal from a gas stream 3
  1.2.1 State of the art 7
1.3 Scope of this gas/liquid separation study 9
1.4 Structure of the thesis 9
  Literature 10

2 EXPERIMENTAL FACILITIES 11

2.1 The wind tunnel experiments 11
  2.1.1 Smoke generation 12
  2.1.2 The pitot-tube 12
2.2 Laser Doppler velocity measurements 15
  2.2.1 Laser Doppler Anemometry, L.D.A. 15
  2.2.2 The water loop 18
2.3 Separation experiments 18
  2.3.1 The atmospheric rig 18
  2.3.2 The high pressure rig 22
  2.3.3 Efficiency calculations 26
2.4 Droplet size measurements 26
  List of symbols 27
  Literature 28
3 VANE-TYPE DEMISTERS

3.1 Introduction 29
3.2 Geometry parameters 31
3.3 Gas flow 34
  3.3.1 Simulation of the gas flow 34
    3.3.1.1 Theoretical background 34
    3.3.1.2 Numerical studies 38
  3.3.2 Experimental verification 44
    3.3.2.1 Visualisation experiments 44
    3.3.2.2 Velocity and pressure drop measurements 47
  3.3.3 Adapted geometry 48
  3.3.4 Discussion 49
  3.3.5 Conclusions 50
3.4 Motion of droplets 50
  3.4.1 Modelling of droplet motion 51
    3.4.1.1 Numerical studies 54
  3.4.2 Prediction of droplet separation 56
    3.4.2.1 Simulation of droplet separation 57
  3.4.3 Droplet separation in the adapted geometry 60
  3.4.4 Experimental verification 61
  3.4.5 Discussion 65
  3.4.6 Conclusions 66
3.5 Liquid flow 66
  3.5.1 Liquid flow models 67
    3.5.1.1 Film flow 67
    3.5.1.2 Flooding 70
    3.5.1.3 Re-entrainment 72
    3.5.1.4 Evaluation of film, flooding and re-entrainment models for vane-type demisters 74
  3.5.2 Experimental results 77
    3.5.2.1 Vanes without drainage system (type I) 77
    3.5.2.2 Vanes with drainage system (type II and III) 78
  3.5.3 Discussion 82
  3.5.4 Conclusions 87
  List of symbols 88
  Literature 90
4 AXIAL FLOW CYCLONE

4.1 Introduction 93
4.2 The geometry of an axial flow cyclone 94
  4.2.1 Swirl inlet devices 95
  4.2.2 Liquid drainage systems 97
4.3 The gas flow 98
  4.3.1 Rotating gas flow 99
  4.3.2 Simulation studies 101
    4.3.2.1 Theoretical background 101
    4.3.2.2 Numerical studies 106
  4.3.3 Experimental verification 113
    4.3.3.1 Laser doppler measurements 113
    4.3.3.2 Pressure drop experiments 115
  4.3.4 Discussion 115
  4.3.5 Conclusions 116
4.4 Motion of droplets 117
  4.4.1 Modelling of droplet motion 117
  4.4.2 Predicting droplet separation by a simplified model 119
  4.4.3 Numerical prediction of droplet separation 120
  4.4.4 Discussion 123
  4.4.5 Conclusions 124
4.5 Liquid flow 125
  4.5.1 Rotating liquid flow 125
  4.5.2 Re-entrainment 127
  4.5.3 Experimental investigation 128
  4.5.4 Discussion 135
  4.5.5 Conclusions 138
List of symbols 138
Literature 140
5 THE (PREDICTED) PERFORMANCE OF AN AXIAL FLOW CYCLONE AND VANE-TYPE DEMISTERS

5.1 Introduction 143
5.2 Vane-type demisters 144
  5.2.1 The maximum gas-liquid throughput 144
  5.2.2 Droplet separation 147
  5.2.3 Pressure drop 149
5.3 Axial flow cyclone 149
  5.3.1 The maximum gas-liquid throughput 150
  5.3.2 Droplet separation 152
  5.3.3 Pressure drop 152
5.4 Conclusions 153
  List of symbols 153
  Literature 154

SAMENVATTING
Voor Wilma
DANKWOORD

Ik wil iedereen die bijgedragen heeft tot de verwezenlijking van het Offshore project bedanken. In het bijzonder:

Jan de Graauw en Žarco Olujic voor hun stimulerende en vruchtbare discussies en begeleiding. Aat Bos en Wim Wiemer die dit project belangstellend en nauwlettend hebben gevolgd.

Alle medewerkers van het laboratorium Apparatenbouw voor de Proces Industrie. Zij gaven de ondersteuning en zorgden voor de totstandkoming van de faciliteiten die bij dit project nodig bleken te zijn.

De studenten die het vele experimentele werk hebben verricht.

De Industriële Raad van de Oceanologie (IRO) die in de persoon van Frans van Dongen voor de financiële en contractuele zaken zorg droeg.

De bedrijven en instellingen die dit project financieel ondersteunden:
Shell I.P.M.
Stork Ketels
Ministerie van Economische Zaken
Comprimo
Tebodin
Fluor Daniel
Protech
Technische Universiteit Delft
SUMMARY

Inertial gas/liquid separators are found in various designs and applications in the process industry. The know-how is based mostly on proprietary experiments and field experience of the manufacturers. The high pressure (offshore) applications make new demands and existing empirical design rules and practices proved to be insufficient.

Two types of inertial separators were investigated, the vane-type separator and an axial flow cyclone. This research project was started with the objective to improve the knowledge of the phenomena governing gas/liquid separation and to study the performance of these demisters under high pressure conditions.

The "PHOENICS" computer code was adopted to enable the simulation of the gas and droplet flow and the effects of geometry and physical properties of gas and liquid on the efficiency of the separation of droplets. Experiments at various gas densities have been carried out to verify simulations and to study the phenomena causing a loss in separator performance.

The gas flow field in vane-type separators was calculated by the "PHOENICS" computer code using a body fitted coordinate system and the k-ε turbulence model. The calculated velocities agree with pitot-tube measurements except at recirculation zones. The size of the calculated recirculation zones is smaller than observed during smoke tracer experiments. A droplet trajectory model was made using the calculated gas flow field. This model enabled the prediction of the separation efficiency. The results show that blade distance and bend angle have a larger effect on the separation than expected from literature models. The negative effect of the gas density proved to be more pronounced than expected. Based on these simulations the geometry of a vane-type separator was modified so that the energy consuming recirculation zones disappeared. This resulted in a reduction (about 50%) of the pressure drop without loss of efficiency. The major feature of the modified design is the incorporation of gas shielded liquid drainage compartments into the
space occupied earlier by recirculation zones. This type of liquid drainage allows higher gas velocities compared to those in conventional separators in which the liquid drains counter-currently to the gas flow. Experiments with a known droplet size distribution showed that the separation model, based on numerical simulation, predicts very well the separation efficiency. The experiments also showed that it was not possible to separate droplets below 8 micron at atmospheric conditions. The operating range of the vanes with a shielded drainage is limited due to the onset of re-entrainment. The re-entrainment originates from a liquid build-up in front of the drainage compartment. The gas throughput at which re-entrainment is incepted determines the maximum value of the so called gas-load factor, $\lambda_{\text{max}}$. This major design parameter is influenced negatively by a lower liquid surface tension and/or a higher liquid density. Liquid viscosity exhibits no influence on the onset of re-entrainment.

A similar approach was used to design and evaluate an effective axial flow cyclone. The three dimensional gas flow was simulated numerically by incorporation of the geometry of the swirl inducing device into the calculation domain of the "PHOENICS" code. Because of the absence of an anisotropic turbulence model a constant turbulent to laminar viscosity ratio was used. Laser doppler verification experiments indicated that the simple turbulence model gives a reasonable prediction of that part of the gas flow in the wall zone, where the droplets are concentrated. A droplet trajectory model indicated that droplets above 7 micron can be completely separated within the operating range of the cyclone. This occurs also at higher gas densities. Because of a complete separation of droplets above 7 micron the experimental work was focused on the liquid drainage and carry-over. The sharp longitudinal slits proved to be the most favourable design for liquid drainage. However, it was observed that even at low gas velocities the liquid flows in form of a rivulet along the slit to the exit of the cyclone where it is entrained by the gas leaving the cyclone. The amount of liquid carry-over proved to be proportional to the gas-load. Therefore the separation performance has been described in terms of the amount of liquid carry-over instead of an separation efficiency based on the efficiency of catching the droplets.

Vane-type separators have to be designed at their maximum gas-load factor. This means a decreasing gas velocity at higher gas densities and consequently a loss of separation efficiency. At natural gas processing pressures (about 100 bar) the vane-type separators considered here are not expected to separate droplets below 20 micron, whereas the axial flow cyclone is able to separate droplets above 7 micron. The maximum gas flow through the cyclone is determined by a critical pressure drop for a proper liquid drainage or by a critical liquid carry-over. The operation flexibility of a cyclone is much larger than that of vane-type separators. Hence, the axial flow cyclone seems to be a more appropriate device for droplet separation from gases at higher pressures.
CHAPTER 1

INTRODUCTION

Natural gas is and will be in the near future a major basis of the Dutch economy and every days life [1]. Besides financial profits (about 4% of the national income [2]) the natural gas offers also environmental benefits in contrary to other energy sources (oil and coal). Considering future stricter environmental regulations (lower level of maximum allowable emission of SO₂, etc.) energy producers and consumers have enough reasons to give natural gas preference to other fuels.

An increased use of natural gas will shift production toward the exploitation of smaller reservoirs on- and offshore. Especially offshore development of so called "marginal" fields requires cheap and highly efficient production facilities [3]. This study on compact gas/liquid separators gives a contribution to achieve these goals.

1.1 Offshore production of natural gas

The first Dutch offshore gas fields were found and explored in the sixties [1]. The exploitation of a gas field is more difficult offshore than onshore because more logistic and technical effort is needed and extra investment is necessary for the
construction of a platform and a pipeline for transportation to shore. Also the operation of the offshore production facilities and the infrastructure for transport of the gas increases the price threshold for economical production.

The pressure of a natural gas well can be very high (up to 600 bar) and the gas is basically methane with small quantities of heavier hydrocarbons and inorganic gases. A typical composition of a North Sea gas is listed in Table 1.1. The produced gas is contaminated with salt water and sand. During the production life of the well pressure will decrease and the amount of water produced with the gas increases significantly. On the platform the pressure of the gas is generally reduced to 100 - 200 bar (the inlet pressure of the pipeline) after which sand and liquid hydrocarbons are removed and the water dewpoint of the gas is reduced to a sufficient low value. This prevents condensation of corrosive water and formation of stable gas hydrates [4] in the pipeline [5].

A pressure and temperature reduction of the well stream causes not only condensation of water but also the condensation of heavier hydrocarbons [6]. The retrograde condensation [7] of the hydrocarbons is caused by the decrease of solvating power of the supercritical methane when the pressure is reduced or the temperature is increased. Figure 1.1 shows the phase envelope of the water free gas mixture presented in Table 1.1. Between the gas and the liquid phases there is a two-phase region where liquid and gas co-exist. Quality lines represent the relative fraction of each phase. In the retrograde condensation region, to the right of the critical point, decreasing pressure at constant temperature or increasing temperature at constant pressure causes an increase of the amount of liquid. This is in contrast with the other regions.

<table>
<thead>
<tr>
<th>Component</th>
<th>Fraction (mol%)</th>
<th>Component</th>
<th>Fraction (mol%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C₁ Methane</td>
<td>87.55</td>
<td>C₈ Octane</td>
<td>0.32</td>
</tr>
<tr>
<td>C₂ Ethane</td>
<td>4.32</td>
<td>C₉ Nonane</td>
<td>0.16</td>
</tr>
<tr>
<td>C₃ Propane</td>
<td>1.69</td>
<td>C₁₀ Decane</td>
<td>0.10</td>
</tr>
<tr>
<td>iC₄ Iso-butane</td>
<td>0.45</td>
<td>C₁₁ Undecane</td>
<td>0.06</td>
</tr>
<tr>
<td>C₄ Butane</td>
<td>0.73</td>
<td>C₁₂ Dodecane</td>
<td>0.04</td>
</tr>
<tr>
<td>iC₅ Iso-pentane</td>
<td>0.25</td>
<td>&gt; C₁₅ .......</td>
<td>0.072</td>
</tr>
<tr>
<td>C₅ Pentane</td>
<td>0.21</td>
<td>N₂ Nitrogen</td>
<td>1.49</td>
</tr>
<tr>
<td>C₆ Hexane</td>
<td>0.42</td>
<td>CO₂ Carbon dioxide</td>
<td>0.80</td>
</tr>
<tr>
<td>C₇ Heptane</td>
<td>0.69</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1.1: Composition of a North Sea gas.
Figure 1.1: Phase envelope of the gas mixture presented in Table 1.1.

The liquid phase consisting of water and a moderate quantity of hydrocarbons has to be separated from the gas flow to protect downstream equipment and to achieve pipeline transport specifications.

Figure 1.2 shows a simplified process diagram of an installation for gas treatment offshore. In this case the gas enters the wellhead with a pressure of about 230 bar and a temperature of 78 °C. It passes the wellhead separator where free water, liquid hydrocarbons (A in Figure 1.1) and sand are separated. Releasing the pressure down to 101 bar and cooling the gas down to 30 °C (B in Figure 1.1) causes again water and hydrocarbons condensation. This liquid mixture is separated from the gas by the separator housed in the bottom section of the glycol contactor. The glycol absorbs the residual water contained in the gas up to a water dewpoint of -5 to -10 °C. After this treatment the "dry" gas leaves the platform by pipeline to the shore. Further treatment is needed to conform the sale specifications.

### 1.2 Liquid removal from a gas stream

The main reasons for the offshore separation of liquid (droplets) from the gas are listed below [8]:

- Removal of water and sand
- Protection of downstream equipment
- Lowering hydrocarbon and water dewpoint
- Recovery of used liquids (glycol)

In the flowsheet presented in Figure 1.2 gas/liquid separators appear at three positions. First the wellhead separator where liquid and solid particles are separated from
Figure 1.2: Schematic representation of the offshore gas treatment facilities and the corresponding physical implications on the gas stream.
the gas. After a pressure reduction and a temperature decrease liquid condensate appears in the gas and is separated in the bottom section of the glycol contactor. The last separator is placed in the top of the glycol contactor. It removes entrained glycol droplets from the gas before it enters the pipeline.

Typical devices used in the process industry for droplet removal from gases at severe conditions are of the impingement type. Examples are cyclones and vane-type demisters. Wire mesh demisters are mostly used for less severe applications (clean, solid free service).

![Diagram](image)

**Figure 1.3:** Wire mesh demister.
- a) direct interception on a wire of a wire mesh demister
- b) liquid drainage and texture of a knitted wire mesh

In a wire mesh demister direct interception causes droplets to touch a wire when a streamline of the gas passes very close to it as illustrated in Figure 1.3a. The separated droplets coalesce and the liquid then flows counter-currently to the gas to the bottom of the mesh in the case of a horizontally oriented mesh pad which is often used in a vertical mist flow (Figure 1.3b). The liquid drains in the form of large droplets falling down from the mesh to the bottom of the vessel.

In vane-type demisters larger droplets can not follow a sudden change in the direction of the gas flow and collide against the wall (Figure 1.4a). The liquid film formed on the wall has to drain either counter-currently to the gas flow, as in the case of the horizontal oriented wire mesh demisters or shielded from the gas flow via a special drainage system (Figures 1.4b,c).

A cyclone creates a centrifugal gas flow field which forces heavy droplets to deviate from the streamlines and to collide against the wall (Figure 1.5a). The liquid film formed from these droplets flows together with the gas to the exit in the case of a once-through, axial flow cyclone (Figure 1.5b). In the reverse flow cyclone the gas flow reverses and leaves the cyclone through a vortex finder (Figure 1.5c). The driving force for separation in a cyclone is inversionally proportional to the cyclone diameter. When cleaning large gas streams several cyclones are placed therefore in parallel in a so-called multi-cyclone package.
Figure 1.4: Vane-type demister.
   a inertial deposition of droplets on the walls of a vane-type demister
   b liquid drainage from a simple vane-type demister
   c vane-type demister with an internal drainage system

Figure 1.5: Cyclone separator.
   a inertial separation of droplets in a centrifugal velocity field
   b axial flow cyclone
   c reverse flow cyclone
The benefits and drawbacks of these conventional separators are discussed in a publication of Štorch [9].

The know-how of these demisters is mostly restricted to the manufacturers and a small group of users and it is based on a long period of field experience. Nevertheless, misdesign of demisters as result of not enough knowledge still occurs and the costs of making changes are high, especially offshore. The future development of "marginal" fields obliges to use more tailor-made demisters and a more thorough information on design and operation rules is needed to use them. And to exploit the "marginal" fields in an economic way a lighter and cheaper production platform is needed. This can be achieved by a size reduction of the separation equipment.

The objectives stated above gave rise to the start of this research project with the goal to gain more profound insight into the phenomena that restrict the efficiency and capacity of conventional impingement gas/liquid demisters.

1.2.1 State of the art

In 1988 Swanborn [10] reviewed commonly used gas/liquid separation techniques and projected these on the different offshore applications. It was shown that impingement demisters can be found on many places but that their design practice is based on empiricism. A better insight in the physical phenomena that occur in the demister could lift the use of this type of demisters out of this stage.

An overview by Swanborn of the possible phenomena in the demisters showed that gas/liquid interactions, such as flooding and re-entrainment, may take place in these demisters. They influence the separation performance at a high gas throughput. The applicability of the models describing the onset of flooding or re-entrainment for the vane-type demisters and axial flow cyclone was discussed.

Numerical and physical experiments showed promising improvements with respect to separation performance and pressure drop. This led to two improved demister types (a vane-type demister and an axial flow cyclone) which are presented below.

The geometry of commonly used vane-type demisters could be improved, with respect of pressure drop, in a way that energy-consuming recirculation zones are abandoned. To achieve this, the blade distance downstream of the bend was narrowed (Figure 1.6a). By creating a hollow space in the vane blades, a drainage system, separated from the gas channel, was formed. This broadened the operating range considerably. However, this geometry reduces (about 50%) the cross-sectional area available for the mist flow. Also the construction of this geometry is difficult.

A once-through, axial flow cyclone with a recycling gas flow (Figure 1.6b), was designed and optimized. It should replace reverse flow cyclones. This axial flow cyclone showed a good separation performance even at high gas density conditions and a lower pressure drop compared with the conventional reverse flow cyclones. In practise the recycling mechanism is difficult to incorporate in the cyclone geometry and the resistance
Figure 1.6: Newly developed demisters [10].

a global geometry of a newly developed vane-type demister with drainage system
b axial flow cyclone with a recycling gas flow

to plugging or fouling will be high.

Other recent contributions towards the application of impingement demisters were made by Oranje [11] and Bürkholz [12].

Oranje investigated most recently the performance of several impingement gas/liquid demisters, partly under increased gas density conditions, varying from knock-out drum to a recently developed cyclone (the Gasunie cyclone). This cyclone appeared to have an extremely wide operating range together with a high separation efficiency.

Bürkholz reviews the major physical effects in the collection of droplets in various impingement demisters presently used in industry. With several parameters a selection of the demister type can be made and the separation efficiency can be predicted.

Preliminary studies of impingement demisters carried out by the author [13] confirmed that vane-type demisters equipped with a drainage system are very suitable to achieve the desired reduction in weight and height of separator vessels.
1.3 Scope of this gas/liquid separation study

The demisters developed by Swanborn [10] were the basis for a further optimization of the geometry. Also the three phenomena dominating the operation of these impingement demisters had to be investigated more profoundly.

First the gas flow in which the droplets are dispersed. The gas flow pattern affects the separation performance and pressure drop of the demister. For this purpose the use of the commercially available fluid dynamics software package "PHOENICS" was investigated. The results of the gas flow simulations, both velocity profiles and pressure drop, were compared with experimental data for the axial flow cyclone and vane-type demisters.

In the gas flow the droplets are dispersed and travel through the demister channel. The geometry of this channel has a strong influence on the chance for these droplets to get separated from the gas stream. This was studied by means of the simulation of droplet trajectories through the demister and experimentally by determining the separation efficiency.

After the droplets are collected on the wall, a liquid film is created which can leave the demister either counter-current to the gas flow or by a shielded drainage system. Under certain circumstances the gas interferes with the liquid film which can result in the carry-over of liquid to the exit of the separator. The mechanisms causing the liquid carry-over were studied and verified experimentally.

1.4 Structure of this thesis

Experiments have been carried out to investigate gas, droplet and liquid film flow. A wind tunnel was built for gas flow field visualisation and velocity measurements. A water loop to perform laser doppler velocity measurements in the axial flow cyclone. A large scale rig was used to carry out atmospheric experiments on demisters of real scale under different liquid and gas conditions. Scaling up of demisters to higher gas density applications was studied in a pressurized rig. Chapter 2 describes the rigs and experiments.

The emphasis of the work described in this thesis is on two types of impingement demisters. Chapter 3 is concerned with a vane-type demister which has a low pressure drop and a good performance within an operating range much larger than that of the conventional vane-type demisters. The second type is the axial flow cyclone which has a very high separation efficiency and considerably low pressure drop in comparison to a reverse flow cyclone. This demister is investigated in Chapter 4.

Finally in Chapter 5 the performance results of two promising demister types, with respect to droplet separation and operating range, have been extrapolated to increased gas density (high pressure) conditions.
Literature

CHAPTER 2

EXPERIMENTAL FACILITIES

To investigate the behaviour of gas and liquid in vane-type demisters and axial flow cyclones four experimental installations were used. The gas flow in vane-type demisters was investigated in a wind tunnel, the flow field in axial flow cyclones in a water driven loop and the separation performance of demisters of full size was studied in an atmospheric rig. To achieve information about the behaviour of these demisters at increased gas density conditions a pressurized rig was built with a much smaller volume than the atmospheric rig.

2.1 The wind tunnel experiments

To visualise the gas flow in vane-type demisters a small horizontally oriented wind tunnel was built. This installation, shown schematically in Figure 2.1, consists of a ventilator, a long channel to achieve a constant uniform gas velocity profile and a test section containing the vane blades. The complete wind tunnel was built of Perspex which made it visually accessible (e.g. for photography).
The vane blades were made of 1 mm thick aluminium and could be placed on every desired distance from each other.

2.1.1 Smoke generation

A way to visualise the gas flow between the vane blades was to introduce smoke. Smoke was generated by vaporizing a glycerine droplet on a thin resistance wire placed perpendicularly between the blades as can be seen in Figure 2.2. During the evaporation of the droplet a smoke was created which was carried by the passing gas flow. A light source was used to illuminate the smoke track which could be recorded with a photo camera. The best photographs were made at superficial gas velocities around 2 m/s because at higher gas velocities the smoke diffused very quickly.

2.1.2 The pitot-tube

The pitot-tube is an often used, simple and robust pressure measuring device which enables velocity measurements. This device is normally designed with two small radial holes, one oriented frontally to the gas flow and a second one parallel to the flow.
The pressure measured with the tap frontal to the flow contains not only the dynamic pressure (due to the velocity) but also the static pressure. With the second tap only the static pressure is measured. The velocity then can be calculated as follows:

\[
U_g = \sqrt{\frac{P - P_{st}}{\frac{1}{2} \rho_g}} \quad (2.1)
\]

where \(P\) is the total pressure, \(P_{st}\) the static pressure, \(U_g\) the gas velocity and \(\rho_g\) the gas density. To measure the two pressures the pitot-tube needs two separate tubes from each pressure tap which means that in practice the pitot-tube has a diameter above 3 mm. Because of the small distances between the vane blades (i.e. 1 - 3 cm) this type of pitot-tube would block a considerable part of the cross-section and creates therefore a considerable disturbance of the main flow.

An other option was to use a pitot-tube as described by Glaser [1] having only one tap which is used for both measuring static and total pressure. In this case the pitot-tube could be very small (1 mm diameter). The technique of measuring the velocity with one pressure tap is based on the pressure distribution on the wall of a cylinder when it is exposed to a gas flow. This pressure distribution in the idealised situation, for a streamlined flow and an infinite small pressure tap, can be written as:

\[
P = P_{st} + \frac{1}{2} \rho U^2 (1 - 4 \sin^2 \alpha) \quad (2.2)
\]

in which \(U\) the perpendicular velocity, \(\rho\) the density of the medium and \(\alpha\) the angle over which the measuring tap has been rotated relative to the flow direction. Figure 2.3 shows

**Figure 2.3**: Pressure distribution on a cylinder at different Reynolds numbers.

- - - - - - ideal situation

--- Re = 1.9 \(10^5\)

--- Re = 6.7 \(10^5\)
the pressure distribution on a cylinder as a function of the angle, $\alpha$, for both the theoretical case and in the case that the flow is turbulent [2]. It can be seen in this picture that the static pressure can be found at the angle of 30° and is almost independent of the Reynolds number of the flow (the Reynolds number, Re, based on the diameter of the cylinder). For obtaining the velocity the pressure is measured perpendicular to the flow (total pressure) and the static pressure when rotating the cylinder for 30°.

Due to a not ideal boundary layer the static pressure can not be found at the ideal place (30°). This effect causes that the static pressure location moves to higher angles where for example Thom [1] measured 34.4°, Glaser [1] 34.8° and Massey [3] 39.3°. Also the tap diameter influences the angle at which the correct static pressure can be measured and a correction for this angle has to be made. The real location, $\alpha$, for the static pressure measurement is:

$$\alpha = \alpha_o + \frac{90d}{\pi D}$$

where $\alpha_o$ is the angle at which the static pressure on a closed cylinder can be found in practise and d and D are the diameters of the tap and cylinder respectively.

The pitot-tube used in our experiments has a diameter of 1 mm and a tap diameter of 0.35 mm. For this pitot-tube the static pressure can be measured at an angle of 49.3° ($\alpha_o = 39.3°$). Testing the pitot-tube in a wind tunnel, where the exact velocity was known, provided the pressure distribution at different Reynolds numbers for the used pitot-tube (Figure 2.4). The static pressure angle ($\alpha$) was found around 50°.

![Figure 2.4: Measured dynamic pressure distribution at different pitot-tube angles.](image)

- $\Box$ Re = 636
- $\times$ Re = 935
- $\bullet$ Re = 1260
To measure the velocity over the cross-section of the vane-type demister slits were made on the top of the test section. After determining the orientation with the highest pressure, which means that the air is flowing from that direction, the pitot-tube was rotated 49.3° and the static pressure was measured. With Eq. 2.1 the velocity can then be calculated.

The use of this pitot-tube cylinder introduces some errors. First the diameter of the tube causes a reduction in flow cross-section of about 3 - 5%. Secondly, at low gas velocities the measured pressures will be very small. Therefore, at low gas velocities a measuring error in either \( P \) or \( P_s \) will cause a large error in the calculated velocity.

### 2.2 Laser Doppler velocity measurements

A pitot-tube is very often, successfully used for measuring velocities (see Chapter 2.1) but introduces a number of problems. The main disadvantage is that the pitot-tube has to be positioned in the flow and introduces therefore a disturbance of the flow field. Furthermore, the velocity range, which can be measured, is limited and the dynamic response is much slower. These reasons make the use of a pitot-tube for measuring a high swirling flow unattractive and no reliable data can be assumed [4].

The availability of laser beams introduces a new technique for measuring velocities [5,6]. This method works on the principle of a frequency shift of scattered light coming from particles which are lightened by the laser beam. The advantages of this measuring technique are [7]:
- no disturbance of the flow
- simple relation between frequency shift and velocity
- both positive and negative velocities can be measured
- simultaneous measuring of more velocity directions is possible
- possibility to measure at difficult conditions
- measuring of turbulence

However, a disadvantage is the price of this apparatus.

In the next chapters a short introduction of the principle of the L.D.A. technique is given and the measurements in a specially designed rig are described.

#### 2.2.1 Laser Doppler Anemometry, L.D.A.

Velocity measurements in an axial flow cyclone were performed with the reference beam method. A schematic representation of the measuring set-up is given in Figure 2.5a. The laser creates a narrow, monochromatic beam which is split into two coherent beams. The two beams are diffracted by a lens \( L_1 \) and pass in parallel to a second lens \( L_2 \) which focuses the two beams in a small measuring volume. These beams have not the same intensity. The one with the high intensity is the illumination beam and...
Figure 2.5: Schematic diagram of the laser Doppler measuring technique.

a measuring set-up
b the principle of velocity measurement

The other one the reference beam. When a particle flows through the measuring volume it will scatter the light of the illumination beam. Due to the Doppler effect the frequency of the scattered light is different from the frequency of the illumination and reference beam. The frequency difference is proportional to the velocity of the particle. The reference beam and the scattered light of the particle interfere with each other and the resulting light pattern is collected on the detector, a photodiode. This photodiode translates the collected light frequencies into an electric frequency which is then processed by a frequency tracker. The exit voltage of the tracker is used for calculating the velocity.

A particle moving with a certain velocity \( u_p \) through a measuring volume is shown in Figure 2.5b. Due to this motion it observes light with a different frequency than emitted by the laser. The observed frequency is:

\[
v_p = v \left(1 - \frac{\mathbf{e}_1 \cdot \mathbf{u}_p}{c}\right)
\]

(2.4)

where \( v \) and \( v_p \) the frequency of the laser and the frequency observed by the particle respectively, \( \mathbf{e}_1 \) the vector in the direction of beam 1, \( \mathbf{u}_p \) the velocity vector of the particle and \( c \) the velocity of light. The non-moving detector detects the scattered light of the particle \( (v_p) \) and observes a frequency given by:

\[
v_d = \frac{v_p}{\left(1 - \frac{\mathbf{e}_2 \cdot \mathbf{u}_p}{c}\right)}
\]

(2.5)

where \( v_d \) the frequency of the collected light and \( \mathbf{e}_2 \) the vector in the direction of the detector. To calculate the velocity of the particle the frequency difference between the scattered light of the particle and the frequency of the reference beam has to be
determined. The frequency difference is as follows:

\[ v_{rel} = \left| v_d - v \right| \]  \hspace{1cm} (2.6)

where \( v_{rel} \) the frequency of the signal leaving the detector. Substituting Eq. 2.5 into Eq. 2.6 gives the following relation between the frequency of the reference beam and the signal leaving the detector:

\[ v_{rel} = v - \frac{v \left( 1 - \frac{e_1 u_p}{c} \right)}{1 - \frac{e_2 u_p}{c}} \]  \hspace{1cm} (2.7)

After some rearrangements and in the case that the particle velocity is much smaller than the velocity of light the following relation appears:

\[ v_{rel} = v \left( e_2 - e_1 \right) \frac{u_p}{c} \]  \hspace{1cm} (2.8)

This equation can be transformed to:

\[ v_{rel} = \frac{v}{c} u_b \sin \theta = \frac{u_b}{\lambda} 2 \sin \theta \]  \hspace{1cm} (2.9)

in which \( \theta \) the half angle between the two laser beams, \( \lambda \) the wavelength of the laser light and \( u_b \) the velocity component of the particle in the plane of \( e_1 \) and \( e_2 \), which is perpendicular to the bisecting line of those vectors. The absolute values in Equation 2.7 make the direction of the particles unknown. When change of the direction is expected a constant frequency is added to one of the laser beams (in this case the reference beam). This means that an artificial velocity is added to the measured velocity. At zero particle velocity now the measured signal represents the artificial added velocity. A larger value of the signal means a particle velocity in a certain direction and a lower signal into the opposite.

To measure the velocity the axial flow cyclone has to be accessible for the laser beams. When using a transparent tube the actual place and shape of the measuring volume is distorted. This because of the refraction of the laser beams caused by the thick curved walls of the tube. Therefore, a measuring section has been developed to minimize these effects (Figure 2.6). A thin wall section of transparent plastic sheet (A) is surrounded by a rectangular chamber (B) filled with the same fluid as pumped through the cyclone and kept under the same pressure. This minimizes the optical distortion when the beams enter the cyclone. The chamber was equipped with plane optical glass windows (C). These
plane windows refract the light in a way that the place and shape of the measuring volume do not change during traversing the measuring volume through the cyclone cross-section. Laser doppler anemometry can be applied only with particles in the fluid with approximately the same density. For this reason flow patterns in cyclones can be measured with this technique in hydro-cyclones and not in gas cyclones.

2.2.2 The water loop

The water loop is schematically presented in Figure 2.7. Pump (P) transports the water to the measuring section (M) placed about 100 tube diameters above the pump. The amount of water is controlled by a valve \( V_1 \) and a liquid return system and is measured by a magnetic flowmeter (F). A valve \( V_2 \) not only controls the flow but also keeps the system under pressure to avoid the formation of an air core. This air core makes measurements with this L.D.A. set-up impossible. A flexible connection (B) is placed downstream of the pump to avoid the influence of vibrations on the measurements. The water storage vessel (S) is cooled to minimize the effect of heat input by the pump. The L.D.A. system is mounted on a traversing system separated from the rig an enables both vertical and horizontal traversing through the measuring section.

2.3 Separation experiments

2.3.1 The atmospheric rig

The design of this rig (Figure 2.8) is based on the wish to investigate gas/liquid demisters on a real scale under various flow conditions. Figure 2.9 shows a simplified flowsheet of the rig which was built for this purpose. The central part of this rig consists
of two test columns connected in parallel and using the same periphery. Air is blown by
compressor (C) through a gas cooler (H) to one of the test columns. These test columns
are mainly made of Perspex and are therefore visually accessible. Their set-up is such that
various types of demisters (Figure 2.10) can be tested in the column.

The liquid is pumped by a centrifugal pump (P) from the feed tank (D) to a set
of nozzles which disperse it into the air flow. The experiments have been carried out with
air as gas phase and either water, a soap-water mixture or glycol as the liquid phase. The
physical properties of the used gases and liquid phases are presented in Table 2.1. The
created mist flow enters the test section in which one of the demisters has been installed.
The droplets will be separated from the air flow and the outlet air passes through a
cartridge filter (F) where the remaining droplets are separated. This cleaned air is recycled
to the compressor. In the test column provisions were made upstream and downstream of
the demister to enable the measurement of droplet size distributions with a Malvern
particle sizer.
Figure 2.8: Atmospheric rig.
Figure 2.9: Simplified flowsheet of the atmospheric rig.

<table>
<thead>
<tr>
<th></th>
<th>$\mu$ (kg/ms)</th>
<th>$\rho$ (kg/m$^3$)</th>
<th>$\sigma$ (kg/s$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>$1.82 \times 10^{-3}$</td>
<td>1.25</td>
<td>-</td>
</tr>
<tr>
<td>SF$_6$</td>
<td>$1.56 \times 10^{-5}$</td>
<td>6.5</td>
<td>-</td>
</tr>
<tr>
<td>Water</td>
<td>$1.0 \times 10^{-3}$</td>
<td>1000</td>
<td>$69.5 \times 10^{-3}$</td>
</tr>
<tr>
<td>Glycol</td>
<td>$21.0 \times 10^{-3}$</td>
<td>1113</td>
<td>$41.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Butanol</td>
<td>$4.2 \times 10^{-3}$</td>
<td>810</td>
<td>$27.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Soap-water mixture</td>
<td>$1.0 \times 10^{-3}$</td>
<td>1000</td>
<td>$32.0 \times 10^{-3}$</td>
</tr>
</tbody>
</table>

Table 2.1: Physical properties of the used gases and liquids (at ambient conditions).

The maximum available air flow rate was 2400 m$^3$/hr and was measured by a vortex flowmeter (FR1). The maximum liquid flow rate, available from the nozzles, was about 0.5 m$^3$/hr, and was measured by a magnetic flowmeter (FR2). The generated mist flow contained between 0.0013 to 0.013 % liquid by volume (40 - 400 l/hr). A part of the sprayed liquid will come onto column walls and will not reach the demister. This liquid
drains to the bottom of the column where it is measured (FR4) and pumped back into the feed tank. Flowmeter FR3 measures the flow rate of separated liquid when the demister is provided with an internal drainage system. The quantity of the liquid passing the demister can be measured with FR5 after separation from the gas flow in the cartridge filter (F). The pressure drop ($\Delta p$) is measured by means of a differential pressure cell.

2.3.2 The high pressure rig

This rig (Figure 2.11) has been built for the investigation of gas/liquid demisters under increased pressure (increased gas density) conditions as found in natural gas processing. Because of the stringent safety requirements this rig is much smaller than the previous one and may be used up to pressures of 1000 kPa. By using the heavy gas (SF$_6$) a gas density of 70 kg/m$^3$ is theoretically achievable. As is evident from Fig. 2.12 the rig consists of two units. One of the units is supplying the gas and liquid flows and the other one contains the test vessel and the measuring devices.

The gas flow producing unit contains a compressor (C) for recycling the gas through the rig. This compressor was placed in a vessel because of problems with the shaft seals. After leaving the compressor the gas flows through a gas cooler (H) to the test vessel. The test vessel contains a small unit of a vane-type demister or a single axial flow cyclone as shown in Figure 2.13. In the test vessel the liquid is sprayed into the entering gas. The liquid is pumped by a positive displacement pump (P) from the storage vessel (D)
Figure 2.11: The pressurized rig.
to the installed nozzle. Experiments have been carried out with either water, butanol or glycol as liquid phase and air or SF$_6$ as gas phase. The physical properties of the liquids and gases are given in Table 2.1. The mist flows into the separator and the droplets will be removed from the gas. The outlet gas passes through a cartridge filter unit (F) for the removal of the remaining droplets. The clean gas re-enters the first unit to be recycled.

The rig was kept under pressure by connecting it with a gas supplying system which could deliver air at a maximum pressure of 7.5 bar. In case of the SF$_6$ it was not possible to achieve a stable operation of the rig above a pressure of 3.5 bar. Therefore the maximum achievable gas density was 21 kg/m$^3$. The maximum available gas flow rate was 150 m$^3$/hr which was measured by a vortex flowmeter, FR1. The liquid rate can be up to 60 l/hr and was measured by a magnetic flowmeter, FR2. A part of the sprayed liquid does not enter the demister and flows into a device FR4 which measures the flow in batches. The increase of liquid level in a calibrated tube is measured and when the tube is filled up the liquid is released to the storage vessel (D). The liquid collected in the demister was measured in a similar way by FR3. The non separated liquid is captured by a filter unit placed in the strip vessel (F) and is measured by FR5 (also batch-wise). The pressure and pressure drop were measured by a differential pressure cell.
Figure 2.13: Separators for pressurized rig.
a vane separator with shielded drainage system
b axial flow cyclone with different slits
2.3.3 Efficiency calculations

For the vane-type demisters without a shielded drainage compartment it is difficult to calculate separation efficiencies because of the unknown liquid quantity entering the demister (e.g. a part from the liquid which enters the vane will drain to the flowmeter FR4). However, preliminary measurements without the demister in the test section enables the assessment of the quantity of liquid supplied to the demister. The separation efficiency of the vane demister, \( \eta \), can be expressed as:

\[
\eta = \left(1 - \frac{FR5}{FR5'}\right) \times 100
\]  

(2.10)

where \( FR5' \) is the liquid flow rate measured during the operation with an empty test section.

For the vane-type demister equipped with a separate drainage system and the axial flow cyclone where the liquid also drains shielded from the gas flow the drained liquid flow is measured by flowmeter FR3 so that this quantity added to that measured in FR5 gives the total amount of liquid which enters the demister. Thus the efficiency, \( \eta \), can be expressed as:

\[
\eta = \left(1 - \frac{FR5}{FR5 + FR3}\right) \times 100
\]  

(2.11)

The rigs are equipped with a data logger and a PC. A "Basic" program was written to perform the measurement and to control all data and test settings automatically.

2.4 Droplet size measurements

In order to know the droplet size distribution entering the demisters a Malvern particle sizer was used. Size distributions were measured for the different nozzles and for different liquids and liquid loadings. The ability to determine the droplet sizes is based on the diffraction of a laser beam by droplets moving through the measuring section as presented in figure 2.14. A full description of this measuring technique is given by Boxman [4]. The diffracted and not diffracted light is focused by a Fourier transform lens on a detector consisting of 30 semi-circular photo sensitive elements. The scattered light pattern is recorded as a function of the angular distribution and deconvoluted to a droplet size distribution.
Figure 2.14: Droplet size measurement through laser diffraction.

List of symbols

\[
\begin{align*}
\text{c} & \quad \text{Velocity of light} \\
\text{D, d} & \quad \text{Diameter} \\
\text{e} & \quad \text{Vector} \\
\text{P, p} & \quad \text{Pressure} \\
\text{Re} & \quad \text{Reynolds number} \\
\text{U, u} & \quad \text{Velocity} \\
\alpha & \quad \text{Angle of rotation} \\
\eta & \quad \text{Efficiency} \\
\theta & \quad \text{Half angle between laser beams} \\
\lambda & \quad \text{Wavelength} \\
\mu & \quad \text{Kinematic viscosity} \\
\nu & \quad \text{Frequency} \\
\rho & \quad \text{Density} \\
\sigma & \quad \text{Surface tension}
\end{align*}
\]

(m/s) \hspace{2cm} (m) \hspace{2cm} (-) \hspace{2cm} (Pa) \hspace{2cm} (-) \hspace{2cm} (m/s) \hspace{2cm} (°) \hspace{2cm} (-) \hspace{2cm} (°) \hspace{2cm} (m) \hspace{2cm} (kg/ms) \hspace{2cm} (1/s) \hspace{2cm} (kg/m^3) \hspace{2cm} (kg/s^2)

indices

\[
\begin{align*}
b & \quad \text{perpendicular} \\
d & \quad \text{detector} \\
g & \quad \text{gas} \\
o & \quad \text{normal} \\
p & \quad \text{particle} \\
rel & \quad \text{relative}
\end{align*}
\]
Literature

CHAPTER 3

VANE-TYPE DEMISTERS

3.1 Introduction

The first attempt for a more streamlined vane (baffle, chevron) type demister was presented around 1939 by Houghton and Radford [1]. However, this type has found only limited application. Still the basic form consists of simple plates mounted under a certain angle (Figure 3.1).

A gas stream loaded with droplets enters either horizontally or vertically the vane blades. These zig-zag blades force the gas flow to make several sharp bends. The droplets cannot follow the gas streamlines because of the momentum and collide against the target blades where they coalesce and are incorporated into a liquid film. This liquid has to be drained continuously from the vane-type demister.

The performance of vane-type demisters is not only based on the primary separation of droplets but also on the maximum allowable gas throughput. Depending on the type of drainage the maximum capacity is limited due to flooding or re-entrainment. Flooding occurs when the forces on the liquid film in the separator are too high to drain it counter-currently to the gas flow. The liquid accumulates in the separator and ultimately re-enters into the gas stream. Applying a separate drainage system the onset of re-
entrainment from the co-current gas and liquid flow limits the capacity. Liquid droplets are sheared from the film and re-enter the gas stream. Figure 3.2 shows a characteristic performance diagram of a vane-type demister from which it can be seen that at low gas velocities the performance improves with increasing throughput due to an improving elimination of droplets from the gas stream. After a certain $U_{g,c}$ the separation efficiency decreases because of flooding or re-entrainment.

In this chapter first the gas flow in vane-type demisters is investigated. This gas flow is important for the separation of the droplets, the pressure drop and for the flooding and/or re-entrainment behaviour in the demister. The flow pattern of the gas is studied with a computational fluid dynamics program, with smoke trace experiments and with pitot-tube velocity measurements. For a study of the separation of droplets their motion in a gas flow has to be known. Results of the simulations of the droplet separation are compared with models found in literature and with experiments carried out with the

Figure 3.2: Performance diagram of a vane-type demister.
rigs described in Chapter 2. The third part of this chapter deals with the behaviour of the liquid film created by deposition of the droplets on the vane blades. Film flow, flooding and re-entrainment models from literature are discussed and compared with our experimental work.

### 3.2 Geometry parameters

A vane-type demister consists basically of a number of parallel placed multiple bended blades, oriented either horizontally or vertically. The most important geometry parameters influencing the separation efficiency of droplets and pressure drop are shown in Figure 3.3.

The bend angle \(2\alpha\) of the vane blades varies mostly between 40 to 100 degrees. By increasing the angle, the droplet-loaden gas stream is forced to make a sharper bend which increases the effect of inertial forces on the droplet. The gas velocity between the blades becomes higher at increasing bend angle due to a smaller perpendicular blade distance. This means that smaller droplets will be separated but the pressure drop per bend will be higher.

![Diagram of vane-type demister with labels](image)

**Figure 3.3**: Important geometry parameters of vane-type demisters.

The distance between two successive parallel placed blades \(s\) determines the maximum sideward displacement of the droplets needed for collision on the wall. A smaller distance shortens the path of the droplets and therefore the possibility for a droplet to become separated increases.

The influence of the length \(l\) between two successive bends, on the separation of a droplet was noticed by Wörrlein [2]. When the blade length is too short the inertial forces on a droplet, induced by a change of gas flow direction, are not used maximally before entering the next bend. When, however, the length of the blades is too long all inertia working on a not separated droplet will be used and then the droplet moves along with the gas parallel between the blades. Also, from a geometrical point of view, a minimum length is necessary and is determined by the bend angle, \(2\alpha\), and blade distance, \(s\). Wörrlein [2] states that a minimum blade length is found when all the tops of the
successive bends of two parallel placed blades are on one line. A minimum length is given by:

\[ l_{\text{min}} = \frac{s}{\sin \alpha} \]  \hspace{1cm} (3.1)

Another geometry factor is the bend radius, \( r \), of the vane blades. A round bend, instead of a sharp bend, causes the gas flow to make a less abrupt change of the flow direction i.e. reducing the effect of the inertial forces. Both, separation efficiency and pressure drop will decrease.

The number of bends (\( n \)) increases the chance for droplets to be separated. A bend more means another chance for inertial forces to force the droplet to collide against the wall. Some authors [3,4] assume redistribution of the not separated droplets over the complete cross-section after each bend.

Surface textures often found on the blades of commercial demisters are made mainly to provide a good drainage of separated liquid. According to Bürkholz [5], who investigated several types of textured blades, a positive effect on the separation efficiency may be expected only if these textures reduce considerably the gas flow cross-section.

Table 3.1 gives a résumé of the influence of the geometry parameters on the separation efficiency and pressure drop as described by several authors.

<table>
<thead>
<tr>
<th>Increase in</th>
<th>Separation efficiency (( \eta ))</th>
<th>Pressure drop (( \Delta p ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>&gt;</td>
<td>&gt;</td>
</tr>
<tr>
<td>( s )</td>
<td>&lt;</td>
<td>=</td>
</tr>
<tr>
<td>( l )</td>
<td>=</td>
<td>&gt;</td>
</tr>
<tr>
<td>( n )</td>
<td>&gt;</td>
<td>&gt;</td>
</tr>
<tr>
<td>( \rightarrow )</td>
<td>&lt;</td>
<td>&lt;</td>
</tr>
</tbody>
</table>

Table 3.1 : Effects of geometry on the efficiency and pressure drop of vane-type demisters.

With:  
>= increase  
= constant  
< decrease

All these parameters, except the influence of surface textures, have been considered in the investigation of gas and droplet flow which is described below.

After the droplets are deposited on the walls a liquid film is formed which flows counter-currently towards the gas flow, in case of a vane-type demister with a vertical passage. When this separator is equipped with a separate drainage system the liquid film may flow co-currently until it enters a slit. Preliminary tests have shown that shielding of the liquid film from the gas stream by a separate drainage system offers
considerable capacity advantages over conventional single blade vane designs [6]. Both types are investigated in this chapter.

Figure 3.4a shows schematically a vane-type demister without a separate drainage system, type I, made of thin curved metal blades which are placed parallel to each other. The blades are fixed by means of perpendicularly installed and evenly spaced straight sheets. The surface of the blades is, in this case, provided with a texture of V-shaped profiles forcing the collected liquid to flow to the side walls. The collected liquid drains counter-currently to the gas flow and the maximum allowable gas throughput is limited due to flooding.

![Diagram of vane-type demisters](image)

Figure 3.4: Vane-type demisters with different liquid drainage systems.

- a counter-current, type I
- b hollow blade, type II

A vane-type demister equipped with a separate drainage system is presented in Figure 3.4b. Its geometry was optimized by using a blade spacing which varies alongside the gas channel. By narrowing of the gas channel after the bend big recirculation zones are avoided. This type (type II, based on the geometry presented in the work of Swanborn [7]) is made of hollow metal blades connected in parallel which are provided with lateral narrow slits placed equidistantly on both sides. This provides the possibility for the liquid to drain vertically downward without contacting the gas flowing upward. Due to this construction, the effective flow area for the gas stream is reduced to 50% of the frontal area of the package. The main feature of a closed drainage system is avoiding the contact between draining liquid and the gas flow. The liquid may flow co-currently until it reaches a lateral slit. Even with the 50% reduced flow area a higher gas throughput can be achieved because of the absence of flooding. The onset of re-entrainment will limit the gas handling capacity.
3.3 Gas flow

An impression of the gas flow field within a vane-type geometry can be obtained by means of numerical simulation. Wörlein [2] introduced recently a numerical approach to calculate streamlines but nowadays it is possible to simulate such flow in a more advanced way. To verify own numerical simulation results, visual observations of the gas flow and velocity measurements have been carried out.

3.3.1 Simulation of the gas flow

For the simulation of the gas flow field in vane-type demisters a computational fluid dynamics software package, called "PHOENICS", was used. A brief description is given of the governing equations and boundary conditions used for these simulations.

3.3.1.1 Theoretical background

In a basic case of a stationary, incompressible two-dimensional flow, two Navier-Stokes (N.S.) equations and a continuity equation have to be solved to obtain the gas flow field in a vane-type demister geometry. One N.S. equation in the main flow direction and one perpendicular to this main flow to ensure a complete transport of momentum (Figure 3.5).

\[ U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + g_i \]  

(3.2)

where \( \rho \) is the density, \( \nu \) the kinematic viscosity, \( P \) the instantaneous static pressure, \( U_i \) the instantaneous velocity component in the direction \( x_i \), \( U_j \) the instantaneous velocity component in the direction \( x_j \) and \( g_i \) the gravitational acceleration in direction \( x_i \). In this

34
study $g_i$ is neglected. The left hand term in Equation 3.2 represents the convection of momentum, the first term on the right the pressure gradient inducing the flow and the second term on the right indicates the diffusion of momentum. The continuity equation describes the flow field on basis of the law of conservation of mass and is written as (also in tensor notation):

$$\frac{\partial U_i}{\partial x_i} = 0$$ (3.3)

At high Reynolds numbers, often the case in industrial systems, the flow becomes turbulent and the velocity and pressure are constantly changing due to turbulent fluctuations. A way to describe isotropic turbulence, e.g. equal turbulent fluctuations in all directions, is to use a statistical approach which is suggested by Reynolds [9]. The velocity is divided into a mean and a fluctuating component:

$$U_i = \bar{U}_i + u_i$$ (3.4)

with $\bar{U}_i$ the mean and $u_i$ the fluctuating velocity. The average of this fluctuating velocity is zero. When these turbulent velocities are incorporated into Eq. 3.2 and the Boussinesque hypothesis is used, in which turbulent fluctuations are transformed into an added turbulent kinematic viscosity, the following expressions are obtained (for brevity the overbars indicating the averaged values are dropped):

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial U_i}{\partial x_j} - \bar{u}_i \mu_j \right)$$ (3.5)

according to Boussinesque:

$$-\bar{u}_i \mu_j = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}$$ (3.6)

where $\nu$ and $\nu_t$ are the laminar and turbulent viscosity respectively, $\delta_{ij}$ is the Kronecker delta and $k$ the kinetic energy per unit of mass of the fluctuating velocity components. In Eq. 3.5 the turbulent fluctuations appear in the diffusion term of the N.S. equation and contribute to the diffusion of momentum.

Solutions of the Navier-Stokes equations (adapted for turbulent flows) together with the continuity equation have been discussed by Patankar [10] and are incorporated by Spalding [11] into a software package called "PHOENICS". This package enables the calculation of heat, mass and momentum transfer problems.

To calculate the gas flow field using this software package the zig-zag channels have to be divided into a number of finite volumes. For vane-type demisters the basic domain is two-dimensional and the axis are in y- and z-direction (see Figure 3.6). The kind
of grid used in this case is called "Body Fitted" because the orientation of the cells is mostly parallel to the gas flow direction. This is usually done to minimize numerical diffusion. At the bends this leads to non rectangular grid cells.

Because of the elliptic nature of the momentum equation, boundary conditions must be specified at all boundaries of the domain considered. Boundary conditions have to be placed at the inlet, outlet and walls (Figure 3.6). At the inlet a constant mass flow enters the domain which has to leave at the outlet as a fully developed flow. This means only small velocity gradients in z-direction at the outlet. To create developed flow a number of extra cells are placed at the outlet region. The gas flow in the domain flows along the walls where the no-slip condition holds causing a developing boundary layer. For turbulent flows a developed boundary layer reveals three regions, shown in Figure 3.7 [12].

For the turbulent core, when the $Y^+$ value of the cell near the wall is above 11.5, the following relation can be used to couple the velocity profile to the wall friction without solving the flow equations within the boundary layer:

$$U^+ = \frac{1}{k} \ln(E \cdot Y^+)$$

(3.7)

![Schematic representation of a boundary layer in a turbulent flow](Figure 3.7)
where $U^*$ is the dimensionless velocity defined as the ratio of the velocity parallel to the wall and the wall friction velocity, represented as $(\tau/\rho)^{0.5}$, and $Y^*$ the dimensionless distance from the wall defined as a ratio of the wall distance times the wall friction velocity and the kinematic viscosity. $\kappa$ is the von Karman constant and $E$ a wall roughness parameter. Literature values for $\kappa$ and $E$ are presented in Table 3.2. For the viscous sublayer, where viscous effects are dominant, a laminar wall friction relation has to be used.

<table>
<thead>
<tr>
<th></th>
<th>$C_p C_d$</th>
<th>$C_{1e}$</th>
<th>$C_{2e}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\epsilon$</th>
<th>$\kappa$</th>
<th>$E$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Launder [14]</td>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
<td>0.435</td>
<td>9.0</td>
</tr>
<tr>
<td>Schlichting [12]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.4</td>
<td>9.0</td>
</tr>
<tr>
<td>Malin [15]</td>
<td></td>
<td></td>
<td></td>
<td>1.0</td>
<td>1.314</td>
<td>0.41</td>
<td>8.6</td>
</tr>
<tr>
<td>Hussain [16]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.41</td>
<td>7.77</td>
</tr>
<tr>
<td>Mongia [17]</td>
<td></td>
<td></td>
<td></td>
<td>0.9</td>
<td>1.17</td>
<td>0.41</td>
<td>9.0</td>
</tr>
<tr>
<td>Noat [18]</td>
<td></td>
<td></td>
<td></td>
<td>1.22</td>
<td>1.22</td>
<td>0.42</td>
<td>9.0</td>
</tr>
<tr>
<td>Svenson [19]</td>
<td></td>
<td></td>
<td></td>
<td>1.22</td>
<td>1.22</td>
<td>0.42</td>
<td>9.0</td>
</tr>
</tbody>
</table>

Table 3.2: Constants used in the k-\(\varepsilon\) model and wall friction functions [13].

Equation 3.5 and the boundary conditions at the inlet, outlet and walls are sufficient to describe a turbulent flow assuming that the turbulent viscosity does not depend on the flow conditions and is constant in the flow field. This is seldom valid and the turbulent viscosity ($\nu_t$) has therefore to be estimated on basis of a turbulence model. A number of turbulence models exists and they are mainly based on a velocity and length scale [9]. The k-\(\varepsilon\) model is a most commonly used one and has proven its validity for channel flow [13].

The k-\(\varepsilon\) model is a two differential equation model in which the turbulent kinetic energy per unit of mass (k) represents the velocity scale and where with \(\varepsilon\), the dissipation of kinetic energy per unit of mass, a length scale is introduced. Both the transport equations of k and \(\varepsilon\), which are derivatives from the N.S. equations, have to be solved. The transport equation of k is given as:

\[
U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \nu_t \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) \frac{\partial U_i}{\partial x_j} - \varepsilon
\]  

(3.8)

where $k$ is the kinetic energy of the fluctuating components, $\varepsilon$ the dissipation of energy, $\nu_t$ the turbulent viscosity and $\sigma_k$ a constant. For the transport equation of $\varepsilon$ the following expression is used:
\[ U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{v_i}{\sigma} \frac{\partial \varepsilon}{\partial x_i} \right) + \frac{C_{\text{ke}}}{k} \frac{\varepsilon}{k} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - C_{2\varepsilon} \frac{\varepsilon^2}{k} \] (3.9)

The empirical constants, \( \sigma, C_{\text{ke}} \) and \( C_{2\varepsilon} \), used in these expressions are derived from basic experiments, including for instance measuring turbulence in boundary layers. Table 3.2 represents values derived from experiments by several investigators [14-19].

These two differential equations provide the turbulent kinetic energy (k) and the dissipation of kinetic energy (\( \varepsilon \)). They are necessary to calculate the turbulent viscosity (\( \nu_t \)) appearing in Eq. 3.6. The expression for calculating this viscosity is given as:

\[ \nu_t = C_p C_d \frac{k^2}{\varepsilon} \] (3.10)

where \( C_p \) and \( C_d \) are constants for which numerical values can be found in Table 3.2.

Near the wall there is an important dissipation and production of kinetic energy due to wall friction. Roston derived [20] a relation between the dissipation and production of kinetic energy near the wall. This relation (boundary condition) reads:

\[ \varepsilon = \frac{2 d_w}{C_p C_d k^{1.5}} \ln \left( \frac{E}{\nu} \frac{d_w (C_p C_d)^{0.25} k^{0.5}}{\nu} \right) \] (3.11)

where \( \kappa \) and \( E \) are constants as used in Eq. 3.7 and their values can be found in Table 3.2. \( d_w \) is the distance from the wall considered. This relation gives a more accurate description of dissipation of energy near the wall than Eq. 3.7.

The expressions 3.2 up to 3.11 are the basic equations contained in the software package "PHOENICS". It has been used to calculate the turbulent flow in vane-type demisters of which several cases are described below.

### 3.3.1.2 Numerical studies

The possibilities of numerical simulation were investigated considering some characteristic cases. In these cases a basic vane geometry having four bends with sharp edges and bend angles (2\( \alpha \)) varying between 60° and 90° has been investigated. The blade length was kept 6 cm, air was used as fluid and no obstructions were placed on the walls or in the gas stream. The k-\( \varepsilon \) model was used to describe the turbulence. Instead of the use of Eq. 3.7 for the wall friction Eq. 3.11 was applied as wall boundary condition. For Eq. 3.11 the constants of Malin [15] were used and for the k-\( \varepsilon \) turbulence model (formulated by Eq. 3.8 and Eq. 3.9) the constants of Noat [18] and Svenson [19] were adopted [13].

The first case was a basic vane geometry with a blade distance of 2 cm and a bend angle of 90°. The resulting velocity vectors in a part of the domain are shown in
Figure 3.8: Vector plot of a part of the simulated gas flow field, case 1, $2\alpha = 90^\circ$, $s = 2.0$ cm, $U_s = 5.0$ m/s, $\rho_s = 1.2$ kg/m$^3$ and $\eta_s = 1.8 \times 10^{-5}$ kg/ms.

Figure 3.9: Vector plot of a part of the simulated gas flow field, case 2, $2\alpha = 60^\circ$, $s = 2.0$ cm, $U_s = 5.0$ m/s, $\rho_s = 1.2$ kg/m$^3$ and $\eta_s = 1.8 \times 10^{-5}$ kg/ms.
Figure 3.10: Vector plot of a part of the simulated gas flow field, case 3, $2\alpha = 60^\circ$, $s = 3.0 \text{ cm}$, $U_g = 5.0 \text{ m/s}$, $\rho_g = 1.2 \text{ kg/m}^3$ and $\eta_g = 1.8 \times 10^{-5} \text{ kg/ms}$.

Figure 3.11: Distribution of 'k' along the flow path in a vane-type demister, $2\alpha = 90^\circ$, $s = 2.0 \text{ cm}$, $U_g = 5.0 \text{ m/s}$, $\rho_g = 1.2 \text{ kg/m}^3$ and $\eta_g = 1.8 \times 10^{-5} \text{ kg/ms}$. 

40
Figure 3.8. In this figure a recirculation zone in the upper corner and one behind the bend can be noticed. The later occupies about 15% of the channel cross-section.

A second case, with a bend angle of 60° as shown in Figure 3.9 (case 2), shows a recirculation zone behind the bend occupying only 10% of the cross-section. As expected this recirculation zone is smaller than in the first (90°) case. The same applies for the recirculation zone in the upper corner.

Since the velocity vectors are very small at the recirculation zones and difficult to be seen, the corresponding values of two cross-sections are given in Table 3.3. The negative values indicate a back flow.

Expanding the blade distance from 2 to 3 cm for a bend angle of 60° (case 3) results in larger recirculation zones as shown in Figure 3.10. These zones are stretched from corner to corner and the core of the gas flow almost follows a straight line.

The manifestation of the turbulence in the flow channel is illustrated for case 1. As shown in Figure 3.11 the turbulent fluctuations (distribution of k) are increasing along the flow path. Especially near the wall and at the bends the amount of fluctuating kinetic energy appears higher than in the bulk which is due to the generation of k at the walls.

Figure 3.12 shows at several cross-sections the turbulence intensity, $\sqrt{u'^2/U_i}$. At the places were recirculation zones are expected (cross-section -B- and -C-, for $Y/s = 0.9 - 1.0$) the turbulent intensity increases. This tendency is also found at the places with high velocities and large velocity gradients (e.g. at the bend, -C-, $Y/s = 0.0 - 0.1$). For turbulent pipe flow a value of 0.11 [21] for the turbulence intensity is usually found. In this geometry a much higher intensity can be seen. When lowering the bend angle (2\(\alpha\)) to 60° a small decrease in k will be found but the turbulent intensity is still higher than 0.11.

The regions of high dissipation of turbulent kinetic energy ($\epsilon$) are illustrated in Figure 3.13. As expected a maximum dissipation is found near the walls and at the bends, a place with large velocity gradients.

Figure 3.14 illustrates the simulated effect of increased gas density on the gas velocities at a certain cross-section in the domain for case 1. As can be seen obviously there is no influence on the gas velocity. The same was observed for k and $\epsilon$.

![Figure 3.13: Distribution of $\epsilon$ along the flow path in a vane-type demister.](image)
<table>
<thead>
<tr>
<th>Y (-)</th>
<th>$U_g$ (m/s)</th>
<th>$U_g$ (m/s)</th>
<th>$U_g$ (m/s)</th>
<th>$U_g$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-A-</td>
<td>-B-</td>
<td>-A-</td>
<td>-B-</td>
</tr>
<tr>
<td>1</td>
<td>14.4</td>
<td>-3.3</td>
<td>10.9</td>
<td>-1.2</td>
</tr>
<tr>
<td>2</td>
<td>10.3</td>
<td>-0.4</td>
<td>9.1</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>12.2</td>
<td>1.8</td>
<td>9.6</td>
<td>2.0</td>
</tr>
<tr>
<td>4</td>
<td>11.5</td>
<td>3.6</td>
<td>9.2</td>
<td>3.3</td>
</tr>
<tr>
<td>5</td>
<td>10.7</td>
<td>5.0</td>
<td>8.7</td>
<td>4.5</td>
</tr>
<tr>
<td>6</td>
<td>9.7</td>
<td>6.1</td>
<td>8.1</td>
<td>5.4</td>
</tr>
<tr>
<td>7</td>
<td>8.6</td>
<td>6.9</td>
<td>7.7</td>
<td>6.1</td>
</tr>
<tr>
<td>8</td>
<td>7.5</td>
<td>7.5</td>
<td>7.1</td>
<td>6.7</td>
</tr>
<tr>
<td>9</td>
<td>6.4</td>
<td>8.0</td>
<td>6.5</td>
<td>7.1</td>
</tr>
<tr>
<td>10</td>
<td>5.3</td>
<td>8.4</td>
<td>5.8</td>
<td>7.3</td>
</tr>
<tr>
<td>11</td>
<td>4.2</td>
<td>8.7</td>
<td>5.0</td>
<td>7.5</td>
</tr>
<tr>
<td>12</td>
<td>3.1</td>
<td>9.0</td>
<td>4.1</td>
<td>7.5</td>
</tr>
<tr>
<td>13</td>
<td>2.0</td>
<td>9.1</td>
<td>3.3</td>
<td>7.5</td>
</tr>
<tr>
<td>14</td>
<td>1.0</td>
<td>9.3</td>
<td>2.5</td>
<td>7.3</td>
</tr>
<tr>
<td>15</td>
<td>0.1</td>
<td>9.4</td>
<td>1.7</td>
<td>7.1</td>
</tr>
<tr>
<td>16</td>
<td>-0.5</td>
<td>9.5</td>
<td>1.1</td>
<td>6.9</td>
</tr>
<tr>
<td>17</td>
<td>-1.0</td>
<td>9.6</td>
<td>0.5</td>
<td>6.6</td>
</tr>
<tr>
<td>18</td>
<td>-1.3</td>
<td>9.8</td>
<td>0.1</td>
<td>6.3</td>
</tr>
<tr>
<td>19</td>
<td>-1.4</td>
<td>9.9</td>
<td>-0.2</td>
<td>6.0</td>
</tr>
<tr>
<td>20</td>
<td>-1.4</td>
<td>9.7</td>
<td>-0.5</td>
<td>5.5</td>
</tr>
</tbody>
</table>

Table 3.3: Calculated velocities perpendicular to the cell surface for two cross-sections in the domain (-A-, -B-), $Y =$ cell number.
Figure 3.12: Distribution of the turbulence intensity for three cross-sections in a vane-type demister, $2\alpha = 90^\circ$, $s = 2.0 \text{ cm}$, $U_g = 5.0 \text{ m/s}$.

- A-
- B-
- C-

Figure 3.14: Velocity distribution for the cross-section A-A' in the calculation domain for two different gas densities, $2\alpha = 90^\circ$, $s = 2.0 \text{ cm}$, $\eta_g = 1.8 \times 10^5 \text{ kg/m/s}$.

- $\rho_g = 1.2 \text{ kg/m}^3$
- $\rho_g = 50 \text{ kg/m}^3$
3.3.2 Experimental verification

In the next paragraphs results of the above described simulations are compared with visualisation experiments and pitot-tube velocity measurements. These experiments were carried out in a small wind tunnel described in Chapter 2.

3.3.2.1 Visualisation experiments

Sharp bends of a basic vane-type demister cause discontinuities in the flow pattern and induce recirculation zones (Figure 3.15). Several authors [2,5] observed these recirculation zones and assigned a positive influence on the separation efficiency because of higher velocities in the main gas stream.

![Figure 3.15](image)

Figure 3.15: The locations of possible recirculation zones.

Presence of recirculation zones was visualised by smoke tracer photos. As shown in the Figures 3.16a and 3.16b a vane geometry with a bend angle of 90° induces two recirculation zones. The one behind the bend covers about 40% of the cross-section. The recirculation zone has a finite length because of the diverging main gas flow. Compared with the simulation (Figure 3.8) the observed one is twice as large but the length is in good agreement (in both cases about 2.2 cm). At the top of the bend there is a small circular recirculation zone (Figure 3.16b) similar to the one found in the simulation.

When the bend angle is decreased to 60° the large recirculation zone becomes smaller (Figure 3.16c) and covers only 30% of the cross-section. The simulation shows the same trend (Figure 3.9), however it occupies only 10% of the cross-section. The length of the recirculation zone is approximately the same as found in the 90° case.

If the blade distance is increased from 2 to 3 cm the recirculation zone becomes much larger and stretches to the small recirculation zone of the next corner (Figure 3.16d). This is also shown in the simulation (Figure 3.10). The flow lines in the centre become more and more straightened.

With rounded bends neither recirculation zones nor defined discontinuities were found.
Figure 3.16a: Smoke tracer photo visualising the recirculation zone in the upper corner of a bend of 90°, $s = 2$ cm.

Figure 3.16b: Smoke tracer photo visualising the recirculation zone behind a bend of 90°, $s = 2$ cm.
Figure 3.16c: Smoke tracer photo visualising the recirculation zone behind a bend of 60°, $s = 2$ cm.

Figure 3.16d: Smoke tracer photo visualising the change in recirculation zone when increasing the blade distance from 2 to 3 cm, $2\alpha = 60^\circ$. 
3.3.2.2 Velocity and pressure drop measurements

Figure 3.17 shows the comparison between measured and calculated velocities in the vane-type demister in the cross-section indicated in Fig. 3.17. The agreement is reasonably good. In other measurements larger deviations were observed on the places with very low velocities, especially in the neighbourhood of recirculation zones.

![Graph showing velocity comparison](image)

Figure 3.17: Comparison between pitot-tube measurements and calculated velocities in a bend of the vane-type demister, $2\alpha = 60^\circ$, $s = 3.0$ cm.

- ○ measured
- calculated

Another way of evaluating the quality of simulations is to compare calculated and measured pressure drops. For the basic geometries considered experimental pressure drop data are available in literature [4,5]. Table 3.4 shows these pressure drop data, compared with those calculated from simulated flow patterns. The observed discrepancy has to be attributed to the poor simulation of the recirculation zones by "PHOENICS", which were too small compared with those observed on the smoke tracer photos.

<table>
<thead>
<tr>
<th></th>
<th>Experimental</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop (Pa) [4]</td>
<td>188</td>
<td>118</td>
</tr>
<tr>
<td>Pressure drop (Pa) [5]</td>
<td>244</td>
<td>200</td>
</tr>
</tbody>
</table>

Table 3.4: Comparison of measured and calculated pressure drop, $\rho_g = 1.2$ kg/m$^3$ and $\eta_g = 1.8 \times 10^{-3}$ kg/m/s.

- [4] $2\alpha = 60^\circ$, $s = 2.0$ cm, $U_g = 7.0$ m/s
- [5] $2\alpha = 90^\circ$, $s = 2.8$ cm, $U_g = 5.3$ m/s
3.3.3 Adapted geometry

In previous paragraphs the gas flow in the basic geometry of a vane-type demister was discussed. One of the first things observed was the presence of recirculation zones in the upper corner and after a bend (see Figure 3.8). These recirculation zones are energy-consuming (resulting in extra pressure drop) and create an undesirable amount of turbulence. A possibility to avoid the occurrence of recirculation zones is to adapt the geometry in such a way that streamlined flow is ensured. This can be done in a way as shown in Figure 3.18.

![Adapted geometry](image)

Figure 3.18: Adapted geometry.

The gas flow field in the new geometry is simulated with a grid presented in Figure 3.19. The sub-domains are divided into a number of cells and using the boundary conditions, described in Chapter 3.3.1.1, a gas flow field as presented in Figure 3.20 is obtained. It shows that no separation zones are formed and that at almost all places a relatively large velocity vector can be found.

Comparing the new demister geometry (adapted bends, $2\alpha = 60^\circ$ and $s = 2$ cm) with the basic geometry (sharp bends, $2\alpha = 60^\circ$ and $s = 2$ cm) the following differences can be observed. The pressure drop of the new geometry is twice as low as for the basic geometry and equal to experimental results, as presented in Table 3.5. In contrast to the simulation of the basic vanes (Table 3.4, with poorly predicted separation zones) the prediction of pressure drop is good. Table 3.5 indicates also a considerable improvement with respect to the turbulence intensity.

![Calculation domain for the adapted geometry](image)

Figure 3.19: Calculation domain for the adapted geometry.
Figure 3.20: Vector plot of a part of the simulated gas flow field for the adapted geometry, $2\alpha = 60^\circ$, $s = 2.0$ cm, $U_g = 7.0$ m/s, $\rho_g = 1.2$ kg/m$^3$ and $\eta_g = 1.8 \times 10^{-5}$ kg/ms.

<table>
<thead>
<tr>
<th></th>
<th>Simulation of basic geometry</th>
<th>Simulation of adapted geometry</th>
<th>Experiments with adapted geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta P$ (Pa)</td>
<td>118</td>
<td>61</td>
<td>61</td>
</tr>
<tr>
<td>$U_g = 7$ m/s</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\sqrt{u_i^2/U_i}$</td>
<td>0.14 - 0.25</td>
<td>0.10 - 0.15</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 3.5: Comparison between simulation and experiment for two demister types which are basically identical ($2\alpha = 60^\circ$, $s = 2.0$ cm).

3.3.4 Discussion

Numerical simulation showed smaller recirculation zones than actually observed in smoke tracer experiments. A reason for this deviation has not only to be allocated to the relative simplicity of the $k$-$\varepsilon$ model. Apparently the numerical method used by "PHOENICS" is not able to calculate these recirculation zones with sufficient accuracy. Other "PHOENICS" users [22] have found the same underprediction of recirculation zones which points to imperfections in the computer code.
The main flow velocity appears to be calculated well when the simulation results are compared with pitot-tube measurements. This measuring technique is easy and appeared to be adequate for getting an insight into the gas flow field in a vane-type geometry. However, the error made by the pitot-tube becomes larger when the velocities are smaller (as already explained in Chapter 2) or when the pitot-tube is placed in the recirculation zone. The relatively big pitot-tube will disturb the small recirculation zone significantly. At such places and with the low velocities L.D.A. (Laser Doppler Anemometry) measurements are probably more suitable. With these techniques it is also possible to obtain information about turbulent fluctuations (k).

The simulations indicate that the gas velocity profiles and the turbulent properties are not significantly influenced by an increase in the gas density. This is caused by two factors; the absence of the gas density in the k-ε model and the fact that the diffusion term dominates the convection term (see Eq. 3.2). A large diffusion term compared to convection term eliminates the gas density from the Navier-Stokes equation.

An adaption of the channel geometry eliminates the recirculation zones. In contrast to the basic examples (where recirculation zones are underpredicted) the calculated pressure drop is now in good agreement with the measured one. Improving the geometry also resulted in the decrease of the turbulent fluctuations.

3.3.5 Conclusions

Software package "PHOENICS" proved to be a useful tool for the simulation of the gas flow in vane-type demisters. However, in cases where recirculation zones occur their size is underpredicted.

Smoke tracer experiments gave a good insight into the gas flow field in vane-type demisters. Pitot-tube gas velocity measurements are in good agreement with the simulations at higher gas velocities.

Simulation of the gas flow field in the adapted geometry showed the expected absence of recirculation zones. Because of the better aerodynamic geometry a lower pressure drop and a reduced turbulence intensity was achieved.

With respect to high pressure applications of the vane-type demisters it was noticed that simulations with an increased gas density did not result in a change of the gas flow field obtained under atmospheric conditions.

3.4 Motion of droplets

In a vane-type demister droplets which are carried by a gas flow experience inertial forces which lead to a motion of the droplet relative to the gas. The droplet motion can result in a collision with the vane blades. Simulation of the motion of droplets passing through a vane geometry enables therefore the investigation of the separation efficiency.
The governing equations for the motion of the droplets are described and used to calculate the separation performance. The results are compared with literature models and experimental work.

### 3.4.1 Modelling of droplet motion

Motion of a droplet [9,23,24,25] in a gas flow field is usually described using the Lagrangian approach. Discrete droplets are followed through the domain assuming no distortion of the gas flow. A force balance determines the behaviour of the droplets in the gas flow and the forces acting on the droplet are respectively:

**The drag force, \( F_d \).**

This is one of the major forces acting on the frontal surface of a droplet. It describes the frictional energy necessary for the droplet to travel through the gas flow field. This force can be expressed as:

\[
F_d = \frac{\pi}{4} d_p^2 \; Cd \; \frac{\rho_g}{2} \; U_{rel} \; |U_{rel}|
\]  

(3.12)

where \( d_p \) the droplet diameter, \( \rho_g \) the gas density, \( U_{rel} = (U_g - U_p) \) the slip velocity between gas \( (U_g) \) and droplet velocity \( (U_p) \) and \( Cd \) a friction factor defined, according to Kaskas [26], as:

\[
Cd = \frac{24}{Re_p} + \frac{4}{Re_p^{0.8}} + 0.4
\]  

(3.13)

\( Re_p \) denotes the characteristic Reynolds number related to the droplet diameter:

\[
Re_p = \frac{\rho_g \; |U_{rel}| \; d_p}{\mu_g}
\]  

(3.14)

in which \( \mu_g \) the gas viscosity.

**The inertial force, \( F_I \).**

This force appears when the droplet trajectory is altered due to a change of the gas velocity and/or of the gas flow direction. The different velocity vectors cause the droplet to change its direction and velocity. An expression for this force is:
\[ F_i = - \frac{\pi}{6} \rho_p d_p^3 \frac{dU_p}{dt} \] \hspace{1cm} (3.15)

with \(dU_p/dt\) the velocity change of a droplet during a time step.

**The pressure force, \( F_p \).**

This is another important force caused by a pressure gradient over the droplet resulting from the acceleration of the fluid and can therefore be written as:

\[ F_p = \frac{\pi}{6} \rho_g d_p^3 \frac{dU_g}{dt} \] \hspace{1cm} (3.16)

with \(dU_g/dt\) the velocity change of the gas during a time step. In this relation the viscous effects of the gas are neglected.

**The virtual mass force, \( F_a \).**

This force contains the relative acceleration of the gas with respect to the droplet. This force is written as:

\[ F_a = - k \frac{\pi}{6} \rho_g d_p^3 \left( \frac{dU_p}{dt} - \frac{dU_g}{dt} \right) \] \hspace{1cm} (3.17)

where \(k\) is a constant varying between 0 and 1. For spherical droplets a value of 0.5 is often used.

From the force balance \( F_d + F_i + F_p + F_a = 0 \) the following expression for the motion of a single droplet in a gas flow is obtained:

\[ \left( \frac{\rho_p + k \rho_g}{\rho_g} \right) \frac{dU_p}{dt} - (k + 1) \frac{dU_g}{dt} = \frac{3}{4} \frac{Cd}{d_p} |U_{rel}| \] \hspace{1cm} (3.18)

Other forces like gravity, the so called Saffman lift [27] and Basset term [28], etc. are forces of minor importance in this kind of flow [9]. Gravity is neglected because of the large difference in the absolute values of the gas velocity and the falling velocity of a droplet (small size). The Saffman lift [27] only acts in a high shear region of the gas flow and the Basset term [28] describes the viscous resistance of the fluid due to the motion of the droplet. Since the gas viscosity is low this force can be neglected. Turbulent interactions and particle-particle interactions are neglected also.

When the droplet/gas density ratio is large, in the order of 100-1000, the pressure force and added mass force can be neglected. The two major forces (drag and
Figure 3.21: Forces acting on a droplet in a changing gas flow.

Inertial forces acting on a droplet ($d_p$) in a gas flow with velocity ($U_g$) are presented in Figure 3.21. The droplet has at that particular point a velocity ($U_p$) which causes a relative velocity of the gas ($U_g - U_p$) and a drag force. The inertial force can only act in the opposite direction of the force. The force balance can then be written as:

$$\frac{dU_p}{dt} = \frac{3}{4} \frac{Cd}{d_p} \frac{\rho_g}{\rho_p} U_{rel} |U_{rel}|$$

(3.19)

However, when the droplet/gas density ratio decreases below 100 the pressure and added mass forces will play a moderate role and Eq. 3.18 has to be used for predicting the motion of the droplet. This relation can be simplified when assuming that, at higher densities, the acceleration of the gas and of the droplet are of the same magnitude (which is the case when the droplet trajectories and gas streamlines do not differ too much). In that case Eq. 3.18 can be rewritten in:

$$\frac{dU_p}{dt} = \frac{3}{4} \frac{Cd}{d_p} \left( \frac{\rho_g}{\rho_p - \rho_g} \right) U_{rel} |U_{rel}|$$

(3.20)

From this equation, which is nearly identical to Eq. 3.19, it can be seen that only the relative mass of the droplet ($\rho_p - \rho_g$) has to be accelerated.

To calculate the droplet trajectories in a two-dimensional case the velocity components have to be expressed in a two dimensional Cartesian coordinate system.

When the gas and droplet velocity are known at the start location of the droplet the relative velocity between droplet and gas can be calculated and used to give the velocity change of the droplet according to Eq. 3.20.

The force balance can be solved numerically; for instance by a Runge-Kutta procedure. This method has the disadvantage that it creates very small time steps for small droplets which means an extremely long solution time. Another way to solve this balance is to linearise the equation as described by Durst [29]. For a short time step, assuming an instantaneous constant gas velocity, the velocity change of the droplet becomes:
\[ U_{p,t+\Delta t} = U_g - (U_g - U_{p,t0}) \exp\left( -\frac{\Delta t}{\tau} \right) \] (3.21)

where \( U_g \) is the gas velocity, \( U_{p,t0} \) the droplet velocity of the previous time step, \( U_{p,t+\Delta t} \) the new droplet velocity and \( \Delta t \) the time step during which the gas velocity is constant. The
time constant, \( \tau \), is defined as:

\[
\tau = \frac{4}{3} \frac{\left( \rho_p - \rho_g \right) \diameter_p^2}{\mu_g \Cd \Re_p} \] (3.22)

The droplet position after the time step \( \Delta t \) can be calculated with:

\[
x = x_0 + \left( U_{p,t+\Delta t} + U_{p,t0} \right) \frac{\Delta t}{2} \] (3.23)

with \( x \) and \( x_0 \) as the new and old droplet position respectively.

Droplets travelling through a gas flow field transfer and receive momentum due
to the drag force [30]. This momentum change has an influence on the gas flow field
which has not been taken into account here. This is valid only for low droplets
concentrations which is normally the case in vane-type demisters.

3.4.1.1 Numerical studies

The equation of motion and the solution procedure described above were used
in a computer program which enables the prediction of the droplet trajectories using the
gas flow field calculated with the "PHOENICS" computer code. Below a few examples are
given of calculated trajectories in a vane demister.

The calculated trajectories of droplets with a diameter of 25 micron entering
the geometry on different places are depicted in Figure 3.22. Several droplets collide
against the wall and the non-separated droplets are directed by the gas flow to one main
stream from which they can not escape. These results show, in an idealised case which
does not take into account the effect of turbulence on the droplet motion, that a vane-type
demister needs only two bends to separate droplets. The program was further used to make
a parametric study.

The trajectories of different droplets (varying in diameter), entering at one
central position, are depicted in Figure 3.23. It can be seen that they all make a sinusoidal
motion but with a varying amplitude depending on their diameter. It is obvious that in this
situation, this particular vane geometry and given flow conditions, only a droplet of 25
micron or smaller has the chance to pass the flow channel.

A variation of the gas velocity does not affect significantly the trajectories of a
10 micron droplet (Figure 3.24). The gas velocity must be much higher to force a droplet
Figure 3.22: Droplet trajectories for a droplet of 25 micron, $2\alpha = 60^\circ$, $s = 3.0$ cm, $U_g = 5.0$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms.

Figure 3.23: Droplet trajectories of different droplet sizes, $2\alpha = 60^\circ$, $s = 3.0$ cm, $U_g = 5.0$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms.

- - - - - 10 micron
- - - - - 15 micron
- - - - - 20 micron
- - - - - 25 micron

Figure 3.24: Droplet trajectories of a 10 micron droplet at different gas velocities, $2\alpha = 60^\circ$, $s = 2.0$ cm, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms.

- - - - - 5 m/s
- - - - - 7.5 m/s
- - - - - 10 m/s

of this size to collide against the wall in this geometry. A better way to catch this droplet is to reduce the blade distance. This means that the sideward displacement of the droplet is less before colliding against the wall (compare the 10 micron droplet trajectories in the Figures 3.23 and 3.24). An other possibility is to increase the bend angle of the vane geometry so that the inertial forces on the droplet are increased due to a sharper change in direction of the gas flow.
3.4.2 Prediction of separation efficiency

With the model for the droplet trajectories, based on "PHOENICS" gas flow calculations, the separation efficiency can be predicted as a function of physical properties, geometry and droplet size. On the other hand known literature models assume an uniform gas flow field which is a symplifying assumption.

Several authors [3,31,32] used this assumption together with Stokes friction to derive a simple expression for calculating the efficiency. A balance between the drag force and inertial force, as presented in the Equations 3.12 and 3.15 respectively determines the radial displacement of a droplet during its presence in a semi-circular bend. The ratio of the radial displacement of the droplet and the blade distance gives directly the separation efficiency for a droplet of a given size:

$$\eta_p = \frac{(\rho_p - \rho_g) \cdot U_s \cdot d_p^2 \cdot 2 \cdot \alpha}{18 \cdot \mu_g \cdot s \cdot (\cos \alpha)^2}$$  \hspace{1cm} (3.24)

where $U_s$ is the superficial gas velocity, $2\alpha$ the bend angle (radians) and $s$ the distance between the blades normal to the superficial gas velocity (geometry parameters which are presented in Figure 3.3). This relation gives the efficiency for one bend and is realistic only when $0 < \eta_p < 1$.

The droplet diameter which will be separated for 100% ($\eta_p = 1$) can be calculated as follows:

$$d_p^{100} = 3 \left( \frac{\mu_g}{(\rho_p - \rho_g) \cdot U_s} \right)^{0.5} \left( \frac{s \cdot (\cos \alpha)^2}{\alpha} \right)^{0.5}$$  \hspace{1cm} (3.25)

To predict the separation efficiency for two or more bends, Eq. 3.24 was extended. Assuming (partial) remixing of the droplet laden gas flow after each bend, the following expression for the total efficiency can be applied [3]:

$$\eta_t = 1 - (1 - \eta_p)^m$$  \hspace{1cm} (3.26)

where $n$ is the number of bends and $m$ is a remix factor being 1 in the case of total remixing assumed by Calvert [30] and Bürkholz [31] or 0.5 to 0.63 as reported by Gardner [3].

Ushiki [4] neglected the remixing ($m = 1$), and assumed only 50% of contribution of the entrance bend. This resulted in the following expression:

$$\eta_t = 1 - (1 - \eta_p)^{n - 0.5}$$  \hspace{1cm} (3.27)
The assumption of an uniform gas flow used in the above described models can lead to false predictions of the separation efficiency. The "PHOENICS" package enables the simulation of a realistic gas flow field, which, in combination with droplet trajectory calculations results in a more accurate prediction of the separation efficiency.

3.4.2.1 Simulation of droplet separation

The influences of gas velocity, bend angle, blade distance, gas and liquid density and gas viscosity were investigated. The gas flow field in the vane geometry was calculated with "PHOENICS" in the same way and for the same geometry as described in Chapter 3.3.1.2, and the droplet trajectories were calculated using the method of Durst [29].

Figure 3.25 shows a typical example of a separation curve, for vane-type demisters, obtained by numerical simulation. Striking is the sharp increase in efficiency from 22% for a droplet of 25 micron to 100% for a droplet of 26 micron. This abrupt change is experimentally not found [5]. It appears because of the formation of one main droplet stream, as illustrated in Figure 3.22, because we neglect the influence of turbulence on the droplet motion. The droplet trajectories simulations showed that only the first two bends are necessary to separate the droplets.

Figures 3.26a-f show the different relations between physical and geometry parameters and the droplet diameter, which is separated for 100%. The gas flow field was predicted by numerical simulation. In these figures a base case ($2\alpha = 60^\circ$, $s = 2$ cm, $U_g = 5$ m/s, $\rho_g = 1.2$ kg/m³, $\mu_g = 1.8 \times 10^{-4}$ kg/ms, $\rho_l = 1000$ kg/m³, $d_p^{100} = 19$ micron) was chosen around which the parameters were varied. Other simulations showed that when starting

![Figure 3.25](image)

**Figure 3.25:** Typical separation efficiency curve obtained by numerical simulations, $2\alpha = 60^\circ$, $s = 3$ cm, $U_g = 5$ m/s, $\rho_g = 1.2$ kg/m³, $\mu_g = 1.8 \times 10^{-3}$ kg/ms.
Figure 3.26: Influence of the change of process and geometry parameters on the separation of droplets, $d_{p100}$ (simulations with "PHOENICS").

* Base case ($2\alpha = 60^\circ$, $s = 2$ cm, $U_g = 5$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms, $\rho_l = 1000$ kg/m$^3$, $d_{p100} = 19$ micron)

a bend angle d liquid density
b blade distance e gas viscosity
c gas velocity f gas density
from a different base case the same relations between the parameters and the droplet diameter, which is separated for 100%, were found. Combining these relations the following expression was obtained for describing the droplet size which will be separated for 100%:

\[
   d_p^{100} = K \left( \frac{\mu_g}{(\rho_p - \rho_g) U_g} \right)^{0.5} \left( \frac{s^2}{\alpha^2} \right)^{0.5}
\]  

(3.28)

K has the dimension of m^{-0.5}. In this expression the parameters (especially the gas/droplet variables) are almost the same as in expression 3.25. However, the separated droplet diameter decreases more rapidly when the blade distance (s) is decreased or the bend angle (\alpha) is increased. There is a considerable influence of gas density, gas velocity and vane geometry, as presented in the Figures 3.26a - 3.26f. Figure 3.27 shows the variation of

![Graph showing the variation of parameter K with gas density and bend angle.](image)

Figure 3.27: The parameter 'K', in Eq. 3.28, as function of the bend angle (in radians) and gas density.

parameter K, in Eq. 3.28, at increased gas densities. Not only gas density but also the blade angle influences K. It can be seen that the influence of the gas density is larger when the bend angle is increased.

The expression 3.28 is only valid for a sufficient blade length meaning that the inertial force has been utilized completely. Figure 3.28 shows that for different blade lengths the droplet size which can be separated for 100% is constant for conditions where \(l(\sin \alpha)/s > 1\) (see Eq. 3.1).

When the two models, Eq. 3.27 from literature and Eq. 3.28 obtained by numerical simulation, are compared a considerable discrepancy can be observed (Figure 3.29). Expression 3.27 predicts a smooth efficiency curve and, in contrast to expression 3.28, a better separation for the smaller (<15 micron) droplets. Expression 3.28 shows a 100% separation of droplets larger than 15 micron while Eq. 3.25 predicts a 100% efficiency for droplets of 25 micron.
Figure 3.28: Influence of blade length on the separation of droplets.

Figure 3.29: Comparison of two models predicting separation efficiency (2α = 90°, s = 2.8 cm, $U_g = 5.3$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms, n = 2, m = 1).

---

Eq. 3.27, Ushiki [4]
Eq. 3.28, "PHOENICS"

3.4.3 Droplet separation in the adapted geometry

Chapter 3.3.3 shows that the gas flow in a vane geometry can be improved by avoiding recirculation zones. This results in a varying cross-section of the flow channels. The calculated trajectories of a 12 micron droplet in a gas flow field calculated with "PHOENICS" and the equations of droplet motion are shown in Figure 3.30. The droplets enter the demister on different locations and after two bends one main droplet stream, consisting of not separated droplets, is formed. A similar behaviour was found for the not adapted geometry (Figure 3.22).
The separation efficiency of both geometries is shown in Figure 3.31 (both with a blade distance of 2 cm, a bend angle of 60°). This figure shows a calculated 100% separation efficiency for a droplet with a diameter of 16 micron. For smaller droplets the separation efficiency of the adapted geometry is somewhat higher.

![Graph of simulated droplet trajectories](image)

**Figure 3.30**: Simulated droplet trajectories of a 12 micron droplet in the adapted geometry, $2\alpha = 60°$, $s = 2$ cm, $U_g = 7$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms.

![Graph of calculated separation performance](image)

**Figure 3.31**: Calculated separation performance for two geometries, $2\alpha = 60°$, $s = 2$ cm, $U_g = 7$ m/s, $\rho_g = 1.2$ kg/m$^3$, $\mu_g = 1.8 \times 10^{-5}$ kg/ms.

---

### 3.4.4 Experimental verification

The separation efficiency of a prototype vane package with adapted geometry, which appeared to be a promising demister, was investigated in both the atmospheric and pressurized rig. The rigs and the test set-up are described in Chapter 2.

Figure 3.32 shows the adapted geometry (type III) used during the tests. The vane consists of metal sheets connected in parallel and on the places where the geometry was adjusted drainage compartments were incorporated. Lateral narrow slits enable the
separated liquid to enter the drainage compartments and the liquid drains horizontally without a further contact with the gas flow. The vane has three bends and the blades are placed 2 cm from each other (type IIIa). The blade length, from bend to bend, is 6 cm which satisfies the criterion of Eq. 3.1, i.e. \( l(\sin \alpha)/s > 1 \).

Figure 3.33 shows the separation efficiency as a function of the superficial gas velocity for different liquid loads in the gas. At a fixed liquid load an increasing gas velocity increases the separation efficiency. But above a certain gas velocity no further increase in efficiency is found. During each run with a constant liquid load the droplet size distribution was constant by keeping the liquid flow through the spray nozzle at a constant value. It is characterized by the droplet diameter, \( d_{50} \), which means that 50% of the total droplet volume is below this diameter. At a higher liquid flow through the nozzle the droplet size distribution shifts towards larger droplets, and consequently therefore a larger \( d_{90} \) is found. The decrease in the amount of small droplets causes a higher total efficiency.

Figure 3.34 also shows that an increase in gas velocity, from 6 - 8 m/s, influences marginally the separation efficiency. Probably, at a gas velocity of 6 m/s the separation efficiency is at its maximum for this type of vane and cannot be further increased. According to Eq. 3.28 a droplet with the diameter below 17 micron then cannot be separated.

When the separation results of the demister investigated (type IIIa) are interpreted in terms of the smallest droplet size separated for 100% the minimum size of the droplets which can be separated can be obtained. Figure 3.35 shows the comparison of the experimental and the calculated data (relation 3.28). The droplet size which could be separated in reality is smaller than the predicted one.

A calculation of the overall efficiency of vane-type IIIa with the models presented in Chapter 3.4.2 (Eq. 3.27 found in literature and Eq. 3.28 based on "PHOENICS" simulations) shows that the model derived from the numerical calculations (Eq. 3.28) is in reasonable agreement with the level of the experimentally obtained efficiency (Figure 3.36). The model tends to over predict the effect of increasing efficiency with increasing gas velocity. This in contrast to Expression 3.27 which predicts a considerable lower efficiency.
Figure 3.33: Separation efficiency of vane-type IIIa as a function of the gas velocity and droplet size distribution (air/water experiments).

<table>
<thead>
<tr>
<th>Liquid load (l/hr)</th>
<th>$d_{50}$ (micron)</th>
</tr>
</thead>
<tbody>
<tr>
<td>400</td>
<td>90</td>
</tr>
<tr>
<td>280</td>
<td>70</td>
</tr>
<tr>
<td>190</td>
<td>50</td>
</tr>
<tr>
<td>150</td>
<td>45</td>
</tr>
<tr>
<td>100</td>
<td>30</td>
</tr>
<tr>
<td>50</td>
<td>15</td>
</tr>
</tbody>
</table>

Figure 3.34: The influence of the superficial gas velocity on the measured separation performance of vane-type IIIa as a function of the drop sizes.

- 4 m/s
- 6 m/s
- 8 m/s
Figure 3.35: Comparison of experimental and calculated separated (for 100%) droplet sizes for vane-type IIIa (air/water experiments).

+ experimental
—— calculated

Figure 3.36: Comparison of the experimental and calculated separation efficiencies for vane-type IIIa (air/water experiments, d_{50} = 50 micron).

☐ experimental
—— Eq. 3.27, literature model
—— Eq. 3.28, based on "PHOENICS" simulations

The droplet size which is separated for 100% was obtained from the droplet size distribution created during the experiments and was compared with the droplets size calculated with the Eqs. 3.25 and 3.28 (see Table 3.6). The calculated droplet sizes (with Eq. 3.28) are in reasonable agreement with the experimental data. The droplet sizes obtained with Equation 3.25 are all considerably larger than the droplet sizes.
experimentally and numerically, via Eq. 3.28, obtained. A change in the blade distance from 2 to 1.5 cm (denoted during the experiments as vane-type IIIb) results in an increase in separation efficiency. The opposite effect (decreasing separation efficiency) can be noticed when gas density and gas viscosity are increased from 1.2 to 6.5 kg/m$^3$ and from 1.56 $10^{-5}$ to 1.82 $10^{-4}$ kg/ms respectively.

<table>
<thead>
<tr>
<th>Gas velocity (m/s)</th>
<th>Blade distance (m)</th>
<th>Gas viscosity (kg/ms)</th>
<th>Gas density (kg/m$^3$)</th>
<th>$d_{p100}$ (exp., micron)</th>
<th>$d_{p100}$ (Eq. 3.28, micron)</th>
<th>$d_{p100}$ (Eq. 3.25, micron)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>2.0 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>1.2</td>
<td>12</td>
<td>16</td>
<td>26</td>
</tr>
<tr>
<td>7</td>
<td>1.5 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>1.2</td>
<td>8</td>
<td>12</td>
<td>22</td>
</tr>
<tr>
<td>3</td>
<td>1.5 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>3.0</td>
<td>20</td>
<td>18</td>
<td>34</td>
</tr>
<tr>
<td>3</td>
<td>1.5 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>4.7</td>
<td>22</td>
<td>19</td>
<td>34</td>
</tr>
<tr>
<td>3</td>
<td>1.5 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>6.5</td>
<td>24</td>
<td>19</td>
<td>34</td>
</tr>
<tr>
<td>2.5</td>
<td>1.5 $10^{-2}$</td>
<td>1.82 $10^{-5}$</td>
<td>9.8</td>
<td>28</td>
<td>22</td>
<td>38</td>
</tr>
<tr>
<td>2.5</td>
<td>1.5 $10^{-2}$</td>
<td>1.56 $10^{-5}$</td>
<td>9.6</td>
<td>22</td>
<td>20</td>
<td>35</td>
</tr>
</tbody>
</table>

Table 3.6: Comparison of the observed and predicted effects of geometry and gas property variations on the droplet size which can be separated for 100%.

### 3.4.5 Discussion

Trajectories of several droplets of a certain size starting from different locations at the inlet of the vane, simulated under assumption of no turbulent interactions, show that the not separated droplets form one main droplet stream. This phenomenon causes the steep separation curves obtained from these simulations. In reality turbulence will cause an extra radial displacement of the droplets. The effect of the turbulence on the separation efficiency will be that a part of the larger droplets for which the model predicts a 100% separation will not reach the wall. On the other hand smaller droplets get an extra opportunity to reach the wall. The total effect will be a less steep efficiency curve.

The geometry factors influencing the separation efficiency were varied and results show that, in contrast to literature models, bend angle and blade distance have a pronounced effect on separation efficiency. Gas density has a negative effect on the separation efficiency because of the increased droplet Reynolds number (increased friction). In the literature a Reynolds number below 1 (Stokes law) is always assumed, while our simulations show values up to 10 during the simulation of droplet trajectories at higher gas densities.

65
The streamlined vane geometry did not affect the core of the gas flow field. Only recirculation zones, causing a considerable decrease in pressure drop, are eliminated but the separation efficiency was not influenced.

It can be seen from experimental results obtained with the atmospheric rig, that an increasing gas velocity increases the separation efficiency of a vane-type demister. This means that smaller droplets, from the droplet size distribution in the feed, will be separated at higher gas velocities. However, remarkable is the effect that from a certain gas velocity (in this case above 6 m/s, at ambient conditions) no increase in efficiency is achieved. At these conditions other forces, such as interaction of the droplets with turbulent fluctuations of the gas flow, control the motion of these droplets.

The overall separation efficiency obtained experimentally is in reasonable agreement with that predicted by Equation 3.28. The calculated droplet sizes which will be separated for 100% differ from the experimental values but this has not a noticeable influence on the total separation efficiency.

3.4.6 Conclusions

The program for simulating droplet trajectories using the gas flow field calculated with "PHOENICS" enables the visualisation of the droplet flow in vane-type demisters and gives a satisfactory prediction of the separation efficiency.

The gas flow field forces the droplets below a critical size to travel on a narrow sinusoidal path through the vane. All droplets just above the critical size collide with the vane blade. This results in a sharp separation.

Simple literature models (Eq. 3.25) generally predict too low separation efficiencies. The influence of blade distance, bend angle and gas density is in these models underpredicted.

The streamlined geometry did not affect the separation of droplets. So it was possible to develop a vane separator with a lower pressure drop and a good efficiency.

As was found during experiments and simulations (Eq. 3.28) an increase in gas density has a small negative effect on the separation efficiency.

3.5 Liquid flow

This part of this chapter deals with the behaviour of the liquid film created by the deposition of droplets on the vane blades. The liquid film behaviour determines the maximum allowable gas throughput of vane-type demisters. Film flow, flooding and re-entrainment models from literature are discussed and compared with our experimental work.
3.5.1 Liquid flow models

The liquid film formed on the wall by deposition of droplets flows either upward or downward depending on the gas-liquid interaction.

In vertically oriented vane-type demisters, without a drainage system, the gas velocity has to be sufficiently low so that the liquid film always can flow downward. Like in other gas/liquid contacting devices, such as wire-mesh demisters and packed columns, the maximum gas load is limited to a value at which flooding occurs. At flooding conditions the average transport of the liquid is zero and the liquid accumulates in the separator.

With an internal drainage system the liquid film can flow upward until it reaches the collecting pockets. This means that considerably higher gas velocities can be achieved. At high velocities, however, droplets can be sheared off from the liquid film causing re-entrainment of liquid by the gas flow. This phenomenon sets bounds to the maximum gas throughput for this type of separator.

3.5.1.1 Film flow

When we consider a falling liquid film with a counter-current gas flow the following forces have to be taken into account. The gravity, the friction at the wall and the shear force of the gas exerted on the liquid. This is illustrated in Figure 3.37. Neglecting the shear force, Nusselt [33] solved the force balance and with the influence of shear Brauer [34] derived a solution for a smooth film. Feind [35] gives the following dimensionless expression for the liquid velocity inside the smooth film:

Figure 3.37: A force balance on a vertical flowing liquid film.
\[ U_i^* = x^* (1 - \frac{1}{2} x^* - G) \]  

(3.29)

with

\[ G = \frac{\tau_i}{g \rho_i \delta} \]

\[ x^* = \frac{x}{\delta} \]

\[ U_i^* = \frac{U_i}{g \rho_i \delta^2 \mu_i} \]

where \( \tau_i \) is the interfacial shear stress, \( g \) the gravitational acceleration, \( \rho_i \) and \( \mu_i \) the density and viscosity of the liquid respectively, \( x \) the distance from the wall, \( U_i \) the local film velocity, \( \delta \) the film thickness and where the superscript * denotes the dimensionless quantity. The dimensionless shear force \( G \), characterises the velocity profiles in the film as presented in Figure 3.38.

![Figure 3.38](image)

Figure 3.38: Possible velocity profiles in a liquid film.

When \( G = 0 \) the film is free falling and relation 3.29 reduces to Nusselt's expression and gives for a certain liquid flow a film thickness, \( \delta_0 \). Increasing \( G \) up to 1/2 a poiseuille flow appears because the liquid velocity at the liquid-gas interface becomes zero. No net transport of liquid is found when \( G \) is increased to 2/3 and at this point flooding occurs. Further increasing \( G \) results in a co-current gas and liquid flow.
For a turbulent gas flow the interfacial shear stress is expressed as:

\[ \tau_i = f_i \frac{1}{2} \rho_g U_g^2 \]  \hspace{1cm} (3.30)

where \( f_i \) is an interfacial Fanning friction coefficient, \( \rho_g \) the gas density and \( U_g \) the average gas velocity. An often used interfacial friction coefficient is derived by Wallis [36]:

\[ f_i = 0.005 \left( 1 + 300 \frac{\delta}{D_h} \right) \]  \hspace{1cm} (3.31)

where \( D_h \) is the hydraulic free diameter of the channel. In case of vane-type demisters \( D_h \) is equal to \( 2(\delta' - \delta) \), two times the blade distance minus the film thickness.

However, in vane-type demisters the liquid film does not flow along a vertical wall but along an inclined wall as shown in Figure 3.39. This means that the gravity acts only partly in the direction of the liquid film (\( g \cos \alpha \)).

Figure 3.39: Liquid film flow along an inclined wall.

Figure 3.40 shows the effect of gas velocity on the film thickness. As expected, a higher gas velocity results in an increase of film thickness until \( G \) reaches flooding limits and an undefined flow status is reached (no net film flow). At higher gas velocities the film flows upward and film thickness decreases.
Figure 3.40: Development of the film thickness with an increasing gas velocity parallel to the film (air/water, Re\(_g\) = 5.1, \(\alpha = 30^\circ\), s = 2 cm).

---

Eq. 3.29, smooth film, Feind [35]

---

Eq. 3.32, wavy film, Feind [35]

3.5.1.2 Flooding

Flooding conditions set the upper operation limit for conventional countercurrent two-phase contactors, for instance separation columns [37] or cooling systems and in this case vertically oriented vane-type demisters. Figure 3.38 illustrates that flooding in a pipe occurs at a shear force value (G) of 2/3 for a smooth film.

Normally, at the flooding point, the liquid film surface is wavy which means a fluctuating liquid film thickness and a varying roughness of the interface. Based on the investigations of flooding conditions for several tube geometries and fluid properties, Feind [35] developed the following empirical correlation for the onset of flooding when the liquid film is wavy:

\[
Re_{sf} = 240.5 \left( \frac{\rho_g}{\rho_l} \right)^{\frac{2}{3}} \left( \frac{\mu_l}{\mu_g} \right)^{\frac{1}{3}} Re_l^{\frac{1}{3}} \left( 0.0391 \left( \frac{D_h}{\delta_0} \right)^{\frac{3}{4}} - 1 \right)
\]

where the indices \(l\), \(g\) denote the liquid and gas phase, and \(f\) the flooding conditions, \(Re\) the Reynolds number, \(\mu\) the viscosity, \(\rho\) the density, \(D_h\) the hydraulic diameter of the channel, \(\delta_0\) the Nusselt film thickness (relation 3.29, \(G = 0\), a free falling film). As shown in Figure 3.40, flooding occurs for a wavy flow at a gas velocity lower than for a smooth film. This means that waves cause a higher gas/liquid interaction.
A general approach to correlate flooding data of counter-current gas/liquid contactors was introduced by Sherwood [39]. The method is based on dimension analysis. Alen'kin [38] made use of this approach to predict flooding conditions for gas/liquid separators. According to Sherwood, flooding conditions may be correlated with two dimensionless parameters. First the "flow parameter", \( \phi \), representing the ratio of liquid and gas momenta:

\[
\phi = \left( \frac{M_l}{M_g} \right)^{0.5} \frac{\rho_g}{\rho_l}
\]  

(3.33)

\( M_l \) and \( M_g \) are the mass flow rates of liquid and gas respectively. The second parameter, \( \Pi \), describes the influence of the gas velocity, geometry and viscosity and is defined as:

\[
\Pi = \lambda \left( \frac{F_p}{g} \right)^{0.5} \left( \frac{p_l}{p_g} \right)^{0.05}
\]  

(3.34)

where \( F_p \) is a geometrical factor and \( \lambda \) the well known capacity or load factor:

\[
\lambda = U_{g,cr} \left( \frac{p_g}{p_l - p_g} \right)^{0.5}
\]  

(3.35)

with \( U_{g,cr} \) the critical superficial gas velocity, marking the onset of flooding.

To describe flooding in vane-type demisters Alen'kin [38] modified relation 3.34 to:

\[
\Pi = \lambda \left( \frac{1}{g F_{ca}^2 D_h} \right)^{0.5} \left( \frac{p_l}{p_g} \right)^{0.08}
\]  

(3.36)

with \( D_h \) the hydraulic diameter and \( F_{ca} \) the free cross-sectional area of the vane. This relation appears to be incorrect regarding the dimensions.

The exponent of the liquid and gas viscosity ratio seems to be an adjusting parameter. Sherwood uses 0.05, Alen'kin 0.08 for vane-type demisters. Bürkholz [40] uses for wire mesh demisters an exponent of 0.10.

The influence of surface tension on the flooding velocity is reported by Fair [41] who modified relation 3.34 into:

\[
\Pi = \lambda \left( \frac{F_p}{g} \right)^{0.5} \left( \frac{p_l}{p_g} \right)^{0.05} \sigma^{-0.2}
\]  

(3.37)

where \( \sigma \) is the liquid surface tension.
3.5.1.3 Re-entrainment

When a liquid film flows along a wall co- or counter-currently to a gas phase, interface interactions between gas and liquid can lead to entrainment of a part of the liquid. At a certain gas velocity waves appear on the liquid surface. From these waves droplets can be sheared off and taken with the gas phase to the exit of the separator. A considerable effort has been made in the past to model this phenomenon.

Kelvin and Helmholtz [42] described the stability of a liquid surface with a gas flow flowing over it. At a certain gas velocity small waves appear on the surface and they can grow when this velocity is increased above a certain limit. The gas velocity causing the transition between stable and unstable waves can be described as:

\[
U_{g,cr} = \sqrt{\frac{2\left(\rho_l \sigma g\right)^{0.5}}{\rho_g}}
\]

(3.38)

Entrainment is possible from these unstable waves.

Careful examination by Ishii and Groimes [43] of numerous entrainment data for annular gas/liquid flow in tubes indicated three characteristic entrainment regimes. At low liquid flow Reynolds numbers (Re < 40) entrainment is caused by the wave undercut mechanism when the gas velocities are sufficiently high. The inception of entrainment can be correlated by a Weber number based on film thickness. The following criterion for the onset of re-entrainment was derived:

\[
\frac{p_l U_{g,cr}}{\sigma} \sqrt{\frac{\rho_g}{\rho_l}} \geq 0.75 Re_l^{-0.5}
\]

(3.39)

The liquid film Reynolds number is defined by:

\[
Re_l = \frac{\rho_l U_l \delta}{\mu_l} = \frac{\Gamma \rho_l}{\mu_l}
\]

(3.40)

in which \(\Gamma\) the volumetric flow rate per unit wetted perimeter. The here described Reynolds number is four times lower than the one used by Ishii [43]. The absolute limit on the onset of entrainment, below which no entrainment is possible irrespective of the gas velocity, is set by a minimum liquid Reynolds number. This minimum Reynolds number is given by:

\[
Re_{l,\text{min}} = 38.7 \left(\frac{\rho_l}{\mu_g}\right)^{\frac{1}{4}} \left(\frac{\rho_g}{\mu_l}\right)^{\frac{1}{2}}
\]

(3.41)

which means for an air/water system a minimum Reynolds number of 14.7.
Figure 3.41: The forces on a droplet in a vertical gas flow.

Two conditions have to be fulfilled for the decrease of efficiency by re-entrainment. First the droplets have to be sheared off from a wave crest. After that the droplets must be carried by the surrounding gas flow. The onset of re-entrainment is related to the transition from annular flow to mist-annular flow in two-phase flow in pipelines. The forces on an entrained droplet in an upward gas flow are presented in Figure 3.41 and the minimum gas velocity to suspend a droplet is determined from the force balance between gravity and drag force [44]:

\[
U_{g,cr} = \frac{2}{\sqrt{3}} \left( \frac{g (\rho_l - \rho_g) d}{\rho_g C_d} \right)^{0.5}
\]  
(3.42)

For these droplets the value of the drag coefficient (\(C_d\)) is assumed to be 0.44. The diameter of a stable droplet in a gas flow is given by a corresponding Weber number:

\[
We_{cr} = \frac{\rho_g U_{g,cr} d_{max}}{\sigma}
\]
(3.43)

For the critical Weber number different values have been reported. Hinze [45] found for a falling droplet a Weber number of 22. Taitel [44] uses for a non-moving droplet exposed to a gas stream a number of 12.

Substituting expression 3.43 into 3.42 and using a critical Weber number of 12 [44] the relation for the critical gas velocity is as follows:

\[
U_{g,cr} = 2.46 \left( \frac{\sigma g (\rho_l - \rho_g)}{\rho_g^2} \right)^{0.25}
\]
(3.44)

We shall call this the Taitel relation and was used to describe the onset of re-entrainment in vane-type demisters [46]. Kataoka [47] and Ishii [48] investigated the droplet diameters created at re-entrainment conditions. Based on a roll-wave shaped wave and numerous experimental data they came to the following relation for the Weber number:
\[ \text{We}_{cr,mv} = 0.028 \left( \frac{\rho_l}{\rho_g} \right)^{1/6} \left[ \frac{\rho_g}{\rho_l} \right]^{2/3} \left( \frac{\mu_g}{\mu_l} \right)^{1/3} \]  \hspace{1cm} (3.45) \]

Neglecting the influence of the liquid Reynolds number they reduced Equation 3.45 to:

\[ \text{We}_{cr,mv} = 0.0099 \left( \frac{\rho_g}{\rho_l} \right)^{2/3} \left( \frac{\mu_g}{\mu_l} \right)^{2/3} \]  \hspace{1cm} (3.46)

with

\[ \text{We}_{cr,mv} = \frac{\rho_g U_{cr}^2 d_{mv}}{\sigma} \]  \hspace{1cm} (3.47)

in which \( d_{mv} \) denotes the volume mean diameter and can be described as:

\[ d_{mv} = \left( \frac{\sum n_i d_i^3}{\sum n_i} \right)^{1/3} \]  \hspace{1cm} (3.48)

where \( n_i \) means the number of droplets and \( d_i \) the diameter of a droplet in a droplet size distribution. The gas Reynolds number can be written as:

\[ Re_{g,cr} = \frac{\rho_g U_{cr} D_h}{\mu_g} \]  \hspace{1cm} (3.49)

Using expression 3.42 in combination with Eqs. 3.45 - 3.49 enables the calculation of a critical gas velocity at which entrained droplets are carried up by the gas flow.

### 3.5.1.4 Evaluation of film, flooding and re-entrainment models for vane-type demisters

Flooding, for a smooth film, occurs at a \( G \) of \( 2/3 \) as can be seen in relation 3.29 and Figure 3.38. With the relations 3.30 and 3.31 the gas velocity at which flooding starts for a given film thickness can be calculated. Figure 3.42 shows the relation between gas velocity and film thickness at the onset of flooding for three different vane geometries. A smaller blade distance causes an earlier onset of flooding, e.g. at smaller film thickness. Increasing the blade angle causes a lower flooding gas velocity. It can be seen that the vane geometry shows only marginal effects on the onset of flooding.
Figure 3.42: The influence of geometry on the onset of flooding for a smooth film (G = 2/3).

- s = 2.0 cm, α = 30°
- s = 1.5 cm, α = 30°
- s = 2.0 cm, α = 45°

Figure 3.43: Presentation of different models predicting flooding in vane-type demisters. (type I, air/water conditions).

- --- --- --- Eq. 3.32 (flooding of rough film, Feind)
- --- --- --- Eq. 3.29 (flooding of smooth film, Feind)

In Figure 3.43 the flooding gas velocity is presented for both smooth and rough film flow as reported by Feind [35]. The flooding model for a rough film (Eq. 3.32) predicts a lower maximum allowable gas velocity through the vane demister than the flooding model for a smooth film. Interfacial interactions increase when waves appear on the liquid surface resulting in a lower flooding gas velocity.

In Figure 3.44 the theoretical gas velocity for onset of re-entrainment according to the models described above can be seen. The Kelvin-Helmholz instabilities (Eq. 3.38)
Figure 3.44: Presentation of different models predicting re-entrainment in vane-type demisters. (type I, air/water conditions).

- - - - Eq. 3.38 (film instabilities, Kelvin - Helmholtz)
- - - - Eq. 3.46 (droplet carry-up, Kataoka)
. . . . . Eq. 3.44 (droplet carry-up, Taitel)

start at superficial gas velocities which are considerably lower than the velocities, at which droplets will be entrained into the gas flow, calculated with the other models (Eqs. 3.42 and 3.44). In other words the velocities calculated with this model are too low for the onset of entrainment but indicate the first occurrence of unstable waves. The flooding gas velocities are lower than the velocities at onset of re-entrainment. This means that re-entrainment occurs above the state of flooding. Table 3.7 presents droplet diameters and Weber numbers calculated with the model of Kataoka containing the influence of liquid Reynolds number (Eq. 3.45). Using relation 3.46 a critical gas velocity of 7.5 m/s is calculated which means a Weber number of 4.7 and a droplet diameter of 2.4 mm. It can be seen that these Weber numbers are of the same magnitude as proposed by Taitel (We = 12, [44]) and that the entrained droplet diameter is in the order of 2 - 4 mm. Ishii’s model (Eq. 3.39) describing the onset of re-entrainment by means of the wave undercut mechanism is only valid at liquid Reynolds numbers between 14.7 and 40. At those low Reynolds numbers entrainment is possible at very high superficial gas velocities (between 300 and 150 m/s).

<table>
<thead>
<tr>
<th>(U_{cr} ) (m/s)</th>
<th>(Re_i) (-)</th>
<th>(\text{We}_{cr,mv}) (-)</th>
<th>(d_{mv}) (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.0</td>
<td>1.8</td>
<td>14.8</td>
<td>4.3 (10^{-3})</td>
</tr>
<tr>
<td>9.3</td>
<td>7.1</td>
<td>11.1</td>
<td>3.7 (10^{-3})</td>
</tr>
<tr>
<td>8.6</td>
<td>28.3</td>
<td>8.5</td>
<td>3.3 (10^{-3})</td>
</tr>
</tbody>
</table>

Table 3.7: Characteristic parameters calculated with the model of Kataoka (Figure 3.44).
3.5.2 Experimental results

Vane-type demisters with and without drainage system, have been investigated with respect to the maximum allowable gas throughput and variations in liquid properties and geometry. These experiments have been performed at atmospheric and increased gas density conditions, with air or SF₆ as gas phase respectively and either water, glycol or a soap-water mixture as liquids. A detailed description of the experimental set-up and procedures is given in Chapter 2.

3.5.2.1 Vane without drainage system (type I)

Figure 3.45 presents the maximum allowable gas velocity for vane type I (see Figure 3.4 for the geometry) as a function of liquid properties and liquid load (liquid Reynolds number). Exceeding the critical gas velocity means that the separation performance decreases to zero. A slight influence of liquid load was found for the range used and higher liquid loadings cause an earlier state of flooding. The flooding model of Feind (Eq. 3.32) predicts this effect. Further it can be seen that compared to the water properties a lower surface tension (soap experiments) has a more pronounced effect than an increase in liquid viscosity (glycol experiments).

As shown in Figure 3.46 the maximum allowable gas velocity decreases as gas density increases. As illustrated in this figure the influence of gas density as found in the flooding correlations (ρ⁺⁻⁰·⁵ and ρ⁺⁻⁰·⁶) tend to over predict the maximum allowable gas velocity at the highest measured gas density.

![Graph](image)

**Figure 3.45:** The onset of flooding for various experiments (type I, ambient conditions).

- * water
- + soap-water mixture
- □ glycol mixture (μₙ = 11.0 \(10^3\) kg/ms)
- * glycol mixture (μₙ = 19.0 \(10^3\) kg/ms)
3.5.2.2 Vanes with drainage system (type II and III)

Three modifications of the basic geometry of vane-type III (see Figure 3.32) including variations in bend angle and blade distance were tested (Figure 3.47).

By blanking off a part of the demister entrance (type IIIa) it was possible to study the separation efficiency at superficial gas velocities which were higher than those reported in Chapter 3.4.4. Figure 3.48 shows a decreasing trend in the efficiency above a velocity of 10 m/s. The angle of declination of the efficiency is constant in the whole range of liquid load (droplet size distribution) investigated. The slight decline of the separation efficiency means that still the bulk part of the liquid is separated. Comparing Figures 3.33 and 3.48 we see that the experiments under the same conditions give a different efficiency. The later ones are higher, probably because of entrance effects.

The influence of geometry on the separating performance is presented in the Figures 3.49a and 3.49b. Figure 3.49a shows that the decrease of blade distance from 2 to 1.5 cm (type IIIa and IIIb respectively) results in a higher primary efficiency at relatively low gas velocities and in a sharp decline in efficiency beyond the critical value. This figure shows also that increasing the blade angle from $2\alpha = 60^\circ$ to $2\alpha = 90^\circ$ (type IIIa and IIIc respectively) has a negative effect on both the efficiency and capacity. Figure 3.59b shows that a decrease in blade distance (comparing type II and IIIa respectively) results in a higher primary efficiency. For both types there is no abrupt loss in the efficiency. However, the point of maximum achievable efficiency for type II is found at a lower
Figure 3.47: Four different configurations of vane-type demisters with drainage system.
   a type IIIa
   b type IIIb
   c type IIIc
   d type II

Figure 3.48: Separation performance as a function of the superficial entrance velocity of vane-type IIIa for different liquid loads (air/water experiments).

<table>
<thead>
<tr>
<th>Liquid load (l/hr)</th>
<th>$d_{50}$ (micron)</th>
</tr>
</thead>
<tbody>
<tr>
<td>400</td>
<td>90</td>
</tr>
<tr>
<td>280</td>
<td>70</td>
</tr>
<tr>
<td>190</td>
<td>50</td>
</tr>
<tr>
<td>150</td>
<td>45</td>
</tr>
<tr>
<td>100</td>
<td>30</td>
</tr>
<tr>
<td>50</td>
<td>15</td>
</tr>
</tbody>
</table>
Figure 3.49: Influence of geometry on the separation performance of vane-type demisters with drainage system (air/water, $1.8 \times 10^{-3}$ vol%, $d_{50} = 30$ micron).

- ○ type II
- □ type IIIa
- * type IIIb
- △ type IIIc

superficial gas velocity. It should be noted that the actual velocity in the demister of type II is about twice the superficial velocity.

The influence of liquid properties on the performance of vane-type II is shown in Figure 3.50. This is reflected through differences in droplet size. The liquid with the higher viscosity (glycol) is separated with the highest efficiency and the decline in efficiency is equal to the water experiments. The higher efficiency is due to a larger droplet size. On the other hand a liquid with a low surface tension has an earlier and more pronounced decline in efficiency beyond the point of maximum efficiency.

Vane-type IIIb exhibits the same characteristics as vane-type II with respect to the influence of liquid properties as can be seen in Figure 3.51. It shows that a higher viscosity has no effect on the critical gas velocity but that the declination is more pronounced.

Figure 3.52 illustrates the relation between the critical gas velocity and the gas density for vane-type IIIb and IIIc. These results show that the critical velocity decreases exponential ($p_g^{-0.5}$) as predicted by the re-entrainment models. This also means that the level of efficiency which can be reached drops.
Figure 3.50: Influence of liquid properties on the separation performance of vane-type II (type II, $1.8 \times 10^{-3}$ vol%).

- * glycol mixture ($\mu_l = 13.0 \times 10^{-3}$ kg/ms, $d_{50} = 50$ micron)
- □ water ($d_{50} = 30$ micron)
- △ soap-water mixture ($d_{50} = 25$ micron)

Figure 3.51: Influence of liquid properties on the separation performance of vane-type IIIb (type IIIb, $1.8 \times 10^{-3}$ vol%).

- * glycol mixture ($\mu_l = 13.0 \times 10^{-3}$ kg/ms, $d_{50} = 50$ micron)
- □ water ($d_{50} = 30$ micron)
- △ soap mixture ($d_{50} = 25$ micron)
Figure 3.52: The influence of gas density ($\rho_g^{0.5}$) on the critical gas throughput (air/water experiments, $1.5 \times 10^{-3}$ vol%).

- ■ — ■ — ■ — ■ — type IIIb
- × — — — — — type IIIc

3.5.3 Discussion

In contrast to expectations as described by Førde and Nørstrud [49] who predicted a delay in the formation of waves for liquids with a lower surface tension, experiments showed that a decrease in liquid surface tension promotes earlier flooding. This observation is often neglected in flooding models. However, this phenomenon was reported by Kamei [50]. Incorporating a correction term for the experimentally observed influence of surface tension into the model of Feind (Eq. 3.32) results in an expression suitable to predict the flooding conditions of the vane-type demister (type I). After also adapting the constant at the right hand side of the equation (based on our experimental data) the following relation is found:

$$Re_{gf} = 1.14 \times Re_{gf} \ (Eq. \ 3.32) \times \left(\frac{\sigma_l}{\sigma_w}\right)^{0.25}$$  \hspace{1cm} (3.50)

where $\sigma_w$ is the surface tension of water. Figure 3.53 illustrates the accuracy of the modified Feind model.

Liquid viscosity has only a minor effect on the onset of flooding. The model of Feind (Eq. 3.32) gives an exponent of the liquid viscosity varying between 2.25, when the Nusselt film thickness ($\delta_n$) is large, down to -0.25, for a lower Nusselt film thickness. In Sherwoods expression the exponent of the viscosity varies between 0.05 and 0.1.
Figure 3.53: Comparing experimental data with two flooding models (vane-type I).

- Experiments
- Eq. 3.32, Feind
- Eq. 3.50, Feind modified

a) air/water experiments
b) air/soap-mixture experiments
c) air/glycol experiments
Also a good impression of the flooding behaviour of these vane-type demisters can be found when the results are presented according to the dimensionless expressions introduced by Sherwood (Eqs. 3.33 - 3.35). With exception of the low surface tension data, a master curve arises when the exponent of the viscosity group is 0.075 (Figure 3.54a). This is in the same range as several authors [38,39,40] observed. When the surface tension effects are incorporated in the capacity factor all results spread around one curve (Figure 3.54b). The exponent of the surface tension effect (exponent of -0.25) is somewhat higher than reported by Fair [41] who used an exponent of -0.20. The influence of viscosity has to be decreased to get one master curve and the exponent becomes 0.04.

The exponent of the gas density, which describes the effect of gas density on the onset of flooding, lies around -0.6 for a vane-type demister without drainage system. The model of Feind predicts an exponent of -0.6 and according to the Sherwood correlation an exponent of -0.5 has to be found. Figure 3.47 shows that the different powers found only slightly influence the maximum allowable gas throughput.

The vane-type demisters equipped with a drainage system (type II and III) allow co-current gas/liquid flow which means that such demisters operate well at velocities above the flooding limits of conventional demisters. The thickness of the film is so small that no entrainment is possible. However, as illustrated in Figure 3.55 at the edge of the entrance of the liquid drainage compartment a liquid build-up occurs. In this way a wave crest is created from which large droplets are entrained. This situation corresponds to entrainment from crests of roll waves as observed by Ishii [43]. Assuming that the shear stress between wall and liquid film can be neglected, a re-entrainment criterion can be obtained based on drag and surface tension forces. The following relation now appears:

\[
\frac{\mu_l U_{gr}}{\sigma} \sqrt{\frac{\rho_g}{\rho_l - \rho_g}} \geq C N^p_{\mu}
\]  

(3.51)

where \(N_{\mu}\), the viscosity number:

\[
N_{\mu} = \frac{\mu_l}{\left( \rho_l \sigma \left( \frac{\sigma}{g (\rho_l - \rho_g)} \right) \right)^{0.5}} = \frac{\mu_l}{\left( \rho_l \sigma^3 \right)^{0.25}} = M_0^{-0.25}
\]  

(3.52)

in which \(M_0\) the Morton number. A log-log plot of the left hand side of expression 3.51, a dimensionless gas velocity, versus the viscosity number determines the proportionality (exponent \(p\)) between those two numbers. It gives also a value of \(C\). This is presented in Figure 3.56 and a linear relation (\(p=1\)) between the dimensionless gas velocity and the viscosity number is found. With this result and some rearrangements the following expression for the onset of re-entrainment, occurring at the entrance of the slits, can be written:

84
Figure 3.54: Performance of vane demister type I presented according to:

a Sherwoods model

b adjusted Sherwoods model

- water
- glycol mixture ($\mu_l = 11.0 \times 10^{-3}$ kg/ms)
- glycol mixture ($\mu_l = 19.0 \times 10^{-3}$ kg/ms)
- soap-water mixture
\[ U_{g,cr} \sqrt{\frac{\rho_g}{\rho_l - \rho_g}} \geq \lambda \geq C \left( \frac{\sigma g}{\rho_l} \right)^{0.25} \]  

(3.53)

with \( C \), presented in Table 3.8, a constant which is different for all types of demisters. This equation (3.53) is similar to Equation 3.44 of Taitel and the model describing Kelvin-Helmholz instabilities. This could be expected because of an equal type of force balance used for all re-entrainment criteria (Taitel, Kelvin-Helmholz, Ishii). The constant \( K \) for the Taitel model is 2.47 and for the Kelvin-Helmholz instabilities 1.41. The influence of liquid viscosity disappears. The results of experiments with a glycol mixture are, for type II and III, thus only affected by its liquid density and surface tension and not by its high viscosity (Figures 3.50 and 3.51). An equal relation as 3.53 was used by Monat [21] to predict the maximum allowable superficial gas velocity through vane-type demisters without drainage compartment. Knowing now that viscosity and liquid load influence the capacity of such demisters this relation is erroneously used by Monat [21].

![Figure 3.55: Illustration of a liquid hold-up in front of the entrance of a liquid drainage compartment.](image)

<table>
<thead>
<tr>
<th>Vane-type</th>
<th>( \lambda ) (m/s) Eq. 3.35</th>
<th>C-value (-) Eq. 3.53</th>
</tr>
</thead>
<tbody>
<tr>
<td>II</td>
<td>0.25</td>
<td>1.20</td>
</tr>
<tr>
<td>IIIa</td>
<td>0.36</td>
<td>2.38</td>
</tr>
<tr>
<td>IIIb</td>
<td>0.33</td>
<td>2.22</td>
</tr>
<tr>
<td>IIIc</td>
<td>0.24</td>
<td>1.60</td>
</tr>
</tbody>
</table>

Table 3.8: Characteristic parameters of investigated vane-type demisters based on experiments, \( \lambda \) based on air/water experiments.

86
Figure 3.56: Relation between the dimensionless gas velocity and the viscosity number.

- □ type II
- • type IIIa
- + type IIIb
- * type IIIc

The model of Kataoka takes into account an effect of viscosity on the formation of the droplet, which was not observed during the experiments.

The effect of gas density on the onset of re-entrainment can be described with an exponent of -0.5 as found experimentally and in the models.

3.5.4 Conclusions

The maximum allowable gas throughput for a simple vane-type demister (type I, with a counter-current liquid drainage) is limited by flooding.

Surface tension effects play an important role. Correlations of Feind and Sherwood have been adapted to hold for this observed effect.

Vane-type demisters equipped with an internal drainage system (type II and III) can handle a much higher gas throughput than the conventional ones. The onset of re-entrainment determines the capacity limits of these devices.

The onset of re-entrainment was not affected by the liquid viscosity and liquid load within the range investigated. The grade of decrease in the efficiency after reaching the maximum point is dependend on the bend angle and blade distance.

The approach of Taitel can be used for predicting the point of inception of re-entrainment.

The critical gas velocity for all demisters decreases significantly with an increasing gas density.
List of symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Parameter</td>
<td>(-)</td>
</tr>
<tr>
<td>$C_{1e,2e}$</td>
<td>Constants of k-ε model</td>
<td>(-)</td>
</tr>
<tr>
<td>$C_{p,d}$</td>
<td>Constants of k-ε model</td>
<td>(-)</td>
</tr>
<tr>
<td>Cd</td>
<td>Drag coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>d</td>
<td>Droplet diameter, distance</td>
<td>(m)</td>
</tr>
<tr>
<td>$d_i$</td>
<td>Droplet size class</td>
<td>(m)</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Hydraulic diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>E</td>
<td>Roughness parameter</td>
<td>(-)</td>
</tr>
<tr>
<td>f</td>
<td>Fanning friction coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>$F_a$</td>
<td>Added mass force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>$F_{ca}$</td>
<td>Cross-sectional area</td>
<td>(m²)</td>
</tr>
<tr>
<td>$F_d$</td>
<td>Drag force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>$F_g$</td>
<td>Gravitational force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>$F_i$</td>
<td>Inertial force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>$F_p$</td>
<td>Pressure force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>$F_p$</td>
<td>Geometrical parameter</td>
<td>(1/m)</td>
</tr>
<tr>
<td>g</td>
<td>Gravitational acceleration</td>
<td>(m/s²)</td>
</tr>
<tr>
<td>G</td>
<td>Dimensionless shear force</td>
<td>(-)</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy (per unit of mass)</td>
<td>(m²/s²)</td>
</tr>
<tr>
<td>k</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>K</td>
<td>Parameter</td>
<td>(1/m⁰.⁵)</td>
</tr>
<tr>
<td>l</td>
<td>Blade length</td>
<td>(m)</td>
</tr>
<tr>
<td>m</td>
<td>Remix coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>M</td>
<td>Mass flow</td>
<td>(kg/s)</td>
</tr>
<tr>
<td>Mo</td>
<td>Morton flow</td>
<td>(-)</td>
</tr>
<tr>
<td>n</td>
<td>Number of bends</td>
<td>(-)</td>
</tr>
<tr>
<td>$n_i$</td>
<td>Number of droplets in a droplet size class</td>
<td>(-)</td>
</tr>
<tr>
<td>$N_p$</td>
<td>Viscosity number</td>
<td>(-)</td>
</tr>
<tr>
<td>p</td>
<td>Exponent</td>
<td>(-)</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
<td>(Pa)</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
<td>(m)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>(-)</td>
</tr>
<tr>
<td>s</td>
<td>Blade distance</td>
<td>(m)</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
<td>(s)</td>
</tr>
<tr>
<td>u</td>
<td>Turbulent fluctuations</td>
<td>(m/s)</td>
</tr>
<tr>
<td>U</td>
<td>Velocity</td>
<td>(m/s)</td>
</tr>
<tr>
<td>We</td>
<td>Weber number</td>
<td>(-)</td>
</tr>
<tr>
<td>$X,x$</td>
<td>Location</td>
<td>(m)</td>
</tr>
<tr>
<td>Y</td>
<td>Distance</td>
<td>(m)</td>
</tr>
</tbody>
</table>
\( \alpha \) Bend, inclination angle \((^\circ, \text{rad})\)

\( \Gamma \) Volumetric flow rate per unit wetted perimeter \((\text{m}^3/\text{s})\)

\( \delta_{ij} \) Kronecker delta \((-\))

\( \delta, \delta_o \) Film thickness, Nusselt film thickness \((\text{m})\)

\( \varepsilon \) Dissipation rate of turbulent energy (per unit of mass) \((\text{m}^2/\text{s}^3)\)

\( \eta \) Collection efficiency \((-\))

\( \kappa \) von Karman's constant \((-\))

\( \lambda \) Capacity factor \((\text{m/s})\)

\( \mu \) Dynamic viscosity \((\text{kg/ms})\)

\( \nu \) Kinematic viscosity \((\text{m/s}^2)\)

\( \Pi \) Parameter \((-\))

\( \rho \) Density \((\text{kg/m}^3)\)

\( \sigma \) Surface tension \((\text{kg/s}^2)\)

\( \sigma_{k, \varepsilon} \) Constants of k-\( \varepsilon \) model \((-\))

\( \tau \) Shear stress \((\text{kg/ms}^2)\)

\( \tau \) Time \((\text{s})\)

\( \phi \) Flow parameter \((-\))

Indices:

\( cr \) critical

\( f \) flooding

\( g \) gas

\( i \) interface, number

\( i,j \) direction

\( l \) liquid, laminar

\( \text{max} \) maximum

\( \text{min} \) minimum

\( \text{mv} \) mean volume

\( p \) droplet, particle

\( \text{rel} \) relative

\( t \) turbulent, total

\( w \) wall, water

50 50%, based on volume

100 100% separation

+ dimensionless

* dimensionless
Literature


19 Svenson U.; "A note on the value of the Prandtl/Schmidt number for turbulent kinetic energy as used in the K-ε turbulence model", Report n° 7001, Dept. of water resources engineering, University of Lund, (1977).


CHAPTER 4

AXIAL FLOW CYCLONE

4.1 Introduction

Cyclones are widely used and proven devices for the separation of particles or droplets from a fluid flow which can either be a gas or a liquid. For more than 100 years the reverse flow cyclone is in use and can be found in numerous designs [1-6]. Figure 4.1a shows a schematic representation of this type of cyclone. These cyclones are mainly used in solid-liquid and solid-gas systems but they can also be used for gas-liquid systems [7].

A gas stream laden with droplets enters tangential the cyclone body creating a vortex which causes the droplets to migrate to the wall. The cleaned gas moves inwardly into the central core and leaves through the exit duct (vortex finder). The separated droplets create a film which flows to the bottom of the cyclone and enters a liquid drainage compartment.

A more recent type is the uni-flow, straight-through, once-through or axial flow cyclone. It is mainly applied for dust separation [8-10]. Recently Swanborn [11] investigated this type of cyclone for the separation of droplets from a gas stream. An example of such a cyclone geometry is presented in Figure 4.1b. An inlet device forces the
gas to spin around the axis in a tube and to move longitudinally in one direction. The spinning gas flow forces the droplets to deposit on the wall where a created liquid film moves along with the gas. In most types of axial flow cyclones the liquid is drained through an annulus in between the cyclone wall and gas exit tube (vortex finder). To handle large capacities at acceptable pressure drop several smaller cyclones are used in parallel, assembled in compact packages. Figure 4.1c shows a multi-cyclone package.

This chapter deals with the separation of droplets from a gas stream in an axial flow cyclone. Numerical simulations were performed to predict the gas flow field and droplet motion in this type of cyclone tubes.

For a good performance of an axial flow cyclone besides the separation of droplets also the drainage of the liquid film is important. Experiments have been performed to investigate the liquid behaviour and eventually to improve the geometry. The results are discussed in the second part of this chapter.

4.2 The geometry of an axial flow cyclone

As shown in Figure 4.1b, the axial flow cyclone consists basically of a swirl generation device situated in the inlet section and a separation section with discharges for gas and liquid at the outlet. Many variations in the design of swirlers and drainage systems are possible. The use of a secondary, scavenging gas flow for helping the liquid to flow through the liquid discharge has been described by Swanborn [11] (see Chapter 1).
The axial flow cyclone (both experimentally and numerically) considered in this study has a length of 25 cm and a diameter of 5 cm. Figure 4.2 shows a vertically oriented cyclone tube. To clean large gas streams, several cyclones must be used in parallel.

Figure 4.2: An axial flow cyclone tube.

4.2.1 Swirl inlet devices

To generate a spinning, rotating gas flow different inlet devices have been developed, as illustrated in Figure 4.3. These devices can be divided into two groups: stationary deflected vanes and those with a tangential in flow (d) like the entrance of a reverse flow cyclone [2,10]. The stationary deflected vanes vary from a flat (a), to a more streamlined geometry (b). Also variations are found in the number of vanes, swirl angle and the shape of the body. Swirl device c was developed by Swanborn [11] for the use in an axial flow cyclone for separating droplets.

Several swirl inlet devices have been used during the simulations and experiments. The body diameter and vane blade angle were varied. The vane blade angle varied between 30° and 60° and a body diameter of 1 and 3 cm was used. The swirler with a swirl angle of 30° was equipped with eight vane blades. The other swirlers contained six blades. Figure 4.4 presents the two types used.
Figure 4.3: Geometry of an axial flow cyclone and different swirl inducing devices.

Figure 4.4: Two swirl inducing devices with different body diameter.
a 1 cm diameter
b 3 cm diameter
4.2.2 Liquid drainage systems

All types of axial flow cyclones create a droplet free gas stream near the axis and a liquid film on the cyclone wall. There are several ways to remove liquid and often an annular discharge is used as shown in Figure 4.5a. The liquid flows along the wall to the exit where it enters the discharge compartment and the gas is forced into the vortex finder. Figure 4.5b shows a radial discharge using the film rotation to press the liquid through the slits. Swanborn [11] used vertical slits in the cyclone tube between the swirl device and the exit. This is shown in Figure 4.5c. Figure 4.5d shows horizontal slits placed alternating above each other.

As in dust cyclones [3], in gas-liquid cyclones a secondary gas flow can be used to help the liquid to leave the cyclone (through the discharge). This gas flow (sometimes up to 20% of the main gas flow) has to be cleaned before it can be reinjected into the droplet free gas stream. This can be done either by recycling [3,11] the secondary gas flow, or by cleaning it in another device.

Figure 4.5: Several types of liquid drainage systems.
- a annular, co-axial
- b tangential
- c radial with vertical slits
- d radial with horizontal slits
The advantage of a radial discharge, with vertical or horizontal slits, is a fast and continuous drainage of liquid preventing any liquid hold-up or re-entrainment in the cyclone. Swanborn [11] used the vertical radial discharge type in conjunction with a secondary gas flow. A ring was placed at the end of the cyclone to prevent liquid to reach the exit of the cyclone. There it can re-enter the dry gas.

In our study the horizontally and the vertically oriented slits were investigated, both with and in most cases without secondary gas flow. For the vertically oriented slits two designs of openings (see Figure 4.6) were used. The tangential outlets of a slit forces the liquid film to flow into the liquid drainage compartment. In the case of sharp edges the liquid is forced to make a bend before entering the drainage compartment.

Figure 4.6 : Slit designs.
   a tangential
   b sharp edge

4.3 The gas flow

A rotating gas flow has besides an axial velocity component \( U_z \) also a tangential \( U_\theta \) and a radial \( U_r \) component. They correspond to the directions in the coordinate system presented in Figure 4.7.

First the rotating gas flow will be described after which simulation studies will be discussed and compared with laser doppler velocity measurements.

Figure 4.7 : Cylindrical coordinate system.
4.3.1 Rotating gas flow

Basically two types of tangential velocity profiles can be found in a swirling flow.

A free vortex flow, as shown in Figure 4.8a, is induced when for example a rotating jet enters a large chamber. Centrifugal forces create a radial velocity and the jet will spread over the chamber. The tangential velocity changes with the radial distance according to:

\[ U_\theta \times r = \text{constant} \]  \hspace{1cm} (4.1)

with \( r \) the radial position (\( r = 0 \) in the centre).

A forced vortex has a tangential velocity distribution as in a rotating solid body (often called a solid body rotation) and is presented in Figure 4.8b. There exists no radial velocity and the tangential velocity changes with the radial distance according to:

\[ \frac{U_\theta}{r} = \omega = \text{constant} \]  \hspace{1cm} (4.2)

with \( \omega \) being the angular velocity.

In an axial flow cyclone a combination of a forced vortex at the axis and a free vortex near the wall can be found (Figure 4.8c). This type of flow is often called the rankine vortex. It is shown schematically in Figure 4.8c. In reality the transition from the forced to free vortex is less sharp as shown in Figure 4.8d [12] which may be caused by the precession of the forced vortex [13].

An important parameter to characterise the rotating gas flow is the swirl number, \( S_\alpha \), defined as the ratio of the flux of angular momentum (\( M_\theta \)) and the product of the flux of axial momentum (\( M_z \)) and the radius \( r_o \):

\[ S_\alpha = \frac{M_\theta}{M_z r_o} \]  \hspace{1cm} (4.3)

For the flux of angular and axial momentum the following expressions, can be used [14]:

\[ M_\theta = 2 \pi \rho_s \int_0^r U_\theta U_\theta r^2 dr \]  \hspace{1cm} (4.4)

and

\[ M_z = 2 \pi \rho_s \int_0^r U_z^2 r dr \]  \hspace{1cm} (4.5)
in which $r$, $r_0$ the radial distance and radius of the cyclone body respectively and $\rho_g$ the gas density. The measured swirl numbers [11] for several swirl generation devices are listed in Table 4.1. A higher tangential velocity due to a larger blade angle or a thicker body causes an increase in swirl number.

Figure 4.8: Characteristic vortex forms.
- a free vortex
- b forced vortex
- c Rankine vortex
- d experimental Rankine vortex [12]

<table>
<thead>
<tr>
<th>Swirl inducing device</th>
<th>Swirl number (-)</th>
<th>Swirl angle (degrees)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.3a</td>
<td>1.0</td>
<td>60</td>
</tr>
<tr>
<td>4.3b</td>
<td>1.2</td>
<td>60</td>
</tr>
<tr>
<td>4.3c</td>
<td>1.2</td>
<td>45</td>
</tr>
<tr>
<td>4.4a</td>
<td>0.5</td>
<td>45</td>
</tr>
</tbody>
</table>

Table 4.1: Swirl numbers of several swirl inducing devices [11].
4.3.2 Simulation studies

4.3.2.1 Theoretical background

As described in Chapter 3.3.1 a detailed description of the gas flow in the cyclone can be obtained by solving the Navier-Stokes momentum equations together with the continuity equation. For this purpose the cylindrical coordinate system is used in which the circumferential- (θ), radial- (r) and axial- (z) direction can be defined (Figure 4.7).

In the case of a three-dimensional rotating, stationary, incompressible gas flow the momentum conservation equations are written as follows:

\[ U_z \frac{\partial U_z}{\partial z} + U_r \frac{\partial U_z}{\partial r} = - \frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \left( \frac{\partial^2 U_z}{\partial z^2} + \frac{\partial^2 U_z}{\partial r^2} + \frac{1}{r} \frac{\partial U_z}{\partial r} \right) \]  \hspace{1cm} (4.6)

r-direction

\[ U_z \frac{\partial U_r}{\partial z} + U_r \frac{\partial U_r}{\partial r} - \frac{U_\theta^2}{r} = - \frac{1}{\rho} \frac{\partial P}{\partial r} + \nu \left( \frac{\partial^2 U_r}{\partial z^2} + \frac{\partial^2 U_r}{\partial r^2} + \frac{1}{r} \frac{\partial U_r}{\partial r} - \frac{U_r}{r^2} \right) \]  \hspace{1cm} (4.7)

θ-direction

\[ U_z \frac{\partial U_\theta}{\partial z} + U_r \frac{\partial U_\theta}{\partial r} + \frac{U_r U_\theta}{r} = \nu \left( \frac{\partial^2 U_\theta}{\partial z^2} + \frac{\partial^2 U_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial U_\theta}{\partial r} - \frac{U_\theta}{r^2} \right) \]  \hspace{1cm} (4.8)

where \( U \) is the instantaneous velocity component and the indices θ, r and z, denote the direction, P the instantaneous pressure and \( \nu \) the kinematic viscosity of the gas. All derivatives of θ, i.e. \( \partial/\partial \theta \), are zero because of symmetry in the θ-direction.

The continuity equation, to conserve all mass, can be described as:

\[ \frac{\partial U_z}{\partial z} + \frac{1}{r} \frac{\partial U_r}{\partial r} = 0 \]  \hspace{1cm} (4.9)

Similar as in Chapter 3.3.1.1 the Reynolds decomposition method is used to introduce the turbulent fluctuations occurring at high Reynolds numbers. This results in the following formulations of momentum equations:
z-direction

\[ U_z \frac{\partial U_z}{\partial z} + U_r \frac{\partial U_z}{\partial r} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \left( \frac{\partial^2 U_z}{\partial z^2} + \frac{\partial^2 U_z}{\partial r^2} + \frac{1}{r} \frac{\partial U_z}{\partial r} \right) + \frac{\partial u_z^2}{\partial z} + \frac{1}{r} \frac{\partial ru_z u_r}{\partial r} \]  

(4.10)

r-direction

\[ U_z \frac{\partial U_r}{\partial z} + U_r \frac{\partial U_r}{\partial r} - \frac{U_r^2}{r} = -\frac{1}{\rho} \frac{\partial P}{\partial r} + \nu \left( \frac{\partial^2 U_r}{\partial z^2} + \frac{\partial^2 U_r}{\partial r^2} + \frac{1}{r} \frac{\partial U_r}{\partial r} - \frac{U_r}{r^2} \right) - \frac{\partial u_r u_r}{\partial r} + \frac{1}{r} \frac{\partial ru_r^2}{\partial r} - \frac{u_r^2}{r} \]  

(4.11)

θ-direction

\[ U_r \frac{\partial U_\theta}{\partial z} + U_r \frac{\partial U_\theta}{\partial r} + \frac{U_r U_\theta}{r} = \nu \left( \frac{\partial^2 U_\theta}{\partial z^2} + \frac{\partial^2 U_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial U_\theta}{\partial r} - \frac{U_\theta}{r^2} \right) - \frac{\partial u_\theta u_\theta}{\partial z} + \frac{\partial u_\theta u_r}{\partial r} + \frac{2 u_r u_\theta}{r} \]  

(4.12)

where \( \bar{u} \) denotes the turbulent fluctuations in the three directions.

According to Boussinesque's hypothesis the turbulent fluctuations appearing in the equations 4.10 - 4.12 can be calculated by introducing an apparent turbulent viscosity (\( \nu_t \)) \[15,16\] (Chapter 3.3.1).

These expressions, together with the continuity equation, available in the software package "PHOENICS" \[17,18\], have been used as basis for simulation studies.

The axial symmetry of the cyclone makes it possible to use a two-dimensional coordinate system as presented in Figure 4.9. In the two-dimensional cylindrical partition (sub-divided in a number of elements) the three dimensional flow can be solved because of the absence of gradients in \( \theta \)-direction. This means that no equations in that direction have to be solved.

Because of the elliptic nature of the momentum equations, which means that conditions at a given location are influenced by changes in conditions on either side of that location, boundary conditions must be specified for the domain considered.
Figure 4.9: Calculation volume (two-dimensional) used for numerical modelling.

The inlet boundary condition is used to define the mass flux through the cyclone with or without an initial rotation. The conditions at the outlet are normally of the non-gradient type presenting a fully developed flow. However, a swirling flow never becomes fully developed because of a constant decay of the swirl. To avoid a very long domain the characteristic radial pressure gradient, present in rotating flows, is used as a boundary condition at the exit [19]:

\[ \frac{\partial P}{\partial r} = -\rho \frac{U_0^2}{r} \]  \hspace{1cm} (4.13)

This boundary condition can be derived from Eq. 4.7 by assuming that in axial direction there is a developed flow which means there is no gradient in that direction. Further assumptions are that radial velocities and radial velocity gradients are very small.

Wall friction is applied on the outer wall of the cyclone geometry and can be calculated using a log-law approximation of the velocity distribution close to the wall (Figure 3.7 [20]). This way of solving wall friction avoids calculations with a fine grid near the wall. Nevertheless, wall friction at a curved wall is different from the wall friction at a flat wall. An adjustment for curved walls was recently developed by Kind [21] but has not been incorporated into "PHOENICS" as yet.

At the centre line, an axis of symmetry, also the non-gradient condition can be used as well as a zero tangential velocity, \( U_0 \).

In isotropic turbulent flows, when all fluctuating components are equal in all directions, the turbulent components can be solved by using Boussinesque's hypothesis. A turbulent viscosity (\( \nu_t \)) is then calculated. If for swirling flows the k-\( \varepsilon \) model is applied erroneous results are found as shown in Figure 4.10 [13,22,23]. Duggins [23] used a modified k-\( \varepsilon \) model introducing an anisotropy in the radial direction when calculating swirling flows and gained a much better agreement with the experimentally observed profile of tangential velocity (Figure 4.10).

A different way to handle turbulent flows is to derive the exact differential equations for the transport of the fluctuating components (the Reynolds stresses, \( \bar{u}_i \bar{u}_j \)). Solving all these differential equations increases enormously the computational task. Rodi [24] replaced the transport terms by an algebraic approximation. Besides the algebraic relations for the Reynolds stresses the transport equations for k and \( \varepsilon \) still have to be
Figure 4.10: Tangential velocity distribution for a rotating gas flow [23].

- - - - - - k-ε turbulence model

- - - - - - anisotropic k-ε turbulence model

- - - - - - experimental results

Figure 4.11: Comparison of a simulated axial and tangential velocity distribution, using an ASM model, with measurements [25].

- - - - - - axial velocity, calculated

- - - - - - axial velocity, measured

- - - - - - tangential velocity, calculated

- - - - - - tangential velocity, measured
solved and dissipation of turbulence (ε) is still taken isotropic. These ASM-models (Algebraic Stress Models), used by Ayers [25], Fu [26] and Colenbrander [13] for rotating flows, predict the flow field very well as can be seen in Figure 4.11. For the "PHOENICS" code these models were announced four years ago but are still not available. Other computational fluid dynamics software packages already incorporated the ASM models.

The most simple way to introduce the effect of turbulence is to use the constant turbulent viscosity concept over the complete domain. This in contrast to the k-ε model which was applied successfully for the simulation of the gas flow in vane-type demisters. For swirling flows Scott [15] investigated the turbulent viscosities and found for a swirl angle of 45° a sharp decrease of turbulent viscosity near the wall (in both axial and tangential direction). This is shown in Figure 4.12. Moving away from the wall the tangential turbulent viscosity becomes constant (about 200 times νt) and the axial turbulent viscosity still increases. Algifri [27] compared various models for calculating the turbulent viscosity for swirling flows which are based on experimental results of a decaying turbulent swirling flow. The empirical expression for calculating an average turbulent viscosity has the form:

\[
\frac{ν_t}{ν_l} = a \text{Re}^b
\]  

(4.14)
The coefficients 'a' and 'b' are normally taken as a constant and the values proposed by several authors are presented in Table 4.2. However, based on experimental work Algifri found the following dependence of 'a' and 'b' on the swirl number, $S_n$:

$$a = 7.65 \times 10^{-4} - 2.25 \times 10^{-3} S_n^{0.5} \exp(-2.3 S_n)$$

(4.15)

$$b = 0.89 + 0.75 S_n^{0.5} \exp(-2.4 S_n)$$

(4.16)

These expressions were derived for a swirl number between 0.05 and 0.45. The swirl number of the used 45° swirl inducing device as presented in Figure 4.3c is about 1.2. Using the above described expressions for a gas Reynolds number of 50000 and a swirl number of 1.2 the ratio of turbulent and laminar viscosity is in the range of 15 to 214. Table 4.2 presents the calculated values. Considerable differences were found using these relations, even higher values than for a swirl-free case as described by Hinze [16].

<table>
<thead>
<tr>
<th>Author</th>
<th>a</th>
<th>b</th>
<th>$v_t / v_1$ (-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hinze [16]</td>
<td>$6.96 \times 10^3$</td>
<td>0.875</td>
<td>90</td>
</tr>
<tr>
<td>Kreith et al.</td>
<td>$4.15 \times 10^3$</td>
<td>0.86</td>
<td>46</td>
</tr>
<tr>
<td>Wolf et al.</td>
<td>8.32</td>
<td>0.30</td>
<td>214</td>
</tr>
<tr>
<td>Khalatov</td>
<td>$1.78 \times 10^3$</td>
<td>0.93</td>
<td>42</td>
</tr>
<tr>
<td>Rochino et al.</td>
<td>$2.46 \times 10^3$</td>
<td>1.00</td>
<td>123</td>
</tr>
<tr>
<td>Algifri et al.</td>
<td>Eq. 4.15</td>
<td>Eq. 4.16</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 4.2: Constants used in Eq. 4.14 for calculating the turbulent viscosity [27] (Re = 50000, $S_n = 1.2$).

4.3.2.2 Numerical studies

The axial flow cyclone was simulated using the "PHOENICS" computer code with a two dimensional calculation domain. Figure 4.13 shows the domain used together with the cyclone geometry developed for gas/liquid separation experiments. The cyclone has a diameter of 5 cm and a length of approximately 25 cm. Between the grid cells $z = 30$ and $z = 50$ the geometry of the swirl inducing device was simulated. Swirl inducing devices having a blade angle of 45° and a body diameter of 1 and 3 cm respectively were simulated. At $z = 130$, the exit of the cyclone, a ring is introduced to avoid liquid carry-over (see Chapter 4.5). Simulations were performed with air at ambient pressure and with a constant viscosity model to introduce the effect of turbulence. For the simulation
Figure 4.13:  Calculation domain and tested geometry of the axial flow cyclone.
a calculation domain with slender body
b calculation domain with thick body
c geometry used during experiments

studies initially a ratio of turbulent to laminar viscosity of 200 was used. This choice was based on the experiments of Scott [15].

The effect of the device geometry on the flow of the gas has been simulated in the two dimensional partition. First the geometry of the swirler was constructed by blockage of cells. The rotation of the gas flow was introduced by changing the direction of the gas flow according the angle of the swirler in five steps (total angle of 45°). This means an artificial creation of the tangential velocity component. Because of the creation of this tangential velocity, the axial velocity has to be adjusted by adapting the pressure gradient in the energy balance equation.

The gas flow field in an axial flow cyclone with a thick body device is depicted in Figure 4.14a. The red colour indicates the highest velocities and low velocities are coloured blue. The highest overall velocity (the sum of the axial, radial and tangential velocity) is observed at the end of the blades where the gas flow reaches its maximum tangential velocity. At the ring the gas flow has to flow inward which causes an increase of the overall velocity component.

The body diameter of the swirl inducing device strongly affects the amount of swirl generated as presented in Figure 4.15 for the cross-section corresponding to z = 68. The thick body (3 cm diameter) causes a decreased free area and induces higher axial and tangential velocities. Near the axis a pronounced forced vortex is present which changes for the thick body into a free vortex near the wall. For the thin body (1 cm diameter) a
Figure 4.14: Gas flow field and pressure distribution in an axial flow cyclone.
a  gas flow field
b  pressure distribution
Figure 4.15: The influence of the body diameter on the velocity profile behind the swirl inducing device (swirl angle of 45°, $U_g = 10$ m/s, $z = 68$).

1 cm body diameter
- :: :: tangential velocity
- :: :: axial velocity

3 cm body diameter
- :: :: tangential velocity
- :: :: axial velocity

<table>
<thead>
<tr>
<th>Body diameter (cm)</th>
<th>Superficial gas velocity (m/s)</th>
<th>Gas density (kg/m³)</th>
<th>$v_t / v_t$</th>
<th>$S_n$ (z = 68) (-)</th>
<th>$S_n$ (z = 125) (-)</th>
<th>$\xi$ (-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>1.2</td>
<td>200</td>
<td>0.89</td>
<td>0.52</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>1.2</td>
<td>200</td>
<td>0.85</td>
<td>-</td>
<td>6.6</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>1.2</td>
<td>200</td>
<td>1.09</td>
<td>0.9</td>
<td>6.5</td>
</tr>
<tr>
<td>3</td>
<td>15</td>
<td>1.2</td>
<td>200</td>
<td>1.99</td>
<td>-</td>
<td>7.0</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>1.2</td>
<td>100</td>
<td>1.33</td>
<td>-</td>
<td>7.4</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>1.2</td>
<td>40</td>
<td>1.53</td>
<td>-</td>
<td>8.3</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>25</td>
<td>200</td>
<td>1.77</td>
<td>-</td>
<td>9.9</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>50</td>
<td>200</td>
<td>1.77</td>
<td>-</td>
<td>12.8</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>75</td>
<td>200</td>
<td>1.77</td>
<td>-</td>
<td>14.7</td>
</tr>
</tbody>
</table>

Table 4.3: Swirl numbers for a number of calculated gas flow fields.
more constant tangential velocity profile was calculated. The axial flow profile is also different in both cases. With a thin body a more constant axial velocity profile was generated and for the thick body a strong gradient arises direct behind the blades \((z = 68)\). At the axis a reversed flow appears indicating a recirculation zone. The swirl number \((S_a)\) of the gas flow field simulated are listed in Table 4.3. A thick body (3 cm diameter) gives a higher swirl number due to higher axial and tangential velocities.

The simulated profiles of axial and tangential velocities behind the swirler \((z = 68)\) and near the outlet \((z = 125)\) show that an amount of swirl is lost (see Figures 4.16a and 4.16b). Due to wall friction the tangential velocity has decreased about 30\% for the thin body swirler and about 20\% for the thick body geometry. In both cases the axial flow velocity profile changes into a plug flow. Table 4.3 shows that the swirl numbers near the outlet are 40\% and 20\% smaller for the thin and thick body swirl inducing device respectively.

A stronger velocity gradient is created when the entrance velocity is increased. A more pronounced recirculation zone at the centre of the cyclone is found (Figure 4.17a). The maximum of the axial velocity profile moves from the wall towards the centre at higher superficial gas velocities. The tangential velocities are presented in Figure 4.17b and increase due to the higher axial velocities. A steeper forced vortex can be found at the centre of the cyclone. The Rankine vortex shape of the tangential velocity becomes more evident at higher velocities. Also the maximum of the tangential velocity shifts inwardly.

Figure 4.16: Development of the gas flow inside the cyclone (swirl angle of 45°, \(U_z = 10\) m/s).

- **a** 1 cm body diameter
- **b** 3 cm body diameter
- **(z = 68)**
  - Tangential velocity
  - Axial velocity
- **(z = 125)**
  - Tangential velocity
  - Axial velocity
Figure 4.17: Influence of the inlet gas velocity on the velocity profiles \((z = 68, 3 \text{ cm body diameter, swirl angle of } 45^\circ)\).

\begin{itemize}
  \item [a] axial velocity
  \item [b] tangential velocity
  \begin{itemize}
    \item \(5 \text{ m/s}\)
    \item \(10 \text{ m/s}\)
    \item \(15 \text{ m/s}\)
  \end{itemize}
\end{itemize}

As can be seen from Table 4.3 at higher superficial gas velocities (10 and 15 m/s) considerably higher values of the swirl number are reached.

For most of the simulations a turbulent to laminar viscosity ratio of 200 was used to simulate the effects of turbulence. However, when changing this ratio to 100 only a small influence on the velocity profiles is found as illustrated in Figure 4.18. At lower turbulent to laminar viscosity ratios the maxima of the axial and tangential velocity move towards the cyclone centre. The resulting increase of swirl number from 1.09 to 1.31 is mainly caused by a decrease of axial momentum (Table 4.3).

Simulations of increased gas density conditions, up to 75 kg/m\(^3\), and with equal inlet gas velocities show a drastic change in both the axial and tangential velocity profiles (Figure 4.19a and 4.19b). The swirl becomes stronger and the shape of the tangential velocity profile changes into a rankine vortex profile. The maximum of the tangential velocity moves to the axis of the cyclone. The same was found when the gas velocity was increased. The axial velocity profile changes in the same way as the tangential velocity profile. The maximum moves inwardly which was also seen in Figure 4.17a. The changes in tangential and axial velocity profiles effect only marginally the axial and tangential momentum because these changes occur at the centre of the cyclone. So at high densities the swirl number is practically constant.

Figure 4.20 shows the streamlines from the beginning of the domain to the outlet. Downstream of the body of the swirl inducing device and behind the ring recirculation zones are found.
Figure 4.18: Gas velocity profiles of calculations using a different turbulent viscosity, $\nu_t$, ($z = 68$, 3 cm body diameter, swirl angle of $45^\circ$, $U_g = 10$ m/s).

$\nu_t / \nu_i$
- - - - - - - 100
- - - - - - - 200

Figure 4.19: Influence of gas density on the calculated velocity profiles ($z = 68$, 3 cm body diameter, swirl angle of $45^\circ$, $U_g = 10$ m/s).

a Axial velocity
b Tangential velocity
- - - - - - - 1.2 kg/m³
- - - - - - - 25 kg/m³
- - - - - - - 50 kg/m³
- - - - - - - 75 kg/m³

112
Figure 4.20: Streamlines in an axial flow cyclone with a thick swirler body (3 cm diameter, swirl angle of 45°, $U_s = 10$ m/s).

During the simulations also the local pressures were calculated. A pressure field is visualised in Figure 4.14b. The pressure drop ($\Delta p$) between the inlet and the exit of the cyclone can be expressed in the following, commonly used form:

$$\Delta p = \xi 0.5 \rho_s U_s^2$$  \hspace{1cm} (4.17)

in which $U_s$ the superficial gas velocity and $\xi$ a flow resistance coefficient. For the calculated values of the flow resistance coefficient ($\xi$) see Table 4.3. At higher gas densities this coefficient increases considerably. The same is found when changing the turbulent to laminar viscosity ratio from 200 to 100. The changed tangential velocity profile (see Figure 4.18) causes a different flow resistance coefficient. A practical description of the pressure drop is found when $\xi$ is related to the Reynolds number of the gas. The following relation for a turbulent to laminar viscosity ratio of 200 then holds:

$$\xi = 1.2 \text{Re}_g^{0.17}$$  \hspace{1cm} (4.18)

in which $\text{Re}_g$ the Reynolds number of the gas which can be written as:

$$\text{Re}_g = \frac{\rho_s U_s 2 r_o}{\mu_g}$$  \hspace{1cm} (4.19)

### 4.3.3 Experimental verification

The simulations of the axial flow cyclone with a swirl inducing device with a thick central body (3 cm diameter, swirl angle of 45°, see Figure 4.4) were compared with laser doppler velocity measurements (see Chapter 2). Also pressure drop over the cyclone was measured and compared with the predicted values.

#### 4.3.3.1 Laser doppler measurements

The measurements were performed in a cyclone of the same geometry (3 cm diameter, swirl angle of 45°) but operated with water instead of gas. (see also Chapter 2). Figure 4.21 shows data obtained from L.D.A. measurements and the related simulations of
Figure 4.21: Comparison of measured and simulated liquid velocities in an axial flow cyclone (3 cm body diameter, swirl angle of 45°, \( U_i = 0.32 \) m/s).

- a: axial velocity (\( z = 68 \))
- b: axial velocity (\( z = 110 \))
- c: tangential velocity (\( z = 68 \))
- d: tangential velocity (\( z = 110 \))

- Simulation, \( v_i / v_i = 200 \)
- Simulation, \( v_i / v_i = 40 \)
- Measured

The axial and tangential velocities 1 cm and 12 cm downstream the swirl inducing device. Radial velocities could not be measured. It can be seen in Figure 4.21a that 1 cm above the body of the swirler and near to the centre a very sharp peak of the axial velocity is measured. The axial velocities are much lower near the wall. At the centre a small area with negative axial velocities is found indicating a recirculation zone. More downstream of the swirl inducing device (Figure 4.21b) the peak disappears but the velocities near the centre remain low. For the tangential velocities a similar impression is obtained (Figures 4.21c and 4.21d). Near the axis, just above the swirl inducing device, a maximum
tangential velocity is found which falls abruptly to zero at the centre line. Toward the wall the tangential velocity decreases slowly. At the end of the measuring section the tangential velocity profile smoothens out except near the axis of the flow. As can be seen in Figure 4.21 the velocity profiles are not symmetric. This may be caused by a not proper position of the swirl device (a small angle difference between the centre of the cyclone and the swirl device can give such deviations).

The measured velocities are compared with simulations within an equal geometry and under equal flow conditions (water). In the previous simulations a turbulent to laminar viscosity ratio of 200 was used. These simulation results, however, do not follow the measured velocity values. The simulated axial velocity shows a wider recirculation zone and the velocities in the centre zone are much lower. These simulations, depicted in the Figures 4.21a and 4.21b, also show that the highest axial velocities appear closer near the wall. Decreasing the turbulent to laminar viscosity ratio (in this case 40) causes a shift of the velocity maximum to the centre but not enough to fit with the experimentally observed one. The simulated tangential velocities (for $\nu_t/\nu_l = 200$) are much lower than measured. With a lower turbulent to laminar viscosity ratio (40) the discrepancy between predicted and measured tangential velocities becomes smaller. Further downstream of the cyclone (Figure 4.21d) the tangential velocities corresponding to the turbulent to laminar viscosity ratio of 40 approach the measured values, except in the neighbourhood of the centre line.

### 4.3.3.2 Pressure drop experiments

With the experimentally obtained pressure drop data (at ambient conditions) an average value of the flow resistance coefficient ($\xi$) of 8.0 was calculated. For the simulations with a turbulent to laminar viscosity ratio of 40, as expected to give the best velocity profiles, the following relation for $\xi$ was found:

$$\xi = 0.35 \text{ Re}^{0.3}$$

(4.20)

This relation is different to the one for the high turbulent to laminar viscosity ratio. In Eq. 4.18 the influence of the Reynolds number is much lower. The experimentally found coefficient, however, is approximately equal to those obtained with simulations at ambient conditions. The two flow resistance relations, Eqs. 4.18 and 4.20, are depicted in Figure 4.22 together with the experimental data.

### 4.3.4 Discussion

The velocity profiles measured with the L.D.A. technique differ considerably from those obtained by simulations. Especially near the axis the deviation is large but it only affects a very small part of the gas throughput.
Figure 4.22: Comparison of simulated and measured flow resistance coefficient.

- experimental
- Eq. 4.18, numerical \( \nu / \nu_i = 200 \)
- Eq. 4.20, numerical \( \nu / \nu_i = 40 \)

The discrepancy between measured and calculated velocities is mainly due to a not proper simulation. The incorporation of the swirler into the calculation domain and the use of an inaccurate turbulence model provide a faulty prediction of the gas velocities.

The best agreement between simulations and the L.D.A. measurements was found using a turbulent to laminar viscosity ratio of 40. This value is lower than normally found for pipe flow. An even lower value causes a better development of the flow in radial direction but worsens the prediction of the flow in axial and tangential direction.

In contrast to the velocities near the centre, the velocities near the wall are calculated reasonably good. The major droplet concentration can be found near the wall and therefore these gas flow fields can be used for droplet trajectory simulations.

The gas velocity profile influences not only the swirl number but also the flow resistance coefficient (\( \xi \)). This means that the tangential velocity profile determines the pressure drop. The numerically obtained and measured flow resistance coefficients are almost equal at ambient conditions. At higher densities, where the tangential velocity profile changes, the flow resistance coefficient probably changes to higher values.

4.3.5 Conclusions

With the present turbulent models the software package "PHOENICS" is not able to simulate accurately the gas flow field in an axial flow cyclone.
An isotropic turbulence model with a constant turbulent to laminar viscosity ratio of 40 gives a more or less satisfactory prediction of the gas flow field in the separation zone near the wall of the cyclone.

An increase in body diameter of the swirl inducing device (from 1 to 3 cm) and an increase in gas throughput both strongly influence the generated swirl.

The gas flow field determines the pressure drop of the cyclone. The calculated flow resistance coefficient was not constant but could be described as a function of the Reynolds number.

### 4.4 Motion of droplets

The simulation of droplet motion in an axial flow cyclone can be performed on the same way as for vane-type separators, see Chapter 3.4.1. The force balances are now set up in a cylindrical domain. In the following chapters the equations of motion in a cylindrical coordinate system are presented and are then used for calculating droplet trajectories and separation efficiency.

#### 4.4.1 Modelling of droplet motion

The force balance between the drag, inertial and pressure forces working on a droplet in a cylindrical coordinate system [25] gives the next three equations for the droplet motion (a more detailed discussion about droplet motion can be found in Chapter 3.4.1):

**z-direction**

\[
\frac{\partial U_{z,\rho}}{\partial t} = \frac{1}{\tau} \left( U_{z,\rho} - U_{z,\rho} \right)
\]  

(4.21)

**r-direction**

\[
\frac{\partial U_{r,\rho}}{\partial t} = \frac{(\rho_p - \rho_s)}{\rho_p} \frac{U_{0,\rho}^2}{r} + \frac{1}{\tau} \left( U_{r,\rho} - U_{r,\rho} \right)
\]  

(4.22)

**θ-direction**

\[
\frac{\partial U_{\theta,\rho}}{\partial t} = -\frac{2}{r} U_{r,\rho} U_{\theta,\rho} + \frac{1}{\tau} \left( U_{\theta,\rho} - U_{\theta,\rho} \right)
\]  

(4.23)
in which
\[
\tau = \frac{4}{3} \frac{(\rho_p - \rho_g) \, d_p}{\rho_g \, C_d \, U_{rel}} \tag{4.24}
\]

and for the relative velocity, \(U_{rel}\), between droplet and gas:
\[
U_{rel}^2 = (U_{\theta,g} - U_{\theta,p})^2 + (U_{r,g} - U_{r,p})^2 + (U_{z,g} - U_{z,p})^2 \tag{4.25}
\]

The drag coefficient in relation 4.24 can be described by the correlation of Kaskas [28]:
\[
C_d = \frac{24}{Re_p} + \frac{4}{Re_p^{0.5}} + 0.4 \tag{4.26}
\]

where
\[
Re_p = \frac{\rho_g \, U_{rel} \, d_p}{\mu_g} \tag{4.27}
\]

and \(U_{r,z,\theta,g,p}\) the velocities of the gas and droplet in either \(r, \theta\) and \(z\) direction and \(d_p\) the droplet diameter. In the Equations 4.22 and 4.23 a centrifugal acceleration term and a coriolis acceleration term are found which are corrections of the droplets velocity change in a cylindrical domain. The calculated droplet velocities are used to calculate its new position with the following equations:
\[
\frac{\partial x_t}{\partial t} = U_{z,p} , \quad \frac{\partial x_r}{\partial t} = U_{r,p} , \quad \frac{\partial x_\theta}{\partial t} = U_{\theta,p} \tag{4.28}
\]
in which \(x_{z,r,\theta}\) the droplet location in the cyclone domain.

Durst [29] and Crowe [30] linearised the velocity equations. They so prevent very long solution times when the above described differential equations are solved with a Runge-Kutta method. The assumption is made that the droplet velocity is constant during a small time step. The following expressions then appear:
\[
U_{z,p,j+1} = U_{z,p,j} + (U_{z,g} - U_{z,p,j})(1 - \exp{-\frac{\Delta t}{\tau}}) \tag{4.29}
\]
\[
U_{r,p,j+1} = U_{r,p,j} + (U_{r,g} - U_{r,p,j} + \frac{(\rho_p - \rho_g) \, U_{r,p,j}^2}{\rho_p}) (1 - \exp{-\frac{\Delta t}{\tau}}) \tag{4.30}
\]
\[
U_{\theta,p,j+1} = U_{\theta,p,j} + (U_{\theta,g} - U_{\theta,p,j} + \frac{U_{\theta,p,j} \, U_{r,p,j}}{r}) (1 - \exp{-\frac{\Delta t}{\tau}}) \tag{4.31}
\]
where the indices $j$ and $j+1$ refer to the previous and new velocity respectively and $\Delta t$ the time step used.

When the gas flow field is known, for instance from simulations with "PHOENICS", it is possible to calculate droplet trajectories. In this case turbulent interactions and the change of the gas flow field because of momentum interaction between the droplets and the gas flow, are neglected. The critical droplet trajectory ($r_{crt}$) determines the region in the cyclone in which all droplets are separated. The separation efficiency can then be calculated as follows:

$$\eta = \left(1 - \frac{r_{crt}^2}{r_o^2}\right) \times 100$$  \hspace{1cm} (4.32)

in which $r_o$ the radius of the cyclone.

### 4.4.2 Predicting droplet separation by a simplified model

To avoid the difficulties of rigorous simulation of droplet trajectories as described above, a simplified model was proposed [8]. It was used to calculate dust separation in axial flow cyclones. This model uses the radial displacement of the particles as described by Eq. 4.22. Following assumptions are made to simplify Eq. 4.22. The radial gas velocity ($U_{r,g}$) is zero, the particle moves with the same tangential velocity of the gas and Stokes law is used for calculating the drag coefficient. Further, for the gas a solid body rotation ($\omega$ = constant, Eq. 4.2) is assumed and there is a plug flow in axial direction. With these assumptions Eq. 4.22 is reduced to:

$$\frac{\pi}{6} d_p^3 \rho_p \frac{\partial r}{\partial t^2} = \frac{\pi}{6} d_p^3 (\rho_p - \rho_g) \omega^2 r - 3 \pi \mu_g d_p \frac{\partial r}{\partial t}$$  \hspace{1cm} (4.33)

where $\partial r/\partial t$ is the radial velocity of the droplet, $\partial^2 r/\partial t^2$ the radial acceleration of the droplet, and $\omega$ the angular velocity of the gas rotation. The analytical solution of Equation 4.33 is:

$$\frac{r_i}{r_f} = (1 - B) \exp(n_1 t) + B \exp(n_2 t)$$  \hspace{1cm} (4.34)

where

$$B = \frac{n_1}{(n_1 - n_2)}$$
\[ n_1 = \frac{(-p + (p^2 - 4q)^{0.5})}{2} \quad n_2 = \frac{(-p - (p^2 - 4q)^{0.5})}{2} \]

\[ p = \frac{18 \frac{\mu_s}{\rho_d}}{d_p^2} \quad \quad q = -\omega^2 \]

The angular velocity for a swirl angle of 45° and travelling time, t, through the axial flow cyclone respectively:

\[ \omega = \frac{U_{r_0}}{r_o} \quad \quad t = \frac{L_c}{U_z} \]

in which \( U_z \) the axial velocity, \( U_{r_0} \) the tangential velocity at the cyclone radius \( r_o \), \( L_c \) the cyclone length, \( r_{1_i} \) the initial and local radial position of the droplet respectively.

The droplets enter the cyclone uniformly distributed over the inlet. From the calculations follows for each droplet size a critical initial position, \( r_{1c} \), which divides the entrance in two areas. One area where no droplets of a given size are separated and a second one where all droplets were separated. The separation efficiency of an axial flow cyclone with a body radius, \( r_b \), of the swirler then can be written as follows:

\[ \eta = \left( \frac{r_o^2 - r_{1c}^2}{r_o^2 - r_b^2} \right) \times 100 \quad (4.35) \]

when the \( r_{1c} \) is smaller then the body radius, \( r_b \), the separation efficiency is 100%. In the next paragraph droplet trajectories and separation efficiencies are calculated with the above described equations (Eqs. 4.34 and 4.35).

### 4.4.3 Numerical prediction of droplet separation

Figure 4.23 shows the separation efficiency of an axial flow cyclone as a function of droplet diameter for different gas velocities, as calculated by the simplified model. A higher gas velocity enables a better separation of smaller droplets. The same results also from an increase in the radius of the body of the swirl inlet device (Figure 4.24), because the droplets enter the cyclone closer to the wall. Calculated droplet trajectories are shown in Figure 4.25 for two droplet diameters, 5 and 10 micron respectively.

In the next cases the gas velocity fields calculated by "PHOENICS" were used in conjunction with the rigorous droplet trajectory model to evaluate the separation performance. A turbulent to laminar viscosity ratio of 40 was used. Calculation examples of these flow fields are presented in Chapter 4.3.2.2.
Figure 4.23: Separation performance of an axial flow cyclone with a thick body, calculated by the simplified model (3 cm body diameter, swirl angle of 45°, $\rho_g = 1.2$ kg/m$^3$).

- - - - - - - 5 m/s
- - - - - - - 10 m/s
- - - - - - - 15 m/s

Figure 4.24: Influence of the body diameter on the separation efficiency (simplified model, swirl angle of 45°, $U_g = 10$ m/s, $\rho_g = 1.2$ kg/m$^3$).

- slender body (1 cm)
- - thick body (3 cm)

Figure 4.25: Calculated droplet trajectories with Eq. 4.34 (swirl angle of 45° $U_g = 10$ m/s, $\rho_g = 1.2$ kg/m$^3$).

- - - - - - - $d_p = 5$ micron
- - - - - - - $d_p = 10$ micron

121
In Figure 4.26 the trajectories of droplets with a different size are shown, all entering from the same position. Larger droplets can be separated more easily than smaller ones. Near the ring the small droplets move inwardly due to a strong radial velocity in that area.

The separation efficiency of the axial flow cyclone with the thick body is very good even at low gas velocities (Figure 4.27). Droplets of 8 micron or larger are all separated. By increasing the gas velocity from 5 to 10 m/s the droplets larger than 5 micron can be separated completely and also the separation efficiency for smaller droplets is improved considerably. The calculated separation efficiency is better than the one predicted by the simplified model described in Chapter 4.4.2.

![Droplet Trajectories](image)

**Figure 4.26**: Droplet trajectories in a by "PHOENICS" calculated velocity field (\(U_g = 10\) m/s, \(\rho_g = 1.2 \text{ kg/m}^3\), note the different scales for length and radius of the cyclone!).

As summarized in Table 4.4 for 5 micron droplets, an increase in gas density has an adverse effect on separation efficiency. On basis of the velocity profiles calculated with "PHOENICS" (see Figure 4.19b) a positive effect of gas density could be expected. However, the observed increase in tangential velocity is limited to the core and the decrease of density difference between droplet and gas and an increased gas friction seem to have an overruling negative effect.

Figure 4.28 shows the droplet trajectories for 5 micron droplets travelling under different gas densities starting from a critical radius above which 100% separation occurs. At higher gas densities this critical radius moves to the wall of the cyclone indicating a lower separation efficiency. For the droplet travelling at ambient conditions the critical radius could not be found. After the inward motion of the droplets, behind the swirl inducing device, they travel along a nearly straight line to the cyclone wall.
Figure 4.27: Separation performance by numerical simulation ($\rho_g = 1.2$ kg/m$^3$).

- 5 m/s, "PHOENICS" gas flow field
- 10 m/s, "PHOENICS" gas flow field
- 5 m/s, Eqs. 4.33 - 4.35
- 10 m/s, Eqs. 4.33 - 4.35

<table>
<thead>
<tr>
<th>$\rho_g$ (kg/m$^3$)</th>
<th>Efficiency (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>100</td>
</tr>
<tr>
<td>25</td>
<td>90</td>
</tr>
<tr>
<td>50</td>
<td>77</td>
</tr>
<tr>
<td>75</td>
<td>69</td>
</tr>
</tbody>
</table>

Table 4.4: Calculated separation efficiency of a 5 micron droplet at different gas densities ($U_g = 10$ m/s).

4.4.4 Discussion

It was noticed that the simple model (with an analytical solution) predicts a lower separation efficiency compared to that obtained by numerical solution of the equations for droplet motion. The droplet sizes which will be separated for 100% differ approximately 50%. This means that the simple model gives only a rough approximation of the separation performance. The main reason for this discrepancy is the assumption of a solid body flow field in the simple model. This does not agree with the numerical predicted gas flow field (see Chapter 4.3.2.2) which shows an almost constant tangential velocity in the separation zone of the cyclone. An additional error is caused by the assumption that the droplet Reynolds number is below 1, which is certainly not the case for higher gas densities.

The diameter of the body of the swirl inducing device strongly affects the trajectories of the droplets. The positive effect of delivering the droplets to a position more
Figure 4.28: Calculated droplet trajectories at different gas densities ($U_g = 10 \text{ m/s}, d_p = 5$ micron, note the different scales for the length and radius of the cyclone!).

- $\rho_g = 25 \text{ kg/m}^3$
- $\rho_g = 50 \text{ kg/m}^3$
- $\rho_g = 75 \text{ kg/m}^3$

near to the wall of the cyclone is partly undone by the radial gas velocities behind the body. This effect is stronger at increased densities and increased gas velocities. Also the ring at the outlet of the cyclone negatively influences the motion of the droplets. The small droplets which are under the ring and very near to separation are forced to the centre of the cyclone by the inward gas flow. A longer cyclone will therefore give a better separation of very small droplets.

Droplets below 3 micron can hardly be separated which marks the separation limits of the axial flow cyclone.

4.4.5 Conclusions

According to the droplet trajectory calculations utilizing gas flow field calculated with "PHOENICS" the axial flow cyclone enables at atmospheric conditions complete separation of droplets as small as 5 microns.

At increased gas densities (natural gas separation conditions) the simulation of the gas flow field shows the creation of a strong swirl. The change in velocity profile has only a small effect on the separation efficiency of the cyclone. All droplets larger than 7 micron will be separated when the gas has a density up to 75 kg/m$^3$ and a superficial velocity of 10 m/s.

The simple model produces values of the droplet sizes which are approximately 50% higher.
4.5 Liquid flow

For the performance of the axial flow cyclone the behaviour of the liquid film can be critical in a way similar to that described for vane-type demisters. First film and re-entrainment models will be discussed after which experiments are presented. They show the performance of the axial flow cyclone under different geometry and gas/liquid conditions.

4.5.1 Rotating liquid flow

The liquid flow in an axial flow cyclone is slightly different from the flow on a vertical blade of a vane-type separator. In the cyclone the rotation of the gas will influence film flow and re-entrainment.

The gas flow rotates due to the swirl inlet device. This causes an axial and a tangential velocity component both acting on the liquid film. The forces on the liquid film act into two directions as illustrated in Figure 4.29 and forces the film flow to rotate. For the axial film flow direction in the cyclone the dimensionless liquid velocity equation for a smooth film is presented in Eq. 3.29 [31]. In this equation the gas and liquid velocities, interfacial shear and the gravitational force are oriented in axial direction (Eqs. 3.30, 3.31).

![Diagram of forces acting on a liquid film](image)

Figure 4.29: Forces acting on a liquid film.

Perpendicular to the axial shear of the gas a tangential component of the shear stress acts on the liquid film. This will cause a rotation of the film. No other forces act in tangential direction. This results in a Couette flow, a laminar flow between the wall and the rotating gas flow. For the tangential liquid velocity the following relation then holds:
\[ U_{\text{tg}} = \frac{\tau_{\text{tg}} \delta}{\mu_l} \]  
(4.36)

in which \( U_{\text{tg}} \) is the tangential velocity component of the liquid at the interface. In analogy to the axial shear (see Eq. 3.30) the interfacial shear stress in tangential direction can be written as:

\[ \tau_{\text{tg}} = f_i \times \frac{1}{2} \rho_g U_{g\tau g}^2 \]  
(4.37)

where \( U_{g\tau g} \) is the tangential gas velocity and the Fanning friction factor, \( f_i \), equal in both axial and tangential direction, is given in Eq. 3.31.

The two mean liquid velocity components in a vertically oriented axial flow cyclone, calculated by the above described relations, are presented in Figure 4.30. Constant axial and tangential gas velocity are used in this example. A negative axial film velocity, occurring at low superficial gas velocities, indicates a counter-current gas-liquid flow. Increasing the gas velocity causes a thicker liquid film and a lower axial film velocity. At zero axial liquid velocity (an unstable situation which is comparable with the state of flooding in Chapter 3.5.1.2) a maximum film thickness appears causing a maximum tangential liquid velocity (see expression 4.36). The tangential liquid velocity for the upwardly flowing liquid film is higher than the axial one which means that the swirl angle of the liquid is higher than that of the gas (which is 45°).

![Graph](image)

**Figure 4.30:** Calculated liquid velocity of a rotating film (air/water, swirl angle of 45°, \( \text{Re}_l = 70 \), \( \rho_g = 1.2 \text{ kg/m}^3 \)).

---

126
4.5.2 Re-entrainment

Similarly to two-phase channel flow, interfacial interactions in cyclones can lead to re-entrainment i.e. shearing off droplets from created wave crests. The models describing the onset of re-entrainment are described in Chapter 3.5.1.2 for vane-type demisters. The adaptions which have to be made to include the effects of the rotation of the gas and liquid flow are discussed below.

The instability of a liquid surface with a gas flowing over it was described by Kelvin-Helmholz [32]. The gas velocity creating unstable waves can be calculated with Eq. 3.38. For horizontal flow gravity acts perpendicular on the liquid film. In a vertical rotating flow the force acting perpendicular to the film is created by the centrifugal acceleration of the liquid film. The acceleration will affect the onset of unstable waves on the film surface and the gravitational acceleration must be replaced by:

$$g = \frac{U_{i/s}^2}{r_o}$$  \hspace{1cm} (4.38)

in which \(r_o\) denotes the radius of the cyclone. Figure 4.30 shows relatively low tangential liquid velocities resulting in this case in a centrifugal acceleration in the film of about 0.5 - 3.6 m/s². These values are considerably lower than the normal gravitational acceleration and a less stable liquid surface and early Kelvin-Helmholz instabilities appear at equal gas velocities. This is in contrast with the statement of Swanborn [11]. He predicted a stabilizing effect of the rotation of the liquid film on the onset of unstable waves but overestimated the tangential liquid velocities.

The entrainment of droplets from a liquid film, as described by Ishii and Grolmes [33], may decrease the performance of gas-liquid separators when the liquid Reynolds number is above a minimum value (Eq. 3.40 [33]). For air/water conditions this Reynolds number should exceed 14.7. The phenomenon of entrainment from a liquid film was described for vane-type demisters in Chapter 3.3.3.2. In the relations describing the onset of entrainment a gravitational acceleration is present in the viscosity number. The centrifugal acceleration acts perpendicular on the liquid film. This means that the gravitational acceleration has to be replaced by Eq. 4.38.

Figure 4.31 shows the influence of the centrifugal acceleration of the liquid film on the onset of re-entrainment for different liquids, such as glycol, water and butanol. At low acceleration values the possibility of re-entrainment increases considerably and the low surface tension of butanol also induces an early entrainment of droplets. Viscosity has a stabilizing effect.

The model of Taitel [34] and Kataoka [35] (Eqs. 3.42 - 3.48) describes the forces acting on an entrained droplet floating in an upward gas flow. Small droplets are carried to the exit of the separator. This model describes very well the onset of re-entrainment in vane-type demisters.
Figure 4.31: Influence of liquid film velocity, Eq. 4.38, on the onset of re-entrainment according to Ishii (air/water, swirl angle of 45°, $Re_t = 70$, $\rho_l = 1.2$ kg/m³).

- — — — water
- — — — glycol
- — — — butanol

However, the droplets sheared off from a wave crest in a cyclone are exposed to a high swirling gas flow and the centrifugal force will throw them again on the wall. A typical drop size, which can be calculated with Eqs. 3.45 - 3.48, entrained from a liquid film in an axial flow cyclone is about 2 - 4 mm. Droplets of that size will be separated immediately.

### 4.5.3 Experimental investigation

The axial flow cyclone has been investigated under different conditions. Experiments have been carried out in the pressurized test rig, described in Chapter 2, equipped with one cyclone tube. Variations were made in the geometry of the swirl inducing device, swirl angle and liquid drainage geometry (described in Chapter 4.2). Various liquids and liquid loads as well as different gases (air or SF₆) at different pressures were used during the experiments. Also the effect of a secondary gas flow was investigated. The physical properties of the used liquids and gases are given in Chapter 2.

As presented in Chapter 4.4.3 the separation efficiency of the axial flow cyclone is high and it appears that droplets above 7 micron will all be separated. With the spray nozzles used it was not possible to create a considerable amount of droplets below 10 micron. This means that all droplets entering the cyclone are separated. This was assumed for all experiments.

The diameter of the body of the swirl inducing device influences considerably the amount of swirl, e.g. the tangential velocity, in the cyclone. The high swirl has a positive effect on the separation of droplets and the drainage of the liquid film. An increased tangential gas velocity increases the tangential velocity of the liquid film and forces more easily the liquid through the slits. Figure 4.32 shows a considerably better performance when the body diameter is changed from 1 to 3 cm. At a high gas throughput an increase in efficiency is found for the slender body as expected for higher tangential velocities.
Figure 4.32: Influence of body diameter on the separation performance of the axial flow cyclone (air/water experiment, blade angle of 45°, \( \rho_s = 1.2 \text{ kg/m}^3 \), 0.8 \( 10^{-3} \) vol%).

- □ 1 cm diameter
- △ 3 cm diameter

Figure 4.33: Influence of the blade angle of the performance of the axial flow cyclone (air/water experiment, 3 cm body, \( \rho_s = 1.2 \text{ kg/m}^3 \), 0.8 \( 10^{-3} \) vol%).

- □ swirl angle of 30°
- △ swirl angle of 45°
- × swirl angle of 60°
Not only the diameter of the body but also the blade angle of the swirl inducing device (relative to the vertical direction) influences the swirl component of the gas flow. Figure 4.33 shows an improved separation performance when using a blade angle of 45° or 60°. An almost twice as good performance (with respect to the not separated liquid) can be achieved by using a 60° swirl inducing device instead of one with a swirl angle of 45°.

The influence of the secondary gas flow is illustrated in Figure 4.34. A constant secondary gas flow of 20 m³/hr causes a lower amount of liquid carried to the exit at a low gas throughput. At increasing gas flow rates, but with a constant secondary gas flow, this improvement becomes less and disappears at 80 m³/hr. The effect of this scavenging gas flow is, for this cyclone configuration, only significant when it is above 20% of the total gas throughput.

![Graph](image)

**Figure 4.34:** Influence of a constant scavenging gas flow, (20 m³/hr), on the amount of liquid carried through the cyclone, (air/water experiment, horizontal slit, \( \rho_g = 1.2 \) kg/m³, 0.8 \( 10^{-3} \) vol%).

- no secondary gas flow
- \( 20 \) m³/hr secondary gas flow

The geometry of the liquid drainage of the cyclone has a considerable influence on the separation performance. Figure 4.35 shows that horizontally oriented slits, as presented in Figure 4.5, cannot drain the liquid effectively. Due to the non circumferential slit geometry a certain amount of liquid passes the slit and can reach the gas outlet of the cyclone. Longitudinal slits with a tangential orientation, shown in Figure 4.6a, are not that effective as expected. At a moderate gas throughput a considerable loss of efficiency can be seen. At a low tangential velocity the liquid will not pass the slits into the liquid drainage compartment. At the inlet of the slit the liquid velocity reduces to zero, because of the absence of tangential shear, causing a local build-up of liquid. The liquid will consequently be carried up to the outlet by the axial velocity component which is still present at that location. Longitudinal slits with a sharp edge, as presented in Figure 4.6b,
Figure 4.35: Performance of the axial flow cyclone equipped with different slit geometries, (air/water experiment, blade angle of 45°, 3 cm body diameter, \( \rho_s = 1.2 \) kg/m³, 0.8 \( 10^{-3} \) vol%).

- △ horizontal slits, Figure 4.5d
- • longitudinal slits with sharp edge, Figure 4.6b
- ◊ longitudinal slits with tangential orientation, Figure 4.6a

work very well. When the liquid is forced beyond the edge it is shielded completely from the gas flow and enters the liquid drainage compartment. In this case there is no axial velocity in the slit so the liquid cannot be carried to the exit. Visual observations, however, show the presence of a small liquid rivulet (about 1 mm thick) in front of the sharp edge slit in which the liquid can be carried up by the axial shear causing a small performance reduction.

Figure 4.36a shows the influence of the liquid load on the separation performance. The separation of droplets is not important here because all droplets are above 10 micron thus only the liquid drainage plays a role. An increasing liquid load of the gas increases the separation performance of the cyclone. The influence of the gas throughput on the not separated liquid is presented in Figure 4.36b. This figure shows that the liquid load has only a minor influence on the amount of liquid carried up to the outlet. Only at very high liquid loads, over 0.1 vol%, the drainage capacity of the cyclone is insufficient and a sharp increase in amount of liquid carried to the outlet is found.

The influence of liquid properties, such as liquid viscosity (experiments with glycol), liquid density and surface tension (experiments with butanol), on the performance of an axial flow cyclone is shown in Figure 4.37. These properties will influence the flow inside the liquid rivulet located upstream of the slit. Liquid viscosity may influence the rivulet thickness and the surface tension the shape of the rivulet.

The effect of the gas density on the performance of the axial flow cyclone, equipped with longitudinal sharp edge slits, can be seen in Figure 4.38a. An increase of gas density causes an efficiency decrease. When plotting the not separated liquid as a function of the gas-load factor, \( \lambda \), (Figure 4.38b) a continuous increase is found except for
Figure 4.36a: Influence of liquid load on the separation efficiency of the axial flow cyclone with sharp edge longitudinal slits (air/water experiment, $\rho_g = 7.2 \text{ kg/m}^3$).

Figure 4.36b: Liquid carried through the cyclone for different liquid loads (air/water experiment, $\rho_g = 7.2 \text{ kg/m}^3$).

- $\square$ 0.15 - 0.05 $10^2 \text{ vol\%}$
- $\times$ 0.95 - 0.60 $10^2 \text{ vol\%}$
- $\triangle$ 4.50 - 1.30 $10^2 \text{ vol\%}$
- $\bullet$ 10.6 - 3.30 $10^2 \text{ vol\%}$
Figure 4.37: Influence of liquid properties on the separation performance of the axial flow cyclone (air experiment, sharp edge longitudinal slit, $\rho_g = 1.2$ kg/m$^3$, $0.8 \times 10^{-3}$ vol%).

- water
- glycol mixture ($\mu_l = 14.2 \times 10^{-3}$ kg/ms)
- butanol

Figure 4.39: The influence of the liquid properties on the not separated liquid.

- water
- glycol mixture ($\mu_l = 14.2 \times 10^{-3}$ kg/ms)
- butanol
Figure 4.38: Effect of the gas density on the separation efficiency of the axial flow cyclone (water experiment, 0.8 $10^{-3}$ vol%).

a) efficiency

b) overhead

- $\rho_g = 1.2$ kg/m$^3$, air
- $\rho_g = 7.5$ kg/m$^3$, air
- $\rho_g = 13$ kg/m$^3$, SF$_6$
- $\rho_g = 23$ kg/m$^3$, SF$_6$

maximum value
the experiments at a gas density of 13 kg/m³ which all gave a lower carry-over.

Figure 4.39 shows the carry-over for all the liquids used up to gas densities of 25 kg/m³. The trends of the curves are the same. The highest liquid carry-over was found for the liquid with the lowest surface tension (butanol).

4.5.4 Discussion

The geometry of the cyclone, i.e. swirl inducing device and slit geometry, strongly influences the separation performance. A stronger swirl improves the separation performance. The slits with the sharp edges ensure a much better performance then horizontal slits and longitudinal slits with tangential orientation. In the case of horizontal slits the liquid passes the slits, because they are not completely circumferential, and a continuous liquid film can flow to the exit. A large amount of carry-over was also found when using the tangential slits. The liquid film enters the slit and faces no more tangential gas velocity and the liquid velocity decreases. In the slit there is an axial gas velocity which carries the liquid, present in the slit, to the exit. With the sharp edge slit this all is not possible. There is no tangential or axial gas velocity in the slit carrying the liquid to the exit.

A remarkable result was the constant liquid carry-over without any relation to the amount of liquid sprayed into the cyclone. Only when this liquid flow was too high the drainage capacity of the used slits was not enough. This indicated that the performance of an axial flow cyclone could not be evaluated by percentages of the liquid entering the cyclone but only by the liquid carry-over. Using percentages would give a wrong picture of the performance of the cyclone when comparing different liquid conditions.

Liquid properties influenced also the amount of carry-over. An increased viscosity or lower surface tension causes a decrease in separation performance.

Experiments also showed that the liquid carried to the exit increases with increasing gas-load factor, which is proportional to an increased shear of the gas flow. A maximum carry-over curve is obtained for each liquids used.

A secondary gas flow, to help the liquid to enter the slit, has a marginal effect on the performance. Only when a large amount (> 20%) of the gas flow is pressed through the slits it helps the liquid to flow through the slits. A secondary gas flow is only useful when it can interact with the liquid film, i.e. when the interfacial friction coefficient is high enough. When the slits have a sharp edge the secondary gas flow has to make a bend to enter the slit. It is uncertain whether this gas flow will push the liquid also beyond the edge. For the tangential slits it is easier to force the liquid through the slit with the secondary gas flow. However, to do this on a controlled way, a high, constant gas flow has to be forced through the slits to create enough shear on the liquid film. Swarborn [11] generated a the secondary gas flow by a venturi or by the low pressure region behind the swirler (Figure 1.8). This means that the secondary gas flow is proportional to the cyclone gas throughput. The secondary gas flow may therefore be too low and another way of
generating this secondary flow has to be used. Apart from using the secondary gas flow, this flow has also to be cleaned before it can re-enter the main gas flow. This can be done by another cleaning device or by a venturi or the low pressure region behind the swirl inducing device [11] (Figure 1.8).

The rotational film flow model described in Chapter 4.5.1 shows that the tangential velocity of the film is higher than the axial film velocity and increases with an increase in film thickness. The maximum tangential liquid velocity occurs around the flooding point. The film velocities are considerably lower than the gas velocities (both tangential and axial).

In horizontal film flow the gravity counteracts the formation of waves. In a vertical rotating film flow the centrifugal acceleration influences the stability of the liquid surface, however, this force is much lower than gravity and therefore an early formation of waves can be expected. From the crests of the waves droplets can be sheared off (they will be in the order of magnitude of 2 to 4 mm, as found for vane-type demisters). These droplets are exposed to a rotating gas flow which will push them directly to the wall. When the formation of a droplet occurs near a slit it is possible that the droplet will jump over the slit and will flow to the next one. In this way a droplet may reach the outlet of the cyclone (this is equal to the behaviour of sheared off droplets in vane-type demisters, Chapter 3.5). However, this phenomenon has not been observed in the cyclone.

Visual observations in an axial flow cyclone with longitudinal slits with a sharp edge show small rivulets (about 1 mm) near the slit inlet as illustrated in Figure 4.40. This means that the bulk of the liquid leaves the cyclone through the slits and the remaining liquid, in form of a small rivulet, flows upstream the slit to the gas exit. The liquid reaching the top ring accumulates and flows over the edge of the ring where it is re-entrained by the gas.

The amount of liquid carried up to the exit by four rivulets was calculated by assuming a semi-cylindrical rivulet geometry with a radius δ, and is illustrated in Figure
4.41 for three rivulet thicknesses. A higher gas velocity results in an increased liquid transport through the rivulet. At low gas velocities the rivulet flow does not exists.

The rivulet thickness calculated from measured carry-over (see Chapter 4.5.3) is shown in Figure 4.42, for glycol experiments at different gas densities. The rivulet thickness decreases at increasing gas velocities and is hardly influenced by the gas density at higher gas velocities.

Figure 4.41: Rivulet throughput for different rivulet thickness (water).
- - - - \( \delta = 0.5 \times 10^{-3} \) m
- - - - \( \delta = 1.0 \times 10^{-3} \) m
- - - - \( \delta = 2.0 \times 10^{-3} \) m

Figure 4.42: Rivulet thickness at different gas densities for glycol experiments (\( \mu_i = 14.2 \times 10^{-3} \) kg/ms).
- - - - 5 kg/m\(^3\)
- - - - 10 kg/m\(^3\)
- - - - 20 kg/m\(^3\)
### 4.5.5 Conclusions

The geometry of both the swirl inducing device and the liquid drainage system has a major effect on the performance of an axial flow cyclone. A swirl inducing device with a blade angle of 45° and a cyclone tube with longitudinal oriented sharp edge slits appears to be a powerful combination.

In contrast to vane-type demisters the separation performance of the axial flow cyclone is practically independent of droplet size. Hence, it seems more appropriate to express it as a maximum liquid carry-over.

The drainage capacity of the sharp edge slits is sufficient up to liquid loads of 0.1 vol%. Below this value the liquid carry-over seems to be independent of the liquid load of the gas.

Surface tension and liquid viscosity contribute in a similar way to an increased liquid carry-over. Gas density influences negatively the performance of the cyclone.

The limiting factor with respect to the capacity of an axial flow cyclone seems to be the formation of rivulets upstream the liquid drainage slits.

### List of symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>b, B</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>Cd</td>
<td>Drag coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>d</td>
<td>Particle, droplet diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>D_h</td>
<td>Hydraulic diameter</td>
<td>(m)</td>
</tr>
<tr>
<td>f</td>
<td>Friction coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>F_d</td>
<td>Drag force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>F_i</td>
<td>Inertial force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>F_p</td>
<td>Pressure force</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>g</td>
<td>Gravitational acceleration</td>
<td>(m/s²)</td>
</tr>
<tr>
<td>G</td>
<td>Dimensionless shear force</td>
<td>(-)</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy</td>
<td>(m²/s²)</td>
</tr>
<tr>
<td>L</td>
<td>Length</td>
<td>(m)</td>
</tr>
<tr>
<td>M_θ</td>
<td>Flux of angular momentum</td>
<td>(kgm²/s²)</td>
</tr>
<tr>
<td>M_x</td>
<td>Flux of axial momentum</td>
<td>(kgm/s²)</td>
</tr>
<tr>
<td>n</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>N_p</td>
<td>Viscosity number</td>
<td>(-)</td>
</tr>
<tr>
<td>p</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
<td>(Pa)</td>
</tr>
<tr>
<td>q</td>
<td>Constant</td>
<td>(-)</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
<td>(m)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>(-)</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------</td>
<td>------------</td>
</tr>
<tr>
<td>$S_n$</td>
<td>Swirl number</td>
<td>(-)</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>(s)</td>
</tr>
<tr>
<td>$u$</td>
<td>Turbulent fluctuations</td>
<td>(m/s)</td>
</tr>
<tr>
<td>$U$</td>
<td>Velocity</td>
<td>(m/s)</td>
</tr>
<tr>
<td>$We$</td>
<td>Weber number</td>
<td>(-)</td>
</tr>
<tr>
<td>$x$</td>
<td>Location</td>
<td>(m)</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Film thickness</td>
<td>(m)</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Dissipation rate of turbulent energy</td>
<td>(m$^2$/s$^3$)</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Collection efficiency</td>
<td>(-)</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>(kg/ms)</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>(m$/s^2$)</td>
</tr>
<tr>
<td>$\xi$</td>
<td>Flow resistance coefficient</td>
<td>(-)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>(kg/m$^3$)</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Surface tension</td>
<td>(kg/s$^2$)</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress</td>
<td>(kg/ms$^2$)</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Time</td>
<td>(s)</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Angular velocity</td>
<td>(1/s)</td>
</tr>
</tbody>
</table>

Indices:

- $ax$: axial
- $b$: body
- $c$: cyclone
- $cr$: critical
- $g$: gas
- $i$: interface, initial
- $j$: number
- $l$: liquid, laminar, local
- $mv$: mean volume
- $o$: outer
- $p$: droplet, particle
- $r$: radial direction
- $rel$: relative
- $t$: turbulent
- $tg$: tangential
- $z$: axial direction
- $\theta$: tangential direction
- *: dimensionless
Literature


CHAPTER 5

THE (PREDICTED) PERFORMANCE OF AN AXIAL FLOW CYCLONE AND VANE-TYPE DEMISTERS

5.1 Introduction

Previous chapters contain results of experimental and numerical evaluations of the performance of gas-liquid separators considered in this study. Numerical results gave a general insight into the phenomena appearing in a droplet separator. The experimental results, however, were obtained under conditions different from those in high pressure natural gas treating. Here, an attempt will be made to extrapolate these results to the conditions under which a natural gas treating plant operates. This means that the gas density is considerably higher than in our experiments.

The studies carried out with "PHOENICS" showed that the gas flow field in a vane-type demister is not affected by gas density. Simulation of droplet trajectories has shown that the separation of droplets is negatively affected by the gas density. The operating range is increased by using a drainage system and the maximum throughput is limited by the occurrence of re-entrainment from a liquid hold-up upstream of this drainage system. This re-entrainment limit can be characterised by a critical gas-load factor. However when this gas-load factor is exceeded, the maximum performance does not
decrease dramatically because a major part of the liquid will still be drained by the slits. Unlike surface tension and liquid density the liquid viscosity does not affect the onset of re-entrainment.

The gas flow field in an axial flow cyclone is influenced by the amount of swirl created by the swirl inducing device. A change of gas density or superficial gas velocity changes the velocity profiles. This, however, has only a minor effect on the separation of droplets which appears to be 100% for droplets above 7 micron. For the axial flow cyclone no operation limits were found with respect to gas load. Only when the liquid load is very high the drainage capacity of the slits is insufficient. The separation performance is not 100%. This is caused by a liquid rivulet flowing along the slit to the cyclone exit where it is again entrained into the gas flow. Liquid and gas properties influence the behaviour of this liquid rivulet but it was impossible to quantify this.

From the point of view of high gas density effects on the performance of gas-liquid separators there is no information available in open literature. Recently Oranje [1] presented some experimental results on the separation efficiency of several types of gas-liquid separators at moderate gas densities (up to gas pressures of 39 bar). The maximum density considered in Oranje's work is about 30 - 50% of that occurring at natural gas production facilities.

In this chapter the design rules for the improved vane-type and axial flow cyclone demisters are evaluated.

5.2 Vane-type demisters

In Figure 5.1 three vane-type demisters are shown which proved to be most suitable for separation of droplets at a high gas throughput.

The parameters, which are most important for the design and the operation of such devices are the maximum gas throughput (capacity), the efficiency of separation of droplets and the pressure drop.

5.2.1 The maximum gas-liquid throughput

The maximum allowable gas throughput can be expressed in form of a maximum gas-load factor, \( \lambda_{\text{max}} \), as was presented in Chapter 3.5.2.2. This gas-load factor can be written as:

\[
U_{g,\text{max}} = \lambda_{\text{max}} \left( \frac{\rho_l - \rho_g}{\rho_g} \right)^{0.5}
\]

(5.1)
Various vane-type gas-liquid separators.

- a separator vessel
- b vane-type II
- c vane-type IIIa
- d vane-type IIIb

here $U_{g,max}$ denotes the maximum superficial gas velocity at the entrance of the separator package and $\rho_l$ and $\rho_g$ are the liquid and gas density, respectively.

The maximum gas-load factor was determined experimentally (Chapter 3.5.2.2) at atmospheric conditions and the characteristic values for each separator are presented in Table 5.1. Vane-type IIIa can handle the highest gas throughput and type II about 30% less. Figure 5.2 illustrates the separation performance of vane-type II, IIIa and IIIb under atmospheric conditions. It shows that in contrast with type IIIb, type II and IIIa achieve a

<table>
<thead>
<tr>
<th>Vane-type</th>
<th>$\lambda_{max}$ (m/s)</th>
<th>$C$ (-)</th>
<th>$\xi$ (-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>II</td>
<td>0.25</td>
<td>1.20</td>
<td>5.5</td>
</tr>
<tr>
<td>IIIa</td>
<td>0.36</td>
<td>2.38</td>
<td>3.5</td>
</tr>
<tr>
<td>IIIb</td>
<td>0.33</td>
<td>2.22</td>
<td>6.1</td>
</tr>
</tbody>
</table>

Table 5.1: Constants of vane-type demisters considered in this study.
Figure 5.2: Separation performance as a function of the gas-load factor (atmospheric conditions, air/water experiments).

- vane-type II
- vane-type IIIa
- vane-type IIIb

high separation performance at a gas throughput above the maximum gas-load factor. A good primary separation efficiency (before reaching the maximum gas-load factor), was found for type II and IIIb, because of their small blade distance.

The gas-load factor, $\lambda$, incorporates the influence of gas density on the maximum allowable gas velocity and appeared to be for these demisters a function of surface tension, liquid density and geometry. The following expression was experimentally derived in Chapter 3.5.3 (Eq. 3.53):

$$
\lambda_{\text{max}} = C \left( \frac{\sigma g}{\rho_l} \right)^{0.25}
$$

(5.2)

where $\sigma$ is the surface tension, $g$ the gravitational acceleration and $\rho_l$ the liquid density. The influence of liquid viscosity was not observed. The parameter $C$ is presented in Table 5.1 for the different vane-type demisters.

The experimental results of vane-type IIIb are compared with the separation efficiency of the gas-liquid separators tested by Oranje (Figure 5.3) [1]. It can be seen that the vane demisters with a shielded liquid drainage have a larger operating range than the wire-mesh and coalescer separator which have no such drainage system. It also appears that the separation performance of the vane-type demister tested by Oranje (having a $\lambda$-factor of 0.25) is equal to that of vane-type II, used in our experiments. However, our experiments were performed up to a gas-load factor of 0.5.
Figure 5.3: Operation range of several types of demisters [1].

Oranje [1]:
- Gasunie cyclone
- vane demister (equal to type II)
- multi cyclones
- coalescer separator (equal to type I)
- wire mesh
- gravity separator

This work:
- vane demister (type IIIb)
- axial flow cyclone

The liquid load in the gas flow can be expressed in form of the well known flow parameter, \( \phi \):

\[
\phi = \frac{\dot{M}_l}{\dot{M}_g} \left( \frac{\rho_g}{\rho_l} \right)^{0.5} = \frac{U_i}{\lambda} \tag{5.3}
\]

where \( \dot{M}_l \) and \( \dot{M}_g \) the mass flow rates of liquid and gas respectively and \( U_i \) the superficial liquid velocity based on the vane surface. The vane-type demisters were tested up to a flow parameter (based on the liquid entering the separator) of \( 8.0 \times 10^3 \) without loss of efficiency. As can be seen in Eq. 5.3 the liquid flow, which can be drained, remains constant when designing a vane-type separator at its maximum gas-load factor. The influence of liquid load is often observed by demisters without drainage system (see Chapter 3.5.2.1).

### 5.2.2 Droplet separation

A vane-type separator operates best at the maximum gas-load factor (\( \lambda_{\text{max}} \)) which is given either by flooding or excessive re-entrainment. For higher gas pressures this
means that the superficial gas velocity will be correspondingly lower. This decreases the inertial forces working on the droplet and therefore only larger droplets can be separated. The probability for a droplet to impinge on the vane blades for 100% is strongly influenced by its diameter, the geometry of the vane, the gas density and gas velocity. The following expression, derived in Chapter 3.4.2.2 (Eq. 3.28), defines the smallest droplet size for which a 100% separation is achieved:

\[
d_{p,100} = K \left( \frac{\mu_g}{(\rho_p - \rho_g) U_g} \right)^{0.5} \left( \frac{s^2}{\alpha^3} \right)^{0.5}
\]  

(5.4)

where \(d_{p,100}\) is the droplet diameter, \(\alpha\) the bend angle of the blades, \(\mu_g\) the gas viscosity, \(s\) the blade distance, \(\rho_p\) the droplet density and \(K\) an empirical value which has to be derived from Figure 5.4.

![Figure 5.4: K value for Eq. 5.4, using radians.](image)

The influence of gas density on the droplet separation efficiency of the three described demisters, designed at their maximum gas-load, is shown in Figure 5.5. Vane-type II separates at natural gas treating conditions (gas density of about 70 kg/m\(^3\) at a pressure of 100 bar) droplets bigger than 20 micron where the other two demisters only can separate droplets with a diameter above 40 - 50 micron. The good separation efficiency of type II, although it has the lowest gas-load factor, is caused by the small blade distance and the high gas velocity between the vane blades. All demisters show that at a higher gas density the droplet size which can be separated for 100% increases.
Figure 5.5: The influence of gas density on the droplet size which can be separated for 100% at the maximum allowable gas throughput (calculated).

--- vane-type II
--- vane-type IIIa
--- vane-type IIIb

5.2.3 Pressure drop

The pressure drop of a vane-type demister follows from:

$$\Delta p = \xi \frac{1}{2} \rho_s U_g^2 = \xi \frac{1}{2} \rho_l \lambda^2$$  \hspace{1cm} (5.5)

Where $\xi$ denotes the pressure drop coefficient of the demister and $\rho_l$ the liquid density. The values of $\xi$ are given in Table 5.1 and it can be seen that the lowest pressure drop coefficient was measured for vane-type IIIa and that the pressure drop coefficient for type II is slightly lower than for type IIIb.

Designing a vane-type separator at its maximum gas-load factor gives a maximum pressure drop value. For type II the lowest pressure drop ($\Delta p = 172$ Pa) and for type IIIb the highest pressure drop ($\Delta p = 332$ Pa) was found.

5.3 Axial flow cyclone

The schematic representation of the axial flow cyclone, used during the experiments is shown in Figure 5.6a. The dimensions of this cyclone are presented in Figure 4.2. An optimized swirl inducing device with a body diameter of 3 cm and a total
Figure 5.6: Axial flow gas-liquid separator.
- separator vessel
- swirl inducing device
- slit geometry

The bend angle of the blades of 45° (Figure 5.6b) is used together with a v-shaped, sharp edge, longitudinal slit geometry (Figure 5.6c).

For handling large amounts of gas a number of axial flow cyclones has to be used in parallel.

### 5.3.1 The maximum gas-liquid throughput

Figure 5.7 shows that a higher liquid carry-over, by transport of liquid through small rivulets, occurs at an increasing gas-load factor (Chapter 4.5.2). This carry-over depends also on the physical properties of the liquid but was not affected by the amount of liquid entering the cyclone. A correlation describing the dependence of the liquid carry-over on the liquid properties could not be set-up.

When the amount of liquid carry-over is a design criterion a choice has to be made whether to use for example one cyclone at a gas-load factor of 1.6 or to use two tubes at a gas-load factor of 0.8 (at a gas density of 70 kg/m³). In the first case 10 gram
per actual cubic meter of gas remains in the gas and in the second case only 5 gram. This means a twice as good performance when using two cyclone tubes.

The maximum drainage capacity of a single cyclone tube is reached at a liquid flow rate (based on the liquid entering the cyclone) of 50 l/hr. At atmospheric conditions the flow parameter ($\phi$) becomes $2.1 \times 10^2$ at 70 m$^3$/hr gas throughput. For natural gas treating conditions (gas density of 70 kg/m$^3$ at 100 bar) the flow parameter at a gas throughput of 70 actual cubic meter gas decreases to $1.6 \times 10^3$.

![Graph](image)

Figure 5.7: Liquid carry-over in the axial flow cyclone as a function of gas-load factor and liquid properties.

- • water
- △ glycol
- □ butanol

Oranje [1] presented experimental data of a newly developed cyclone separator (the Gasunie cyclone) as a function of the gas-load factor, $\lambda$ (Figure 5.3). In the case of the Gasunie cyclone the gas-load factor is based on the diameter of the vessel (this in contrast with the other demisters tested). A separation efficiency of above 99% is achieved with this separator up to a $\lambda$-factor of 1.0. The experimental data of the axial flow cyclone expressed as a function of the gas-load factor (based on the entrance area of the cyclone) indicates a high performance up to a $\lambda$ of 2.0. When, however, a number of axial flow cyclones are placed in parallel the vessel based $\lambda$-factor decreases to 1 (50% of the vessel area is used for axial flow cyclones) and is equal to the $\lambda$-factor of the Gasunie cyclone. The efficiency of the axial flow cyclone was calculated with the liquid flows entering and leaving the cyclone. This in contrast with the efficiency calculated for the Gasunie cyclone, where all the liquid entering the vessel is used for calculating the efficiency. Pre-separation of droplets, before entering the axial flow cyclone, was not taken into account. A pre-separation of 80 - 90% of the liquid entering the separator vessel (not unrealistic) would increase the efficiency of the axial flow cyclone (at a $\lambda$ of 2.0) from 97% to 99.7%.
5.3.2 Droplet separation

A strong acceleration of the gas in the swirl inducing device causes an initial separation of larger droplets and the droplets remaining in the gas are separated downstream the swirl inducing device. Calculation of the droplet trajectories in the cyclone using "PHOENICS", as presented in Chapter 4.4, indicates a separation efficiency of 100% for droplets of 7 micron at a gas throughput above 35 m³/hr (5 m/s) at atmospheric conditions (Figure 5.8). At increased gas densities (up to 75 kg/m³) droplets of 7 micron are still separated at an actual gas throughput of 70 m³/hr per cyclone tube (Table 5.2).

![Figure 5.8](image)

**Figure 5.8**: Calculated droplet size which can be separated at atmospheric conditions.
- 5 m/s (35 m³/hr per cyclone tube)
- 10 m/s (70 m³/hr per cyclone tube)

<table>
<thead>
<tr>
<th>$\rho_g$ (kg/m³)</th>
<th>Efficiency (%) 5 micron</th>
<th>Efficiency (%) 7 micron</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>25</td>
<td>90</td>
<td>100</td>
</tr>
<tr>
<td>50</td>
<td>77</td>
<td>100</td>
</tr>
<tr>
<td>75</td>
<td>69</td>
<td>100</td>
</tr>
</tbody>
</table>

**Table 5.2**: Calculated separation efficiency of 5 and 7 micron droplets at different gas densities.

5.3.3 Pressure drop

Two pressure drops have to be distinguished over the axial flow cyclone. The first is the pressure drop from inlet to the gas outlet and the second from the gas inlet to the liquid drainage compartment. Pressure drop over the cyclone can be calculated from
Eq. 5.5 using the corresponding pressure drop coefficient which amounts to 8.0 over the full length of the cyclone and to 4.7 over the liquid drainage system. It is important that the pressure drop from the gas inlet to the liquid drainage system is low enough to drain the liquid to the bottom of the vessel by a stand pipe as presented in Figure 5.6. A typical acceptable pressure drop from gas inlet to the drainage system is 20000 Pa. This means a maximum design gas-load factor of 2.9.

5.4 Conclusions

At increased gas densities, the gas throughput of the vane-type demisters is limited to a maximum value of the gas load factor, \( \lambda \), above which excessive re-entrainment occurs. The vane-types investigated can be used up to a gas-load factor of 0.36 for air/water (type IIIa). The other vane configurations have a gas-load factor of 0.25 (type II) and 0.33 (type IIIb). The maximum gas-load factor is influenced by surface tension and liquid density, but no influence of liquid viscosity was found.

Designing the vanes on basis of this \( \lambda \)-value cause a decrease in separation efficiency at increasing gas densities. Droplets below 20 micron cannot be separated at natural gas treating conditions (type II). The other types investigated (type IIIa, IIIb) can only separate droplets above 50 - 40 micron respectively.

Vane-type demisters have, at their maximum gas load factor, a pressure drop of 170 (type II) - 332 Pa (type IIIb). It was found experimentally that up to a flow parameter of 8.0 \( 10^3 \) the vane demister is able to drain the liquid.

Axial flow cyclones achieve a separation efficiency of 100% for droplets above 7 micron at natural gas treating conditions.

The axial flow cyclone can be used up to a very high gas throughput. At a gas-load factor of 2 the cyclone tube showed a good performance. The liquid carry-over increases at a higher gas-load factor. Also liquid properties influence the carry-over.

Depending on the amount of liquid carry-over which is acceptable, a proper gas-load factor must be chosen.

The pressure drop which can be allowed over a multi-axial flow cyclone package can be also a limiting factor for the maximum gas throughput. A gas-flow factor up to 2.9 can be used for a stand pipe of 2 meter.

An cyclone tube can drain 50 l/hr of liquid by the slits before the drainage capacity becomes insufficient. This means that the allowable liquid load decreases at a higher gas-load factor.

List of symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Parameter</td>
<td>(-)</td>
</tr>
<tr>
<td>d</td>
<td>Particle, droplet diameter</td>
<td>(m)</td>
</tr>
</tbody>
</table>
\( g \) Gravitational acceleration \((m/s^2)\)
\( K \) Parameter \((1/m^{0.5})\)
\( \rho \) Pressure \((Pa)\)
\( l \) Blade length \((m)\)
\( M \) Mass flow \((kg/s)\)
\( Re \) Reynolds number \((-)\)
\( s \) Blade distance \((m)\)
\( U \) Velocity \((m/s)\)

\( \alpha \) Bend angle \((^\circ)\)
\( \mu \) Dynamic viscosity \((kg/ms)\)
\( \lambda \) Capacity factor \((m/s)\)
\( \xi \) Pressure drop coefficient \((-)\)
\( \rho \) Density \((kg/m^3)\)
\( \sigma \) Surface tension \((kg/s^2)\)
\( \phi \) Flow parameter \((-)\)

Indices:

\( g \) gas
\( l \) liquid
\( max \) maximum
\( min \) minimum
\( p \) particle, droplet
\( sup \) superficial
100 100% separation

**Literature**

SAMENVATTING

Gas-vloeistof scheiders die zijn gebaseerd op traagheidskrachten worden in vele uitvoeringsvormen en op verscheidene plaatsen in de procesindustrie toegepast. De praktische kennis is vooral verkregen door eigen experimenten en praktijkervaring van de scheiderfabrikanten. Aan hoge druk (offshore) toepassingen worden andere eisen gesteld en bestaande toepassingsregels zijn ontwikkeld.

Twee traagheidsscheiders waren onderzocht, de vane-type scheider en de axiaal cycloon. Het onderzoek was gestart om de kennis omtrent de fenomenen die bij gas-vloeistof scheiding een rol spelen te verbeteren en de werking van deze scheiders bij hoge druk te onderzoeken.

Het software pakket "PHOENICS" was gebruikt om de gas- en druppelstroming, de effecten van geometrie en de fysische eigenschappen van gas en vloeistof op de scheiding van druppels te simuleren. Bij verschillende gasdichtheden waren experimenten uitgevoerd om simulaties te verifiëren en om de fenomenen, die de scheidente werking verslechteren, te bestuderen.

Het gassnelheidsveld in vane-type scheiders was berekend met het "PHOENICS" simulatiepakket door gebruik te maken van een "body fitted" coördinaten-stelsel en het k-ε turbulentie-model. De berekende snelheden komen overeen met pitot-buis metingen, behalve in de buurt van loslatingsgebieden (wervels). De omvang van de berekende wervels is kleiner dan de omvang van de wervels waargenomen tijdens rookexperimenten. Een druppelbaanmodel dat gebruik maakt van het berekende snelheidsveld was gemaakt. Met dit model kon de scheidingsefficiency voorspeld worden. Resultaten laten zien dat de bladafstand en de afbuighoek een grotere invloed hebben op de scheiding dan zou worden verwacht volgens literatuur-modellen. Het negatieve effect van de gasdichtheid was groter dan verwacht. Naar aanleiding van deze simulaties is de vane geometrie aangepast.
Daardoor verdwenen energieverslindende wervels. Dit resulteerde in een lagere drukval (50%) zonder de druppelscheiding te beïnvloeden. Binnen deze verbeterde geometrie is het mogelijk om beschutte vloeistofafvoer compartimenten aan te brengen op plaatsen waar vroeger de wervels zaten. Dit type vloeistofafvoer maakt gasdoorzetten mogelijk die hoger zijn dan gebruikelijk voor conventionele scheiders, waar de vloeistof tegen de gastroom in afgevoerd wordt. Experimenten met een bekende druppelgrootte laten zien dat het scheidingsmodel, gebaseerd op numerieke simulatie, de scheiding goed voorspelt. Tevens zien we dat het niet mogelijk is om druppels kleiner dan 8 micron te scheiden onder atmosferische omstandigheden. Het werkingsgebied van de vanes met een beschutte afvoer wordt beperkt door het ontstaan van re-entrainment. De re-entrainment ontstaat vanuit een vloeistofophoping voor de afvoer compartiment. De gasdoorzet, waarbij re-entrainment optreedt, bepaalt de maximum waarde van de gas-load factor, $\lambda_{\text{max}}$. Deze belangrijke ontwerp parameter wordt negatief beïnvloed door een lagere oppervlaktespanning en/of een hogere vloeistofdichtheid. Vloeistofviscositeit heeft geen invloed op het ontstaan van re-entrainment.

Een vergelijkbare benadering was toegepast voor het ontwerp en de evaluatie van een effectieve axiaal cyclus. De driedimensionale gasstroming was gesimuleerd door het inbouwen van de vorm van het swirl-element in het rekendomein van de "PHOENICS" software. Omdat er geen anisotroop turbulentie-model aanwezig was, is gebruik gemaakt van een constante turbulentie-viscositeit. Laser doppler verificatie-metingen laten zien dat dit simpele turbulentie-model een redelijke voorspelling geeft van het gasstromingsveld in de buurt van de wand. In dit gedeelte van het gasstromingsveld concentreren de druppels zich. Een druppelbaanmodel laat zien dat alle druppels groter dan 7 micron gescheiden kunnen worden binnen het werkgebied van de cyclus. Dit gebeurt ook bij hogere gasdichtheden. Omdat alle druppels groter dan 7 micron gescheiden worden, richtten de experimenten zich op de vloeistofafvoer en de vloeistofdoorslag. Scherpe longitudinale sleuven zijn het best voor een goede vloeistofafvoer. Het bleek echter dat reeds bij lage gasdoorzetten een vloeistofstroom in de vorm van een riviertje naast de sleuf naar de gasuitlaat stroomt, afwaa het terug in de gastroom verspoeiid wordt. De vloeistofhoeveelheid die met dit riviertje omhoog stroomt, is evenredig met de gasdoorzet. Dit is de reden waarom de scheidende werking beschreven wordt door middel van vloeistofdoorslag in plaats van een scheidings efficiency, gebaseerd op de efficiency van druppelscheiding.

Vane-type scheiders moeten ontworpen worden op hun maximale gas-load factor. Dit betekent een lagere snelheid bij een hogere gasdichtheid en een verslechtering van het scheidingsrendement. Bij aardgasverwerkingsdrukken (ongeveer 100 bar) wordt verwacht, dat de onderzochte scheiders geen druppels onder de 20 micron vangen, daarentegen zal de axiaal cyclus alle druppels groter dan 7 micron scheiden. De maximum gasdoorzet wordt bepaald door een kritische drukval, waarbij de vloeistof nog goed afgevoerd wordt naar de bodem van het vat, of wordt bepaald door de kritische vloeistofdoorslag. De flexibiliteit van de cyclus is groter dan die van de vane-type scheider. Het lijkt dus meer toepasselijk om, voor het verwijderen van druppels uit gasstromen bij hogere drukken, de axiaal cyclus te gebruiken in plaats van de vane-type scheiders.