Compact Centrifugal Separator of Dispersed Phases

PROEFSCHRIFT

ter verkrijging van de graad van doctor aan de
Technische Universiteit Eindhoven, op gezag van de
Rector Magnificus, prof.dr.ir C.J. van Duijn, voor een
commissie aangewezen door het College voor
Promoties in het openbaar te verdedigen
op maandag 12 december 2005 om 16.00 uur

doors

Eva Mondt

geboren te Goes
Dit proefschrift is goedgekeurd door de promotor:

prof.dr.ir. J.J.H. Brouwers

Copromotor:
dr.ir. H.P. van Kemenade

Copyright © 2005 by E. Mondt
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form, or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior permission of the author.

Printed by the Eindhoven University Press.

CIP-DATA LIBRARY TECHNISCHE UNIVERSITEIT EINDHOVEN

Mondt, Eva

Proefschrift. – ISBN 90-386-2907-9
NUR 978
Subject headings: two-phase flow / centrifugal separation / creeping film flow
ACKNOWLEDGEMENTS

The following organizations are acknowledged for their contribution:

The Ministry of Economic Affairs (EET) for funding this research.

The company Romico Hold for providing know-how and information on the design and the design principles of the rotational particle separator.

The company CDS Engineering for making available testing facilities and process data.
Contents

Summary 1
Samenvatting 3
Nomenclature 5

1 Introduction 9
1.1 Background 9
1.2 Rotational Particle Separator 11
1.3 Combined expansion and separation 14
1.4 Goal and outline 16

2 Creeping film model 19
2.1 Introduction 19
2.2 Existing models describing liquid films 20
2.3 Current analysis 23
2.4 Vertical configuration 24
2.4.1 Reynolds number in liquid film 35
2.5 Horizontal configuration 36
2.5.1 Horizontal configuration with shear stress at interface 40
2.5.2 Maximum film thickness in vertically versus horizontally orien-
tated filter element 44
2.6 Closure 45

3 Conceptual design 47
3.1 Introduction 47
3.2 Design parameters 47
3.2.1 Condensation 47
3.2.2 Separation performance 57
3.2.3 Angular momentum 63
3.2.4 Pressure loss 67
3.3 Design 70
3.3.1 Design criteria 70
3.3.2 Design process 71
Summary

With longer exploration of offshore gas wells, the amount of liquid, especially water, in the product stream increases. This is mainly due to the fact that water is usually used to keep the well under pressure. The increase in liquid contaminants requires improvement of current separation methods. In commonly used separators gravity is used to separate the dispersed phase. To allow for sufficient settling time, gas velocities in these separators must be low. As a result these devices are voluminous, heavy and expensive. With increasing liquid amount, the capacity of these devices is no longer sufficient. As a result more and/or heavier separation devices are needed and sometimes it even requires heavier and more expensive supporting structures. In some cases, although considerable gas reserves are still present, the exploitation of a well has to be stopped, as current gas treatment techniques are not economically viable. In order to make the exploitation of older wells profitable, the offshore industry is searching for efficient, compact phase separation devices.

The goal of this study is to design and test a compact and efficient device for separating condensed liquid phases from natural gas. For this purpose the Rotational Particle Separator (RPS) principle is used. Due to the high centrifugal force created in the separator and the small radial distance the particles have to travel to reach a collecting wall the separator can be designed compact, while at the same time a high efficiency can be reached.

Before the separator was designed, two issues were resolved. At first, the behavior of the liquid film inside the channels of the filter element was investigated by using an analytical model based on the Nusselt analysis. The model is valid for small liquid loads and with the model the amount of liquid leaving the filter at the in- and outlet can be determined. Secondly, the order of magnitude of the smallest droplets present in the gas phase was determined by considering nucleation, condensation and coagulation of water vapor in methane. For the conditions considered the droplets reach a minimum size of 1 micron, provided that the coagulation time is sufficiently large.

For the design of all main components of the separator general design relations were derived. With these relations the pressure drop, angular speed and separation efficiency of the separator can be predicted as a function of the flow rate. Based on the design relations, the design criteria and the offshore process conditions, a full-scale prototype was built, which is capable to handle the volume flow of one wellhead under high pressure (80 bar) and which separates particles down to 2 micron. The
design involves a so-called naturally driven RPS, which means that the separator is driven by a swirl generated in the flow upstream of the separator. As a consequence no external motor and therefore no shaft, which needs to be sealed, is required.

The performance of the prototype was tested in two test loops. In a low pressure loop, which was built at the Technical University Eindhoven, both the hydrodynamic and separation performance of the prototype were measured. In a high pressure loop, which was available at CDS Engineering, the hydrodynamic performance of the prototype was measured. In this test loop the offshore operating conditions can be simulated. Comparing the hydrodynamic performance of the prototype with the theoretical model shows good agreement. The separation performance of the prototype was determined by injecting dust particles in a size range between 1-10 micron in the low pressure test loop. Subsequently the particle size distribution both up- and downstream of the separator was measured with an impactor. From those measurements the efficiency of the prototype as a function of the separated particle size was calculated. The measurement results were compared with both an analytical model and with previous measurement, which were performed with an externally driven RPS. The current measurement results show good agreement with both the previous results and the analytical predictions.
Samenvatting

Met langere exploitatie van offshore gasbronnen neemt de hoeveelheid vloeistof, voornamelijk water, in de productiestroom toe. Dit wordt hoofdzakelijk veroorzaakt door het feit dat er meestal water gebruikt wordt om de bron onder druk te houden. De toename in vervuiling vraagt verbetering van huidige scheidingsmethoden. In de huidige scheidingsapparatuur wordt gebruik gemaakt van de zwaartekracht om de gedispergeerde fase af te scheiden. Om voldoende bezinktijd te creëren moeten de gassnelheden in deze scheiders laag zijn. Dit leidt tot grote, zware en duur scheiders. Omdat de hoeveelheid vloeistof toeneemt, is de capaciteit van deze apparaten niet langer toereikend. Dit leidt tot meer en/of nog zwaardere scheiders. Bovendien zijn er soms zwaardere en duurdere ondersteuningen voor de platformen nodig. In sommige gevallen moet, alhoewel er nog aanzienlijke reserves in de bron aanwezig zijn, de exploitatie gestopt worden omdat de huidige scheidingsmethoden niet economisch rendabel zijn. Om het exploiteren van oudere bronnen winstgevend te maken is de offshore industrie op zoek naar efficiënte en compacte scheidingsapparatuur.

Het doel van deze studie is het ontwerpen en testen van een compact en efficiënt apparaat voor het scheiden van gecondenseerde componenten uit aardgas. Voor dit doel wordt het Roterend Deeltjes Principe (RDS) gebruikt. Door het creëren van hoge centrifugaalkrachten in de scheider en het feit dat de deeltjes slechts een kleine radiale afstand hoeven te overbruggen voordat ze een scheidingswand bereiken, kan de scheider compact uitgevoerd worden terwijl er tegelijkertijd een hoge efficiëntie bereikt kan worden.

Voor het ontwerpen van de belangrijkste componenten van scheider zijn er eerst twee zaken opgehelderd. Ten eerste is het gedrag van de vloeistoffilm in de kanalen van het filter element bestudeerd. Hiervoor is een analytisch model gebaseerd op de Nusselt analyse gebruikt. Dit model is geldig voor kleine vloeistofbelastingen. Met het model kan de hoeveelheid vloeistof, die het filter aan de in- en uitlaat verlaat, bepaald worden. Ten tweede is de orde van grootte van de kleinste druppels aanwezig in de gasfase bepaald door het beschouwen van condensatie en coagulatie van waterdamp in methaan. Hieruit blijkt dat de druppels een minimale grootte van 1 micron bereiken mits de coagulatie tijd voldoende lang is.

Voor het ontwerpen van de belangrijkste componenten van scheider zijn algemene ontwerprelaties afgeleid. Met deze relaties kunnen de drukval, toerental en efficiëntie van de scheider voorspeld worden als een functie van het debiet door de scheider. Samen met de ontwerprelaties, de ontwerpeisen en de offshore procescondities is een
full-scale prototype gebouwd. Dit prototype is geschikt om de totale volumestroom van één put te verwerken bij een druk van 80 bar. Onder deze condities worden deeltjes tot 2 micron afgescheiden. Het ontwerp betreft een zogenoemde zelf aangedreven RPS. Dit houdt in dat de scheider wordt aangedreven door een wervel, die stroomopwaarts van de scheider in de stroming wordt gegenereerd. Hierdoor is er geen externe motor en geen as, die afgedicht moet worden, nodig.

De werking van het prototype is getest in twee testopstellingen. In een lagedruk opstelling, die gebouwd is op de Technische Universiteit Eindhoven, zijn zowel het hydrodynamische als het scheidingsgedrag van het prototype gemeten. In een hogedruk opstelling, aanwezig bij CDS Engineering, is het hydrodynamische gedrag van de prototype gemeten. In deze testopstelling kunnen offshore condities gesimuleerd worden. De hydrodynamische metingen zijn in goede overeenstemming met het theoretische model. Het scheidingsgedrag van het prototype is gemeten door stofdeeltjes met een grootte tussen de 1 en 10 micron in de lagedruk testopstelling te injecteren. Vervolgens is met een impactor de deeltjesverdeling zowel stroomop- als stroomafwaarts van de scheider gemeten. Met deze metingen is de efficiëntie van het prototype als een functie van deeltjesgrootte bepaald. De meetresultaten zijn vergeleken met analytische relaties en met eerder metingen, die uitgevoerd zijn met een extern aangedreven RDS. De huidige metingen zijn in goede overeenstemming met zowel de eerdere metingen als de analytische voorspellingen.
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Cross sectional area</td>
<td>m$^2$</td>
</tr>
<tr>
<td>$A$</td>
<td>($= \tau_i/\eta$) Coefficient in creeping film model</td>
<td>s$^{-1}$</td>
</tr>
<tr>
<td>$a$</td>
<td>Acceleration</td>
<td>m s$^{-2}$</td>
</tr>
<tr>
<td>$b$</td>
<td>Coefficient in expression for axial velocity through filter element</td>
<td>s$^{-1}$</td>
</tr>
<tr>
<td>$b$</td>
<td>Base width</td>
<td>m</td>
</tr>
<tr>
<td>$b_{sg}$</td>
<td>Ratio of inner to outer radius of swirl generator</td>
<td></td>
</tr>
<tr>
<td>$C$</td>
<td>Concentration</td>
<td>kg m$^{-3}$</td>
</tr>
<tr>
<td>$C_1$</td>
<td>Integration constant</td>
<td></td>
</tr>
<tr>
<td>$C_c$</td>
<td>Cunningham slip correction factor</td>
<td></td>
</tr>
<tr>
<td>$C_m$</td>
<td>Torque coefficient</td>
<td></td>
</tr>
<tr>
<td>$C_T$</td>
<td>Coefficient in expression for tangential velocity of free vortex</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter</td>
<td>m</td>
</tr>
<tr>
<td>$D$</td>
<td>Diffusion coefficient</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Hydraulic diameter</td>
<td>m</td>
</tr>
<tr>
<td>$d$</td>
<td>Droplet diameter</td>
<td>m</td>
</tr>
<tr>
<td>$d_c$</td>
<td>Channel height</td>
<td>m</td>
</tr>
<tr>
<td>$d_p$</td>
<td>Particle collected with 100% probability for a uniform fluid velocity</td>
<td>m</td>
</tr>
<tr>
<td>$E$</td>
<td>Collection efficiency</td>
<td></td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
<td>N</td>
</tr>
<tr>
<td>$f$</td>
<td>Dimensionless variable</td>
<td></td>
</tr>
<tr>
<td>$f$</td>
<td>Friction factor</td>
<td></td>
</tr>
<tr>
<td>$f_e$</td>
<td>Enhancement factor</td>
<td></td>
</tr>
<tr>
<td>$G_θ$</td>
<td>Angular momentum</td>
<td>N m</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravitational acceleration (9.81)</td>
<td>m s$^{-2}$</td>
</tr>
<tr>
<td>$g_c$</td>
<td>Centrifugal force</td>
<td>m s$^{-2}$</td>
</tr>
<tr>
<td>$h$</td>
<td>Height</td>
<td>m</td>
</tr>
<tr>
<td>$J$</td>
<td>Nucleation rate</td>
<td>m$^{-3}$ s$^{-1}$</td>
</tr>
<tr>
<td>$J$</td>
<td>Moment of inertia</td>
<td>kg m$^2$</td>
</tr>
<tr>
<td>$K$</td>
<td>Kinetic factor</td>
<td>m$^{-3}$ s$^{-1}$</td>
</tr>
<tr>
<td>$K$</td>
<td>Coagulation coefficient</td>
<td>m$^3$ s$^{-1}$</td>
</tr>
<tr>
<td>Kn</td>
<td>Knudsen number</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------------------------------------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>$k_B$</td>
<td>Boltzmann’s constant ($1.38 \times 10^{-23}$)</td>
<td>J K$^{-1}$</td>
</tr>
<tr>
<td>$L$</td>
<td>Length</td>
<td>m</td>
</tr>
<tr>
<td>$L_{ch}$</td>
<td>Channel length</td>
<td>m</td>
</tr>
<tr>
<td>$L_{hy}$</td>
<td>Dynamic entrance length</td>
<td>m</td>
</tr>
<tr>
<td>$L_w$</td>
<td>Penetration depth</td>
<td>m</td>
</tr>
<tr>
<td>$M$</td>
<td>Molar mass</td>
<td>kg mole$^{-1}$</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
<td>-</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
<td>kg</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Rate of mass flow</td>
<td>kg s$^{-1}$</td>
</tr>
<tr>
<td>$N$</td>
<td>Number density</td>
<td>m$^{-3}$</td>
</tr>
<tr>
<td>$n$</td>
<td>Rotational speed</td>
<td>rev min$^{-1}$</td>
</tr>
<tr>
<td>$n_a$</td>
<td>Inverse of molecular area</td>
<td>m$^{-2}$</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$P$</td>
<td>Perimeter</td>
<td>m</td>
</tr>
<tr>
<td>$P_0$</td>
<td>Static pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$P_l$</td>
<td>Langmuir pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius</td>
<td>m</td>
</tr>
<tr>
<td>$R$</td>
<td>Gas constant</td>
<td>J kg$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$R_{univ}$</td>
<td>Universal gas constant (8.314)</td>
<td>J mole$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$r$</td>
<td>Droplet radius / Radial position</td>
<td>m</td>
</tr>
<tr>
<td>$S$</td>
<td>Saturation ratio</td>
<td>-</td>
</tr>
<tr>
<td>$s_g$</td>
<td>Gap size</td>
<td>m</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>K</td>
</tr>
<tr>
<td>$Ta$</td>
<td>Taylor number</td>
<td>-</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$U$</td>
<td>Volume flow per unit width</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
<tr>
<td>$u$</td>
<td>Axial velocity</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$u'^2$</td>
<td>Square of the velocity difference between two points</td>
<td>m$^2$ s$^{-2}$</td>
</tr>
<tr>
<td>$v$</td>
<td>Radial velocity</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$v_0$</td>
<td>Velocity of droplets towards liquid film</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$We$</td>
<td>Weber number</td>
<td>-</td>
</tr>
<tr>
<td>$w$</td>
<td>Tangential velocity</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$x$</td>
<td>Dimensionless particle diameter</td>
<td>-</td>
</tr>
<tr>
<td>$x_0$</td>
<td>Turning point</td>
<td>m</td>
</tr>
<tr>
<td>$x^*$</td>
<td>Ratio of pipe length to hydraulic diameter</td>
<td>-</td>
</tr>
<tr>
<td>$\pi$</td>
<td>Dimensionless distance along plate</td>
<td>-</td>
</tr>
<tr>
<td>$y$</td>
<td>Molar vapor fraction</td>
<td>-</td>
</tr>
<tr>
<td>$Z$</td>
<td>Compressibility factor</td>
<td>-</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Blade angle</td>
<td>-</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Coefficient in creeping film model</td>
<td>m$^2$</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Coefficient in creeping film model</td>
<td>-</td>
</tr>
<tr>
<td>$\Gamma$</td>
<td>Magnitude of circulation of free vortex</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
</tbody>
</table>
\( \gamma \) Coefficient in creeping film model \( \text{m} \)
\( \gamma \) Isentropic exponent \( \text{-} \)
\( \Delta G_n \) Gibbs free energy of droplet formation \( \text{J} \)
\( \Delta P \) Pressure loss \( \text{Pa} \)
\( \delta \) Film thickness \( \text{m} \)
\( \delta_0 \) Maximum film thickness \( \text{m} \)
\( \tilde{\delta} \) Dimensionless film thickness \( \text{-} \)
\( \epsilon \) Energy dissipation per unit mass \( \text{W kg}^{-1} \)
\( \epsilon \) Reduction of the effective cross sectional area due to finite wall thickness of the channels \( \text{-} \)
\( \epsilon_1 \) Dimensionless variable \( \text{-} \)
\( \epsilon_2 \) Dimensionless variable \( \text{-} \)
\( \epsilon_3 \) Dimensionless variable \( \text{-} \)
\( \eta \) Dynamic viscosity \( \text{Pa s} \)
\( \eta \) Dimensionless variable \( \text{-} \)
\( \eta_0 \) Dimensionless turning point \( \text{-} \)
\( \nu \) Kinematic viscosity \( \text{m}^2 \text{s}^{-1} \)
\( \lambda \) Kolmogorov length scale \( \text{m} \)
\( \lambda \) Gas mean free path \( \text{m} \)
\( \xi \) Pressure loss factor at entrance channel \( \text{-} \)
\( \phi \) Volume flow \( \text{m}^3 \text{s}^{-1} \)
\( \rho \) Density \( \text{kg m}^{-3} \)
\( \sigma \) Surface tension \( \text{N m}^{-1} \)
\( \sigma_0 \) Surface tension of pure substance \( \text{N m}^{-1} \)
\( \tau \) Shear stress \( \text{Pa} \)
\( \Omega \) Angular velocity \( \text{rad s}^{-1} \)

**Superscript**
- \( \text{eq} \) Equilibrium
- \( \text{-} \) Second time derivative
- \( \text{-} \) Mean
- \( \text{*} \) Sonic conditions

**Subscript**
- \( a \) Average
- \( ax \) Axial
- \( bearing \) Bearing
- \( c \) Critical
- \( c \) Centrifugal
- \( cor \) Corrected
- \( d \) Droplet
- \( d \) Duct
- \( ds \) Downstream
- \( f \) Fluid
- \( fe \) Filter element
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$g$</td>
<td>Gas</td>
</tr>
<tr>
<td>$gap$</td>
<td>Gap between filter element and housing</td>
</tr>
<tr>
<td>$horz$</td>
<td>Horizontal</td>
</tr>
<tr>
<td>$i$</td>
<td>Inner</td>
</tr>
<tr>
<td>$i$</td>
<td>Interface / Interfacial</td>
</tr>
<tr>
<td>$imp$</td>
<td>Impactor</td>
</tr>
<tr>
<td>$l$</td>
<td>Liquid</td>
</tr>
<tr>
<td>$lam$</td>
<td>Laminar</td>
</tr>
<tr>
<td>$main$</td>
<td>Main stream</td>
</tr>
<tr>
<td>$max$</td>
<td>Maximal</td>
</tr>
<tr>
<td>$mean$</td>
<td>Mean</td>
</tr>
<tr>
<td>$o$</td>
<td>Outer</td>
</tr>
<tr>
<td>$p$</td>
<td>Particle</td>
</tr>
<tr>
<td>$pipe$</td>
<td>Pipe</td>
</tr>
<tr>
<td>$pre$</td>
<td>Pre-separator</td>
</tr>
<tr>
<td>$post$</td>
<td>Post-separator</td>
</tr>
<tr>
<td>$s$</td>
<td>Saturated</td>
</tr>
<tr>
<td>$sample$</td>
<td>Sample tube</td>
</tr>
<tr>
<td>$sg$</td>
<td>Swirl generator</td>
</tr>
<tr>
<td>$shaft$</td>
<td>Shaft</td>
</tr>
<tr>
<td>$turb$</td>
<td>Turbulent</td>
</tr>
<tr>
<td>$us$</td>
<td>Upstream</td>
</tr>
<tr>
<td>$v$</td>
<td>Vapor</td>
</tr>
<tr>
<td>$vert$</td>
<td>Vertical</td>
</tr>
<tr>
<td>$w$</td>
<td>Wall</td>
</tr>
<tr>
<td>$0$</td>
<td>Initial</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>Angular</td>
</tr>
</tbody>
</table>

**Abbreviations**

- ACFM: Actual Cubic Feet per Minute
- BCNT: Binary Classical Nucleation Theory
- CMAG: Condensate Monodisperse Aerosol Generator
- CNT: Classical Nucleation Theory
- DEHS: Di-2-Ethyl Hexyl Sebacate
- ICCT: Internally Consistent Classical Theory
- RPS: Rotational Particle Separator
- SF6: Sulfur Hexafluoride

**Coordinates**

- $(x,y,z)$: Cartesian
- $(x,r,\theta)$: Cylindrical
Chapter 1

Introduction

1.1 Background

Natural gas is a vital component of the world’s supply of energy. It is found in porous rock formations or cavities deep beneath the earth’s surface. It may occur alone in separate reservoirs, but more commonly it forms a gas cap entrapped between petroleum and an impervious, capping rock layer in a petroleum reservoir. Under high-pressure conditions, it is mixed with or dissolved in crude oil [45].

Natural gas used by consumers is almost entirely composed of methane. However, natural gas extracted from offshore gas deposits, although still composed primarily of methane, is by no means pure. It contains oil, different semi-liquid hydrocarbons, water in the form of reservoir brine and condensate (vapor), sand and other large particle impurities [45]. The removal of these contaminants is the most important process step before the gas can be distributed as sales gas. The solid particles have to be removed from the gas because of erosion problems. Water has to be removed for two reasons. First, in the presence of CO$_2$ and H$_2$S, two other possible components of natural gas, it forms a highly corrosive mixture. Second, water together with hydrocarbon components can form hydrates, flaky solids, which could cause plugging. Finally, the hydrocarbon condensates are removed from the gas before it is delivered to the customers. To prevent erosion, corrosion and plugging of process equipment, water and the solid particles have to be removed in the earliest stages of production. Hydrocarbon condensates can be removed in a later stage as they form less hazards during processing [67].

The major transportation pipelines and in particular the compressors, used to transport the gas to shore, impose restrictions on the amount of solid and liquid contaminants. Offshore platforms are therefore built to remove those contaminants from the gas in a number of steps, depending on the composition and quality of the gas extracted. A schematic overview of the separation steps is given in figure 1.1. Natural gas extracted from a reservoir reaches the surface at pressures up to several hundred bar. Subsequently the gas is transported along a high-pressure pipeline into the first separation step, also called wellhead separation. In this stage separation
takes place in a conventional separator, a knock-out drum. It consists of a closed tank, where gravity serves to separate the heavier solid contaminants and already condensed liquids, like oil and water under high pressure.

As the pressure of the gas is too high for the transportation pipelines, the pressure must be reduced. This is done in a so-called throttling (Joule-Thompson) valve. In a throttling valve the gas is expanded. Due to this expansion not only the pressure, but also the temperature of the gas is reduced. The reduction in temperature causes condensation of the hydrocarbon vapors and water vapor. When the well pressure is too low for cooling the gas by expansion, the gas is mechanically refrigerated. Besides condensation, sometimes condensable vapors are removed by adding solid or liquid desiccants. These desiccants absorb the liquid phases and are subsequently regenerated. The condensed liquids and/or desiccants are removed in the second separation step. This step usually consists of several interdependent treating steps. In these steps the gas is brought to the required specifications for pipeline transportation to onshore facilities. Common separation mechanisms applied are sedimentation techniques (gravity settling), inertial separation (mesh, vane and cyclone type separators) and diffusion (filter cartridge) [67].

After the second separation step the gas, oil and condensed hydrocarbons are transported to an onshore treatment facility. A more detailed overview of the different separation steps and separation devices used for the purification of natural gas is given by Medici [45] or Swanborn [67].

Conventional separation devices for purifying natural gas are mostly based on gravitational settling. With longer exploration of the wells, the amount of liquid, especially water, in the product stream increases. This requires improvement of current separation methods. The main problem is the size of currently used separators. In
gravitationally based separators, gas velocities have to be low to allow for sufficient settling time. As a result these devices are voluminous, heavy and expensive. With increasing contaminant amount, the capacity of these devices is no longer sufficient. This leads to more or larger and heavier separation devices. Besides, it requires heavier and more expensive supporting structures. In some cases, although considerable gas reserves are still present, the exploitation of a well has to be stopped, as the current gas treatment techniques are economically not viable. In order to make the exploitation of older wells profitable, the offshore industry is searching for compact and more efficient phase separation devices. Furthermore, compact, light-weight devices will also make the exploitation of smaller wells economically feasible by using floating platforms. Another way to reduce the exploitation costs is achieved by transferring the separation processes undersea. When the contaminants are separated down-hole, the expensive conventional techniques on the platforms become redundant.

To arrive at compact separation, high g-devices, like cyclones, are frequently used. To cater for increasing water cuts and reduce operating costs a new type of high g-separator based on the Rotational Particle Separation (RPS) principle was designed and tested [16]. This separator combines high centrifugal forces and small radial collection distances to separate micron sized particles. A detailed explanation of the working principle of this separator is given in the next section.

To identify the possibilities of the RPS as a compact device for separating micron sized liquid droplets from natural gas under high pressure, a new project started. This project resulted in this thesis. The project was supported by the governmental programme Economy, Ecology and Technology (E.E.T.), and involved a collaboration of the Technical University of Eindhoven and CDS Engineering, one of the main suppliers of offshore phase separation equipment in the Netherlands.

1.2 Rotational Particle Separator

In the nineties, the Rotational Particle Separator (RPS) was introduced as a new technique for separating solid and/or liquid particles from gases [13, 14, 15, 34]. In this patented technique [10, 11] the principles of centrifugation are exploited to enhance separation of small-sized components and particulate matter, with a different density compared to the carrier fluid. The core component is the rotating filter element (figure 1.2), which consists of a multitude of axially oriented channels, which rotate as a whole around a common axis. Particles or droplets flowing in the fluid in a laminar motion are centrifuged to the outer walls of each individual channel and adhere to the collecting walls as a result of the centrifugal force, van der Waals forces and/or forces due to surface tension. The purified fluid leaves the channels at the exit and the channels can be cleaned periodically. As the radial distance over which the droplets have to move to arrive at a collecting surface is small, the RPS is capable of separating particles of small sizes: e.g. solid and liquid particles entrained in gases with sizes down to 0.5 micron at relatively low angular speeds and small channel lengths [15]. Practical designs of the RPS include equipment for purifying gases of industrial processes, and portable air cleaners for domestic appliances [15].

In this research project a new separator, based on the RPS principle, is designed
Figure 1.2. Left: filter element of the rotational particle separator consisting of a multitude of axial channels. Right: while the channels rotate around a common axis, particles entrained in the gas flowing through these channels are centrifuged towards the channel wall.

and tested for purifying natural gas from offshore wells. The design involves a so-called naturally driven RPS; i.e. the filter element rotates as a result of a swirl, which is generated in the flow approaching the element. As a consequence, there is no need for an external motor and therefore no shaft, which needs to be sealed, is required. In the separator high centrifugal forces are generated, which not only results in a good separation performance, but also makes a compact design possible. The housing of the present design is a thick-walled tube, which can sustain high pressures and which is aligned to the pipe transporting the fluid (in-line installation).

A schematic representation of the separator is given in figure 1.3. After passing the inlet part, which mainly serves as a bearing support, the contaminated fluid (gas/liquid) enters a swirl generator where the fluid is brought in a rotational motion as it passes its static vanes.

After the swirl generator the mixture enters the pre-separation chamber, which serves as a kind of axial cyclone. In this separation area large contaminants are swept out of the flow, due to the centrifugal motion of the fluid. The contaminants leave the separator through the pre-separator outlets, which are situated before the filter element at the outside of the separator.

The filter element is brought into rotation by the angular momentum of the fluid. Due to the centrifugal force, the droplets in the fluid are driven towards the wall of the channels where a liquid film is formed. This film breaks up at the end of the filter element and larger droplets are created which can easily be separated in the post-separator (figure 1.4).

If the dispersed phase is the lighter medium, as is the case for oil droplets dispersed in water, it is swept to the core of the post-separator [23, 30]. In the case of liquid
1.2 Rotational Particle Separator

Figure 1.3. Schematic representation of the RPS-based separator, which is designed to purify natural gas under high pressure.

Figure 1.4. The filter element operates as a coalescer when separating dispersed liquid phases from gas. Entering droplets are centrifuged towards the outer walls of each channel, creating a liquid film. At the end of each channel the liquid film breaks up in larger droplets. These droplets are collected at the outer wall of the post-separation area.

droplets in a gas flow, the droplets move towards the outer wall.

The outer wall of the filter element is extended into the post-separation chamber in order to reduce the shear stresses on the liquid film at the outer wall of the post-separator (figure 1.5). The liquid film, which develops at the extended wall of the filter element, co-rotates with the gas and in this way re-entrainment of liquid droplets in the gas flow downstream of the filter element is minimized. The liquid separated from the fluid flow leaves the separator through the post-separator outlets, which are positioned at the outer radius of the separator. The purified gas leaves the separator...
Figure 1.5. The outer wall of the filter element is extended into the post-separation area in order to minimize re-entrainment of liquid droplets downstream of the filter element.

axially through the main outlet.

Downstream of the post-separator a de-swirler is placed. The stator vanes convert a large part of the rotational energy of the fluid into static pressure. Without the de-swirler the rotational energy is completely dissipated in heat and thus wasted. Recovering the major part of the pressure loss, means that the separation process generates the waste stream at high pressures. This enables re-injection of the contaminants back into the gas reservoir from which they originally came.

1.3 Combined expansion and separation

Before natural gas extracted from offshore wells can be processed and transported, the pressure of the gas has to be reduced. This pressure reduction usually takes place in a throttling (Joule-Thompson) valve. The available energy, which is released during expansion over the valve, is dissipated and thus wasted. Useful energy can be recovered by replacing the throttling valve by a turbine. Expansion over a turbine is also more efficient compared to a throttling valve, as the expansion is isentropic instead of isenthalpic. This causes a larger temperature drop and as a result the dewpoint of the mixture decreases. This leads to the formation of more condensate and as a consequence the amount of liquid, which can be removed from the gas stream, increases. Although it involves only a slight increase in the amount of condensate removal, further gas treatment facilities can be simplified significantly. To prevent erosion and corrosion of downstream equipment it is necessary to remove the liquid as soon as possible. The combination of a turbine and an RPS-based separation device in a single unit, is therefore an attractive solution to arrive at a more compact and more efficient phase separation device [16].

A schematic representation of such a device is given in figure 1.6. The device consists of three parts. The first part is the turbine, consisting of a nozzle/rotor combination. In this turbine a part of the pressure of the gas is converted into
1.3 Combined expansion and separation

Electrical energy. The gas is expanded over a nozzle, transferring the pressure into kinetic energy. The accelerated mixture subsequently impacts on a turbine, where the kinetic energy of the mixture is transferred to a rotor. Finally, the rotor drives an electric generator. The second part of the device is the condensation area. After expansion, condensate particles are formed. These particles are typically very small in size (diameters in the submicron range; generally these particles are designated as mist or fog [24]). Therefore they must be given enough time to grow, such that they are large enough to be separated. Besides, the cooled mixture must reach an equilibrium state, to prevent condensation subsequent to the separation step. The third part of the device is the separator. In this part the gas and liquid phases are separated in an RPS-based separator, as described in section 1.2 (figure 1.3).

![Diagram of an RPS-based separation device combined with a turbine.]

Figure 1.6. Schematic representation of an RPS-based separation device combined with a turbine.

Companies have already developed devices for the separation of oil and gas in which the expansion and separation step are integrated in a single device. Dresser-Rand in cooperation with Kvaerner developed a Bi-phase Rotary Separation Turbine [68]. The Bi-phase Turbine is mainly focussed on the energy generation from a two- or three-phase mixture. The device is applied in industrial refrigeration and geothermal applications, although also developments into the separation of oil and gas are made [56]. The general operating principle is comparable to the RPS – turbine combination as depicted in figure 1.6.

A big difference with the RPS – turbine, however, is that the separation and energy generation step are combined in a single compartment. The rotor which generates energy, also produces the centrifugal force for the separation. This means that no condensation area, which allows the condensate particles to grow, is present. As a result the Bi-Phase separator is not able to remove the smallest sized condensate droplets with a high efficiency. In the RPS – turbine combination the rotors for the expansion, and separation step are separated, which allows optimal design for both stages. Besides, the design of the condensation area can be adapted to its specific
Twister B.V. has also developed a device in which expansion and separation are combined; the supersonic separator [49, 80]. In this tubular device a Laval nozzle is used to expand the feed gas to a supersonic velocity. As a result pressure and temperature are reduced, which causes the formation of a mist of water and hydrocarbon condensate droplets. Static vanes placed downstream in the supersonic flow regime, generate a vortex. The droplets in the flow are swirled to the outside of the tube and can subsequently be removed (cyclonic separator). After the separation step a diffusor recovers some of the initial pressure.

One of the disadvantages of this separator is the way of expansion, which provides cooling of the gas stream. Cooling by an expansion turbine is preferred above techniques employing expansion by acceleration, such as the Laval nozzle. This is because after the turbine the velocity of the gas can be kept relatively small as cooling occurs by withdrawing power from the gas rather than from the gas speed. This avoids the risk of evaporation of the ultra-fine condensate droplets due to heating up of the gas by internal friction. By using an expansion turbine a stable situation is created whereby nucleation and growth of droplets can take place.

Another disadvantage of both the Twister and Bi-phase separator compared to the RPS – turbine, is the way in which the droplets are separated. In the RPS the radial distances, which the droplets have to travel to reach a collecting wall, are small. This enhances the separation of micron sized droplets, at the price of adding a moving component; i.e. the filter element. In the other two separators, however, radial distances are much larger. Due to the disadvantages mentioned above, the separation efficiency of small sized particles in the Twister and the Bi-Phase separator will probably be less compared to the RPS – turbine.

1.4 Goal and outline

The aim of this study is to design a compact and efficient device for separating condensed liquid phases from natural gas. For this purpose the rotational particle separator principle is used. The device allows reduction of the size of phase separation equipment on space and weight constrained platforms and can possibly even allow for down-hole separation. In this way the exploration of older and smaller wells becomes economically feasible.

Besides a compact design, also the separation efficiency of the device should be competitive with current separation techniques. The separation performance of settling tanks is limited to a $d_{p,50\%}$ of 100 micron [2, 67] (the $d_{p,50\%}$ represents the diameter of a particle separated with 50% probability). In order to enhance separation performance of these tanks, mesh pads or vane packs are usually inserted. A properly designed tank with baffles is able to reach a $d_{p,50\%}$ of 5 micron [67]. The separation efficiency of centrifugal separators is comparable to mesh or vane pack separators [2, 67] at the advantage that they are less prone to plugging and the capacity per unit area is larger (as higher fluid velocities are allowed). By applying the RPS separation technique in the current design, it becomes possible to achieve a better efficiency compared to current techniques.
Furthermore, it is investigated whether the functions of expansion and separation step can be combined in a single device or not. This also contributes to a more compact way of processing natural gas. Besides, the dissipated pressure can be converted into useful energy. In this study it is only investigated if the expansion and separation chamber can be combined in a single device. The design of the turbine (nozzle/rotor) and the subsequent conversion to electrical energy are not treated in this study.

In close consultation with CDS Engineering the design parameters of the prototype were fixed [59]. It was decided to design the RPS-based separator such that droplets with diameters down to 1 micron are removed from the gas flow with 100% probability \(d_{p,100\%}=1\ \mu m\). These droplets are created by condensation and coagulation, as discussed in section 3.2.1. Furthermore, the separator should be able to handle the volume flow of one wellhead, which is typically 2335 m\(^3\) hr\(^{-1}\) (at \(P=80\) bar and \(T=340\) K). Besides the separator should be designed such that it can be installed in-line. It was therefore decided to restrict the outer diameter of the separator to 0.2 m (\(\sim 8\) inch). Initially the separator is designed to replace the second processing stage (downstream of the knock-out drum) and possibly the Joule-Thompson valve. This means that the fluid properties for which the separator should be designed, depend on the conditions after wellhead separation. According to Swanborn [67], pressures at wellhead separation may range up to 600 bar and temperatures vary between 80 and 100°C. For the current design a pressure of 300 bar and a temperature of 100°C after wellhead separation is assumed. Conditions at the separation step depend on the way of expansion (throttling / expanding over a turbine). The amount of liquid contaminants in the gas before the separator is assumed to be smaller than 1% volume fraction, as bulk separators have already separated the major part of the contaminants [59]. An overview of the preliminary design criteria is given in table 1.1. In section 3.3.1 the final criteria will be determined.

Table 1.1. Design parameters for the RPS – turbine.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(d_{p,100%})</td>
<td>1 micron</td>
</tr>
<tr>
<td>(D_o)</td>
<td>0.2 m</td>
</tr>
<tr>
<td>Volume flow gas</td>
<td>0.18 m(^3) (s^{-1}) (\rho_g \sim 180\ \text{kg} \text{m}^{-3})</td>
</tr>
<tr>
<td>Volume fraction contaminants</td>
<td>&lt; 1%</td>
</tr>
<tr>
<td>Pressure</td>
<td>300 bar</td>
</tr>
<tr>
<td>Temperature</td>
<td>373 K</td>
</tr>
</tbody>
</table>

To design the separator two issues are resolved. First the behavior of the liquid film inside the channels of the filter element is investigated. In figure 1.4 the working principle of the filter element is explained. It can be seen that the filter element channels act as a coalescer. The micron sized droplets, which enter the channels, are centrifuged towards the outer wall of each channel. At the wall the droplets coagulate and form a liquid film. The motion of the droplets towards the channel wall is well understood. The way in which the liquid film develops over the channel wall is however not clear. It is interesting to know how the liquid is distributed under the influence of the centrifugal and shear force. Will all liquid leave the filter element
at the end of the channels or is part of the liquid forced towards the inlet? This is important for the design of both the pre- and post-separator outlets, as they must be large enough to drain the offered liquid. In chapter 2 a model describing the behavior of the liquid film in the channels of the filter element is presented. Both vertically and horizontally placed filter elements are discussed.

A second crucial parameter in the design of the separator is the size of the droplets which appear right after expansion. The size of these droplets determines the dimension of the condensation area. At the exit of the condensation area these droplets must be large enough to be separated in the RPS. The size of the smallest droplets is determined by nucleation and growth of the vapor phases present in the gas flow. The group Gas Dynamics of the Physics Department at the Technical University Eindhoven has done considerable research in this area [40, 47, 53]. Their work is examined in section 3.2.1, to see if it is feasible to combine the expansion and separation area in one device.

After these two issues are resolved, the design process can be started. The equations describing separation performance of the filter element, pre- and post-separator are treated (section 3.2.2). In order to drive the filter element, the gas stream is brought in rotation by upstream placed (fixed) vanes. The angular momentum generated by these vanes is not only used to turn the element but also to overcome losses in other parts of the separator (like the bearings). Relations for the generated and dissipated angular momentum are given in section 3.2.3. (The conversion of part of the rotational energy of the fluid which leaves the post-separator, to static pressure by means of a de-swirler, as mentioned in section 1.2, is not treated in this study.) The last important design parameter, the pressure drop over the different parts of the separator is discussed in section 3.2.4. Using these design relations together with the design criteria, stated in section 3.3.1, the individual components of the separator are designed (section 3.3.2). At the end of chapter 3 the theoretical performance of the designed prototype is presented. This includes angular speed, pressure loss and separation performance as functions of flow rate.

The experiments performed with the prototype are discussed in chapter 4. The measurements can be subdivided into hydrodynamic and separation performance tests. During the hydrodynamic tests the pressure drop over the separator and the angular speed of the filter element were measured at different flow rates. These tests were performed at both atmospheric and high-pressure conditions. With the separation performance tests, the efficiency of the separator as a function of droplet size was measured. All measurement results are compared with a theoretical performance model, which is based on the design relations derived in chapter 3.

Finally, the conclusions of the findings of this study are given in chapter 5.
Chapter 2

Creeping film model

2.1 Introduction

The filter element of the Rotational Particle Separator (RPS), as depicted in figure 1.4, acts as a coalescer, when separating liquid phases from a gas. Droplets, which enter the channels, are centrifuged towards the outer wall of each channel. At the wall these droplets coalesce and form a liquid film.

The motion of the droplets towards the channel walls is well understood. Analytical expressions for parameters describing the separation performance in the filter element were derived by Brouwers [14], see section 3.2.2. Brouwers derived an expression for the diameter of a particle or phase, which reaches a collecting wall with 100% probability. Particles smaller than this diameter are also collected, if they are at a shorter distance from the collection surface at the channel inlet and do not have to cross the entire channel width. Brouwers [14] found relations for the entire filter element efficiency as a function of a dimensionless particle diameter, which is defined as the particle diameter divided by the particle diameter separated with 100% probability. Furthermore, measurements with a number of filter elements, which removed solid phase contaminants and were subject to different conditions (angular speed, flow rate, particulate matter, etc.), have been performed. The result of these measurements showed good agreement with theory [13].

When the liquid film reaches the end of the channels of the filter element, the liquid film breaks up into droplets. In the post-separator, which is situated downstream of the filter element, the gas flow is still in a rotational motion and due to this rotation the droplets are centrifuged towards the outer wall of the post-separator. However, there is a risk of breaking up of these droplets into smaller droplets as a result of the centrifugal or turbulent forces acting in the post-separation area (section 3.2.2). Besides, there is a chance of re-entrainment of the liquid film, which develops at the extended inner wall of the filter element, into the gas flow. These two processes would lead to a decrease in the efficiency of the separator.

To minimize the decrease in efficiency, the filter element could be placed vertically, such that the main flow enters the element at the bottom and the cleaned gas leaves
the element at the top. In such a configuration most of the liquid in the filter element moves downwards due to the gravitational force acting on the liquid film. However, as will be shown in the following sections, part of the liquid film is forced upward due to the centrifugal force. This means that part of the liquid leaves the filter element together with the purified gas stream. This may again result in a decrease in filter efficiency.

To predict the decline in filter efficiency, the amount of liquid leaving the filter element at the top must be known. This amount can be estimated from the behavior of the liquid film, which develops along the channel walls. The behavior of this film is not known and therefore relations have to be derived, which are valid for the specific situation encountered in the filter element of the RPS. In this chapter a model is developed to describe the liquid film behavior. The model enables determination of the thickness of the liquid film as a function of the liquid influx. Furthermore, with the model the amount of liquid, which leaves the filter element at the top, can be predicted and also the parameters determining this amount are derived.

With the results of the analysis, better design of future RPS-based separators becomes possible.

2.2 Existing models describing liquid films

Before we start with the analysis of the liquid film inside the channels of the filter element, existing models describing liquid film behavior are discussed.

A well-known and thoroughly studied example of film flow, is the liquid film arising on a cold surface due to condensation of the surrounding saturated vapor. Nusselt [48] was the first to derive an analytical relation for the film thickness as a function of the plate length for a vertically orientated plate. He considered the situation in which the flow of condensate in the film is laminar and both the plate and the vapor have a constant temperature, respectively \( T_w \) and \( T_s \) (figure 2.1). Also the fluid properties are assumed constant. The vapor is regarded stationary and exerts no drag on the downward motion of the condensate. Furthermore a slender film, implying that the thickness of the liquid film \( \delta \) is much smaller than the length of the plate \( L \) is considered. This means that the flow in the liquid film can be regarded as a boundary layer-type flow [5, 6]. In this case the condensate momentum equations in \( x \)- and \( y \)-direction reduce to

\[
x\text{-component conservation of momentum} \\
\frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \frac{\partial^2 u}{\partial y^2} + g \\
\]

(2.1)

\[
y\text{-component conservation of momentum} \\
0 = -\frac{1}{\rho} \frac{\partial P}{\partial y} \\
\]

(2.2)

where \( x \) and \( y \) are respectively the coordinate along the plate and perpendicular to the plate, \( u \) and \( v \) are respectively the velocity of the liquid along the plate and
2.2 Existing models describing liquid films

perpendicular to the plate, $P$ is the pressure in the liquid film, $g$ is the gravitational force acting on the film and $\rho$ and $\nu$ are respectively the density and the kinematic viscosity of the liquid. Because of the slenderness of the liquid film, it is assumed that the vertical pressure gradient in the liquid is equal to the hydrostatic pressure gradient in the surrounding vapor $\frac{\partial P}{\partial x} = \rho_v g$, where $\rho_v$ is the density of the vapor. Substituting this expression in equation (2.1) gives

$$
\rho \left( \frac{\partial u}{\partial x} + \nu \frac{\partial u}{\partial y} \right) = \eta \frac{\partial^2 u}{\partial y^2} + (\rho - \rho_v) g
$$

(2.3)

Subsequently it is assumed that the inertia effects are small compared to the effects of friction (creeping flow is considered). This means that the left side of equation (2.3) can be disregarded. The resulting equation is integrated twice over $y$, with the boundary condition of no slip at the wall ($u = 0$ at $y = 0$) and zero shear at the vapor-liquid interface ($\frac{du}{dy} = 0$ at $y = \delta$). This gives the liquid velocity profile along the plate

$$
u = \frac{g}{\eta} (\rho - \rho_v) \left( \delta y^2 - \frac{y^2}{2} \right)
$$

(2.4)

Applying the first law of thermodynamics to a control volume in the liquid film, with size $\delta \times dx$, assuming zero film thickness at the beginning of the plate ($\delta = 0$ at $u = 0$) and a linear temperature profile inside the liquid film, the thickness of the laminar film can be derived [5, 6]. It follows that the film thickness $\delta$ increases with the longitudinal length $x$ raised to the power $1/4$.

Figure 2.1. Laminar film of condensate, which is created by the condensation of a saturated stationary vapor with temperature $T_s$ on a cold wall with temperature $T_w$. 
In the above given analysis derived by Nusselt the effect of inertia is assumed negligible. Sparrow and Gregg [64] included inertia forces within the condensate film and used a similarity transformation to obtain the complete boundary layer solution to the same problem. They found that the inclusion of the inertia terms does not alter the finding that $\delta$ is proportional to $x^{1/4}$. Furthermore, they found that for Prandtl number larger than 1 the inclusion of the inertia terms has only little effect on the heat transfer results. For Prandtl number smaller than 1 and relatively thick condensate films the inclusion of the inertia terms plays a more important role.

Chen [17] extended the model of Sparrow and Gregg by including the influence of the drag exerted by the vapor on the liquid film. Chen found that in this case the film thickness $\delta$ is again proportional to $x^{1/4}$. Furthermore, he derived velocity and temperature profiles in the liquid film by using perturbation methods.

Leppert and Nimo [35] used a Nusselt-type analysis to study laminar film condensation on a surface, which is normal to body or inertial forces (condensation on a horizontal surface or inside a rotating drum). They assumed that the condensate flow was driven by the hydrostatic pressure gradient due to changes in the film thickness along the surface. Vapor-liquid interfacial shear was neglected. As boundary conditions were taken: maximum film thickness in the middle of the plate and a minimum finite film thickness at the edge of the plate. Bejan [4] treated a similar problem: film condensation on an upward facing plate with free edges. In this Nusselt-type analysis it is assumed that the condensate flows away tangentially from the central region of the plate and spills over the edge. He assumed that the liquid film thickness reaches a maximum at the middle of the plate and is zero at the edges. Both Bejan [4] and Leppert and Nimo [35] found that $\delta$ decreases with $x$ to the power $1/5$ from the middle of the plate towards the edges.

Sparrow and Gregg [65] also treated laminar film condensation on a rotating disk. They solved the complete Navier-Stokes, mass and energy equations and found that the film thickness decreases with increasing rotational speed of the disk, $\delta = \Omega^{-1/2}$ for fixed fluid properties and a fixed temperature difference between the disk and the surroundings.

All analyses mentioned above mainly focus on heat transfer between a cold plate and the surrounding vapor. The goal of these analyses is to find Nusselt relations or heat transfer coefficients to describe the heat transfer process. In the filter element of the RPS mass transfer is however not due to condensation but due to the gravitational and centrifugal forces, which act on the droplets present in the gas stream. Nevertheless, as is shown in the next sections, many of the assumptions used in the Nusselt analysis also apply to the situation encountered in the RPS. Therefore, in the current analysis of the liquid film in the channels of the filter element, the Nusselt analysis is adapted, such that it is valid for our situation.

It must be mentioned that also considerable research has been done on liquid films in rotating gas/liquid systems, like cyclones. These analyses however, are mainly focused on the stability of the liquid film. In these devices re-entrainment of the separated liquid from the film into the gas must be avoided. The aim of these studies, therefore, is to find stability criteria in terms of the maximum allowed gas velocity along the liquid film or the film Reynolds number. A survey of re-entrainment and
related effects in gas-liquid separators for the offshore industry is given by Swanborn [67]. In the current analysis, however, it is assumed that the liquid film in the channels of the filter element does not break up. This is acceptable as the Reynolds number of the liquid film in the channels of the RPS is small (section 2.4.1) and also the gas velocity in the channels of the filter element is relatively low.

2.3 Current analysis

In this chapter a model describing the liquid film behavior at the channel walls of the filter element is presented. With this model, which is based on the Nusselt approach (section 2.2), the development of the height of the liquid film over the length of the channels is calculated for a given influx of liquid towards the film. The goal of this analysis is to find expressions for the amount of liquid leaving the channels of the filter element at the in- and outlet and to find the parameters determining this amount.

Two configurations are regarded, one in which the filter element is placed vertically and one in which the element is placed horizontally. In the model it is assumed that the transfer of droplets from the gas towards the wall (due to the action of the centrifugal and gravitational force) and in this way feeding the liquid film at the wall affects the mass transfer in the film. Other influences, such as the disturbance of the velocity profile at the interface between gas and liquid, are supposed to be negligible. The properties of the liquid film, like density and viscosity, are assumed constant.

The channels of the filter element are assumed to be concentric in shape. In reality the filter element is manufactured by winding up layers of corrugated and

![Figure 2.2](image_url)

*Figure 2.2.* Cross-section of a channel of the filter element. Left: in reality the channels are created by winding up layers of corrugated and non-corrugated metal sheets, which results in a triangular/sinusoidal cross-section. Right: in the model the channels are regarded as concentric rings.
non-corrogated sheet material around a shaft. In this way triangularly or sinusoidally shaped channels are created (figure 2.2). The radial walls inside the annulus stabilize the flow in case of rotation. However, due to the centrifugal force, the liquid film will only arise on the outer wall of each channel. Therefore, as a first approximation the channels of the filter element can be regarded as concentric rings.

Furthermore, a stationary situation, in which the complete outer surface of the channels is wetted, is considered. Besides, the influx of liquid towards the film is assumed constant over the length of the channel. In practice this is generally not true, as a diversity of large and small droplets enter the filter element channels. The larger droplets will reach the channel walls sooner than the smaller droplets. Usually the amount of the larger droplets is larger than the amount of the smaller droplets. This leads to a higher mass flow rate towards the walls at the entrance region of the channels than further downstream of the channels. It is assumed, however, that this does not have a large influence on the liquid film behavior.

2.4 Vertical configuration

When a gas, containing droplets, flows through a vertically placed rotating filter element, the droplets are subjected to gravity and to the centrifugal force. Due to these forces, the droplets migrate relatively to the gas towards the channel walls. A liquid film of arrested droplets arises at the wall. A schematic representation of this situation is given in figure 2.3. The gas flow with the entrained droplets enters the channel from the bottom. This is the standard operating condition in the case the filter element is placed vertically [13, 15]. For describing the liquid film distribution, a single channel of the filter element is regarded. The channels of the filter element are assumed to be concentric, as was discussed in the previous section. Furthermore the centrifuged droplets, which feed the film, the gas and the liquid film co-rotate with the same circumferential speed. Therefore, the wall of the concentric channels can be described as a flat plate with length $L$ and an infinite width. A schematic representation of this situation is given in figure 2.4. The liquid droplets, which enter the channel, are evenly distributed over the length of the plate and move towards the plate with a constant velocity, $v_0$. It is assumed that the liquid load is small, i.e. the film thickness $\delta$ is much smaller than the thickness of the channel $R$. Therefore, the centrifugal force $g_c$ acting on the liquid film can be regarded constant, $g_c = \Omega^2 r$, with $r$ the radius at which the channel is located. Another implication of a small liquid load is that the Reynolds number based on the liquid film thickness is small (section 2.4.1). This means that the liquid film can be treated as a laminar layer with negligible inertia effects. Besides, the flow inside the film can be regarded as a boundary layer-type flow. This is a general assumption in describing slender liquid films. For example, in describing condensation of a saturated vapor on a cooled vertical surface (section 2.2), this assumption is also used [5, 6]. Taking all the considerations mentioned above into account, the equations for conservation of mass and momentum for a steady incompressible laminar liquid film on a vertical flat plate under the action of gravitational and centrifugal forces are given by
conservation of mass
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\]  

(2.5)

\(x\)-component conservation of momentum
\[
0 = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \frac{\partial^2 u}{\partial y^2} + g
\]  

(2.6)

\(y\)-component conservation of momentum
\[
0 = -\frac{1}{\rho} \frac{\partial P}{\partial y} - \Omega^2 r
\]  

(2.7)

where \(x\) and \(y\) are respectively the coordinate along the plate and perpendicular to the plate, \(u\) and \(v\) are respectively the velocity of the liquid along the plate and perpendicular to the plate, \(P\) is the pressure in the liquid film, \(g\) and \(\Omega^2 r\) are respectively the gravitational and centrifugal force acting on the film and \(\rho\) and \(\nu\) are respectively the density and the kinematic viscosity of the liquid. By integrating equation (2.7) the equation for the pressure distribution in the liquid film can be derived
\[
P(x, y) = -\rho \Omega^2 r y + K(x)
\]  

(2.8)

The integration factor \(K(x)\) is determined by equalling the liquid pressure to that of the gas at their interface \(y = \delta\). Because the gas flow in the channels is laminar and
δ ≪ R, the velocity of the gas at the boundary is negligible. When the pressure drop over the channel is also neglected, there is only a static pressure $P_0$ in the gas at the boundary

$$P_g(x, \delta) = P_0$$  \hspace{1cm} (2.9)

By setting $P_g(x, \delta) = P(x, \delta)$ the integration factor $K(x)$ can be determined

$$K(x) = P_0 + \rho \Omega^2 r \delta(x)$$  \hspace{1cm} (2.10)

and the pressure gradient in $x$-direction can be written as

$$\frac{\partial P}{\partial x} = \rho \Omega^2 r \frac{d\delta}{dx}$$  \hspace{1cm} (2.11)

Substituting equation (2.11) into equation (2.6) yields

$$\nu \frac{\partial^2 u}{\partial y^2} = \Omega^2 r \frac{d\delta}{dx} - g$$  \hspace{1cm} (2.12)

This equation can be integrated twice with the following boundary conditions: on the interface between liquid and gas the shear force is zero

$$y = \delta \frac{\partial u}{\partial y} = 0$$  \hspace{1cm} (2.13)

and at the wall the velocity of the liquid is zero

$$y = 0 \quad u = 0$$  \hspace{1cm} (2.14)

With these conditions the velocity profile along the plate is given by

$$u = \frac{1}{2\nu} \left[ y^2 - 2y\delta \right] \left[ \Omega^2 r \frac{d\delta}{dx} - g \right]$$  \hspace{1cm} (2.15)

This solution resembles the velocity profile for a laminar film of condensate, supplied by a reservoir of stationary saturated vapor, along a cooled vertical surface, as discussed in section 2.2. In that case only gravity has an influence on the liquid film thickness. The velocity along the plate for this situation, assuming that the vapor density is negligible compared to the liquid density, is given by

$$u = \frac{g}{2\nu} \left[ 2y\delta - y^2 \right]$$  \hspace{1cm} (2.16)

In case of a film subjected to gravity the liquid leaves the plate at the bottom ($x = L$), as there is no force counteracting the gravitational force. However, for the case considered here, the centrifugal force also acts on the film. As a result, part of the liquid is forced upwards. This means that there exists a turning point, $x_0$, where the velocity $u$ along the plate is zero, i.e. changes direction. From equation (2.15) it can be deduced that for this point $\frac{d\delta}{dx} = \frac{g}{\nu \Omega^2 r}$ holds. Zero velocity in the liquid film at the
turning point implies that not all the liquid flows downward along the wall but part of the liquid flows upward and leaves the plate at the top, $x = 0$.

A relation for the film thickness can be derived by considering a mass balance over the control volume depicted in figure 2.5. In steady state condition the mass flow rate to the liquid film must be equal to the mass flow rate through the liquid film. Entering the control volume from the right is the liquid load feeding the liquid film. The mass rate of this liquid flow is represented by $\rho v_0 \Delta x$. The velocity in $y$-direction of this liquid load $v_0$ is assumed constant over the length of the plate. At the top of the control volume the liquid, which has already reached the wall, enters. The mass rate of this flow is given by $\rho U$, where $U$ is the volume flow per unit width along the wall defined as

$$
U = U(x) = \int_0^\delta u \, dy
$$

(2.17)

At the bottom of the control volume the liquid supplied from the droplets feeding the film and from the droplets which have already reached the wall are discharged. This mass rate is given by $\rho U + \rho \frac{dU}{dx} \Delta x$, neglecting terms of higher order than $\Delta x$. Balancing these three mass rates gives

$$
\rho U + \rho v_0 \Delta x = \rho U + \rho \frac{dU}{dx} \Delta x
$$

(2.18)
which yields ($\rho$=constant)

$$v_0 = \frac{dU}{dx} \quad (2.19)$$

Integrating both sides with respect to $x$ gives an expression for the film thickness $\delta$. It must however be kept in mind that only the part of the liquid load, which reaches the wall below $x_0$, leaves the plate at the bottom. This means that integrating equation (2.19) gives

$$U(x) = v_0(x - x_0) \quad (2.20)$$

The local volume flow rate of liquid per unit width of the wall, i.e. the left hand side of equation (2.20) follows from equation (2.17) and equation (2.15) as

$$U(x) = \int_0^\delta u \, dy = \frac{1}{3\nu} \delta^3 \left[ g - \Omega^2 r \frac{d\delta}{dx} \right] \quad (2.21)$$

Substituting equation (2.21) in equation (2.20) results in a differential equation describing the film thickness as a function of the channel height

$$\delta^3 \frac{d\delta}{dx} + \alpha(x - x_0) - \beta \delta^3 = 0 \quad (2.22)$$

where the coefficients $\alpha$ and $\beta$ are given by

$$\alpha = \frac{3\nu v_0}{\Omega^2 r} \quad \beta = \frac{g}{\Omega^2 r} \quad (2.23)$$

Equation (2.22) is made dimensionless with the following variables

$$\overline{\delta} = \frac{\delta}{\delta_0} \quad \overline{x} = \frac{x - x_0}{L} \quad \epsilon_1 = \frac{\delta_0}{\beta L} = \frac{\delta_0 \Omega^2 r}{gL} \quad (2.24)$$

where $\delta_0$ is the maximum film thickness yet to be specified. Substituting these variables in equation (2.22) gives

$$\epsilon_1 \overline{\delta} \frac{d\overline{\delta}}{d\overline{x}} + \alpha L \frac{\overline{x}}{\beta \delta_0^2} \overline{\delta} - \overline{\delta}^3 = 0 \quad (2.25)$$

Although the centrifugal force is in general much larger than the gravitational force, $\epsilon_1$ is in most situations much smaller than 1. This is due to the fact that for a small liquid load the maximum film thickness $\delta_0$ is much smaller than the length of the channel $L$.

For $\epsilon_1 \ll 1$ the first term on the left-hand side of equation (2.25) can be disregarded, leaving as the main part of the solution

$$\overline{\delta}^3 = \frac{\alpha L}{\beta \delta_0^2} \overline{\delta}$$

(2.26)
2.4 Vertical configuration

From this result it is seen that the film thickness increases with $\pi^{\frac{1}{4}}$. In comparison, for the case of condensation of a pure vapor on a vertical cooled plate $\delta \sim x^{\frac{1}{4}}$ holds [6]. From this result it can also be derived that the derivative $\frac{d\delta}{dx} \sim \pi^{\frac{1}{2}}$ and becomes infinite as $\pi \to 0$. This indicates that although $\epsilon_1$ is small, the first term of equation (2.25) can not be neglected around the turning point $x_0$. It implies that the main solution (equation (2.26)) is only valid for $x \gg x_0$, in other words far away from the turning point $x_0$. Another conclusion resulting from equation (2.25) is that the centrifugal force does not play a role in the region $x \gg x_0$ and only gravity has an influence. The reason is that although the centrifugal force is in general much larger than the gravitational force, it causes a force in vertical direction only through the slope in the film thickness, $\frac{d\delta}{dx}$. For sufficiently small values of this slope the net effect of the centrifugal force on the vertical transport remains small.

From equation (2.15) it was deduced that for the turning point, $x_0$, $\frac{d\delta}{dx} = \frac{g}{\Omega r}$ holds. At this point both the velocity $u$ and the volume flow $U$ are zero. This indicates that the liquid above the turning point flows in upward direction, whereas the liquid below $x_0$ flows downward along the wall. Considering equation (2.15) this implies that above the turning point $x_0$ the derivative of $\delta$ with respect to $x$ is larger than the ratio of gravity to centrifugal force, whereas below the turning point the opposite holds, $\frac{d\delta}{dx} < \frac{g}{\Omega r}$.

Although the major part of the liquid leaves the filter element at the bottom, a small part will leave the element together with the purified gas stream. It is interesting to find the position of $x_0$ as it determines the amount of liquid leaving the plate at the top. In order to solve equation (2.25) in the local region around $\pi = 0$, two new variables are defined

$$
\eta = \epsilon_1^{\frac{3}{2}} \pi \quad f = \epsilon_1^{\frac{1}{2}} \delta
$$

(2.27)

Substituting these two variables in the dimensionless equation (2.25) gives

$$
f^3 \frac{df}{d\eta} + \eta - f^3 = 0
$$

(2.28)

In order to solve this equation two boundary conditions are required. For the first boundary condition the film thickness on the top of the plate is taken zero*.

$$
x = 0 \quad \delta = 0 \quad \text{or} \quad \pi = \frac{-x_0}{L} \quad \delta = 0
$$

(2.29)

The boundary condition of zero film thickness at the top of the plate does not mean that the liquid flow rate vanishes. From equation (2.25) it can be derived that a mass flow rate at the top of the plate remains possible as the term $\delta^3 \frac{df}{d\pi}$ is not zero at the top of the plate. Rewriting this boundary condition in terms of $\eta$ and $f$ gives

$$
\eta = \eta_0 \quad f = 0
$$

(2.30)

where

$$
\eta_0 = \epsilon_1^{\frac{1}{2}} \frac{-x_0}{L}
$$

(2.31)
Figure 2.7. Liquid film along the channel wall of a vertically placed filter element.

In figure 2.7 the development of the liquid film thickness along the length of the plate is depicted. At the top of the plate the film thickness is zero. From this point onward the film thickness increases, until it reaches its maximum value at the end of the plate, \( x = L \). This means at the bottom of the plate, \( \delta = \delta_0 \) for \( x = L \), holds. As the change in film thickness is the largest at the top of the plate, the turning point

When the liquid reaches the top of the plate, it is forced over the edge of the plate by the centrifugal force. As the plate has a certain thickness the liquid film continues along the thickness of the plate (figure 2.6). The height of the liquid film at the top of the plate, \( \delta(0) \), can be estimated from the momentum equation in \( y \)-direction of the liquid which flows over the edge of the plate (assuming that the liquid film thickness does not change as the liquid flows over the edge of the plate). Regarding the conservation of momentum in \( y \)-direction gives: \( 0 = \nu \frac{\partial^2 v}{\partial x^2} - \Omega^2 r \). Integrating this equation with the boundary conditions that at the interface between liquid and gas the shear force is zero and at the wall the velocity of the liquid is zero gives: \( v = \frac{1}{2\pi} \Omega^2 r(x^2 - 2\delta x) \). The volume flow per unit width of the wall at the top of the plate follows from

\[
U(0) = \int_0^\delta v \, dx = -\frac{1}{3\nu} \Omega^2 r \delta(0)^3.
\]

Thus, the film thickness at the top of the plate is given by \( \delta(0)^3 = -\frac{3 \nu U(0)}{\Omega^2 r} \). From equation (2.20) it follows that \( U(0) = -v_0 x_0 \). Substituting this in the equation for \( \delta(0) \) gives \( \delta(0)^3 = \frac{3 \nu v_0 x_0}{\Omega^2} \). Comparing the film thickness at the top of the plate with the maximum film thickness \( \delta_0 \), which is reached at the end of the plate when only gravity is considered (equation (2.32)), gives \( \left( \frac{\delta(0)}{\delta_0} \right)^3 = \frac{x_0^2}{L^2} \frac{\Omega^2 r}{\nu} \). As the ratio of the gravitational force to the centrifugal force is of the order 0.01 or less and also the ratio of \( x_0 \) to \( L \) is much smaller than 1, the film thickness at the top of the plate is much smaller than the film thickness at the end of the plate. So, the assumption of zero film thickness at the top of the plate is reasonable.
is also situated at the beginning (top) of the plate, \( x_0 \ll L \). This means that \( \delta = 1 \) for \( \bar{x} \approx 1 \). The expression for the maximum film thickness can now be derived from equation (2.26)

\[
\delta_0^3 = \frac{\alpha L}{\beta} = \frac{3 \nu_0 L}{g} \tag{2.32}
\]

Thus, for the second boundary condition the situation at the end of the plate can be taken. This means that for \( \eta \to \infty \) (\( \epsilon_1 \ll 1 \)) the main solution (equation (2.26)) must apply

\[
\eta \to \infty \quad f = \eta^3 \tag{2.33}
\]

Numerical solution of equation (2.28) with boundary conditions (2.33) and (2.30) by means of a fourth order Runge Kutta method gives \( \eta_0 = -0.1831 \). The fraction of liquid leaving the channel at the top is equal to the ratio of the position of the turning point to the channel length. Thus, with the numerical solution the fraction of liquid leaving the channel at the top for situations in which \( \epsilon_1 \ll 1 \) can be expressed as

\[
\text{Fraction}_{\text{top}} = \frac{x_0}{L} = -\eta_0 \epsilon_1^2 = 0.1831 \sqrt{\frac{\alpha}{\beta^2 L}} \tag{2.34}
\]

It follows that the larger the ratio of centrifugal to gravitational force the more the point \( x_0 \) moves downwards. In the local region of \( \bar{x} \sim \epsilon_1^2 \) there is a strong curvature in film thickness. Here the centrifugal force exerts a large force on the film, causing a part of the liquid to flow upward (local solution in figure 2.8). The larger the centrifugal force the larger the mass flow rate that leaves the filter element at the top. In the region far away from the turning point the curvature in film thickness is relatively small and as a result the influence of the centrifugal force on the liquid film can be neglected and only gravity plays a role (main solution in figure 2.8).
**Liquid injected from the top of the channel**

There are also situations in which the liquid phase enters the filter element from the top. For example situations in which solid particles have to be separated from a gas stream. In those situations the channels of the filter element must be periodically cleaned with a liquid or gas in order to prevent blockage of the channels. In case liquid is injected from the top of the channels, a liquid film contaminated with solid particles arises at the wall, see figure 2.9. The centrifugal force acting on the film, will force the liquid to both ends of the plate. The liquid which is forced upward will leave the filter element together with the purified gas stream. As this liquid is contaminated, it is interesting to know the amount of liquid leaving the plate at the top and to find ways to minimize this flow.

When injecting the liquid by a nozzle into the channels of the filter element, the liquid penetrates the channel to a certain depth. The penetration depth of the liquid jet $L_w$ follows from conservation of mass. If an equal distribution of the liquid influx over the penetration length is assumed, $L_w$ is expressed as

$$L_w = \frac{u_I R}{v_0}$$

(2.35)

in which $u_I$ is the vertical velocity of the injected liquid, $R$ the height of the concentric channel and $v_0$ the velocity in $y$-direction of the liquid load. Both $u_I$ and $v_0$ are assumed constant. Besides, the vertical velocity of the injected liquid is assumed equal to the vertical velocity in the liquid film.

![Figure 2.9. Model of a liquid film at the channel wall of a vertically placed filter element for situations in which the liquid is injected at the top of the channel.](image)

To describe the thickness of the liquid film over the length of the channel the same analysis as in the situation the liquid enters the channel from the bottom is
2.4 Vertical configuration

performed. Like in the previous analysis, it is assumed that the channels are concentric and that the gas, the entrained droplets and the liquid film co-rotate with the same circumferential velocity. This means that the wall of the channels can be described as a flat plate with length $L$ and infinite width. The liquid influx is assumed to be evenly distributed over the channel length. Furthermore the liquid load is assumed small, which means that the film thickness $\delta$ is much smaller than the height of the channels $R$. As a consequence, the film can be regarded as a boundary layer-type flow with negligible inertia effects (section 2.4.1) and the centrifugal force can be regarded constant over the radial height of the liquid film. Analogously to the previous section the same velocity profile along the plate is deduced from the momentum equations and the boundary conditions given by equations (2.13) and (2.14) From the mass balance, also derived in the previous section, the differential equation describing the thickness of the liquid film follows, equation (2.22)

$$\delta^3 \frac{d\delta}{dx} + \alpha(x - x_0) - \beta \delta^3 = 0 \quad (2.36)$$

with the coefficients $\alpha$ and $\beta$ defined by equation (2.23)

$$\alpha = \frac{3 \nu v_0}{\Omega^2 r}, \quad \beta = \frac{g}{\Omega^2 r} \quad (2.37)$$

Similar to the derivation in the previous section it is assumed that at the top of the channel the liquid film thickness is zero

$$x = 0 \quad \delta = 0 \quad (2.38)$$

Also a second condition is required because $x_0$ is unknown. This condition is based on the assumption that below the penetration depth no mass is supplied to the liquid film. This means that the film thickness below this point does not change in $x$-direction and thus at $x = L_w$ the derivative of the film thickness is zero. From equation (2.36) it follows that for the second boundary condition holds

$$x = L_w \quad \delta = \sqrt{\frac{\alpha}{\beta} \left( \frac{L_w - x_0}{L_w - x} \right)} \quad (2.39)$$

The differential equation (2.36) can only be solved numerically. First equation (2.36) is made dimensionless by introducing the following variables

$$\delta_0 = \frac{\delta}{\delta_0}, \quad \delta_0 = \sqrt{\frac{\alpha}{\beta} \left( \frac{L_w - x_0}{L_w - x} \right)}, \quad x = \frac{x - x_0}{L_w - x_0}, \quad \epsilon_2 = \frac{\delta_0}{\beta (L_w - x_0)} \quad (2.40)$$

where $\delta_0$ is the maximum film thickness. Substituting these variables in equation (2.36) gives

$$\epsilon_2 \delta_0^3 \frac{d\delta_0}{d\epsilon} + \pi - \delta_0^2 = 0 \quad (2.41)$$
The boundary conditions are

\[
\begin{align*}
\tau &= -\frac{x_0}{L_w - x_0} \quad \gamma = 0 \\
\tau &= 1 \quad \gamma = 1
\end{align*}
\]  

(2.42)

The dimensionless equation (2.41) can be solved numerically for different values of \( \epsilon_2 \) with a fourth order Runge Kutta method. In figure 2.10 the result is shown. In this figure \( x_0/L_w \) is plotted for different values of \( \alpha/(\beta^4 L_w^2) \). The fraction of liquid leaving the top of the plate is given by

\[
\text{Fraction}_{\text{top}} = \frac{x_0}{L_w}
\]  

(2.43)

Consider the following example. Water \((\nu \simeq 1 \cdot 10^{-6} \text{ m}^2/\text{s})\) is injected at the top with a velocity of \( u_l = 1 \text{ m/s} \). The plate is rotating with \( \Omega^2 r = 100 \text{ g} \), the radial thickness of the channel \( R \) is 0.001 m and the liquid load is moving with velocity \( v_0 = 0.01 \text{ m/s} \) towards the wall. From equation (2.35) it follows that \( L_w = 0.1 \text{ m} \). The factor \( \alpha/(\beta^4 L_w^2) \) for this situation is 0.3 and from figure 2.10 it follows that \( \frac{x_0}{L_w} \) is about 0.1. This means that in this case about 10% of liquid leaves the filter element at the top.

From figure 2.10, it can be concluded that the parameter \( \alpha/(\beta^4 L_w^2) \) must have a low value to minimize the amount of contaminated liquid in the purified outlet flow. This can be realized by a low angular speed of the filter element or a high velocity of the injected liquid. The angular velocity has a large influence on the overall collection efficiency of the RPS. In practice, therefore, the angular speed is usually fixed by the design parameters. This means that only the velocity of the injected liquid can be adjusted to minimize the fraction of liquid leaving the filter element at the top.
Experiments have been performed with a tangential version of the RPS (gas enters separator tangentially into a cyclone, which serves as a pre-separator, subsequently the gas enters the RPS filter element [16]). Liquid was injected with a jet nozzle at one radius of the filter element for a certain time. At specific values of \( \alpha/(\beta^4 L_0^2) \) the amount of liquid leaving the filter element at the top outlet as well as the bottom outlet was measured. For values of \( \alpha/(\beta^4 L_0^2) \) up to ten the results of the experiment were in accordance with the model. For values of \( \alpha/(\beta^4 L_0^2) \) above ten, the fraction of liquid leaving the top of the filter element predicted by the model was much higher than the measured values. Thus, for the case considered the model gives an upper limit for the liquid fraction leaving the channels at the top.

In the model it is assumed that the vertical velocity of the injected liquid is equal to the vertical velocity in the liquid film. In reality the velocity of the injected liquid decreases first by friction. This may have a substantial influence on the velocity in the liquid film and subsequently on the fraction of liquid leaving the filter element at the top. The model can be made more accurate by introducing an extra term in the Navier-Stokes equation for the momentum of the injected liquid.

### 2.4.1 Reynolds number in liquid film

In the momentum equation in \( y \)-direction, equation (2.7), the friction and inertia terms have been disregarded because of the assumption that the liquid film is much smaller than the radial thickness of the channel, \( \delta \ll R \). In fact this is the boundary layer approximation as a result of the slenderness of the liquid film [5, 6]. Due to the slenderness of the film the viscous term \( \partial^2 u/\partial x^2 \) in the momentum equation in the \( x \)-direction, equation (2.6), can be neglected as well. In ordinary boundary layers, however, the inertia terms in the momentum equation in the \( x \)-direction can not be disregarded. In fact, they represent the driving force for flow in the boundary layer. For the liquid film in the channels of the filter element, on the other hand, the driving force is provided by the gravitational force, while the inertia terms in the vertical direction can be disregarded, provided the appropriate Reynolds number is sufficiently small. To illustrate this, consider the inertia and viscous terms in the vertical direction

\[
\begin{align*}
\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} & \ll \nu \frac{\partial^2 u}{\partial y^2} \\
\text{Inertia} & \ll \text{Friction}
\end{align*}
\]

From the equation for conservation of mass, equation (2.5), it follows that the velocity in \( x \)-direction scales with

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \implies u \sim \frac{v_0 L}{\delta}
\]

This means that both inertia terms are of the order

\[
\frac{\partial u}{\partial x} = v \frac{\partial u}{\partial y} \sim \frac{v_0^2 L}{\delta^2}
\]

\[ (2.45) \]
and the friction term is of the order
\[
\nu \frac{\partial^2 u}{\partial y^2} \sim \frac{\nu v_0 L}{\delta^3} \quad (2.47)
\]

The ratio of the magnitude of the inertia term to the magnitude of the friction term can now be expressed as
\[
\frac{\text{Inertia}}{\text{Friction}} = \frac{v_0^2 L}{\delta^2} \cdot \frac{\delta^3}{\nu v_0 L} = \frac{v_0 \delta}{\nu} \quad (2.48)
\]

which is the Reynolds number based on the thickness of the liquid film.

Consider the situation in which a small amount of water droplets enter the filter element. The kinematic viscosity \( \nu \) of water is of the order \( 1 \cdot 10^{-6} \) m\(^2\) s\(^{-1}\). The magnitude of the velocity of the droplets towards the liquid film \( v_0 \) can be estimated from conservation of mass. Suppose that 1\% of the total volume flow is liquid. The liquid enters the channels with a constant axial velocity, which is assumed equal to the axial gas velocity, \( u_g \). As the liquid feeding the film is distributed uniform over the length of the plate \( L \), \( v_0 \) can be estimated by \( \frac{0.01 \cdot \text{flow}}{L} \). Here, \( R \) represents the height of the channel. Under normal operating conditions \( u_g \) is about 5 m s\(^{-1}\). The height of the channels \( R \) is about 0.001 m and the length of the channels \( L \) is about 0.2 m. This means that \( v_0 \) is about \( 2.5 \cdot 10^{-4} \) m s\(^{-1}\). An estimation for the film thickness is the maximum film thickness, given by equation (2.32)
\[
\delta_0^3 = \frac{\alpha L}{\beta} = \frac{3 \nu v_0 L}{g} \quad (2.49)
\]

With the values mentioned above it follows that the maximum film thickness \( \delta_0 \) is about \( 1 \cdot 10^{-4} \) m. The Reynolds number in the liquid film, as defined by equation (2.48), for this maximum film thickness and the values for \( \nu \) and \( v_0 \), mentioned above, is of the order 0.1. For smaller values of \( \delta \) the Reynolds number is even smaller, as the Reynolds number is proportional to the film thickness. Thus, in deriving the velocity profile for a small liquid load and consequently a slender liquid film (\( \delta \ll R \)) in the filter element, the inertia terms in the expressions for the momentum equation in the \( x \)-direction can indeed be neglected. Note that for condensation heat transfer and also for falling liquid films, the Reynolds number of the liquid film is often based on the volume flow along the wall, defined as \( \text{Re} = 4 \frac{U}{\nu} \) [5, 7]. However the Reynolds number as defined by equation (2.48) is representative of the ratio of viscous to inertia forces.

### 2.5 Horizontal configuration

Another possible configuration is the one in which the filter element is placed horizontally. For example in situations the RPS operates in-line, like the prototype built for this study. For the design of the pre- and post-separation area in such a configuration the amount of liquid which is expected at both outlets must be known. The post-separator only receives liquid from the filter element. The pre-separation area works
as a kind of axial cyclone and removes the largest particles from the gas before the gas flow enters the filter element. Besides, it is expected that due to the centrifugal force, part of the liquid, which is separated from the gas in the filter element, is forced back into the pre-separator. The fraction of liquid leaving the element on the side adjacent to the pre-separation area is deduced by following the same analysis as for the vertically orientated filter element.

Again the channel of the filter element is regarded as a flat plate with length \( L \) and an infinite width. The liquid influx towards the wall is again evenly distributed over the length of the plate. The gravity force, however, can in this case be disregarded as the centrifugal force is working in the same direction and is generally much larger. A schematic representation of this situation is given in figure 2.11. It is again assumed that the liquid load is small, which means that the film thickness \( \delta \) is much smaller than the height of the channel \( R \). As a result, the centrifugal force on the liquid film can be regarded constant. Due to the slenderness of the liquid film, the flow inside the film can be regarded as a boundary layer-type flow. Besides the liquid film can be treated as a laminar layer with negligible inertia effects, as the Reynolds number of the liquid film is small, see section 2.4.1. The equations for the conservation of mass and momentum in \( x \)- and \( y \)-direction for a steady incompressible laminar film on a rotating horizontal flat plate are given by

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.50}
\]

\[
0 = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \frac{\partial^2 u}{\partial y^2} \tag{2.51}
\]
y-component conservation of momentum

\[ 0 = -\frac{1}{\rho} \frac{\partial P}{\partial y} - \Omega^2 r \]  

(2.52)

Equation (2.52) is exactly the same as the equation for the conservation of momentum in y-direction for a vertical plate, equation (2.7). The pressure in the liquid film at the gas-liquid boundary is again assumed equal to the static pressure \( P_0 \). Therefore, the pressure gradient in x-direction in the liquid film for the horizontal case is equal to the pressure gradient in x-direction for the vertical case, equation (2.11). Substituting this equation in equation (2.51) gives

\[ \nu \frac{\partial^2 u}{\partial y^2} = \Omega^2 r \frac{d\delta}{dx} \]  

(2.53)

Integrating this equation with the boundary conditions that at the wall the velocity is zero, equation (2.14), and at the interface between gas and liquid the shear stress is zero, equation (2.13), gives

\[ u = \frac{1}{2\nu} \left[ y^2 - 2y\delta \right] \left[ \Omega^2 r \frac{d\delta}{dx} \right] \]  

(2.54)

From this result it is seen that the velocity along the plate is zero for \( \frac{d\delta}{dx} = 0 \). This means that for the turning point, \( x_0 \), the derivative of the film thickness is zero. In case the channels are not rotating and only gravity acts on the liquid film, the velocity distribution in the liquid film is also given by equation (2.54) only now the gravitational force \( g \) is substituted for the centrifugal force \( \Omega^2 r \) [4].

A relation for the film thickness is found by considering a mass balance over a control volume inside the liquid film, as was done in section 2.4. In steady state conditions the mass flow rate to the liquid film must be equal to the mass flow rate through the liquid film, or (with \( \rho \) is constant)

\[ U(x) = v_0 (x - x_0) \]  

(2.55)

where \( U(x) \) is the horizontal volume flow per unit width along the wall defined by equation (2.17), \( v_0 \) is the velocity in y-direction of the liquid feeding the film and \( x_0 \) is the turning point. The volume flow along the wall can be derived by substituting the velocity profile given by equation (2.54) into equation (2.17)

\[ U(x) = \int_0^\delta u \, dy = -\frac{\Omega^2 r}{3\nu} \delta^3 \frac{d\delta}{dx} \]  

(2.56)

The expression for the liquid film thickness along a horizontal rotating plate is found by substituting equation (2.56) into equation (2.55)

\[ \delta^3 \frac{d\delta}{dx} + \alpha(x - x_0) = 0 \]  

(2.57)
where \( \alpha \) is a constant defined by equation (2.23). Equation (2.57) is made dimensionless by replacing \( \delta \) and \( x \) with respectively \( \bar{\delta} \) and \( \bar{x} \) defined by equation (2.24). This results in

\[
\frac{\delta^3}{\bar{\delta}^3} \frac{d\bar{\delta}}{d\bar{x}} + \frac{\alpha L^2}{\delta_0^3} \bar{x} = 0 \tag{2.58}
\]

This equation can be solved analytically, giving

\[
\bar{\delta} = \left( -\frac{2\alpha L^2}{\delta_0^3} \bar{x}^2 + C_1 \right)^{\frac{1}{4}} \tag{2.59}
\]

where \( C_1 \) is an integration constant. The value of this integration constant can be retrieved by applying the appropriate boundary conditions to equation (2.59). From equation (2.54) it follows that at the turning point \( \frac{d\bar{\delta}}{d\bar{x}} = 0 \). This implies that the film thickness must have a maximum or minimum at this point. The film thickness can however not reach a minimum value anywhere along the channel wall. Minimum film thickness at a certain position along the wall results in a negative value of \( \frac{d\bar{\delta}}{d\bar{x}} \) left from this position and a positive value of \( \frac{d\bar{\delta}}{d\bar{x}} \) at the right of this position. From equation (2.56) it follows that \( U(x) \sim -\frac{d\bar{\delta}}{d\bar{x}} \). As a result liquid from the left of the point where film thickness is minimal as well as from the right of this point flows in the direction of the point where film thickness is minimal. This is a situation which physically can not exist. The only possible situation therefore is the one in which film thickness has its maximum at the turning point (figure 2.12).

![Figure 2.12.](image)

As this is a symmetrical problem, the turning point at which the film thickness is maximal must be situated in the middle of the channel, \( x_0 = L/2 \). From equation (2.56) it can be deduced that in this situation liquid flows from the middle of the channel towards both edges, which is physically possible. In order to retrieve the integration constant \( C_1 \), boundary conditions at the end of the plate are needed. These can be derived by regarding the pressure distribution inside the liquid film. This pressure distribution is given by equation (2.8)

\[
P(x, y) = P_0 + \rho \Omega^2 r (\delta(x) - y) \tag{2.60}
\]
It is assumed that on both ends of the plate \((x = 0 \text{ and } x = L)\), the pressure inside the liquid film must equal the pressure of the surrounding gas \(P_0\). This implies that the film thickness \(\delta\) is zero at both ends. Bejan [4] used the same boundary condition in deriving the Nusselt number for laminar film condensation on a horizontal plate with free edges and also in the treatment of single phase natural convection on a cold plate facing upward this condition was used [32]. Applying the condition of zero film thickness at the edges to equation (2.59) results in the following value for \(C_1\)

\[
C_1 = \frac{\alpha L^2}{2\delta_0^4}
\]

and the equation for \(\bar{\delta}\) becomes

\[
\bar{\delta} = \sqrt[4]{\frac{\alpha L^2}{2\delta_0^4}(1 - 4\pi^2)}
\]

From this result it is seen that film thickness \(\bar{\delta} \sim x^{\frac{1}{4}}\). As \(U(x) \sim \frac{\bar{\delta}^3 d\bar{\delta}}{dx}\) it follows that the volume flow increases with \(\pi\). Thus, although film thickness is zero at both ends of the channel it does not mean that liquid flow rate vanishes at the edges. On the contrary the volume flow reaches its largest (finite) value at the edges\(^\dagger\). The maximum film thickness \(\bar{\delta} = 1\) is reached at \(\pi = 0\). From equation (2.62) it follows that

\[
\delta_0^4 = \frac{\alpha L^2}{2} = \frac{3 \nu v_0 L^2}{2 \Omega^2 r}
\]

This means that equation (2.62) can be written as

\[
\bar{\delta} = \sqrt{(1 - 4\pi^2)}
\]

From equation (2.62) it can also be derived that the liquid film thickness \(\delta\) is proportional to \(\Omega^{-\frac{1}{2}}\) \((\alpha \sim \frac{1}{\Omega^2})\). This result was also obtained by Sparrow and Gregg [65] for laminar film condensation on a horizontal rotating disk.

### 2.5.1 Horizontal configuration with shear stress at interface

In this section the horizontal configuration as described in section 2.5 is regarded with the modification that the shear stress at the interface between the liquid film and the gas is no longer assumed to be negligible. This is physically more accurate as the volume flow through channels of horizontally orientated filter elements is generally much higher than through vertically orientated elements. This leads to a turbulent

\(^\dagger\)The volume flow along the plate is given by

\[
U(\pi) = -\frac{\Omega^2 r \delta_0^4}{3 r^2 \pi^3} \frac{\partial}{\partial \delta} \delta^3 (\alpha L^2 / 2\delta_0^4 \pi^3) \pi \frac{\partial}{\partial \pi} \pi \pi (\text{equation (2.56)}).
\]

From equation (2.62) follows

\[
\frac{\partial U}{\partial \pi} = -2\pi \left(\frac{\pi^2 \delta^3}{2\delta_0^4} \pi \frac{\partial}{\partial \pi} \pi \pi (1 - 4\pi^2)^{-3}\right)^{\frac{1}{2}}
\]

and

\[
\frac{\partial U}{\partial \pi} = \left(\frac{\pi^2 \delta^3}{2\delta_0^4} \pi \frac{\partial}{\partial \pi} \pi \pi (1 - 4\pi^2)^{-3}\right)^{\frac{1}{2}}
\]

Substituting these expressions in the relation for \(U(\pi)\) gives

\[
U(\pi) = \frac{\Omega^2 r \delta_0^4}{3 r^2 \pi^3} \pi \frac{\partial}{\partial \pi} \pi \pi (1 - 4\pi^2)^{-3}
\]

Thus, the volume flow is linearly dependent on \(\pi\) and reaches its maximal value at both ends of the plate.
flow inside the filter elements channels. Turbulent velocity profiles are more flat in
the middle of the channel but steeper near the wall compared to laminar velocity
profiles. This means that the gradient $\frac{\partial u}{\partial y}$ near the wall and thus also the interfacial
shear stress are much larger than in the laminar case, and can not be neglected. It is
assumed that the shear stress at the interface is described by $\tau_i = 0.5 f_i \rho_g u_g^2$, where
$f_i$ is the interfacial friction coefficient, $\rho_g$ and $u_g$ are respectively the density of the gas
and the mean gas velocity in the gas-droplet core. As a result the boundary condition
given by equation (2.13) no longer holds and is replaced by
\[ y = \delta \quad \frac{\partial u}{\partial y} = \frac{\tau_i}{\eta} = \frac{f_i}{2\eta} \rho_g u_g^2 \quad (2.65) \]
where $\eta$ is the dynamic viscosity of the liquid film. Together with the boundary
condition of no slip at the wall, equation (2.14), equation (2.53) is integrated twice
to give the velocity distribution inside the liquid film in $x$-direction along the plate
\[ u = \frac{1}{2\nu} \left[ y^2 - 2y\delta \right] \left[ \Omega^2 r \frac{d\delta}{dx} \right] + Ay \quad (2.66) \]
where $A = \frac{\tau_i}{\eta}$. In deriving this equation it is assumed that the pressure in the liquid
film at the interface is equal to the pressure of the surrounding gas $P_0$. Substituting
this result in equation (2.17) gives the volume flow per unit width along the plate
\[ U(x) = \int_0^\delta u \, dy = -\frac{\Omega^2 r}{3\nu} \delta^3 \frac{d\delta}{dx} + \frac{1}{2} A \delta^2 \quad (2.67) \]
In steady state condition this volume flow equals the flow rate to the plate given
by: $v_0 (x - x_0)$. The expression for liquid film thickness along the channel wall of a
horizontally placed rotating filter element, with shear stress at the gas-liquid interface,
follows as
\[ \delta^3 \frac{d\delta}{dx} + \alpha(x - x_0) - \gamma \delta^2 = 0 \quad (2.68) \]
where $\alpha$ is given by equation (2.23) and $\gamma$ is represented by
\[ \gamma = \frac{3\nu A}{2\Omega^2 r} = \frac{3\tau_i}{2\rho \Omega^2 r} \quad (2.69) \]
Before solving, the equation is made dimensionless with the following variables
\[ \bar{\delta} = \frac{\delta}{\delta_0} \quad \bar{x} = \frac{x - x_0}{L} \quad \epsilon_3 = \frac{\delta_0^4}{\alpha L^2} \quad (2.70) \]
Substituting these variables in equation (2.68) gives
\[ \epsilon_3 \bar{\delta}^3 \frac{d\bar{\delta}}{d\bar{x}} + \bar{x} - \frac{\gamma \delta_0^2 \epsilon_3^2}{\alpha L^2} \bar{\delta} = 0 \quad (2.71) \]
This equation differs from equation (2.25) by the third term on the left hand side. In equation (2.25) this term results from the gravitational force, whereas in the above equation it results from friction of the gas stream with the liquid film.

When the shear force is disregarded, the third term on the left hand side of equation (2.71) vanishes and the same result as in the previous section is obtained, equation (2.58). Thus, the liquid flows from the center, where the film thickness is maximal, to both edges where the film thickness is minimal.

When there is no centrifugal force acting on the liquid film, but only the shear force is present, the first term on the left hand side of equation (2.71) vanishes. As a result the film thickness is given by

$$\delta^2 = \frac{\alpha L}{\gamma \delta_0^2 \pi} \tag{2.72}$$

and the volume flow along the plate in this case is given by (substituting $\Omega^2 r = 0$ in equation (2.67))

$$U(x) = \frac{1}{2} A \delta^2 \tag{2.73}$$

As $\delta$ and $A$ are both positive everywhere along the plate, it follows that $U(x)$ is always positive. This means that all liquid flows towards the end of the plate ($x_0$ is situated at $x = 0$). This is expected as the shear force also acts in this direction and the shear force is the only force considered. Thus the liquid film thickness has a minimum value at the beginning of the plate and increases as $\delta \sim \pi^{\frac{1}{2}}$ (equation (2.72)) to a maximum value at the end of the plate, $\delta_0$ ($\delta = 1$), given by

$$\delta_0^2 = \frac{\alpha L}{\gamma} = 2 \eta \frac{v_0 L}{\tau_i} \tag{2.74}$$

In figure 2.13 the development of the liquid film thickness on a horizontal plate, which is subjected to a shear stress, is depicted.

The question now arises how the liquid film behaves when both forces are acting on the liquid film. Two limiting situations can occur. If the centrifugal force dominates
2.5 Horizontal configuration

the liquid film behavior, the film will resemble figure 2.12. The shear force shifts the
turning point towards the end of the plate and flattens the liquid film. If the shear
force dominates, a situation which resembles figure 2.13 is obtained, except for the
beginning of the plate, where due to the sharp gradient in the film thickness \( \delta(x) \), the
centrifugal force influences the behavior of the film. This can also be deduced from
equation (2.72). As the film thickness increases with \( x \), it follows that the derivative
\( \frac{d\delta}{dx} \sim \frac{1}{x^{\frac{1}{2}}} \) and becomes infinite at the beginning of the plate. Thus although \( \epsilon_3 \) is
small as a result of the small film thickness, the centrifugal force has an influence on
the behavior of the film at the beginning of the plate, due to the strong curvature
of the liquid film thickness at this place. This implies that although most of the
liquid leaves the plate at the end, part of the liquid is forced by the centrifugal force
towards the beginning of the plate. This is the same situation as was encountered for
the vertically orientated filter element in section 2.4.

In order to see which force dominates, the dimensionless constants in equation (2.71)
are compared.

\[
\epsilon_3 = \frac{\delta_0^4}{\alpha L^2} \quad \text{and} \quad \frac{\gamma \delta_0^2}{\alpha L} = \frac{\tau_i \delta_0^2}{2 \eta v_0 L} \quad (2.75)
\]

where \( \epsilon_3 \) represents the centrifugal force, \( \Omega^2 r \), and the third dimensionless term in
equation (2.71) represents the shear stress \( \tau_i \). Dividing these two terms gives

\[
\frac{\text{centrifugal}}{\text{shear}} = \frac{\delta_0^4}{\alpha L^2} \cdot \frac{\alpha L}{\frac{\gamma \delta_0^2}{\gamma L}} = \frac{2 \rho \delta_0^2 \Omega^2 r}{3 \tau_i L} \quad (2.76)
\]

Now, consider the RPS of the natural gas – water separator, which is designed
in this study (chapter 3). In the filter element the gas velocity is of the order of 10
m s\(^{-1}\). The density of natural gas at operating conditions \( (P = 80 \text{ bar and } T =
340 \text{ K}) \) is about 50 kg m\(^{-3}\). The interfacial friction factor \( f_i \) for small values of the
ratio of film thickness to the hydraulic diameter of the channel is about 0.005 [70].
This means that the interfacial shear stress \( \tau_i \) is of the order 10 Pa. The centrifugal
force \( \Omega^2 r \) at the outer radius of the filter element is in the order of 10000 m s\(^{-2}\).
The length of the filter element is about 0.2 m. Suppose that the liquid film entirely
consists of water. The density of water at these conditions is about 1000 kg m\(^{-3}\) and
the kinematic viscosity is about 1·10\(^{-7}\) m\(^2\) s\(^{-1}\). The height of the liquid film \( \delta \) can be
estimated from equation (2.74) or equation (2.63). The magnitude of the velocity of
the droplets towards the liquid film, \( v_0 \), can be estimated from conservation of mass.
Suppose that 1% of the total volume flow is liquid. The liquid enters the channels
with a constant axial velocity, which is assumed equal to the velocity of the gas stream
\( u_g \). As the liquid feeding the film is distributed uniformly over the length of the plate
\( L \), \( v_0 \) can be estimated by \( \frac{0.01 R \rho u_g}{L} \). The gas velocity \( u_g \) is about 10 m s\(^{-1}\) and height
of the channel \( R \) is about 0.001 m. This means that \( v_0 \) is about 5·10\(^{-4}\) m s\(^{-1}\). The
dynamic viscosity of water is of the order 1·10\(^{-4}\) Pa s at the operating conditions.
Substituting these values in equation (2.74) and equation (2.63) gives a maximum
film thickness of respectively 4·10\(^{-5}\) m and 1·10\(^{-4}\) m. Thus \( \delta_0 \) is maximal 1·10\(^{-4}\) m.
With this value, equation (2.76) gives for the ratio between the centrifugal and shear
force a value of about $5 \cdot 10^{-2}$. Thus, in this case the centrifugal term is negligible compared to the shear stress term.

Thus, for the case considered the shear force predicts the behavior of the liquid film, except for the beginning of the plate, where due to the sharp curvature in the film thickness, the centrifugal force plays a role. This is similar to the vertical case, where, except for the upper part of the filter element, gravity dominates the behavior of the film. So, in the horizontally orientated natural gas – water separator, the major part of the liquid leaves the filter element at the end and only small fraction of the liquid leaves the element at the beginning $x = 0$. As a result, the turning point $x_0$ is situated at the entrance of the channel, $x_0 \ll L$ (figure 2.14).

![Liquid film, which is subjected to a centrifugal force and the shear stress of the gas flow above the liquid film, at the wall of a channel in a horizontally placed filter element.](image)

In order to calculate the amount of liquid leaving the channels at $x = 0$, equation (2.71) has to be solved completely. This can be done similarly to the approach used in section 2.4. The boundary conditions in this case are: at the beginning of the plate film the thickness is zero and at the end of the plate the film thickness approaches its maximum value $\delta_0$, given by equation (2.74)

$$\bar{x} = -\frac{x_0}{L}, \quad \bar{\delta} = 0$$
$$\bar{x} = 1, \quad \bar{\delta} = 1$$

Using these two boundary conditions, equation (2.71) can be solved with a fourth order Runge Kutta method.

### 2.5.2 Maximum film thickness in vertically versus horizontally orientated filter element

Besides the distribution of the liquid along the plate, also the maximum film thickness, which is reached in the channels of the filter element is an important parameter with respect to the operation of the filter element. If the film gets too thick, there is a higher possibility that the channels of the filter element get blocked. This results in a decrease in filter efficiency, which is not wanted. It is therefore interesting to
determine the influence of the filter orientation on the maximum film thickness. For this purpose the ratio of the maximum film thickness in a vertically orientated filter element to the maximum film thickness in a horizontally placed filter element, in case the shear force can be neglected, is calculated.

For the vertical case the maximum film thickness is (equation (2.32))

\[ \delta_{0_{vert}}^{3} = \frac{\alpha L}{\beta} \]  

(2.78)

For the horizontal case without shear force the maximum film thickness is (equation (2.63))

\[ \delta_{0_{horz}}^{4} = \frac{\alpha L^2}{2} \]  

(2.79)

The ratio of \( \delta_{0_{vert}} \) to \( \delta_{0_{horz}} \) can now be calculated as

\[ \frac{\delta_{0_{vert}}}{\delta_{0_{horz}}} = \left( \frac{24\nu v_0 (\Omega^2 r)^3}{g^4 L^2} \right)^{\frac{1}{12}} \]  

(2.80)

Consider the example of the natural gas – water separator, as mentioned in section 2.5.1. A liquid film of water (\( \nu \approx 10^{-6} \text{ m}^2/\text{s} \)) is formed on a channel wall, with length \( L \) of 0.2 m. The plate rotates with \( \Omega^2 r = 1000 \text{g} \). The liquid load feeding the film has a velocity \( v_0 \) of \( 5 \cdot 10^{-4} \text{ m s}^{-1} \) towards the wall. In this case equation (2.80) gives for the ratio of \( \delta_{0_{vert}} \) to \( \delta_{0_{horz}} \) a value of about 1.3. This means that for this particular situation the orientation of the filter element has hardly any influence on the maximum film thickness. For each particular situation in which the interfacial shear force is negligible equation (2.80) predicts the most favorable orientation of the filter element with respect to minimum film thickness.

2.6 Closure

In this chapter, relations, which describe the development of the liquid film in the channels of the rotating filter element, are derived. With these relations the amount of liquid leaving the element on both sides is predicted and the parameters determining this amount are derived. The model is valid for small liquid loads, as inertia effects are disregarded and a boundary layer-type flow is considered.

Two filter configurations are regarded, a vertically orientated filter element and a horizontally orientated one. The liquid film in the vertically orientated filter element is subjected to a gravitational and a centrifugal force. The shear force of the gas at the gas-liquid interface is disregarded. In the horizontal case the liquid film is only subjected to a centrifugal force, as the gravitational force works in the same direction and is much smaller. For the horizontal configuration also the effect of the shear force on the film behavior is examined.

The model for the horizontally orientated filter element, only subjected to a centrifugal force, shows that the liquid film is forced equally to both sides of the channel.
In the middle of the channel a turning point is situated for which the liquid film velocity along the channel wall is zero. At this point the film thickness is maximal.

The other two models show a more or less similar behavior. The majority of the liquid is forced in the same direction as the force, which acts along the channel wall. For the vertically placed filter element this is the gravitational force and for the horizontally placed element this is the shear force. However, in both cases a small part of the liquid film at the beginning of the channel is forced in the opposite direction. This is due to the fact that in both cases the film thickness has a relatively large curvature at the beginning of the channel. Due to this strong curvature, the centrifugal force exerts a large force on the liquid film in this region. As a result part of the liquid is forced in the direction opposite to the direction of the gravitational or shear force (i.e. towards the beginning of the channel). As a consequence a turning point is situated at the beginning of the channel. At this point the liquid velocity along the channel wall is zero. Thus, although the centrifugal force is much larger than the force acting along the channel wall, it only causes a force in the direction opposite to the force acting along the wall through the slope of the film thickness. For small slopes the influence of the centrifugal force can be disregarded.

In the model it is assumed that the influx of liquid feeding the liquid film at the wall is evenly distributed along the length of the channel wall. This is generally not true, as in practise droplets with different sizes are dispersed in the gas stream. The larger droplets reach the channel walls sooner than the smaller droplets. Usually, the large-sized droplets prevail and as a result the mass flow rate towards the wall at the entrance region of the rotating filter element is much larger than at the exit. The current model can be extended to include varying influx over the length of the channel.

Furthermore, it is assumed that the liquid load is small. The implication of a small liquid load is that the Reynolds number based on the liquid film thickness is small (see section 2.4.1). This means that the liquid film can be treated as a laminar layer with negligible inertia effects, i.e. creeping flow. For a high liquid load the inertia terms can no longer be disregarded. The current model can be extended to be also valid for high liquid loads. A possible approach is to use a similarity transformation to solve the complete boundary layer solution to the same problem, like Sparrow and Gregg [64] did for the problem of laminar film condensation on a vertical plate.
Chapter 3

Conceptual design

3.1 Introduction

The prototype of the natural gas – water separator is based on fluid flow relations and previously developed Rotational Particle Separator (RPS) design principles [16]. The RPS design principles deal with the separation performance of the separator. Fluid flow relations are used to design the swirl generator, which generates the angular momentum to drive the filter element, and to find relations for the pressure losses in the main parts of the separator. In this chapter the methods and criteria for the design of the main components of the separator are given. Furthermore, the final design of the prototype is described and its theoretical operating performance is presented. First, however, condensation and droplet growth of the condensable vapor components in natural gas is regarded. This information is used to assess the necessity to include an expansion (and condensation) area in the design.

3.2 Design parameters

3.2.1 Condensation

The natural gas – water separator can be designed such that the separation unit is integrated with a turbine (figure 1.6). A nozzle will expand the gas in order to reduce its pressure, and a rotor will convert the reduction in pressure to electrical energy. When the natural gas containing the contaminating vapors passes the nozzle, the temperature of the gas drops and the vapors start to condensate. In the area between the turbine and the separator the condensation process must be such that the droplets created are large enough to be separated in the RPS. This requires proper dimensioning of the expansion area. In order to do this, information is required about the creation (nucleation) and growth of condensate droplets in natural gas.
Nucleation

The formation of droplets starts by nucleation. Nucleation can be either homogeneous or heterogeneous. Homogeneous nucleation is the formation of particles in a supersaturated vapor without the assistance of condensation nuclei or ions [1]. This process is also called self-nucleation. During this process, molecular clusters of all sizes are continuously agglomerating and evaporating. From the moment that a cluster is significantly larger than a certain critical size, droplet growth starts. Well-established clusters evolve, gaining new molecules and thereby growing into macroscopic droplets. Thus, homogeneous nucleation is a process of the formation of small droplets that are thermodynamically stable. A necessary condition is that the gas has to be brought into a state of supersaturation. In this supersaturated state, the formation of a liquid phase tends to decrease the free energy of the system. However, as condensation can only take place in the form of droplets, new droplet surface area has to be formed, which requires a positive surface energy. Due to the combined effect of bulk condensation and surface formation, there is an energy barrier in the formation of droplets. The free energy of droplet formation as a function of the droplet radius has a metastable maximum at a critical radius \( r_c \). Droplets smaller than this critical radius tend to evaporate. Droplets exceeding this critical radius tend to grow. This critical droplet size follows from the Classical Nucleation Theory (CNT) and for a pure vapor it is expressed by the Kelvin relation [24, 40]

\[
\frac{r_c}{\sigma} = \frac{2 \sigma M_l}{\rho_l R_{\text{univ}} T \ln S}
\]

where \( r_c \) is the critical radius of the droplet, \( \sigma \) the surface tension, \( M_l \) the liquid molar mass, \( R_{\text{univ}} \) the universal gas constant (\( = 8.314 \text{ J/mole K} \)), \( T \) the temperature, \( \rho_l \) the liquid mass density and \( S \) the saturation ratio. As mentioned above a state of supersaturation is necessary to form stable droplets. A gas-vapor mixture is supersaturated when its saturation ratio is greater than one. The saturation ratio for an ideal vapor is defined as [24, 40]

\[
S = \frac{P_v}{P_s} = \frac{y_v P}{P_s}
\]

where \( P_v \) is the vapor pressure, \( P_s \) the saturated vapor pressure, \( y_v \) the actual vapor fraction and \( P \) the system pressure. A vapor, coexisting in equilibrium with its own liquid, will exert a pressure \( P_v \) that is uniquely determined by the temperature of the system: \( P_v = P_v(T_0) \), and hence the saturation ratio equals one. However if the temperature suddenly decreases (say at constant pressure \( P_v \)), the vapor momentarily contains more molecules per unit volume than would be possible in the new equilibrium state at temperature \( T \), and is therefore supersaturated. Once supersaturated the system will search its way back to equilibrium.

If the saturation ratio is sufficiently large, the vapor itself can form clusters in the gas phase that are large enough to act as condensation nuclei: homogeneous nucleation takes place. However, if the saturation ratio is not too large, vapor will condense on condensation nuclei or ions. This process is called nucleated condensation or heterogeneous nucleation. Whereas homogeneous nucleation usually requires
saturation ratios of 2 – 10, heterogeneous nucleation can occur at a supersaturation of only a few percent [24]. The nucleation process will always follow the path along which the smallest amount of energy is necessary to build a stable cluster. Therefore, heterogeneous nucleation will only take place if the total change in interfacial energy is smaller than the change of interfacial energy in case of homogeneous nucleation.

In equation (3.2) it is assumed that the vapor can be considered as an ideal gas. For the non-ideal situation the saturated vapor pressure of a binary mixture, consisting of one vapor component and a carrier gas, is corrected with the enhancement factor $f_e$. It represents the increase of the partial saturated vapor pressure, which by definition equals the product of equilibrium molar vapor fraction and total pressure, with respect to the pure vapor state [40, 42]

$$f_e \equiv \frac{y_{eq}^v(P, T) P}{P_s(T)}$$ (3.3)

where $f_e$ is the enhancement factor and $y_{eq}^v$ the equilibrium vapor fraction. The saturation ratio for very small vapor fractions can be now be calculated as follows [40, 42]

$$S \approx \frac{y_v}{y_{eq}^v} = \frac{y_v P}{f_e(P, T) P_s(T)}$$ (3.4)

Thus if $S$ is large enough, homogeneous nucleation takes place and stable nuclei are formed from the vapor molecules. The rate of stable clusters formed per unit time and volume is defined by the nucleation rate $J$. The general expression for the homogeneous nucleation rate is [42, 43, 54]

$$J = K \exp \left( -\frac{\Delta G_n}{k_B T} \right)$$ (3.5)

where $\Delta G_n$ is the Gibbs free energy of formation of a cluster consisting of $n$ molecules, $k_B$ the Boltzmann’s constant and the prefactor $K$ is governed by the kinetic process of growth and evaporation. Nowadays there is a considerable agreement upon the form of the prefactor. Besides, its precise form is not too critical in calculations of $J$. For the formation energy, however, many models exist, which differ significantly. As the formation energy appears in the exponential, large deviations between the individual theoretical models are possible. The research on homogeneous nucleation started with the work of Volmer and Weber [69] and Becker and Döring [3]. They developed the Classical Nucleation Theory (CNT) for pure vapors. Since the first introduction of the CNT many modifications have been suggested. One of the corrections deals with the fact that CNT fails to satisfy the law of mass action. Removing this inconsistency results in the introduction of the term $1/S$ in the prefactor of the CNT nucleation rate expression. Nowadays the application of this correction factor is generally believed to be justified [40]. Another, often applied, modification concerns the so-called limiting consistency. In the CNT model the formation energy of monomers is not zero, although it should be in a consistent model valid for all $n$. There are several possibilities to restore consistency and the most widely used correction is the introduction of a dimensionless surface tension into the exponential form
of the free energy. The result of this is the so-called Internally Consistent Classical Theory (ICCT) [40]. The unary classical nucleation theory was extended by Reiss [57] to binary mixtures, the so-called Binary Classical Nucleation Theory (BCNT). In this model the formation energy is a function of the two vapor components present in the mixture and the nucleus. Later Staufer [66] improved the calculation of the prefactor $K$ and Wilemski [71] extended the model to be thermodynamically consistent. For a further discussion about the different nucleation theories, the reader is referred to Kashchiev [29].

Peeters [51] performed high pressure nucleation experiments in a pulse-expansion wave tube to study the nucleation and growth processes in natural gas. The experiments were performed with methane, containing water and/or n-nonane vapor. The choice for methane is evident, as it is the main component of natural gas. The vapour component water is chosen as this phase is abundant in natural gas reservoirs and n-nonane is chosen as being a typical hydrocarbon vapor. Both experiments were performed for the binary systems n-nonane in methane and water in methane as well as for the ternary mixture n-nonane and water in methane. The nucleation experiments were performed at two different conditions; 10 bar and 235 K, and 40 bar and 240 K. Besides for the water – methane system experiments at 25 bar and 235 K were performed. Experimental results from the binary systems were compared with the CNT, the ICCT and the BCNT model. The results from the ternary mixtures were compared with the sum of the experimentally determined binary nucleation rates. From this comparison it was concluded that n-nonane and water nucleate independently. When comparing the experimental nucleation results of the binary mixtures water – methane and n-nonane in methane with the theoretical models at low pressure (10 bar) only qualitative agreement is found. At high pressures the models fail to predict the nucleation rates for the n-nonane – methane system. For the water – methane system at 40 bar agreement is better but still only qualitative. Overall deviations between experimental and theoretical nucleation rates by a factor $10^4$ is not an exception.

Luijten [41] performed nucleation experiments with natural gas, directly received from the Dutch natural gas distribution system. These experiments were carried out for temperatures in the range of 220 – 240 K and pressures up to 25 bar. The nucleation data have been compared to the Binary Classical Nucleation Theory (BCNT). The model calculations do not quantitatively reproduce the experimental results, only qualitatively. The experimental data were also compared to an existing multicomponent nucleation model. Also in this case a large discrepancy between theory and experiments was found.

Based on the above mentioned results, it can be stated that the current nucleation models can not be used to accurately predict nucleation rates in a multicomponent mixture like natural gas. Especially at high pressures, there is a large disagreement between the theoretical and experimental results.
### 3.2 Design parameters

**Droplet growth**

After stable nuclei have been formed the droplets start to grow. Mass is transferred from the vapor molecules to the droplet and simultaneously energy (latent heat) is transferred to the gas phase. Two regimes can be distinguished, depending on the Knudsen number, $\text{Kn}$, which is defined as the ratio of the mean free path of a vapor molecule to the droplet diameter $[52, 55]$.

If the Knudsen number is large, the growth is dominated by the impingement rate of molecules onto the surface of the droplet and growth is described by gas kinetics. This regime (Knudsen regime) is typical for the earliest stage of the growth process, where droplets sizes are in the range of nanometers. For very small Knudsen numbers the growth is best described by the diffusion of vapor molecules through the carrier gas (continuum regime). Especially in situations in which the carrier gas pressure is high, Knudsen numbers are small due to small values of the mean free path. The equations describing the growth process in both limiting regimes are relatively well established $[40]$.

However, when droplets start to grow after the nucleation stage they will grow from a region of very large Knudsen numbers to very small Knudsen numbers, passing the intermediate region where $\text{Kn} = O(1)$. Several growth models describing droplet growth for all Knudsen numbers have been developed. Many of these models are based on the so-called flux matching method $[19, 21]$. This method assumes that the area around the droplet is surrounded by a Knudsen layer, where mass and heat transfer are governed by gas kinetics. This Knudsen layer has a thickness in the order of magnitude of the mean free path of the molecules. Outside the Knudsen layer, continuum relations describe the transfer rates. As steady state conditions are assumed, mass and heat fluxes in both regions can be equated. This evolves in a set of equations describing the growth process for all values of the Knudsen number. Peeters $[52]$ and Luijten $[40]$ have adopted the flux matching model of Young $[75]$ to make it applicable to droplet growth in high pressure gases by incorporating real gas effects. Furthermore, Peeters $[53, 55]$ included dissolution of the carrier gas into the droplet and the condensation of more than one vapor in the model.

Peeters $[53, 54]$ performed growth measurements for the binary systems n-nonane in methane and water in methane and for the ternary mixture n-nonane and water in methane. The experiments were performed at two different conditions; 11 bar and 242 K, and 44 bar and 247 K. He compared the experimental results with his flux matching model. In general there is a satisfactory agreement between experimental results and the theoretical model. Especially for the larger carrier gas pressure (and smaller vapor fractions) there is a good agreement between the predictions of the flux matching model and the experimental results.

Luijten $[41]$ also observed the growth of the vapor fractions present in the natural gas, directly received from the Dutch natural gas distribution system. Experiments were performed at temperatures between 230 and 245 K and pressures in the range of 5 – 25 bar. Most results showed a linear increase of the square of the droplet radius $r_d$ with time. This implies that droplet growth is diffusion controlled. Droplet growth of dilute vapor components in a non-condensing carrier gas in the continuum regime
can be described by

$$\frac{d r_n^2}{d t} = 2 \frac{\rho_g}{\rho_l} \sum D_i (y_i - y_i^{eq})$$  \hspace{1cm} (3.6)$$

where $D_i$ denotes the diffusion coefficient of component $i$ in the carrier gas, $\rho_g$ and $\rho_l$ are the molar densities of respectively the carrier gas and the liquid phase and $y_i$ and $y_i^{eq}$ are respectively the molar fraction and the equilibrium molar fraction of component $i$. Comparing the experimental results with equation (3.6) showed quantitative agreement. In conclusion it can be stated that droplet growth at high pressures ($> 10$ bar) can be well predicted by theoretical models.

Muitjens [47] derived that the final droplet radius $r_{max}$ in a binary mixture of one heavy hydrocarbon in a carrier gas is proportional to the number of nuclei present

$$r_{max} = \left[ \frac{3}{4 \pi N_0^{-1} \rho_l \rho_l} y_{eq}[S(0) - 1] + r_0^3 \right]^{\frac{1}{3}}$$  \hspace{1cm} (3.7)$$

where $N_0$ denotes the initial droplet number density, $S(0)$ the initial supersaturation ratio and $r_0$ the initial radius of the droplet. As the second term on the right hand side of equation (3.7) is many orders of magnitude lower than the first one, the final droplet radius is practically independent on the initial droplet radius.

**Integrating condensation area in separator**

In the expansion turbine a large pressure reduction takes place. (for example expanding from 300 to 80 bar). This leads to a high saturation ratio $S$. As a consequence homogeneous nucleation dominates over heterogeneous nucleation, leading to a large amount of small nuclei after expansion. In the next step, the condensation area, enough time must be given to these nuclei to grow in size, such that they can be separated from the gas stream by the RPS. Equation (3.7) shows that the final droplet radius after growth strongly depends on the initial droplet number density. The crucial parameter in the estimation of the final droplet radius after growth is therefore the droplet number density after expansion.

This is also illustrated by following example. For the binary system water – methane, the growth of water droplets is calculated for the conditions right after throttling; pressure 80 bar, temperature 340 K (temperature follows from isenthalpic change of state from 300 bar and 373 K (table 1.1) to 80 bar, see section 3.3.1) and four different initial number concentrations of nuclei present in the gas phase. Methane is chosen as the continuous phase, because methane is the main gaseous component of natural gas. Water is chosen as the impurity, as it is one of the main impurities and besides, the properties of water are well-known. The physical properties of methane and water are given in appendix A. A vapor fraction of 0.015 mol mol$^{-1}$ is assumed (corresponds to 1.5% mass percentage or 0.075% volume percentage, i.e. a small amount of condensable vapor). This vapor fraction corresponds with the equilibrium vapor fraction of water vapor in methane before throttling (at 300 bar and 373 K, calculated with equation (3.3)). From equation (3.4) it follows that for this situation
$S$ is about 4. The critical droplet size at this saturation ratio can be calculated with equation (3.1) and is of the order $1 \times 10^{-9}$ m.

Equations (3.6) and (3.7) are used for calculating the growth time and the maximal droplet radius after growth. Although equation (3.6) only applies to the continuum region, we can use this equation as in a high pressure carrier gas the Knudsen number is small, even for the earliest stages of the growth process. It is assumed that the droplets start growing with an initial radius of $0 \mu m$. This is a reasonable assumption as the second term on the right hand of equation (3.7) is usually many orders of magnitude lower than the first one. Besides, by assuming an initial radius of $0 \mu m$ the worst case scenario is considered. In figure 3.1 the result of the calculation is shown.

![Figure 3.1](image.png)

Figure 3.1. Droplet diameter versus growth time for different values of $N_0$ for the condensation of water in methane at 80 bar and 340 K.

It can indeed be seen that the amount of initial nuclei has a strong influence on the final diameter of the droplets after growth. A deviation in initial number density with a factor $10^5$ already results in a factor $10^2$ difference in final droplet diameter. The lower the number of nuclei present, the larger the final droplet radius and the more time the droplets need to achieve their maximum size. It can also be seen that for initial number concentrations of $1 \times 10^{15}$ and larger, the droplets are too small for the RPS to remove them from the gas stream (RPS is designed for $d_{p,100\%} = 1 \mu m$). It must be mentioned that in the calculations the temperature of the gas is assumed constant. For a low condensable vapor content this is acceptable as warming up of the gas due to condensation is negligible.
So far, however, an important physical mechanism is disregarded in the growth process; the coagulation of the droplets. Coagulation of particles (or droplets) is a process whereby particles collide with each other due to a relative motion between them and adhere to form larger particles. The net result is a continuous decrease in number concentration and an increase in particle size. The relative motion between the particles is normally due to Brownian motion. In this case the process is called thermal coagulation. Thermal coagulation is a spontaneous and ever-present phenomenon for particles. When the relative motion arises from external forces, such as gravity or electrical forces or from aerodynamic effects, the process is called kinematic coagulation. The simplest case of thermal coagulation is the coagulation of monodisperse spherical particles (all particles have the same diameter). If it is assumed that particles adhere at every collision (also called Smoluchowski coagulation), the rate of change in number concentration is equal to [24]

$$\frac{dN}{dt} = -4 \pi d_p D N^2$$  \hspace{1cm} (3.8)

where $N$ is the particle number concentration, $d_p$ the particle diameter and $D$ the diffusion coefficient of the particles. The diffusion coefficient is expressed by [24]

$$D = \frac{k_B T C_c}{3 \pi \eta_g d_p}$$  \hspace{1cm} (3.9)

where $k_B$ is the Boltzmann’s constant, $T$ the temperature, $C_c$ the Cunningham slip correction factor and $\eta_g$ the dynamic viscosity of the gas phase. The Cunningham slip correction factor for particles of 0.1 $\mu$m and larger is given by [24]

$$C_c = 1 + \frac{2.52 \lambda}{d_p}$$  \hspace{1cm} (3.10)

where $\lambda$ is the gas mean free path. For particles smaller than 0.1 $\mu$m the Cunningham slip correction is represented by the empirical relation [24]

$$C_c = 1 + \frac{\lambda}{d_p} \left[ 2.514 + 0.800 \exp \left( -0.55 \frac{d_p}{\lambda} \right) \right]$$  \hspace{1cm} (3.11)

Substituting equation (3.9) in equation (3.8) gives

$$\frac{dN}{dt} = -4 k_B T C_c \eta_g N^2 = -K N^2$$  \hspace{1cm} (3.12)

where $K$ is the coagulation coefficient. For large particles with negligible slip correction $K$ is independent of particle size ($C_c \simeq 1$). For small particles, however, $K$ increases as particle size decreases, because of the effect of the slip correction factor. For high pressure carrier gases the mean free path is small ($\lambda \ll d_p$). As a result $C_c$ will be of the order 1, also for small droplet sizes. The coagulation coefficient in a high pressure gas is therefore independent of the droplet diameter.
The number concentration as a function of time can be determined by integrating equation (3.12), assuming $K$ to be constant. The number concentration at time $t$ can now be calculated according to

$$N(t) = \frac{N_0}{1 + N_0 K t}$$

(3.13)

where $N_0$ is the initial number concentration.

As a direct consequence of the reduction in number concentration due to coagulation, the particle size increases. Under the assumption that the mass concentration of the particles remains constant, the increase in particle diameter with time is given by

$$d(t) = d_0 (1 + N_0 K t)^{1/3}$$

(3.14)

where $d_0$ is the initial particle diameter. From the above equation it follows that the higher the initial number concentration of particles, the larger the increase in particle diameter due to coagulation.

Until now only monodisperse thermal coagulation is regarded. Kinematic coagulation occurs as a result of relative particle motion caused by mechanisms other than Brownian motion. For example in turbulent flow, the coagulation process is influenced by the eddies produced. These eddies cause relative motion between particles and might enhance the coagulation process. Fuchs [20] derived the following ratio for turbulent to thermal coagulation

$$\frac{b d_p^6}{64 \pi D \left( \frac{\rho_g \epsilon}{\eta_g} \right)^{1/2}}$$

(3.15)

where $b$ is a constant of the order 10 and $\epsilon$ the energy dissipation per unit mass of gas (equation (3.29)). Generally, equation (3.15) predicts that turbulent coagulation is negligible for particles of 0.1 $\mu$m and very important for particles larger than 10 $\mu$m [20]. In the condensation area the flow may be turbulent. However as particle sizes are small, coagulation due to turbulence can be neglected. Also not regarded sofar is polydisperse coagulation, i.e. the coagulation of particles of different sizes. For simplicity however we will only regard monodisperse coagulation.

With the expressions given above, the growth of water droplets due to monodisperse thermal coagulation can be included in the condensation example mentioned above. In practice, the growth due to condensation and the growth due to coagulation are processes which occur simultaneously. In the current example, however, it is assumed that first the droplets grow due to condensation. As soon as the droplets reach their final radius given by equation (3.7), the coagulation process is started. In the calculations the Cunningham slip correction factor $C_s$ is assumed 1. The result of these calculations is given in figure 3.2.

Comparing figures 3.1 and 3.2, the following remarks can be made. For small time scales ($t < 1$ s) the two lowest initial number concentrations ($N_0=1\cdot10^{13}$ m$^{-3}$ and $N_0=1\cdot10^{15}$ m$^{-3}$) hardly show an increase in droplet diameter due to coagulation. This is due to the fact that for the time scales considered the initial concentrations are
Figure 3.2. Droplet diameter versus time for different values of $N_0$ for the system water – methane at 80 bar and 340 K. Droplet growth due to condensation and coagulation.

Almost equal to (or less than) the equilibrium concentration for coagulation. So, for low initial number concentrations and time scales smaller than several seconds, the growth of the droplets is entirely due to condensation. However, for the two highest initial number concentrations ($N_0=1 \times 10^{20} \text{ m}^{-3}$ and $N_0=1 \times 10^{25} \text{ m}^{-3}$) a considerable increase in droplet diameter due to coagulation is observed directly after condensation. For growth times up to several seconds the droplet diameter has increased from $10^{-8}$ m or less to about $10^{-6}$ m. Comparing the results for $N_0=1 \times 10^{20} \text{ m}^{-3}$ and $N_0=1 \times 10^{25} \text{ m}^{-3}$ also shows that the higher the initial number concentration, the faster the droplet grows due to coagulation. Furthermore, it can be seen that regardless of the initial number concentration, after a certain time all droplets reach a certain minimal size. This is due to the fact that after a given time has elapsed, the number concentration cannot exceed a certain maximal value regardless of the initial number concentration (results directly from equation (3.13)). In conclusion, the combination of condensation and coagulation shows that, regardless of the initial number concentration, after a certain time all droplets reach a diameter in the order of 1 $\mu$m or larger. This means that the condensation area can always be given the required length such that at the beginning of the separation area the particles have reached the critical droplet diameter of 1 $\mu$m.

In the RPS – turbine combination homogeneous nucleation will dominate heterogeneous nucleation, due to the high saturation ratio after expansion. This leads to a high initial number density of nuclei. For high pressures, however, the nucleation
rate is not quantitatively predictable (as indicated in the section **Nucleation**) and thus the initial number density cannot be accurately estimated. What we do know is that for high initial number concentrations the growth time scale to reach particle diameters in the order of 1 micron is determined by coagulation. The growth time required to attain droplets of 1 micron and larger by coagulation is of the order of several seconds. As the gas velocities through the separator are in the order of 20 m s\(^{-1}\) (volume flow of 0.65 m\(^3\) s\(^{-1}\) and maximum outer radius of 0.2 m, table 3.3), the condensation area must at least be 20 m long. However, the exact dimensions of the condensation area cannot be determined as the nucleation rate cannot be estimated accurately. Therefore, it was decided not to integrate the expansion step and the condensation area in the prototype. Instead a throttling valve is used to expand the gas flow.

In the next sections the design rules of the separator are given. With these design rules and the results, which were presented in this section, the natural gas – water separator is designed (section 3.3).

**3.2.2 Separation performance**

**Filter element**

In the filter element (figures 1.2 and 1.3) the fluid rotates as a rigid (solid) body and flows in an axial direction parallel to the rotation axis, such that a laminar flow exists [13]. As the droplets present in the fluid flow are very small in size, inertia forces acting on them are small. It is assumed that the droplets follow the streamlines of the fluid flow, except for the radial direction where, as a result of the centrifugal force, they move relatively to the fluid (figure 3.3). The radial droplet velocity \(v_p\) is calculated from the equilibrium between the centrifugal force, the buoyancy force and the Stokes drag force acting on the droplet. It follows that the radial particle velocity is given by [15]

\[
v_p = \frac{(\rho_p - \rho_f)d_p^2 \Omega^2 r}{18 \eta_f} \tag{3.16}
\]

where \(\rho_p\) is the dispersed phase density, \(\rho_f\) the fluid density, \(d_p\) the droplet diameter, \(\Omega\) the angular speed of the filter element, \(r\) the radial position of the channel relative to the rotation axis and \(\eta_f\) the dynamic viscosity of the fluid. Whether the droplets reach the outer wall of the channel depends, besides their radial velocity, on the residence time in the channel and the radial distance the droplets must travel. The smallest particle, which is able to reach the outer collecting wall with 100% probability, is the particle which in the available time moves over the entire height of the channel (figure 3.3). Assuming a uniform axial velocity of the fluid in the channels of the RPS, the radial velocity of such a particle is given by

\[
v_{p,100\%} = \frac{u_{fe} d_c}{L_c} \tag{3.17}
\]

where \(u_{fe}\) is the axial fluid velocity in the channels of the filter element, \(d_c\) the channel height and \(L_c\) the channel length. Equalling equation (3.16) and equation (3.17) gives
an expression for the smallest particle collected with 100% probability in a channel located at position \( r \) from the rotation axis

\[
d_{p,100\%} = \sqrt{\frac{18 \eta_f u_{fe} d_c}{(\rho_p - \rho_f) \Omega^2 r L_c}}
\]  

(3.18)

Provided that only \( r \) in equation (3.18) is decreased, it follows that \( d_{p,100\%} \) increases due to the fact that the centrifugal force on the particle decreases. This increase of \( d_{p,100\%} \) is avoided by designing the inlet and outlet configuration of the filter element such that the axial fluid velocity increases linearly with \( r \). The decrease of the centrifugal force with decreasing \( r \) is then compensated by a longer residence time in the channel. This results in an equal particle separation for all channels. Brouwers [13, 14] derives an expression, valid for the entire filter element, for the smallest droplet, which is just able to reach the outer wall with 100% probability in case of this optimal distribution of the axial flow

\[
d_{p,100\%} = \sqrt{\frac{27 \eta_f \phi_{fe} d_c}{(\rho_p - \rho_f) \Omega^2 L_o \pi (1 - \epsilon) (R_{o,fe} - \epsilon R_{i,fe})}}
\]  

(3.19)

where \( \phi_{fe} \) is the fluid volume flow through the filter element, \( \epsilon \) the reduction of the effective cross-sectional area of the element due to the wall thickness of the channels, \( R_{o,fe} \) the outer radius of the element and \( R_{i,fe} \) is the inner radius. It follows from
3.2 Design parameters

Equation (3.18) that particles entering a separation channel larger than \(d_{100}\) will be collected with 100% probability. However, also smaller particles are able to reach the collecting wall provided that the distance of the particle towards the wall is sufficiently small. Brouwers [14] derived expressions for the particle collection efficiency of an entire filter element for two axial flow distributions; a constant flow distribution over the channels and a flow distribution linearly proportional to \(r\) and two channel shapes; concentric rings and triangles. Besides, a parabolic velocity profile inside the channels is assumed. For a filter element composed of trian-
gularly shaped channels and a linear proportional flow distribution \(u_f \propto r\), the collection efficiency of the whole element is given by

\[
E = \begin{cases} 
1 & x \geq \sqrt{2} \\
\frac{1}{2}x^2 \left(1 - \frac{1}{4} \left(\frac{x^2}{2}\right)^{1/3}\right) & x \leq \sqrt{2}
\end{cases}
\tag{3.20}
\]

where \(x = \frac{d_p}{d_{100}}\).

The smallest fractions, which are separated in the RPS, move with a radial velocity, which is equal to the ratio of the channel height to the channel length times the axial fluid velocity, equation (3.17). In practical applications the ratio between the channel height and channel length is very small, in the order of \(10^{-2}\). Thus the droplets move with a radial velocity, which is only a few percent of the axial fluid velocity \(u_{fe}\). This means that the process of radial migration of the droplets to the channel walls is already disturbed when secondary flows of one percent in magnitude of the axial fluid velocity occur in planes perpendicular to the axial channel axis. To avoid these secondary flows, the flow in the channels of the filter element has to be laminar. It is known that for non-rotating channels such flow only occurs for limited value of the axial Reynolds number

\[
Re_{ax} = \frac{\rho_f u_f d_c}{\eta_f} < 2300
\tag{3.21}
\]

For rotating channels a much stronger condition must be fulfilled to satisfy stability, as rotation is an additional destabilization factor. In the case of a tube rotating around its own symmetry axis, not only condition (3.21) must be satisfied but also the rotational Reynolds number, defined as

\[
Re_\Omega = \frac{\rho_f \Omega d_c^2}{\eta_f}
\tag{3.22}
\]

should not be larger than 108 [44]. If however, \(Re_\Omega\) exceeds 108, the axial Reynolds number should become less than 166 [44], see figure 3.4. Brouwers [12] shows that these considerations are also valid for tubes rotating around an axis not coinciding with but parallel to their symmetry axis. A further consideration is that tubes should be parallel within a few degrees to prevent secondary flow due to Coriolis forces [12].
Figure 3.4. Stability limit of Hagen-Poiseuille flow in a circular pipe with solid-body rotation, after Mackrodt [44]. Notice that both Reynolds numbers are based on the channel radius $R$.

Pre-separator

To prevent blockage of the filter elements channels by coarse particles, large droplets and solid particles are separated from the flow in the pre-separator (figure 1.3). Although the streamlines in the pre-separator partly point to the inner radius, the particles and droplets move, as a result of centrifugal forces, in a radial direction opposite to the fluid stream. Droplets or particles, which have a radial velocity that is larger than the average radial velocity of the gas stream, are driven towards the outer radius of the pre-separator and are collected there. The diameter of the smallest particles or droplets, which are able to reach the wall, is determined analogously to equation (3.16)

$$d_{p, pre} = \sqrt{\frac{18 \eta_f v_{pre} r}{(\rho_p - \rho_f) w_p^2}}$$

(3.23)

where $v_{pre}$ is the average radial gas velocity in the pre-separator and $w_p$ is the tangential velocity of the particle at radius $r$.

The pre-separator can be considered as an axial cyclone. In the core of a cyclone the tangential velocity component can be approximated by a solid body rotation ($w/r = \text{constant}$). While at the outside of a cyclone the tangential velocity component comes close to a free vortex ($w r = \text{constant}$) [27].
The general expression for the tangential velocity component in case of a solid body rotation is \( w = \Omega r \). Substituting this expression in equation (3.23) gives the smallest droplet or particle diameter, which is separated in case the fluid flow in the pre-separator is regarded as a solid body rotation

\[
d_{p_{pre}} = \sqrt{\frac{18 \eta_f v_{pre}}{(\rho_p - \rho_f) \Omega^2 r}}
\]  

(3.24)

In case of a free vortex, the tangential velocity component is given by \( w = C_{\Gamma} / r \) with \( C_{\Gamma} = \Gamma / 2\pi \) [7] and \( \Gamma \) is the magnitude of the circulation. Substituting this expression in equation (3.23) gives the relation for the smallest droplet or particle diameter, which is separated in case of a free vortex

\[
d_{p_{pre}} = \sqrt{\frac{72 \pi^2 \eta_f v_{pre} r^3}{(\rho_p - \rho_f) \Gamma^2}}
\]  

(3.25)

Besides that the pre-separator prevents blockage of the filter element channels, the pre-separator is also designed such that the desired axial fluid velocity profile at the inlet of the filter element is created.

**Post-separator**

The post-separator is a tubular section located after the filter element (figure 1.3). It receives flows from all the individual filter element channels. The droplets, which reach the walls of the filter element, coalesce and form a liquid film. At the exit of the channels this liquid film breaks up into relatively large droplets. In the post-separation area the rotational motion of the gas forces these droplets towards the outer wall, where they are separated from the main flow. However, before the droplets reach the outer collecting wall, they may break-up into smaller droplets as a result of centrifugal forces and/or forces due to the turbulent motion of the fluid. As a result these small droplets may leave the separator together with the gas stream and this deteriorates the separation performance. Therefore, the minimum diameter of the droplets present in the post-separator area is an important quantity for the design of the post-separator.

Hinze [25] assumes that the fluctuations of the turbulent motion determine the size of the largest drops in a turbulent flow. For a low viscosity droplet in a turbulent flow he states that the condition of break-up is achieved when the dynamic pressure force becomes larger than the surface tension force. The dynamic pressure force due to the fluctuations of the turbulent flow is expressed as \( \rho w_f^2 \). Where \( w_f^2 \) is the average value of the square of the velocity difference between two points at a distance \( d_{max} \) and \( d_{max} \) is the maximum stable droplet size. The surface tension force, which tends to resist deformation, is given by \( \sigma / d \). Where \( \sigma \) is the surface tension of the liquid.

The dimensionless group representing the condition of break-up is the ratio of these two forces, known as the (critical) Weber number

\[
W_{c} = \frac{\rho_f w_f^2 d_{max}}{\sigma}
\]  

(3.26)
In the case of isotropic homogeneous turbulence, the main contributions to the kinetic energy are made by the fluctuations in the region of wavelengths where the Kolmogorov energy distribution law is valid. For this region the average value across the whole flow field of the square of velocity differences can be expressed by \[ (3.27) \]

\[ \overline{u'^2_f} \sim (\epsilon d)^{\frac{3}{7}} \]

where \( \epsilon \) represents the energy dissipation per unit mass and \( d \) the diameter of the droplet. It follows that the maximum stable droplet diameter in an isotropic, homogeneous turbulent flow is given by

\[ d_{\text{max}} = \left( \frac{\text{We}_c \sigma}{\rho_f} \right)^{\frac{3}{5}} \epsilon^{-\frac{2}{5}} \]

Relation (3.28) only holds if the viscous forces within the droplets can be neglected. This is the case when the diameter of the droplet is much larger than the internal length scale of local turbulence (usually called the Kolmogorov length scale). For high Reynolds number (turbulent) flows, these local turbulent regions are usually very small compared with the droplet size. This means that in turbulent flow the dynamic pressure forces are generally much larger than the viscous forces. Therefore it is assumed that the disruptive dynamic pressure forces and the stabilizing surface tension force are dominant in the process of droplet break-up in the post-separator.

For several situations, estimations of the critical Weber number have been made. Hinze estimated the critical Weber number to be 0.59 by using the experimental data of Clay [25]. Clay used an apparatus which consisted of two coaxial cylinders, of which the inner rotated [18]. The space between the cylinders was filled with two immiscible liquids of equal densities, one of which formed discrete drops. Sevik and Park [60] were able to predict theoretically the critical Weber number for the case of liquid droplets in a turbulent liquid of equal density. They found a value of \( \text{We}_c = 0.52 \), which is in good agreement with the result of Hinze. They also predicted \( \text{We}_c = 1.24 \) for air bubbles in a water jet, which they verified with experimental work.

In all the studies mentioned above the dispersed phase was of lower density than the continuous phase. This is not representative for the situation in which water and oil droplets have to be separated from natural gas. Lopes and Dukler [39] measured the value of the critical Weber number to be 0.194 for water droplets in a turbulent air stream. Kocamustafaogullari [33] confirmed these experimental results theoretically. For estimating the maximum stable droplet size in the post-separator, equation (3.28) will be used with \( \text{We}_c \) equal to the value found by Lopes and Dukler.

The energy dissipation per unit mass \( \epsilon \) can be estimated by \[ (3.29) \]

\[ \epsilon = \frac{2 f u_{\text{mean}}^3}{D_{\text{pipe}}} \]

where \( f \) is the friction factor, \( u_{\text{mean}} \) the mean velocity in the pipe and \( D_{\text{pipe}} \) the pipe diameter. The friction factor in the transition region, \( 2300 < \text{Re} < 10^5 \), can be estimated by the Blasius equation, provided that the tubes are smooth [8]

\[ f = 0.3164 \text{Re}^{-0.25} \]

(3.30)
3.2 Design parameters

where Re is based on the diameter of the tube. The friction factor in smooth pipes for higher Reynolds numbers, \(10^4 < \text{Re} < 10^6\), can be estimated by [5]

\[
f = 0.184 \text{Re}^{-0.2}
\]  

Since no relation was found in literature for the minimum droplet diameter in turbulent flow it is assumed that the minimum droplet diameter equals half the maximum stable droplet diameter.

Also in the duct of the swirl generator droplet break-up due to turbulence is possible. Therefore also for the design of the swirl generator equation (3.28) is used to calculate the minimum droplet size. Again the minimum droplet size is taken half of the maximum stable size.

The rotational motion of the fluid behind the filter element may also cause droplet break-up. The centrifugal force \(F_c\) on a droplet, which arises from this rotation (assuming solid body rotation), is calculated by

\[
F_c = \frac{1}{6} \pi d_p^3 \rho_p \Omega^2 R_{\text{post}}
\]  

where \(R_{\text{post}}\) is the outer diameter of the post-separator. The interfacial tension \(\sigma\) gives rise to a surface force that counteracts this deformation process. The surface force \(F_i\) of a particle is given by

\[
F_i = \pi d_p \sigma
\]  

The droplet size at which break-up occurs can be found by equalling the centrifugal and interfacial tension force

\[
d_{pc} = \sqrt{\frac{6 \sigma}{\rho_p \Omega^2 R_{\text{post}}}}
\]

The length necessary for the post-separator area to remove the droplets generated by turbulent and/or centrifugal break-up forces is found analogously to equation (3.19). The break-up process that predicts the smallest droplet size, either by centrifugal forces or by turbulent forces, determines the minimum length of the post-separation area.

3.2.3 Angular momentum

In the swirl generator the angular momentum required to drive the filter element is generated. At equilibrium the generated angular momentum equals the drive momentum and the momentum losses in other parts of the separator. The parts that contribute the most to these losses are the pre-separator, the gap between the filter element and the static housing and the bearings. This equilibrium is given by

\[
G_{\theta sg} = G_{\theta fe} + G_{\theta gap} + G_{\theta pre} + G_{\theta bearing}
\]  

where \(G_{\theta sg}\) is the angular momentum generated in the swirl generator and \(G_{\theta fe}\), \(G_{\theta gap}\), \(G_{\theta pre}\) and \(G_{\theta bearing}\) are respectively the loss of angular momentum in the
filter element, the gap between the filter element and the static housing, the pre-
separator and the bearings. The general definition of angular momentum is

\[ G_\theta = \int_A \rho_f u_f w_f r dA \]  (3.36)

where \( u_f \) denotes the axial fluid velocity, \( w_f \) the tangential component of the fluid ve-
locity and \( A \) the cross-sectional area. Applying this equation to calculate the angular
momentum of the flow leaving the filter element gives

\[ G_{\theta fe} = \int_{R_i fe}^{R_o fe} \rho_f u_{fe} w_{fe} r 2\pi r (1-\epsilon) dr = 2\pi \rho_f (1-\epsilon) \int_{R_i fe}^{R_o fe} w_{fe} u_{fe} r^2 dr \]  (3.37)

where \( u_{fe} \) and \( w_{fe} \) are respectively the axial and tangential velocity component at the
outlet of the filter element. The tangential velocity at the outlet of the filter element is
given by \( w_{fe} = \Omega r \). As desired for an optimal performance, the axial velocity should
be linearly distributed in the filter element (section 3.2.2). If we take \( u_{fe} = br \), with \( b \)
a constant, it follows that the volume flow through the filter element can be expressed
as

\[ \phi_{fe} = \int_A u dA = \int_{R_i fe}^{R_o fe} u (1-\epsilon) 2\pi r dr = \frac{2\pi (1-\epsilon) b}{3} \left( R_o^{3 fe} - R_i^{3 fe} \right) \]  (3.38)

Thus, the axial fluid velocity is given by

\[ u_{fe} = br = \frac{3 \phi_{fe}}{2\pi (1-\epsilon) \left( R_o^{3 fe} - R_i^{3 fe} \right)} r \]  (3.39)

Substitution of the equations for the tangential and axial velocity component at the
outlet of the filter element in equation (3.37) gives the generated angular momentum
in the filter element

\[ G_{\theta fe} = \frac{3 \rho_f \Omega \phi_{fe}}{5} \frac{R_o^{5 fe} - R_i^{5 fe}}{R_o^{3 fe} - R_i^{3 fe}} \]  (3.40)

In most practical designs the inner radius of the filter element equals half the outer
radius. This means that for those designs the angular momentum generated in the
filter element is given by

\[ G_{\theta fe} = \frac{93 \rho_f \Omega \phi_{fe}}{140} R_o^{2 fe} \]  (3.41)

In Li and Tomata [36] a relation is given for the axial decay of angular momentum
in a hydraulically smooth tube.

\[ 10 \log \frac{G_\theta}{G_{\theta 0}} = -0.01605 x^{0.8} \]  (3.42)
3.2 Design parameters

where $x^*$ denotes the ratio between the pipe length and hydraulic diameter of the tube, $G_\theta$ the actual angular momentum and $G_{\theta 0}$ the initial angular momentum. With this relation the loss of momentum in the pre-separator can be estimated as

$$G_{\theta \text{pre}} = G_{\theta \text{sg}} (1 - 10^{-0.01605} x^{0.8})$$

(3.43)

Also in the post-separator angular momentum is dissipated as a result of wall friction by the fluid. However the loss of momentum does not have to be taken into account as it has no influence on the required angular momentum to drive the filter element.

The momentum loss in the gap between the filter element and the housing can be determined from the momentum required to turn a cylinder in a static housing [58]

$$G_{\theta \text{gap}} = C_m 0.5 \pi \rho_f \Omega^2 R_{o e}^4 L_c$$

(3.44)

where $C_m$ represents the torque coefficient of the inner rotating cylinder. For a laminar Couette flow ($Ta < 25$) the torque coefficient equals $C_m = 0.67 / Ta$, for a laminar flow with Taylor vortices ($25 < Ta < 400$), $C_m = 0.194 Ta^{-0.58}$ and for a turbulent flow region ($Ta > 400$) the torque coefficient is given by $C_m = 0.019886 Ta^{-0.2}$. The Taylor number $Ta$ is defined as

$$Ta = \frac{\rho_f \Omega \eta_f R_1^2}{s_g^{3/2}}$$

(3.45)

where $s_g$ denotes the gap size between the rotating filter element and the static housing.

The loss of angular momentum in the bearings due to friction is dependent on the type of bearing. The expression for the angular momentum generated in the swirl generator is derived from equation (3.36). In this equation $u_f$ and $w_f$ must be replaced by respectively the axial velocity $u_{\text{sg}}$ and the tangential velocity $w_{\text{sg}}$ at the exit of the swirl generator. In the swirl generator the flow is accelerated in a constant area duct. The axial velocity at the exit of this duct is calculated by using flow relations of an ideal gas with viscous effects in a duct of constant cross-sectional area as described by Owczarek [50], see appendix B (for incompressible gas flows, as is the case in the current design, the axial velocity can also be calculated by considering conservation of mass). At the exit of the duct, static vanes give the flow the required tangential velocity. This tangential velocity is prescribed by the blade angle $\alpha$ (figure 3.5)

$$w_{\text{sg}} = u_{\text{sg}} \tan(\alpha)$$

(3.46)

Furthermore a fixed ratio of inner to outer radius of the swirl generator is assumed $R_{i \text{sg}} = b_{\text{sg}} R_{o \text{sg}}$. Substituting these expressions in equation (3.36) results in

$$G_{\theta \text{sg}} = \frac{2 \pi \rho_f \tan(\alpha) u_{\text{sg}}^3 R_{o \text{sg}}^3}{3} (1 - b_{\text{sg}}^3)$$

(3.47)

In deriving the above expressions for the generated and dissipated momentum in the various parts of the separator single phase flow is considered. In practice the
fluid flow through the separator is not single phase, as the gas is contaminated with different liquid and possibly solid phases. However, the amount of these contaminants is small (table 1.1) such that single phase equations can be used in deriving the design equations for the swirl generator.

Acceleration of filter element

For practical applications the acceleration time of the filter element must be as short as possible, since the element only works properly at its design speed. In order to estimate the order of magnitude of the acceleration time, rotational speed of the filter element during the start-up period is modelled. In this model it is assumed that the full (design) flow rate is applied at once. In practice however this is not an option because high inertia forces may damage the separator when the full flow rate is applied at once.

The model is based on the equilibrium between the generated angular momentum and the required momentum corrected for the moment of inertia \( J \) of the rotating parts

\[
G_{\theta sg} = G_{\theta fc}(\Omega) + G_{\theta gap}(\Omega) + G_{\theta pre} + G_{\theta bearing}(\Omega) + J \frac{d\Omega}{dt} \quad (3.48)
\]

From equation (3.48) it follows that the angular acceleration is given by

\[
\frac{d\Omega}{dt} = -\frac{G_{\theta fc} + G_{\theta gap} + G_{\theta bearing}(\Omega)}{J} + \frac{G_{\theta sg} - G_{\theta pre}}{J} \quad (3.49)
\]
3.2 Design parameters

The moment of inertia of the rotating parts, the shaft and the filter element, can respectively be calculated by

\[
J_{\text{shaft}} = \frac{1}{2} m_{\text{shaft}} D_{\text{shaft}}^2
\]
\[
J_{fe} = \frac{1}{2} m_{fe} (D_{i,fe} - D_{o,fe})^2
\]

(3.50)

3.2.4 Pressure loss

In consideration of energy consumption, it is preferred to keep the pressure drop over the separator as small as possible. The parts of the separator that contribute most to the total pressure loss are the filter element and the swirl generator. In both parts pressure drop is caused by friction and by changes in the direction of the fluid velocity.

In the filter element the pressure drop due to friction at the channel walls, assuming circular channels with diameter \(d_c\), is given by [58]

\[
\Delta P_{fe} = \left( f \frac{L_c}{d_c} + \xi \right) \frac{1}{2} \rho_f u_{fe}^2
\]

(3.51)

where \(f\) is the friction factor and \(\xi\) represents the additional pressure drop at the entrance region of the duct where the transition to a parabolic velocity profile takes place. According to Schlichting [58] the factor \(\xi\) equals 1.16 for circular channels and fully developed flow. Experimental results vary from 1.20 to 1.32 [61]. For a completely developed laminar Poiseuille flow the friction coefficient \(f\) for circular channels equals 64/Re [8], where the Reynolds number Re is based on the diameter of channel. For laminar flow through non-circular cross sections relation (3.51) can be used, provided that the hydraulic diameter of the channel is substituted for \(d_c\) and the correct friction factor is used. In the filter element the channels have the shape of triangles or more specifically sinusoids. Brouwers [14] derived expressions for the parabolic axial velocity profiles in such channels assuming that the width of the channels is much larger than the height as is the case in the filter element of the RPS. Based on these velocity profiles, a force balance reveals that for triangular channels the friction coefficient equals 48/Re and for sinusoidal channels \(f\) equals 38.4/Re, see appendix C. In table 3.1 these results are summarized. Also the factor \(\xi\) for the three channel shapes, in case of a fully developed laminar flow, is given [61]. The values for the triangularly and sinusoidally shaped channels only apply to channels for which the width is much larger than the height. The hydraulic diameter for those channel geometries is equal to the maximum height of the channel (appendix C).

Table 3.1. Friction factor \(f\) and \(\xi\)-value for fully developed laminar flow in different channel geometries [8, 61].

<table>
<thead>
<tr>
<th></th>
<th>circle</th>
<th>triangle</th>
<th>sinus</th>
</tr>
</thead>
<tbody>
<tr>
<td>(f)</td>
<td>64/Re</td>
<td>48/Re</td>
<td>38.4/Re</td>
</tr>
<tr>
<td>(\xi)</td>
<td>1.16</td>
<td>2.971</td>
<td>2.271</td>
</tr>
</tbody>
</table>

The friction coefficient for turbulent flows in circular channels is derived from the Moody diagram [7] or in the range of 2300 < Re < 10^5 the Blasius equation,
equation (3.30), can be used, provided that the tubes are smooth. For turbulent flows through channels of other cross sections it is proved empirically that the Moody diagram or equation (3.30) can be used to define the friction coefficient, provided that the hydraulic diameter of these channels is substituted for \(d_c\). In calculating the pressure loss over the filter element due to friction, \(u_{fe}\) is taken equal to the mean axial fluid velocity through the filter element.

Besides friction, pressure loss also results from the development of a free vortex in the pre-separator to a solid body rotation in the filter element. In the current analysis the flow in the pre-separator is considered as a free vortex. This is not entirely correct as in practice, the fluid in the core of the pre-separator behaves as a solid body. The pressure loss due to the difference in tangential velocity component prior to and in the filter element is estimated from the conservation of mass and momentum over the filter element. The pressure drop due to different tangential velocities follows from the conservation of momentum

\[
\Delta P_{\Delta w} = \rho_f \left( \langle \tau_{pre}^2 \rangle - \langle \tau_{fe}^2 \rangle \right)
\]

where \(\tau_{pre}^2\) and \(\tau_{fe}^2\) denote respectively the mean tangential velocities in the pre-separator and in the filter element. In deriving this expression it is assumed that the cross-sectional area of the pre-separator and the filter element are equal and the density of the fluid is constant. The tangential velocity of a free vortex is given by \(C \Gamma / r\) with \(C \Gamma = \Gamma / 2\pi\) and \(\Gamma\) is the magnitude of the circulation [7]. The tangential velocity in a solid body rotation is \(\Omega r\) [27]. Integration of these tangential velocity distributions over the cross-section of the filter element and substitution in equation (3.52) leads to

\[
\Delta P_{\Delta w} = \rho_f \left( \frac{2C^2}{R_{fe}^2} \ln \left( \frac{R_{fe}}{R_{pre}} \right) - \frac{1}{2} \Omega^2 (R_{fe}^2 + R_{pre}^2) \right)
\]

When conservation of mass is considered the unknown \(C \Gamma\) results

\[
C \Gamma = \frac{\Omega (R_{fe}^3 - R_{pre}^3)}{3 (R_{fe}^2 - R_{pre}^2)}
\]

The pressure drop caused by introducing a swirl component in the flow can be calculated as

\[
\Delta P_{sg} = \frac{1}{2} \rho_f u_{sg}^2 = \frac{1}{2} \rho_f (u_{sg} \tan(\alpha))^2
\]

where \(\alpha\) is the blade angle of the vanes.

All the equations given in this section are valid for single phase flow. Although the fluid flow through the separator for the intended application is not single phase, the amount of liquid phase is small, such that the single phase equations may be used for the pressure drop analysis.
### 3.2 Design parameters

#### Gap between filter element and housing

The size of the gap between the filter element and its static housing must also be determined. On the one hand this gap must not be too small as the angular momentum lost in the gap (equation (3.44)) should be limited. On the other hand the height of the gap should be small enough to minimize leak flow through the gap. In the current design, the maximal allowed gap size is limited to the size for which the pressure drop over the gap equals the pressure drop over the channels of the filter element

\[
\Delta P_{\text{gap}} = \left( f_{\text{gap}} \frac{L_{\text{gap}}}{D_{h \text{ gap}}} + \xi \right) \frac{1}{2} \rho_f u_{\text{gap}}^2 = \left( f_{\text{c}} \frac{L_{\text{c}}}{d_{\text{c}}} + \xi \right) \frac{1}{2} \rho_f u_{\text{fe}}^2 \tag{3.56}
\]

where \( D_{h \text{ gap}} \) is the hydraulic diameter of the gap. When we assume that the pressure loss due to entrance effects is the same for both channels, and that the velocity through the gap \( u_{\text{gap}} \) and the channels of the filter element \( u_{\text{fe}} \) are equal, it follows that

\[
D_{h \text{ gap}} = \frac{f_{\text{gap}}}{f_{\text{c}}} d_{\text{c}} \tag{3.57}
\]

For a laminar flow the friction coefficient for the channels of the filter element is given by 64/Re if circular channels are assumed. For the gap, whose inner wall is rotating and outer wall not, Yamada [73] finds that the friction coefficient for laminar flow is larger than

\[
f_{\text{gap lam}} \geq \frac{48}{\text{Re}_{\text{ax gap}}} \tag{3.58}
\]

where \( \text{Re}_{\text{ax gap}} \) is the axial Reynolds number in the gap defined by

\[
\text{Re}_{\text{ax gap}} = \frac{\rho_f u_{\text{gap}} s_{\text{gap}}}{\eta_f} \tag{3.59}
\]

where \( u_{\text{gap}} \) is the fluid velocity in the gap, which is assumed equal to the fluid velocity inside the filter element channels and \( s_{\text{gap}} \) is equal to \( D_{h \text{ gap}}/2 \). Substituting the equations for \( f_{\text{c}} \) and \( f_{\text{gap}} \) in equation (3.57) gives the maximum gap size in case of laminar flow

\[
s_{\text{gap lam}} = 0.612 d_{\text{c}} \tag{3.60}
\]

For a turbulent flow (transition region) the friction coefficient in the channels of the filter element is given by equation (3.30). For the gap the friction coefficient in the case of turbulent flow is larger than [73]

\[
f_{\text{gap turb}} \geq 0.26 \text{Re}_{\text{ax gap}}^{-0.24} \tag{3.61}
\]

From these relations it follows that for a turbulent flow the maximum gap size is

\[
s_{\text{gap turb}} = \left( 0.13 \left( \frac{\rho_f u_{\text{fe}}}{\eta_f} \right)^{-0.24} \right)^{\frac{1}{0.3164 \text{Re}_{\text{ax fe}}^{-0.25} d_{\text{c}}}} \tag{3.62}
\]

After the filter element is designed, the maximum allowed gap size between the element and the static housing follows from equation (3.60) in case the flow is laminar or from equation (3.62) in case the flow is turbulent.
3.3 Design

3.3.1 Design criteria

Now the general equations for the design of the separator are known, a specific design of the prototype can be made. For this we have to state the design criteria. First of all the fluid properties should be known as they have a large influence on all three design parameters; separation efficiency, pressure drop and required angular momentum.

It was decided not to integrate the turbine and the expansion area in the separator. Instead, pressure reduction takes place by using a throttling valve (section 3.2.1). It is assumed that after throttling the pressure is reduced to 80 bar (the conditions before throttling are stated in table 1.1). Throttling is an isenthalpic process, thus the temperature after throttling can be calculated by regarding an isenthalpic change of state. We used the program FYSCAL [22] to calculate the temperature change and properties of natural gas. With this program the physical and thermodynamic properties of different natural gas compositions can be calculated. We used the composition of Slochteren natural gas, which is representative for the gas found near Groningen, the Netherlands. An isenthalpic change of state from 300 bar and 373 K to 80 bar, results in a temperature of 340 K. The density of Slochteren gas calculated with FYSCAL at 80 bar and 340 K is about 50 kg m$^{-3}$. The dynamic viscosity at this condition is about $1.5 \cdot 10^{-5}$ Pa s. For the entrained phase the properties of water are used (appendix A.4) as they are well-known for the operating conditions and the majority of the contaminants are water droplets [59].

The performance of the prototype is tested in a test loop at the Technical University of Eindhoven and in a test loop at CDS Engineering (section 4.2). The conditions in both tests loops are given in table 3.2. The test loop at CDS Engineering is filled with SF6, a high density gas. The absolute pressure of the gas in the test loop is maximal 8 bar and the temperature of the gas is about 300 K. The density and the dynamic viscosity of SF6 at these conditions are respectively 46.8 kg m$^{-3}$ and $1.55 \cdot 10^{-5}$ Pa s [72]. This indicates that the CDS test loop is suitable for the simulation of field conditions.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Natural gas</td>
<td>SF6</td>
<td>Air</td>
</tr>
<tr>
<td>Pressure (bar)</td>
<td>80</td>
<td>8</td>
<td>1.2</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>340</td>
<td>300</td>
<td>283</td>
</tr>
<tr>
<td>Fluid density (kg m$^{-3}$)</td>
<td>50</td>
<td>46.8</td>
<td>1.4</td>
</tr>
<tr>
<td>Dynamic viscosity (Pa s)</td>
<td>$1.5 \cdot 10^{-5}$</td>
<td>$1.55 \cdot 10^{-5}$</td>
<td>$1.8 \cdot 10^{-5}$</td>
</tr>
</tbody>
</table>

At a typical gas field 0.65 m$^3$ s$^{-1}$ of natural gas is extracted at 80 bar (table 1.1). The prototype is designed for this volume flow. The amount of liquid contaminants in the gas before the separator is assumed to be between 0.1 – 1% volume fraction, as bulk separators have already separated the major part of the contaminants. The maximum pressure drop, allowed over the separator is 1 bar, because the test loop at
Table 3.3. Design parameters for the RPS-based separator, which is designed to separate water droplets from natural gas under high pressure.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{p,100%}$</td>
<td>1 $\mu$m</td>
</tr>
<tr>
<td>$D_o$</td>
<td>$\sim$ 0.2 m</td>
</tr>
<tr>
<td>Ratio of $D_i$ to $D_o$</td>
<td>0.5</td>
</tr>
<tr>
<td>$L_c$</td>
<td>$\leq$ 0.18 m</td>
</tr>
<tr>
<td>Volume flow gas</td>
<td>0.65 m$^3$ s$^{-1}$ ($\rho_g \simeq 50$ kg m$^{-3}$)</td>
</tr>
<tr>
<td>Volume fraction contaminants</td>
<td>0.1 – 1%</td>
</tr>
<tr>
<td>Pressure</td>
<td>80 bar</td>
</tr>
<tr>
<td>Temperature</td>
<td>340 K</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>$&lt; 1$ bar</td>
</tr>
</tbody>
</table>

CDS Engineering enforces this restriction (for field conditions there is no restriction on the pressure drop as there is enough pressure available).

With current techniques like settling tanks, particles down to 5 micron can be captured with 50% probability ($d_{p,50\%}$), section 1.4. This means that also the prototype should be able to achieve this efficiency and preferable remove even smaller particles. As design parameter for the filter element of the prototype, a $d_{p,100\%}$ of 1 micron is taken (table 1.1).

Besides the criteria concerning the operating conditions, there are also demands on the dimensions of the separator. As was explained in section 1.1, the separator has to be designed as compact as possible. In agreement with CDS Engineering the outer diameter of the separator was restricted to 8 inches ($= 0.2032$ m) [59]. Fabrication processes limit the length of the filter element to 180 mm. The ratio between inner and outer diameter of the filter element is fixed at 0.5. This value results from two contradicting effects. On the one hand the inner diameter of the filter element should be as small as possible to increase the flow area and thus the volume flow through the separator. On the other hand the efficiency of the element deteriorates for smaller radii (equation (3.18)). From practical experience with the RPS a ratio between inner and outer diameter of 0.5 has proven to perform satisfactorily [16].

It is preferred that the flow inside the filter element is laminar (section 3.2.2). The whole separator is constructed out of stainless steel as this is best suitable in the offshore industry. All design parameters for the design of the natural gas – water separator are given in table 3.3.

### 3.3.2 Design process

In figure 3.6 the final construction drawing of the whole prototype is shown, with the different parts of the separator tagged. In the next sections, the design process of these parts is discussed. The construction drawings of the different parts are included in appendix D.
Filter element

For the design of the filter element two parameters are important: pressure drop over the entire separator and the droplet diameter which is separated with 100% probability. For the current design pressure drop is limited to 1 bar and the $d_{p,100\%}$ is set at 1 micron (section 3.3.1).

The pressure drop constraint of 1 bar applies to the whole separator. During the design process, the pressure drop over the filter element due to friction, equation (3.51), and the dynamic pressure loss over the swirl generator, equation (3.55), are set to a maximum of 1 bar. Only these two pressure losses are considered as the other two losses, i.e. pressure loss due to friction in swirl generator and pressure loss due to the tangential velocity difference between pre-separator and filter element, are negligible compared to the former two. The pressure drop restraint can however not be realized with an outer diameter smaller than 8 inches ($\approx 0.2$ m). The minimal outer diameter for which the pressure drop is limited to 1 bar at a flow rate of 0.65 m$^3$ s$^{-1}$ is 0.24 m. Due to the high flow rate and the restrictions on pressure drop, outer diameter and length of the filter element, it was not possible to design a filter element with a lower $d_{p,100\%}$ than 2 micron. An outer diameter of 0.24 m is chosen, as this is the most compact design possible. The final dimensions of and flow conditions in the filter element are given in table 3.4.

During the design process of the filter element it appeared to be impossible to design the element such that it was compact ($D_0 \approx 8$ inches) and at the same time
Table 3.4. Final dimensions and flow conditions of the filter element at the operating conditions given in table 3.2.

<table>
<thead>
<tr>
<th></th>
<th>SF6 loop</th>
<th>Field conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume flow</td>
<td>0.65 m³ s⁻¹</td>
<td>0.65 m³ s⁻¹</td>
</tr>
<tr>
<td>Re_{ax}</td>
<td>64000</td>
<td>71000</td>
</tr>
<tr>
<td>ReΩ</td>
<td>970</td>
<td>950</td>
</tr>
<tr>
<td>D_o</td>
<td>0.24 m</td>
<td>0.24 m</td>
</tr>
<tr>
<td>D_i</td>
<td>0.12 m</td>
<td>0.12</td>
</tr>
<tr>
<td>L_c</td>
<td>0.18 m</td>
<td>0.18 m</td>
</tr>
<tr>
<td>n</td>
<td>3000 min⁻¹</td>
<td>2700 min⁻¹</td>
</tr>
<tr>
<td>u_f</td>
<td>21 m s⁻¹</td>
<td>21 m s⁻¹</td>
</tr>
<tr>
<td>w(R₀)</td>
<td>38 m s⁻¹</td>
<td>34 m s⁻¹</td>
</tr>
<tr>
<td>d_{p100%}</td>
<td>2 µm</td>
<td>2 µm</td>
</tr>
<tr>
<td>ΔP_fₑ</td>
<td>0.50 bar</td>
<td>0.52 bar</td>
</tr>
<tr>
<td>ΔP_{Δw}</td>
<td>280 Pa</td>
<td>240 Pa</td>
</tr>
</tbody>
</table>

fulfill the requirement that the flow through its channels remained laminar (ReΩ > 108 ∪ Re_{ax} < 166). Due to the high flow rate and high fluid density, the axial Reynolds number is higher than 2300 and ReΩ could not be restricted to 188 as the angular speed of the filter element must be high enough to separate particles in the order of 1 µm from the gas flow (table 3.4). The demand of laminar flow was therefore rejected. A consequence of turbulent channel flow is that the radial migration of the droplets towards the channel walls may be disturbed (as explained in section 3.2.2). This results in a decrease in the filter efficiency.

Pre-separator

The in- and outlet diameters of the pre-separator are prescribed by the design of the filter element and the swirl generator. The length of the pre-separator should be large enough to remove those particles, which could cause blockage of the filter element channels. The pre-separator of the prototype has a length of 50 mm and a mean radius of about 70 mm. With this design water droplets of about 20 micron and larger are separated in the pre-separator if a solid body rotation is assumed (equation (3.24) with \( r \) equal to the inner diameter of the filter element). In case the gas flow in the pre-separator is regarded as a free vortex, also water droplets of approximately 20 micron and larger are separated from the gas stream (equation (3.25) with \( r \) equal to the outer diameter of the filter element and \( \Gamma \) calculated with equation (3.54)). For sand and other dust particles this diameter is even smaller, as they have a higher density. Solid particles, which predominately originate from mechanical break-up processes, typically have a diameter of 20 micron or larger (appendix E). This means that the pre-separator is capable to remove most of these particles.
Gap between filter element and housing

For the current design the maximum allowed gap size calculated by equation (3.62) is 0.5 mm. As mentioned in section 3.2.4 the height of the gap should be small to minimize leak flow but on the other hand be large to limit dissipated angular momentum in the gap. In figure 3.7 the leak flow in percentage of the total flow through the separator and the lost angular momentum in the gap are plotted versus the gap size for the current prototype. The leak flow is calculated as the product of cross-sectional area of the gap and average axial fluid velocity for the field conditions. The loss of angular momentum in the gap is calculated with equation (3.44). For a gap size $s_g$ of 0.5 mm, figure 3.7 shows that the leak flow is approximately 1% of the total flow and the dissipated angular momentum is about 0.04 Nm. As these are acceptable values a gap size of 0.5 mm is taken for the design of the prototype.

Shaft and bearings

The most important quantities for the design of the shaft are the bending stresses and the maximal deflection. For the choice of the bearings supporting the shaft, the reaction forces in the support joints must be calculated. In order to calculate these quantities the Finite Element Method Program 'Analysis for Windows' version 1.11 was used. The shaft is represented as a solid cylinder with a constant diameter of 120 mm and a length of 600 mm. The filter element is modelled by loading the shaft at the position of the element with a uniform load equal to the mass of the filter element, as if it were solid. In this way both the shaft and the filter element are modelled such that stresses and deflection are maximal. The shaft is supported at the position at which the bearings are mounted. Two ball bearings on either side of the shaft are used for this purpose. Both bearings will be loaded in radial direction, due to the weight of the shaft and the filter element. In addition, one bearing, the
one situated downstream of the filter element, is loaded in axial direction, due to the pressure drop over the element. Ball bearings are chosen because of their low friction and little maintenance.

The calculated maximal bending stress in the shaft is 0.58 MPa and the maximal deflection of the shaft is 0.002 mm. As the bending stress is far below the maximum allowed value for stainless steel and the deflection is much less than the gap size between filter element and the housing, we conclude that the design is safe. The reaction force on both bearings is about 500 N. As the maximal pressure difference over the filter element is about 0.5 bar (table 3.4), the force due to this pressure loss is about 2500 N. Based on these conditions two bearings are chosen. For the bearing situated upstream of the filter element a groove ball bearing (type 6010, SKF catalogue [62]) is used and for the bearing situated downstream of the filter element an angular-contact ball bearing (type 7310 BE, SKF catalogue [62]).

**Swirl generator**

The swirl generator must be designed such that the required angular momentum to drive the filter element and to overcome the losses of angular momentum in the bearings, pre-separator and gap between filter element and the housing is generated (equation (3.35)). In order to generate the required momentum, the gas flow is first accelerated in a constant area duct and subsequently a swirl is induced by static vanes. The height and length of the duct and the blade angle of vanes are the most important design parameters. With the aid of the equations given in section 3.2.3 and appendix B and the dimensions of the filter element, pre-separator, gap and the losses of angular momentum of the bearings given by the SKF catalogue [62] it was calculated that for a duct length of 50 mm, a duct height of 54.5 mm and blade angles of 50 degrees [59] the required angular momentum is generated (in figure 3.5 the geometry of the vanes is depicted. Straight vanes were used as pressure loss was not crucial in the design.). In the design the total duct length is taken twice as long, as the blades also have a length of 50 mm. A schematic representation of the swirl generator is given in figure 3.8.

The pressure loss in the swirl generator at the design conditions is 0.2 bar, of which 3500 Pa is due to friction in the duct.

The maximum stable droplet size in the swirl generator calculated according to equation (3.28) is approximately 400 micron and according to our assumption, the minimum droplet size is therefore 200 micron. As this droplet size is much larger than the average droplet size of the particles to be separated, it can be assumed that no break-up occurs due to turbulence in the swirl generator.

**Post-separator**

Centrifugal and/or turbulent forces acting on the liquid droplets, which are created at the exit of the filter element, may lead to break-up of these droplets, as explained in section 3.2.2. The droplet size at which break-up occurs due to centrifugal forces (equation (3.34)) is about 180 micron. The minimum droplet size due to turbulence (equation (3.28)) is about 200 micron. To separate droplets of these sizes, a minimal
length of 5 mm is required. For security reasons the post-separator is designed with a length of 50 mm.

To avoid re-entrainment of liquid droplets in the gas flow the outer wall of the filter element is extended into the post-separation area (figure 3.9). This reduces the shear stresses on the liquid film at the outer wall of the post-separator.

Figure 3.8. Three-dimensional construction drawing of the swirl generator.

Figure 3.9. Extended wall of the filter element.
Prototype

A schematic representation of the prototype is given in figure 3.10. In order to sustain pressures up to 80 bar a thick-walled tube, with a wall thickness of 0.055 m, is used for the housing. The whole separator is made out of stainless steel. The prototype has an outer diameter of 0.36 m and a total length of about 0.6 m (figure 3.6). For comparison, settling tanks have a typical length of 10 m and a outer diameter of 2 m [59].

Figure 3.10. Exploded view of the prototype of the natural gas – water separator suitable for the removal of small sized liquid droplet from natural gas under high pressure.
3.4 Theoretical performance of the prototype

To predict the hydrodynamic and separation performance of the separator at other flow rates than the design flow rate a numerical model of the separator performance is made. With this model the pressure drop over the separator, the angular speed and the cut-off diameter of the separator as a function of flow rate can be calculated for different conditions. It is assumed that the post-separator has no influence on the performance of the separator. Water is taken as the liquid phase. The performance of the separator is calculated for the field conditions and for the two experimental conditions (table 3.2).

The angular speed of the separator results from the equilibrium between the produced angular momentum in the swirl generator and the losses in the other parts of the separator. This equilibrium is given by equation (3.35). The produced angular momentum in the swirl generator and the losses in the pre-separator are only a function of the flow rate. The other terms in equation (3.35) however also depend on the angular speed. This implies that equation (3.35) has to be solved by iterative calculation. The result of these calculations is given in figure 3.11.

For the SF6 and natural gas conditions the filter element does not rotate for flow rates below $7 \cdot 10^{-3} \text{ m}^3 \text{ s}^{-1}$. This is mainly due to the static bearing friction. For the air loop at the Technical University Eindhoven the element needs even a higher flow rate ($\sim 0.05 \text{ m}^3 \text{ s}^{-1}$) before it starts rotating. This is mainly due to the lower density of the medium.

It can be seen in figure 3.11 that although there is a large density difference between natural gas (or SF6) and air the rotational speed of the filter element is almost the

![Figure 3.11. Rotational speed as a function of flow rate for the three operating conditions of table 3.2.](image)
same for both fluids. This can be expected as both the generated momentum and the loss of angular momentum in the filter element, which has the highest contribution to the total dissipated momentum, are linearly dependent on the gas density. As a result the fluid density does not have a large influence on the rotational speed.

![Figure 3.12.](image)

Figure 3.12. Pressure drop as a function of flow rate for the three operating conditions of table 3.2.

The main pressure loss over the separator is due to friction in the filter element, equation (3.51). This is primarily due to the small channel height combined with a relatively large volume flow through the channels of the filter element. Other contributions to the total pressure loss are the generation of angular momentum in the swirl generator (equation (3.55)), the pressure loss due to friction in the swirl generator (appendix B) and the loss due to the difference in the tangential velocity component prior to and in the filter element, equation (3.53). In figure 3.12 the total pressure drop over the separator versus flow rate is depicted. It shows that the total pressure drop at the design flow rate for the field conditions is approximately 0.8 bar. In the calculations of the pressure loss over the filter element, it is assumed that the channels of the filter element are circular and for calculating the friction coefficient in case the flow is turbulent, the Blasius equation is used (equation (3.30)).

It can be seen that the pressure loss increases quadratically with the flow rate. This is due to the fact that the two main pressure loss contributions, pressure loss due to friction in the filter element and pressure loss due to the generated swirl, both increase with the axial fluid velocity squared. The pressure loss for air as the working fluid is much less compared to the other two due to the much lower fluid density.

The cut-off diameter of the filter element $d_{p100\%}$ depends on the flow rate as well as on the angular speed, see equation (3.19). The relation between angular speed and
flow rate follows from figure 3.11. With this relation the cut-off diameter is calculated and the result is given in figure 3.13. The separation performance deteriorates for decreasing flow rate. In case the flow rate is lower than the minimal flow rate necessary to turn the filter element, the filter element does not rotate and thus no droplets are separated.

![Figure 3.13](image)

**Figure 3.13.** Cut-off diameter of the filter element as a function of flow rate for the three operating conditions of table 3.2.

### 3.5 Closure

In this chapter the design relations to design an RPS-based separator for the removal of condensate from natural gas are presented. With these relations and the design criteria, which are summarized in table 3.3, a prototype is designed to remove liquid droplets of 2 micron and larger from natural gas under high pressure (80 bar). The design involves a so-called naturally driven RPS; i.e. the filter element rotates as a result of a swirl, which is generated in the flow approaching the element. The prototype is designed to operate under field conditions, is capable to handle the volume flow of one wellhead and can be installed in-line.

At first, it was intended to combine a turbine with the separator. By replacing the throttling valve, which is used to release the pressure of the gas prior to separation, by a turbine, useful energy can be recovered. During expansion of the gas, small sized condensate droplets are created. These droplets must grow, such that they are large enough to be separated in the RPS. However, the time these condensate droplets require to grow to sizes large enough for the separator to remove them, could not
be accurately predicted. Therefore, it was decided not to combine the turbine and separator in a single unit.
Chapter 4

Testing of prototype

4.1 Introduction

Two test facilities for measuring the hydrodynamic and separation performance of the prototype are introduced. The first test facility was built up in the laboratory of the section Process Technology at the Technical University of Eindhoven. This setup was used to measure the separation performance and the hydrodynamic performance of the prototype at low (atmospheric) pressure. The second test facility is located at CDS Engineering in Arnhem, the Netherlands. In this test-loop the hydrodynamic performance of the separator was measured at elevated pressures (field test conditions are simulated, see section 3.3.1). After introducing the test facilities, the separation performance measurements are described. At the end of this chapter the measurement results are presented and compared with the theoretical models, which are given in chapter 3.

4.2 Test facilities

4.2.1 Low pressure test facility

The low pressure test facility, which was constructed at the Technical University of Eindhoven, is shown in figure 4.1. In figure 4.2 a schematic overview of the test setup is depicted. In this test loop air is used as the working fluid. The air is supplied at a pressure of about 7 bar, but as the test loop has an open end, the pressure inside the test loop is only slightly higher than atmospheric pressure (pressure is approximately 1.2 bar). The temperature of the air in the setup is about 283 K. The amount of air, which enters the test loop, is controlled by a pneumatic actuated Invensys FoxPak control valve, type V725 DCFNA. At a distance of 1.6 m downstream of the control valve an Invensys I/A series electronic absolute pressure sensor, type 1GP10, is positioned. Volume flow is measured with an Invensys vortex flow meter, type 83F, which is positioned 0.6 m downstream of the pressure sensor. The pressure sensor is able to measure pressures in a range between 0 and 21 bar. The flow meter measures
air flow rates between 0.0042 and 0.45 m$^3$ s$^{-1}$. The accuracy and operating conditions of both sensors are given in table 4.1. At the end of the test loop the prototype of the natural gas – water separator is mounted. Six pipe diameters upstream of the separator, an injection point for droplets/particles is situated.

With an Invensys 1/A series differential pressure sensor, type IDP10 the pressure drop over the filter element is measured. For this purpose 4 equally spaced pressure holes over the circumference of the housing of the separator, before as well as behind the filter element, are present (figure 4.3). The 4 holes were interconnected such that, before as well as behind the element, the average pressure is measured. Besides one pressure hole is located upstream of the swirl generator. By using this pressure hole the total pressure drop over the separator can be measured. The sensor is adjusted to measure pressure differences in the range of 0 – 0.2 bar. Accuracy and operating conditions are given in table 4.1. As only stationary measurements were attempted, both absolute and differential pressure and the volume flow were directly read out from the LCD digital indicators during the measurements.

The angular speed of the filter element was measured with a Turck inductive sensor, type Bi1-EG05-AP6X. This sensor detects a small slot, constructed in the outer wall of the filter element. The sensor creates a high frequency electromagnetic field. When the sensor is positioned above the slot, the energy loss becomes so large that the magnetic field collapses. This breakdown is detected and subsequently the state of the output circuit is changed. In this way the revolutions of the filter element are counted. The location of the inductive sensor is denoted in figure 4.3. The sensor has a maximum switching frequency of 3000 Hz. The inductive sensor is coupled to a
Figure 4.2. Schematic representation of the low pressure test facility.

Figure 4.3. Location of the pressure holes and angular speed sensor on the prototype of the separator.

Brodersen digital to analogue convertor, type PXF-20.230 and subsequently led to a PC (Workbench). The convertor has a NPN/PNP input in 5 ranges from 0 to 5000 Hz. The input is set at 0 – 100 Hz. The output is set at 0 – 10 V and the operating voltage is 230 V. To check the conversion from voltage to rotational speed, the sensor and the convertor were calibrated on a turning lathe. The calibration curve is given in figure 4.4. As was expected the conversion is linear.
Table 4.1. Operating conditions of the measuring devices in the low pressure test loop.

<table>
<thead>
<tr>
<th></th>
<th>Flow meter</th>
<th>Abs. pres. sensor</th>
<th>Diff. pres. sensor</th>
<th>Inductive sensor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operating range</td>
<td>0.0042 – 0.55 m$^3$ s$^{-1}$</td>
<td>0 – 21 bar$^a$</td>
<td>0 – 0.2 bar</td>
<td>0 – 100 Hz</td>
</tr>
<tr>
<td>Temp. medium</td>
<td>255 – 473 K</td>
<td>227 – 394 K</td>
<td>227 – 394 K</td>
<td>not applicable</td>
</tr>
<tr>
<td>Static pressure rating</td>
<td>100 bar</td>
<td>150 bar</td>
<td>150 bar</td>
<td>not applicable</td>
</tr>
<tr>
<td>Accuracy</td>
<td>Re &lt; 20000: 2%</td>
<td>&lt; 0.075%</td>
<td>0.2%</td>
<td>≤ 2%</td>
</tr>
<tr>
<td></td>
<td>Re &gt; 20000: 1%</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

$^a$ absolute or gauge units

Figure 4.4. Calibration curve (solid line) of the inductive sensor, which is used to measure the rotational speed of the filter element. Circles represent the measurement data.

Measurements were performed as follows: the control valve was opened until the volume flow was high enough for the filter element to start rotating. At this point volume flow, absolute pressure, pressure drop over the separator and filter element and the angular speed were read out. Subsequently, the voltage to the control valve was increased in steps of 1 V, which corresponds to an increase in air flow of approximately 0.028 m$^3$ s$^{-1}$. As soon as the angular speed of the filter element reached a constant value, volume flow, absolute and differential pressures and angular speed were read out. The maximum volume flow that could be reached was about 0.42 m$^3$ s$^{-1}$. In this way 16 measuring points were obtained. Furthermore the behavior of the filter element during the start-up period was measured. For this purpose the maximal
volume flow was applied at once, after which the angular speed of the filter element was recorded every second until the element reached its final speed.

### 4.2.2 High pressure test facility

The hydrodynamic performance of the prototype at higher pressures was measured in a test loop located at CDS Engineering in Arnhem. This test loop is filled with Sulfur Hexafluoride (SF6). SF6 is a high density gas, with a molar mass of 7.8 times the molar mass of natural gas (appendix A). This means that at relatively low pressures, field test conditions can be simulated.

The actual volume flow of SF6 was monitored with an Instromet turbine gas flow meter, type Q75. The operating range of this flow meter is $0.05 - 1.8 \text{ m}^3 \text{s}^{-1}$ for SF6 at atmospheric conditions, with an accuracy of ±2%. During the measurements also the temperature and the absolute pressure upstream of the separator were monitored. Temperature was measured with an Invensys 1/A series temperature transmitter, Model RTT20 (Pt100), with an operating range between 233 and 358 K and an accuracy of ±0.05 K. The specifications of the absolute pressure sensor are given in table 4.1. The pressure drop over the filter element was measured by applying the pressure holes on the prototype itself and by using the pressure hole upstream of the prototype the total pressure loss over the separator was measured (figure 4.3). The same differential pressure sensor was used as in the low pressure test setup (table 4.1, operating range adjusted). Angular speed of the filter element was measured with the inductive sensor mentioned in table 4.1. As only stationary measurements were attempted, all process conditions were directly read from the LCD digital indicators during the measurements. Measurements were performed at 3 different absolute mean pressures: 1.9 bar, 2.9 bar and 3.6 bar, corresponding to fluid densities of respectively: 11.2, 17.5 and 22.2 kg m$^{-3}$ [72] (the capacity of the blowers was too low to reach the maximum absolute pressure of 8 bar). The temperature of SF6 during measurements was about 300 K. Two blowers circulated the SF6 through the setup. The volume flow was controlled by changing the capacity of these blowers from 5 to 70%, which corresponds to an actual volume flow between approximately 0.2 and 0.5 m$^3$ s$^{-1}$. As soon as the volume flow reached a constant value, absolute pressure, temperature, volume flow, pressure drop and angular speed were read out from the LCD displays.

### 4.3 Separation performance measurements

The efficiency of the separator was measured in the low pressure test facility. In order to measure efficiency as a function of particle size, it was intended to use condensate droplets as the dispersed phase. The reason is that droplets formed by homogeneous condensation have a typical size around 1 micron (section 3.2.1), which is the size range in which the filter element operates, whereas droplets created mechanically (for example by a nozzle) generally have diameters larger than 10 microns (appendix E.1). Although there are some mechanical processes by which droplets with diameters smaller than 5 micron can be created (nebulizers, electrostatic techniques),
their low droplet concentration, high geometric standard deviation and uncontrolled operation, make them inappropriate to measure the efficiency of the prototype [26].

The condensate droplets were created with a TSI Condensate Monodisperse Aerosol Generator (CMAG), Model 3475 [79]. This generator, based on the Sinclair LaMer principle, creates highly monodisperse liquid particles by controlled heterogeneous condensation. Vaporized aerosol material, di-2-ethyl hexyl sebacate (DEHS) is condensed on small salt nuclei, with sizes of 10 to 100 nm [46]. The sizes of the condensate droplets are adjustable within a size range of 0.1 to 8 micron with a geometric standard deviation smaller than 1.15 micron. Concentrations higher than $10^6$ particles per cubic centimeter can be achieved. Nitrogen is used as a carrier gas. A disadvantage of the CMAG is that the device only works against atmospheric pressure. Although the absolute pressure in the test setup is low, around 1.2 bar, the volume flow is relatively large ($\sim 1500 \text{ m}^3 \text{ hr}^{-1}$). This means that in order to insert the condensate in the test loop, the condensate flow has to be increased in pressure. However, as we want to measure separator efficiency as a function of particle size, it is important that during injection the size and monodispersity of the condensate particles remain undisturbed as much as possible.

Liebrand [37] tested various injection systems. Droplet sizes after the injection systems were measured with a Malvern Mastersizer X [77]. This device uses Fourier Optics and Mie scattering theory to determine the sizes of the particles from their diffraction patterns [28]. Several Fourier transform lenses were available, such that particle sizes in a broad range from 0.1 to 600 micron could be measured. It appeared that injection systems, which provide a continuous condensate flow, like pumps and compressors, did not work. The condensate stuck to the interior of these devices and only the carrier gas was discharged. The only injection system that worked properly was filling a tank with condensate and subsequently discharge the tank by applying compressed air. Although at discharge the diameter of the condensate particles is somewhat larger and the size distribution is more spread than at the inlet of the tank, the mean droplet diameter at discharge is well reproducible for specific operating conditions of the CMAG [37]. In these preliminary tests the volume flows of the condensate, pressurized air and the mixture of air and condensate were controlled with ball valves. These valves were operated manually. For the final efficiency measurements of the prototype, however, this was not an option, as automated operation was necessary. We therefore replaced the ball valves with Asco Joucomatic pneumatic valves, type E290. These valves were chosen because of their large throughput area, while at the same time they can cope with the required flow rates. During injection of the condensate, however, the same problems as with the continuous injection systems arose. Again the condensate stuck to the interior of the valves, which resulted in a liquid film in the outlet tube of the valve instead of individual droplets. It can therefore be concluded that injection of the condensate droplets in the test loop, without changing the size of the condensate droplets, is not feasible. As an alternative, it was decided to use solid particles to measure the efficiency of the separator as a function of particle size.
4.3 Separation performance measurements

4.3.1 Filter efficiency

A cascade low pressure impactor, an eight stage Andersen impactor type Mark II, was used to measure the efficiency of the prototype as a function of separated particle size [78]. The measuring principle of an impactor is based on the selective size distribution of particles, extracted from an accelerated gas stream and impacted on collection plates [24]. This principle is shown in figure 4.5. The sample is drawn through the stages of the impactor by a vacuum pump. Every stage consists of an orifice plate, a collection plate and an O-ring for sealing. Every orifice plate contains multiple precision drilled orifices. The diameter of the orifices is constant for each stage, but is smaller in each succeeding stage. Particles, which as a result of their inertia, can not follow the jet from one stage to another, will impact on a collection plate. As the velocity of the flow increases each succeeding stage (as orifice diameter decreases stage by stage), the collected particles are smaller on each successive collection plate. The range of particle sizes collected on each stage depends on the sample stream velocity of the stage, the distance between the orifices and the collection surface and the cut-off size of the previous stage. Any particle not collected on the first stage, follows the air stream around the edge of the plate to the next stage, where it is either impacted or passed on to the succeeding stage, and so on until the

Figure 4.5. Measuring principle of a cascade low pressure impactor.
jet velocity is sufficient for impaction. Particles too small to be impacted on the last collection plate are collected in a backup filter.

When sampling a gas flow containing particles larger than 10 micron, a pre-separator must be used to prevent particle bouncing and re-entrainment errors. The Mark II pre-separator is an impaction chamber with one 0.0135 m diameter inlet orifice and three outlet tubes. The outlet tubes are situated 0.0254 m above the impaction surface. This design results in very low turbulence and allows collection of several grams of particulate without overloading the pre-separator.

At \( \phi_{\text{imp}} = 4.72 \cdot 10^{-4} \text{ m}^3 \text{ s}^{-1} \) (\(= 1 \text{ ACFM}\)), the particle fractionation of the Andersen Mark II impactor ranges from 10.0 to 0.4 micrometer aerodynamic diameters, see table 4.2. During the experiments, however, sample flow varied around \( 4.72 \cdot 10^{-4} \text{ m}^3 \text{ s}^{-1} \). Small variation in sample flow can be accounted for by adjusting the size ranges as follows

\[
d_{p, \text{cor}} = \sqrt{\frac{4.72 \cdot 10^{-4} d_p^2}{\phi_{\text{imp}}}}
\]

where \( \phi_{\text{imp}} \) is the volume flow through the impactor. The \( d_p \) usually expressed as the \( d_{p,50}\% \) of a certain stage, i.e. the particle diameter, which has 50% probability of being collected, is proportional to the square root of the Stokes number. For an impactor the Stokes number is defined as the ratio of the particle stopping distance at the average orifice exit velocity to the orifice radius [24]. The particle stopping distance is proportional to the volume flow through the impactor, \( \phi_{\text{imp}} \). Therefore the size range of the impactor can be corrected for by the square root of the volume flow.

The powder, which was used in the experiments, is calcium hydroxide (Ca(OH)\(_2\)), also called chalk hydrate or slaked lime. It was supplied by Carmeuse (product name Mekal [76]). The lime has an absolute density of 2249 kg m\(^{-3}\) and a bulk density in the range of 650 – 850 kg m\(^{-3}\). The powder was injected by means of an ejector at the injection point (IP in figure 4.2) in the test setup.
The whole injection system is depicted in figure 4.6. Before injected, the particles were fed in a box. A screw situated at the bottom of the box transported the lime into an ejector. The rotational speed of the screw could be controlled in order to vary the mass flow rate of the lime into the ejector. The ejector has three connection points; the center one is connected to pressurized air and the two outer tubes are used to transport the particles. During the measurements only one outer tube was used. A funnel was placed in this tube to guide the lime into the ejector. In the center tube of the ejector a hollow tube, contracted at the tip, was mounted. The pressurized air, which was supplied to this center tube, is accelerated at the tip, creating an underpressure. As a result, the particles, which are fed into the ejector by the screw, are sucked through the ejector and leave the outlet of the ejector together with the pressurized air. The contracted tip can be moved up- and downwards in the center tube of the ejector. In this way the amount of underpressure and thus the amount of injected lime can be varied. The total amount of injected lime was determined by measuring the box, filled with lime, prior to and after each measurement (with an accuracy of ± 2·10^{-3} kg). The size distribution of the lime particles injected in the setup was measured with the impactor both up- and downstream of the separator.

To ensure that a representative sample of lime particles is extracted from the gas flow, the sample should be extracted isokinetically. Sampling is isokinetic when the inlet of the sampler, in our case a copper tube, is aligned parallel to the gas streamlines and the gas velocity entering the tube is identical to the free stream velocity approaching the inlet [24]. In this way there is no distortion of streamlines in the neighborhood of the inlet, which may lead to measurement errors. Isokinetic sampling does not mean that there are no losses between inlet of the sampler and the impactor. It only ensures that the concentration and size distribution of the particles entering the sampling tube is the same as that of the main stream. In order to meet the requirement that the velocity of the gas, which enters the sample tube, equals the free stream velocity, the diameter of the sample tube $d_{\text{sample}}$ should be

$$d_{\text{sample}} = d_{\text{main}} \sqrt{\frac{Q_{\text{sample}}}{Q_{\text{main}}}}$$  \hspace{1cm} (4.2)

where $d_{\text{main}}$ is the diameter of pipe in the setup and $Q_{\text{sample}}$ and $Q_{\text{main}}$ are respectively the sample volume flow and the volume flow through the setup. Thus, there are two ways to match the sample velocity with the free stream velocity: adjusting the sampling flow rate through the probe, or adjusting the diameter of the sampling probe.

The sample flow was drawn through the impactor by a vacuum pump (Model 10-710 Graseby Anderson, Atlanta Georgia). A valve situated between the impactor and the vacuum pump controls the volume flow (flow during measurements was set at approximately 4.72·10^{-4} m³ s⁻¹). The flow rate through the impactor was monitored by a Brooks Sho-Rate Purge meter, model 1357 F5. The maximum flow of this flow meter for air at normal conditions (293 K and 1.013 bar absolute) is 9.64·10^{-4} m³ s⁻¹. Measurements were performed at the maximum volume flow through the setup, i.e. 0.42 m³ s⁻¹. To achieve isokinetic sampling, a sample tube, with an inner diameter of 5.5·10^{-3} m was positioned at the center line of the main pipe ($d_{\text{main}} \simeq 0.15$ m).
In order to measure the particle size distribution upstream of the separator, the separator was removed from the setup. This was done to increase the distance between the injection point and the sample point. Without removing the separator no representative sample could be taken. This was due to the influence of the bend, which is situated just before the injection point, on the fluid flow pattern. As a result the same sampling tube is used for both the measurements upstream as well as downstream of the separator. The distance between the injection and sampling point during the measurements without separator was about 15 pipe diameters. The distance between the separator and the sample point, during downstream measurements, was about 10 pipe diameters.

Since lime is hygroscopic, all collection media must be preconditioned prior to weighing, i.e. both before and after a sampling cycle. Preconditioning was done by placing the collection media in a oven at 323 K for a period of 12 hours.

After sampling, the collection plates of the impactor were weighed with a Mettler mechanical balance, type B5. This balance has a measuring range of 0 – 0.2 kg, with a standard deviation of $\pm 3 \times 10^{-8}$ kg and an optical accuracy of $\pm 5 \times 10^{-8}$ kg.

The efficiency of the separator is calculated by the following procedure:

- First, the change in weight for each stage of the impactor, including the backup
filter, is determined by measuring all collection media prior to and after sampling.

- All weight changes are added up to obtain the total particulate weight collected by the impactor.

- The fractions of the total collected weight in each stage of the impactor are determined by dividing the weight collected on each stage by the total collected weight. In this way the relative particle size distribution is determined both up- as well as downstream of the separator. As the size ranges of the impactor stages have not the same width (see table 4.2), usually the fraction in each stage is divided by the stage width. In this way the percentage per size interval is retrieved.

- Multiplying these fractions with the total particle concentration present in the gas stream gives the absolute concentration of particles in each size class.

- The efficiency per size class follows from the difference in absolute particle concentration on each impactor plate between the measurements up- and downstream of the separator.

In an ideal situation the particle concentration upstream of the separator \(C_{us}\) measured with the impactor, equals the injected particle concentration. In practice, however, the total particle concentration on the impactor plates upstream of the separator is less than the injected particle concentration. This is due to the loss of lime from the injection point to the impactor. There are several places were losses occur and the most important ones are: the funnel, the inside of the ejector, the vanes of the swirl generator and the bends in the sampling tube. During the measurements the ratio between the injected particle concentration and total impactor concentration upstream of the separator reached a value between 2 and 3.

The total particle concentration downstream of the separator can be determined with an absolute filter measurement downstream of the separator. Measurements performed at the University of Twente, however, showed that the total particle concentration downstream of the RPS-based separator determined with an absolute filter is approximately equal to the total particle concentration on the impactor plates when measuring downstream of the separator [9]. In these experiments also calcium hydroxide and the same Anderson impactor were used. Equal concentrations can be expected, as the impactor works as a kind of absolute filter. In our experiments, therefore, the concentration downstream of the separator \(C_{ds}\) is assumed equal to the total particle concentration on the impactor plates of the downstream measurements. When the total injected concentration and the total impactor concentration downstream of the separator are known, the total efficiency \(E_{total}\) of the prototype can be calculated with

\[
E_{total} = \left(1 - \frac{C_{ds}}{C_{us}}\right) \times 100\% \tag{4.3}
\]
4.4 Measurement results

4.4.1 Hydrodynamic performance at low pressure

In this section the results of the hydrodynamic measurements at low pressure are presented. Both the angular speed and pressure drop are examined as a function of flow rate. Besides, the angular speed, which is measured during acceleration of the filter element from zero to maximum speed (at 0.42 m$^3$ s$^{-1}$), is presented. All experimental results are compared with their corresponding theoretical model, which is presented in chapter 3.

Angular Speed

In figure 4.7 the rotational speed $n$ is depicted as a function of the flow rate through the separator. The solid line represents the model, given in section 3.2.3, the circles, crosses and dots each represent one measurement series. It can be seen that the measurements are well reproducible. Comparing the measurements results with the theoretical model shows, except for a slight deviation at low flow rates, good agreement. For flow rates smaller than about 0.06 m$^3$ s$^{-1}$ the filter element does not rotate. This is mainly due to the static bearing friction. The model also predicts the onset of rotation well. The small bend in the model at small flow rates is due to the fact that the relation for the dynamic bearing friction at low angular speeds differs from the one at higher speeds [62].

![Figure 4.7. Rotational speed of the filter element as a function of flow rate for air at an absolute pressure of 1.2 bar.](image)

The acceleration during start-up of the filter element was measured by applying the maximum flow rate of 0.42 m$^3$ s$^{-1}$ at once. The behavior of the element can
be simulated by equating the generated moment to the required angular momentum corrected for the moments of inertia of the rotating parts (equation (3.49)). The rotating parts are the filter element, including the bushing on the in- and outside, and the shaft. In the model the shaft diameter is taken constant at 0.115 m. (In reality the shaft diameter varies and has a maximum value of 0.115 m). In figure 4.8 the angular speed divided by the maximum angular speed (at full flow rate) is given as a function of time. The solid line represents the model and the crosses represent one measurement series. The filter element does not start rotating at once as the opening of the valve takes some time. It can be concluded that, although the shaft is modelled as a cylinder with constant diameter, the model and measurements are in good agreement.

![Figure 4.8](image)

**Figure 4.8.** Angular speed of the filter element versus time, when full flow rate is applied at once. Working fluid is air at an absolute pressure of 1.2 bar.

**Pressure drop**

The pressure drop over the separator is also measured as a function of flow rate. In figure 4.9 the pressure drop over the filter element is depicted. The crosses and dots each represent one measurement series, while the lines represent the model. In the model the pressure drop due to friction (including entrance losses) in the filter element channels and the pressure loss due to velocity change at the entrance of the filter element are taken into account. Losses are calculated for the three types of channel shapes as discussed in section 3.2.4 and appendix C: circular, triangular and sinusoidal. The values of the friction factor, $f$ and $\xi$ are taken from table 3.1. Figure 4.9 shows that the pressure loss over the filter element is best represented by
the model, which assumes sinusoidal channels. This is also expected, as the filter element channels of the prototype have a more or less sinusoidal shape. The model assumes a fully developed flow in the channels of the filter element. In reality this is not true, as the dynamic entrance length, $L_{hy}$ for the channels is rather large. The dynamic entrance length is defined as the required duct length to achieve a maximal duct velocity of 99% of that for fully developed flow when the entering fluid velocity profile is uniform [61]. For example, for sinusoids channels for which the width is much larger than the height, Shah and London [61] give a dimensionless entrance length $L_{hy}^+ (= L_{hy}/(Dh \text{ Re}))$ of 0.0648. The hydraulic diameter of the channels is $1 \times 10^{-3}$ m and the Reynolds number in the channels during the measurements is of the order $1 \times 10^3$. This means that the dynamic entrance length is about 0.0648 m, which is about one third of the total channel length ($L_c=0.18$ m). Although the model neglects the entrance length, there is a good agreement between the measurements and the model.

![Figure 4.9](image-url)  
**Figure 4.9.** Pressure drop over the filter element as a function of flow rate for air at an absolute pressure of 1.2 bar.

Besides the pressure loss over the filter element also the pressure drop over the entire separator was measured. These results are compared with a theoretical model in which pressure drop due to friction in the filter element (assuming sinusoid channels) and the swirl generator, pressure loss due to velocity changes at the entrance of the filter element and pressure loss due to the added swirl are included (section 3.2.4). Both results are given in figure 4.10 together with the results for the pressure loss over the filter element. It follows that total pressure loss is well predicted by the theoretical model. The pressure loss in the pre- and post-separator and pressure drop in the pipe before the swirl generator are not included in the model, as they are negligible compared to the other losses. Figure 4.10 also shows that pressure loss over
the filter element is the main contributor to the total pressure loss over the separator. In the second place follows the pressure loss due to the generated swirl.

![Graph showing pressure drop over the filter element and the whole separator as a function of flow rate for air at an absolute pressure of 1.2 bar.]

*Figure 4.10.* Pressure drop over the filter element and the whole separator as a function of flow rate for air at an absolute pressure of 1.2 bar.

### 4.4.2 Hydrodynamic performance at high pressure

In this section the results of the hydrodynamic measurements at high pressures are presented. Both the angular speed and pressure drop are examined as a function of flow rate. All experimental results are compared with their corresponding theoretical model, which is presented in chapter 3. As the temperature and absolute pressure in the test setup varied during one measurement cycle, the mean temperature and pressure values of one measurement cycle are used to calculate the theoretical hydrodynamic performance of the separator. The density and viscosity of SF6 are interpolated from the measurement data of Wilhelm [72]. These measurements were performed at a temperature of 300 K and pressures in the range of $3 \cdot 10^4$ to $2 \cdot 10^6$ Pa. The molar mass of SF6 is given in appendix A.3. In this section the results of the measurements at 3.6 bar are presented. The measurement results at 1.9 and 2.9 bar are given in appendix F.

In figure 4.11 the rotational speed is depicted as a function of the flow rate through the separator. The solid line represents the model, represented by equation (3.35), while the crosses represent the measurement data. The theoretical model shows good agreement with the measurements. At lower absolute pressures, results are given in appendix F, the models predict a slightly lower rotational speed than measured.

In figure 4.12 the pressure loss over the filter element and the total pressure loss over the prototype are given as a function of the flow rate. The lines represent the
Figure 4.11. Rotational speed of the filter element as a function of flow rate for SF6 at an absolute mean pressure of 3.6 bar.

Theoretical calculations, while the crosses and circles represent the measurement data. Opposite to the measurements in the low pressure test setup during which the flow in the channels of the filter element was laminar, the channel flow during the high pressure measurement was turbulent (the Reynolds number in the channels varied between $9 \times 10^3$ and $2 \times 10^4$). For calculating the pressure drop over the filter element due to friction, the Blasius relation (equation (3.30)) is used. A value of 2 is assumed for the factor $\xi$, which represents the additional pressure drop at the entrance region of the duct. The line representing the pressure loss over the filter element, also takes into account the pressure loss due to the change in the tangential velocity at the entrance of the filter element. In the model for the pressure loss over the swirl generator, the pressure loss due to the added swirl and due to friction are included. In the line representing the total pressure loss over the separator these four pressure loss contributions are added.

All measurement series show a gradual increase of the pressure loss with increasing flow rate. The filter element has again the highest contribution to the total pressure loss. The measurement data of the pressure loss over the filter element are slightly below the predicted pressure loss. The measurement data for the total pressure loss on the other hand are somewhat higher than predicted. This can be attributed to the fact that in the model some flow phenomena in the swirl generator which contribute to an additional pressure loss are neglected. One of these phenomena is the fact that the angle of incidence of the fluid at the inlet of the vanes differs from the angle of the vanes itself. Furthermore, there will be an additional pressure loss due to the acceleration of the fluid as it enters the vanes (decrease in cross-sectional area due to thickness of the vanes). Besides, wakes may occur at the trailing edges of the vanes,
which cause an extra pressure loss.

The measurements at lower absolute fluid pressures (appendix F) also show a gradual increase of pressure loss with increasing flow rate. Again the measured pressure loss over the filter element is slightly lower than the predicted pressure loss over the filter element and the measured total pressure loss is somewhat higher than predicted. The latter is due to the fact that the model neglects flow phenomena in the swirl generator as indicated before.

![Graph](image)

**Figure 4.12.** Pressure loss over the filter element and the total pressure loss over the prototype as a function of flow rate for SF6 at an absolute mean pressure of 3.6 bar.

Comparing the measurement results at the three different absolute pressures shows that the rotational speed of the filter element hardly depends on the fluid density (or absolute fluid pressure). This is expected as both the generated momentum and the loss of angular momentum in the filter element, which has the highest contribution to the total dissipated momentum, are linearly dependent on the gas density. Besides, it can be seen that for increasing fluid density, the pressure losses increase. This is also predicted by the model as all pressure loss contributions increase for increasing fluid density.

Based on the atmospheric measurements and the measurements at elevated pressures, it can be concluded that the relations derived in section 3.2 give a good estimation of the rotational speed of the filter element and the pressure loss over the separator.

### 4.4.3 Separation efficiency

In order to calculate the mean separation efficiency and the standard deviation per size class, several measurements with the low pressure impactor (section 4.3.1) were
performed. In total 3 representative measurements upstream of the separator and 4 representative measurements downstream of the separator were obtained. It was particularly difficult to obtain good measurement results downstream of the separator, as it appeared quite difficult to inject the appropriate particle concentration at the inlet of the separator. A too high inlet concentration results in blockage of the filter element channels. This causes an increase in the leak flow along the outside of the filter element and a consequential loss in efficiency. By adjusting the rotational speed and pitch of the screw, which transported the lime into the test setup, an appropriate inlet concentration of about $1 \cdot 10^{-4}$ kg m$^{-3}$ was obtained.

The 3 relative size distributions upstream and 4 relative size distributions downstream of the separator are depicted in respectively figure 4.13 and figure 4.14. In these figures the percentage of particles in a certain size class divided by the class width are plotted against the mean particle size of each size class. Figure 4.13 shows that the lime has a relative high concentration of particles between 1 to 3 micron. This is exactly the size range we are interested in, as the $d_{p100\%}$ of the filter element at the full flow rate in the low pressure test setup is about 3 micron.

Multiplying these relative size distributions with the absolute particle concentrations in the gas stream gives the total amount of particles in each size class. This is done as follows. First, the three particle size distributions upstream of the separator are multiplied with the injected particle concentration of one specific measurement downstream of the separator. Subsequently the four particle size distributions downstream of the separator are multiplied with the total impactor concentration of that same specific measurement. For one specific measurement the resulting mean absolute particle concentrations upstream and downstream of the separator are depicted in figure 4.15. Comparing these two concentrations shows a large decrease in particle concentration for all particle sizes. For particles in the size range of 5 $\mu$m and larger,
the absolute concentration almost reaches 0 mg m$^{-3}$. This indicates that almost all particles in this size range were removed by the separator.

By determining the differences in absolute particle concentrations up- and downstream of the separator for each size class, the efficiency of the separator as a function of the particle size follows. The difference in particle concentration can be determined for 12 combinations (3 particle distributions upstream of the separator and 4 distributions downstream of the separator), resulting in the same number of efficiencies per size class. This procedure is repeated for the three other measurements downstream of the separator. This results in 48 efficiencies per size class.

The measured efficiencies per size class are compared with the theoretical filter efficiency given by equation (3.20). In this equation the theoretical efficiency is expressed as a function of the dimensionless particle diameter, $d_p/d_{p100\%}$. Efficiencies, which result from the measurements, are however valid for their specific size range and not for a single particle diameter. We have therefore assigned the measured efficiencies to the mean diameter of each size range. Subsequently this mean diameter is divided by the $d_{p100\%}$ of the filter element. The $d_{p100\%}$ is in this case based on a particle density of 1000 kg m$^{-3}$ instead of the actual density of the lime particles. This is necessary as the impactor measures aerodynamic diameters. The aerodynamic diameter is defined as the diameter of a unit density ($\rho_p = 1000$ kg m$^{-3}$) sphere with the same settling velocity as the particle under consideration [24]. The $d_{p100\%}$ based on a unit particle density of the filter element for the measurement conditions is about 3 $\mu$m (equation (3.18)).

The results of both the measurements and the model are given in figure 4.16. The dotted line represents the separation efficiency predicted by theory for triangularly shaped channels, which have a flow distribution linearly proportional to radial distance $r$ (equation (3.20)). The solid line represents the theoretical separation efficiency for
triangular channels in which the axial flow distribution is constant over the channels, 
\[ u_f \propto 1 \] [14]. In both models a parabolic velocity profile in the channels is assumed (in reality this is not valid, as the dynamic entrance length \( L_{hy} \) for the channels is rather large (section 4.4.1)). Theoretical expressions for triangular channels are used, as they are easier to derive than expressions for sinusoidally shaped channels. Besides, the theoretical results for trianularly shaped channels are a good approximation to those of sinusoidally shaped channels [14]. Furthermore, in practice the channel shape is quite irregular and also the sinusoidal shape is only an approximation of the actual channel shape. It can be seen that the separation performance around \( x=1 (=d_{p100\%}) \) is less than 100\% (equation (3.18)). This is due to the fact that in deriving the expression for the filter element efficiency \( E \), a parabolic velocity profile in the channels is assumed, whereas in deriving the expression for \( d_{p100\%} \) (equation (3.19)) a constant axial velocity profile is assumed. The underlying reason is that for a parabolic velocity profile in channels of varying radial height, the axial velocity in the radial plane where the radial height is largest, is a factor greater than the average axial velocity for the channel as a whole. It is for this reason that the efficiency reaches its maximum of 100\% for values of \( x \) larger than 1.

The mean separation efficiencies from the measurement are represented by circles, while the vertical lines through the circles represent the standard deviation. In figure 4.16 eight measurement points are depicted. These represent all collection plates of the impactor, except the pre-separator and the back-up filter. The pre-separator is not accounted for as it only serves to remove the coarse particles from the flow. The back-up filter is not regarded because, after the measurements upstream of the separator, no weight increase of the back-up filter could be measured. This indicates that the amount of particles smaller than 0.4 \( \mu \text{m} \) created by the ejector is too low to be measured.

Figure 4.16 shows that for a dimensionless particle diameter of 1.5 (corresponding to approximately 5 \( \mu \text{m} \)), an efficiency of about 100\% is reached. This agrees with the result presented in figure 4.15. It can be seen that the standard deviation increases for smaller particle sizes. This is expected as the measurement errors are larger in these size ranges, due to a smaller weight difference between the impactor plates prior to and after the measurements. For particle sizes around \( d_{p100\%} \) and larger, the agreement between theory and experiments is reasonable. However, for particle sizes smaller than \( x=1 \) a large difference between the models and the experiments is observed. Previously performed measurements, using the same Anderson cascade impactor to measure efficiency, showed similar results [13]. The previous measurements were performed with another type of separator, also based on the Rotational Particle Separator principle. The separator is a so-called tangential version of the RPS. In this separator the fluid enters the separator tangentially into a cyclone. The cyclone serves as a pre-separation device, in which coarse particles are separated from the gas. Subsequently the gas enters the filter element. The main difference between the tangential version of the RPS and the RPS-based separator used in this study is the way the filter element is driven. In the tangential RPS the element is driven externally by a motor, while the separator used in this study is naturally driven. In figure 4.17 these measurement together with the current measurements are shown. The circles
**4.4 Measurement results**

![Graph showing separation efficiency as a function of particle size.](image)

**Figure 4.16.** Separation efficiency as a function of particle size. The solid line represents the theoretical efficiency prediction for a constant axial flow distribution, the dotted line represents the theoretical efficiency prediction for an axial flow distribution, which is linearly proportional to \( r \) and the circles represent the measurement data (vertical bars denote the standard deviation).

The volume flow, at which the separation efficiency of the separator is measured,
Figure 4.17. Separation efficiency as a function of particle size. Comparison of current measurements (circles) with previously performed measurements (asterisks and triangles) [13].

is 0.42 m$^3$ s$^{-1}$. The axial Reynolds number in the channels of the filter element for this volume flow is about 900 and the rotational Reynolds number is about 10. Thus, measurements have been performed under laminar flow conditions. In practice the flow in the channels of the filter element is turbulent, see table 3.4. In a turbulent flow secondary flows could negatively influence the separation efficiency, as was explained in section 3.2.2. To get insight in the influence of turbulence on the separation efficiency of the filter element, the presented measurements should be extended to higher Reynolds numbers. Within the period of the current study, however, it was not possible to measure the separation performance of the element under field test conditions.

The separation efficiency of the separator could also be decreased by processes occurring in the post-separation area. In the post-separator droplets, which are created at the end of the filter element, are separated from the gas flow by the centrifugal force. However there is a risk of breaking up of these droplets into smaller droplets by the centrifugal or turbulent forces acting in the post-separation area (section 3.2.2). Besides, there is a chance of re-entrainment of the liquid film, which develops at the extended inner wall of the filter element, into the gas flow. These two processes will lead to a decrease in the efficiency of the separator.

For the maximal volume flow in the low pressure test loop, the flow in the post-separator is turbulent ($Re_{ax} \sim 1 \cdot 10^5$, $Re_\Omega \sim 7 \cdot 10^5$). To see if at this condition droplets would occur in the outlet flow of the separator, water droplets, with diameters between 10 and 100 micron, are injected in the low pressure test loop. The outlet flow of the separator is examined with a Malvern Mastersizer X. During these measurement the
Mastersizer could not detect any droplets. This indicates that at these conditions no droplets are created with sizes smaller than the post-separator can separate. As under field test conditions the Reynolds numbers in the post-separator are even higher, it is recommended that also these measurements are repeated under field test conditions.

4.5 Closure

The hydrodynamic operating performance of the prototype was measured at atmospheric conditions and at elevated pressures. During these measurements the angular speed of the filter element and the pressure drop over the separator were recorded for varying fluid volume flow. At atmospheric conditions also the separation performance of the separator was measured. The concentration of solid lime particles both up- and downstream of the prototype was measured with a low pressure impactor to determine the separation efficiency of the filter element as a function of the diameter of the separated lime particles. All measurement results are compared with theoretical predictions. Overall it can be concluded that the characteristics of the prototype are well predicted by these models.
Testing of prototype
Chapter 5

Discussion

In this study a new type of separator for removing condensed liquid phases from natural gas is designed and tested. More compact separation devices are needed as current gas treatment techniques are not profitable. The Rotating Particle Separation (RPS) principle is used to develop a compact and efficient apparatus. In this chapter the design methods used and the experimental results achieved are reviewed and commented upon.

5.1 Design and design methods

The design of the natural gas – water separator can be divided into three parts. First, the behavior of the liquid film in the channels of the filter element is described by a model based on the classical Nusselt analysis for a condensate film on a vertical cold plate. The model applies to small liquid loads, as inertia effects are disregarded and a boundary layer-type flow is considered. With this model the distribution of the liquid in a single channel of the filter element is determined both for a vertically as for a horizontally placed filter element.

Second, it is examined if the functions of expansion and separation can be combined in a single device, to develop a so-called RPS – turbine. After natural gas is extracted from an offshore well, its pressure has to be reduced in order to process and transport the gas. Nowadays this pressure reduction takes place in a throttling (Joule-Thompson) valve. The available energy, which is released during expansion over the valve, is dissipated and thus wasted. Useful energy can be recovered by replacing the throttling valve by a turbine. A critical parameter in this design step is the size of the condensate particles, which are formed after expansion. As the pressure drop during this expansion is high, a high amount of small-sized nuclei is formed (homogeneous nucleation dominates). These particles must be given enough time to grow such that they are large enough to be separated in the RPS-based separator. This means that the area between the expansion and separation step, the condensation area, must be properly designed.

In the third place, fluid flow relations and RPS design principles are derived, to
predict the hydrodynamic and separation performance of the natural gas – water separator.

The relations, which describe the liquid film thickness along the channel walls, predict for both the vertically and the horizontally placed filter element, a more or less similar behavior. The majority of the liquid is forced in the same direction as the force which acts along the channel wall. For the vertically placed filter element this is the gravitational force and for the horizontally placed element this is the shear force. However, in both cases a small part of the liquid film, which is situated at the beginning of the channels, is forced in the opposite direction. This is due to the fact that in both cases the film thickness has a relatively large curvature at the beginning of the channel. Due to this strong curvature the centrifugal force exerts a large force on the liquid film and causes part of the liquid to flow in opposite direction. Thus, although the centrifugal force is much larger than the force acting along the channel wall, it only causes a force in the direction opposite to the force acting along the wall through the slope of the film thickness. For small slopes the influence of the centrifugal force can be disregarded.

The growth time the condensate droplets, which are generated due to expansion of the gas, require to reach a sufficiently large size for separation, mainly depends on one parameter, the amount of stable nuclei formed per unit time and volume (determined by the nucleation rate). As a result of the expansion a high saturation ratio is achieved. Consequently homogeneous nucleation is the dominating mechanism for producing a large amount of small-sized nuclei. After these nuclei are created they grow due to condensation and coagulation. Both droplet growth due to condensation and coagulation depend on the amount of stable nuclei formed. The maximum droplet size due to condensation increases for a decreasing stable nuclei density, whereas the growth rate due to coagulation increases with increasing droplet density. For the typical process conditions of the natural gas – water separator droplet growth due to condensation and coagulation is calculated. It follows that for a high number density, the droplets created by condensation alone are too small for separation by the RPS. Coagulation however ensures that regardless of the initial number concentration the droplets reach a diameter of 1 micron or larger (the RPS is capable to remove droplets of this size). The growth time, however, depends on the amount of droplets present. The existing models, which predict the homogeneous nucleation rate, are not accurate at high pressures. Deviations between models and experiments reach a factor 10^4 and larger. This means that the required length of the condensation chamber cannot be accurately predicted. It was therefore decided not to combine the expansion and separation step in a single device.

The three crucial parameters in the design of the separator are the angular speed of the filter element, the pressure loss over the separator and the separating efficiency of the filter element, expressed by \(d_p100\%\) (particle size collected with 100% probability). The angular speed of the filter element follows from the equilibrium between the generated angular momentum in the swirl generator and the losses in the other parts of the separator, like the filter element, bearings etc. Relations for the generated and dissipated angular momentum in all relevant parts of the separator are derived. The pressure loss over the separator is mainly due to friction in the filter element channels.
and due to the generated swirl. Besides, expressions for the pressure loss due to friction in the swirl generator and for the pressure loss due to tangential velocity difference prior to and in the filter element are derived. The $d_{p100\%}$ of the filter element is determined by applying a force balance on a particle in the filter element, assuming solid body rotation and a uniform axial fluid velocity. In addition to the $d_{p100\%}$ of the filter element, also the behavior of the droplets in the post-separator, may influence the efficiency of the separator. In the post-separator centrifugal forces and/or dynamic pressure forces, which result from turbulent fluctuations, may lead to droplet break-up. If the resulting droplets are too small to be removed from the flow in the post-separator, the efficiency of the separator decreases. Therefore, relations, which predict the minimum droplet size in the post-separator, are derived.

Most design criteria are based on practical considerations. The operating parameters like operating pressure, temperature and volume flow of both gas and liquid were based on the practical conditions after throttling. The maximum size of the separator was fixed due to fabrication limitations of the filter element and due to the demand that the separator should be installed in-line. The pressure drop over the separator was limited to 1 bar, as the high pressure test setup enforces this restriction. As the separation efficiency should be competitive with current techniques, which typically separate particles down to 5 micron ($d_{p50\%}$), the filter element is designed with a $d_{p100\%}$ around 1 micron.

Based on the above mentioned considerations, design principles and design criteria a prototype of the natural gas – water separator is designed and built. Due to the limitations on size and pressure drop the intended $d_{p100\%}$ of 1 micron could not be achieved. The current prototype has a $d_{p100\%}$ of 2 micron at an operating pressure of 80 bar. The prototype is capable to handle the volume flow of one wellhead and can be installed in-line. The design involves a so-called naturally driven RPS. This means that the filter element rotates as a result of a swirl, which is generated in the flow approaching the element. As a consequence there is no need for an external motor and therefore no shaft seals are required. The prototype has an overall length of 0.6 m and an outer diameter of 0.36 m.

## 5.2 Performance of prototype

The performance of the prototype was determined by measuring its pressure drop, rotational speed and separation performance. The first two parameters were measured at atmospheric conditions as well as at elevated pressures. Separation performance of the separator was measured at atmospheric conditions.

The pressure drop over and the angular speed of the separator were measured as a function of the flow rate through the separator. It appears that the pressure loss due to friction in the channels of the filter element for laminar fluid flow is best represented by a model assuming sinusoidally shaped channels. This is also expected as the filter element channels are more or less sinusoids. Furthermore, it can be concluded that the pressure loss due to friction in the filter element channels and the dynamic pressure loss in the swirl generator are the main contributors to the total pressure loss over the separator. The measured total pressure loss over the separator is somewhat higher
than expected. This is mainly due to the fact that some flow phenomena occurring at the vanes of the swirl generator are neglected in the model.

The measurements of the angular speed of the filter element under atmospheric conditions show, except for a slight deviation at low flow rates, good agreement with the theoretical predictions. Also the onset of rotation, due to static bearing friction, is well predicted by theory. The angular speed of the filter element at elevated pressures is also in good agreement with the predicted speed.

Overall, the hydrodynamic performance of the separator can be well predicted at both atmospheric conditions and at elevated pressures.

Initially it was intended to use condensate droplets to measure the separation performance of the natural gas – water separator as a function of the droplet size. The reason is that by means of condensation droplets can be generated in the size range in which the separator operates. To insert the condensate droplets into the test setup an appropriate injection system had to be found. Various injection systems were tested by measuring droplet sizes after injection by means of Mie scattering. However, no suitable injection system was found. Therefore, it was decided to use solid particles to measure the efficiency of the prototype.

The separation efficiency of the prototype was determined by injecting slaked lime into the low pressure test setup and subsequently measure the size distribution of those particles upstream as well as downstream of the filter element by means of a low pressure impactor. With the impactor particles in the size range of 0.4 – 10 micron can be detected. The injected particles had a peak concentration between 1 and 3 micron, while the filter element of the prototype has a $d_{p100\%}$ of 3 micron at the measurement conditions.

The obtained results are compared with previously obtained experimental results and with theoretical relations predicting the efficiency of the entire filter element. Those previous experiments were performed with an externally driven tangential version of the RPS and with the same type of low pressure impactor. From these comparisons two main conclusions can be drawn. In the first place, the naturally driven RPS has the same particle collection efficiency as the externally driven RPS. Secondly, the measurement results show good agreement with the theoretical predictions.

5.3 Recommendations

In order to integrate the expansion and separation step in a single device, the amount of critical nuclei after expansion should be accurately determined. However, current nucleation models are not able to quantitatively predict the nucleation rate at high pressures. Therefore, more research in this area is required to properly design compact, high pressure, phase separation equipment, which combine expansion and separation in a single unit.

The efficiency of the prototype is determined for laminar flow conditions in the channels of the filter element. Under field test conditions, however, the flow in the filter element is turbulent. Due to turbulence the radial migration of droplets towards the channel walls may be disturbed. To determine the influence of turbulent channel flow on the filter efficiency, measurements should be performed at higher flow rates.
and higher angular speeds. These measurements could be performed with the lime particles as was done in this study. However, it would be better to use droplets. The reason is that not only the influence of turbulence on the radial migration of the droplets in the channels of the filter element may influence the separation efficiency, but also the behavior of the liquid film in the channels of the filter element and at the outer wall of the post-separator may be influenced by the turbulent flow conditions. Furthermore, liquid droplets break-up more easily compared to solid particles. For example, by using liquid as the dispersed phase it can be verified if the relations, which are derived to predict the minimum droplet size in the post-separation area, hold.
Bibliography


[56] Rawlins, H., Ting, F., Testing of an In-Line Rotary Separator (IRIS™) at the Chevron F. Ramirez Gas Production Facility, 52nd Annual Laurence Reid Gas Conditioning Conference, University of Oklahoma, 2002.


[76] www.carmeuse.nl.


[78] www.thermo.com/com/cda/product/detail/0,1055,22485,00.html.


Appendix A

Physical properties

In this appendix the physical properties, which are used in this study, are given.

A.1 Methane

The mass density of the methane is determined by the non-ideal gas law [22]

\[
\rho_g = \frac{PM_g}{Z_gR_{\text{univ}}T}
\]  

(A.1)

where \( P \) is the pressure, \( M_g \) the molar mass of the gas (for methane \( M_g = 0.01604 \) kg mole\(^{-1} \) [45]), \( Z_g \) the compressibility factor (for methane \( Z_g \approx 0.9 \) [22]), \( R_{\text{univ}} \) the universal gas constant (\( = 8.314 \) J mole\(^{-1} \) K\(^{-1} \)) and \( T \) the temperature. The dynamic viscosity of methane is calculated with FYSCAL [22].

A.2 Natural gas

Molar mass: 0.018638 kg mole\(^{-1} \) (Slochteren gas [22]).
Density and kinematic viscosity of natural gas are calculated with FYSCAL [22]. The composition of Slochteren gas is used.

A.3 Sulphur hexafluoride (SF6)

Molar mass: 0.146 kg mole\(^{-1} \) [74]
Density: taken from measurements of Wilhelm [72]
Dynamic viscosity: taken from measurements of Wilhelm [72]

A.4 Water

The data presented in this section are taken from Peeters [51] unless stated otherwise.
Molar mass: 0.018015 kg mole\(^{-1} \).
The temperature dependent density of water is given by
\[ \rho_l(T) = 999.84 + 0.086(T - 273.15) - 0.0108(T - 273.15)^2 \]  
(A.2)

The saturated vapor pressure of water is given by
\[ P_s(T) = 610.8 \exp[-5.1421 \ln(T/273.15) - 6828.77(1/T - 1/273.15)] \]  
(A.3)

**Surface tension**

The liquid surface tension is influenced by the pressure of the gas phase (carrier gas). Due to the adsorption of the gas phase the surface tension decreases. This is expressed as
\[ \sigma(P) = \sigma_0 - n_a k_B T \ln \left( \frac{P + P_l}{P_l} \right) \]  
(A.4)

where \( \sigma(P) \) is the surface tension of the liquid, \( \sigma_0 \) the surface tension of the pure substance, \( n_a \) the inverse of molecular area, \( k_B \) the Boltzmann’s constant (=1.38 \cdot 10^{-23} \text{ J K}^{-1} \), \( T \) the temperature, \( P \) the total pressure and \( P_l \) the Langmuir pressure.

\[ \sigma_0(T) = 0.127245 - 1.89845 \cdot 10^{-4} T \]  
(A.5)

\[ P_l(T) = (-481.95 + 2.1211 T) \cdot 10^5 \]  
(A.6)

\[ n_a \simeq 5.4 \cdot 10^{18} \]  
(A.7)

**Enhancement factor**

The enhancement factor is only a function of temperature and pressure
\[ f_e = \exp[b (P - P_s) + c (P - P_s)^2] \]  
(A.8)

For water in methane the parameters \( b(T) \) and \( c(T) \) are given by
\[ b(T) = (-7.0874 \times 10^{-3} + 3.4131 T^{-1} + 1.40 T^{-2}) \cdot 10^{-5} \]  
(A.9)

\[ c(T) = (-1.2070 \times 10^{-4} + 2.6726 \times 10^{-2} T^{-1} + 2.3585 T^{-2}) \cdot 10^{-10} \]  
(A.10)

The equation for the enhancement factor is valid up to pressures of 100 bar.

**Diffusion coefficient**

The diffusion coefficient for water in methane is given by [53]
\[ D = 2.396 \times 10^{-7}(T)^{0.75} \rho_g^{-1} \]  
(A.11)
Appendix B

Swirl generator

The swirl generator must be designed such that the required momentum is generated to drive the filter element and to overcome losses in the other parts of the separator. In the swirl generator the gas flow is forced through a small annular duct. At the end of the duct vanes are placed, which give a tangential velocity component to the gas. In order to calculate the momentum generated by the swirl generator, the axial gas velocity at the exit of the duct \( u_{sg} \) must be known (equation (3.47)). This velocity can be retrieved from flow relations of an ideal gas with viscous effects in a duct of constant cross-sectional area as described by Owczarek [50]. From these relations also the pressure loss due to friction in the duct can be calculated. In this appendix the relations necessary to derive the exit velocity and pressure drop in the duct are given.

First the Mach number at the inlet of the duct \( M_1 \) is calculated from the rate of mass flow through the duct, assuming a homenergic and homentropic gas flow

\[
\frac{\dot{m} \sqrt{RT_1}}{A_d P_1} = \sqrt{\gamma M_1 \left( 1 + \frac{\gamma - 1}{2} M_1^2 \right)^{\frac{\gamma+1}{2(\gamma-1)}}}
\]

where \( \dot{m} \) is the rate of mass flow through the duct, \( R \) the gas constant (is the universal gas constant divided by the molar mass), \( T_1 \) the inlet temperature, \( A_d \) the cross-sectional area of duct, \( P_1 \) the inlet pressure and \( \gamma \) the isentropic exponent.

When the value of \( M_1 \) is known the velocity and also the Reynolds number at the inlet of the duct can be calculated. With this value of the Reynolds number, the value of the friction coefficient at the inlet of the duct \( f_1 \) can be evaluated from the von Kármán-Nikuradse equation for a fully developed turbulent flow in a smooth duct

\[
\frac{1}{\sqrt{4f}} = -0.8 + 2 \log \text{Re}_Dh \sqrt{4f} \]

where \( \text{Re}_Dh \) is the Reynolds number based on the hydraulic diameter \( D_h \) of the duct

\[
\text{Re}_Dh = \frac{\rho u D_h}{\eta}
\]

Entrance effects are disregarded in the current calculations.

The length of the duct \( L \) over which the Mach number changes from \( M_1 \) to \( M_2 \) is...
given by

\[
\frac{4\overline{f}L}{D_h} = \left(\frac{4\overline{f}L^*}{D_h}\right)_{M_2} - \left(\frac{4\overline{f}L^*}{D_h}\right)_{M_1}
\]  

(B.4)

where \( \overline{f} \) denotes the average friction coefficient in the duct defined by

\[
\overline{f} = \frac{1}{x^* - x} \int_x^{x^*} f \, dx
\]  

(B.5)

and \( L^* \) is the length of the duct, which is sufficient to change the Mach number from any value \( M \) to the value unity (sonic condition). In a constant area adiabatic duct, which is sufficiently long, the friction accelerates the flow to the condition \( M=1 \). Under such conditions the duct is said to be choked because the maximal mass flow, which the duct can pass, is reached for the given inlet conditions. Further friction in a constant area duct merely causes a reduction of mass flow through the duct. The length of the duct \( L^* \) at which sonic flow is reached for a given Mach number \( M \) is given by

\[
\frac{4\overline{f}(L^*)}{D_h} = \frac{1 - M^2}{\gamma M^2} + \frac{\gamma + 1}{2\gamma} \ln \left( \frac{\frac{\gamma + 1}{2} M^2}{1 + \frac{\gamma - 1}{2} M^2} \right)
\]  

(B.6)

Now for a fixed value of the hydraulic diameter and the length of the duct, the Mach number at the outlet of the duct \( M_2 \) can be calculated from equation (B.4) and equation (B.6), provided that the friction factor in the duct remains constant. The friction coefficient is only dependent on the Reynolds number (equation (B.2)). The Reynolds number only changes due to variations of the dynamic viscosity \( \eta \) with the temperature of the gas, as the mass velocity \( \rho u \) is constant because the duct area is constant. The variation of \( \eta \) with \( T \) is small for gases. Furthermore for a turbulent flow the friction factor \( f \) is only a weak function of \( \text{Re}_{D_h} \). This means that the friction factor \( f \) can be assumed constant throughout the duct \( (\overline{f} = f_1) \).

When the Mach number at the duct outlet is known, the pressure at the exit of the duct, \( P_2 \), can be found from the expression for the rate of mass flow in the duct

\[
\frac{\dot{m} \sqrt{RT_1}}{A_d P_1} = \frac{P_2}{P_1} \sqrt{\gamma} M_2 \sqrt{1 + \frac{\gamma - 1}{2} M_2^2}
\]  

(B.7)

The pressure drop due to friction in the duct is the difference between the inlet pressure \( P_1 \) and the pressure at the exit of the duct \( P_2 \).
Appendix C

Friction factor in laminar flow for different channel geometries

In this appendix the laminar friction coefficients for three different channel geometries of the filter element are calculated. It is common to relate the pressure drop as a consequence of wall friction in a channel to the mean flow velocity and the friction coefficient as

$$\Delta P = f \frac{L}{D_h} \frac{1}{2} \rho_f u_a^2 \quad \Delta P = (P_2 - P_1) < 0$$  \hspace{1cm} (C.1)

where $\Delta P$ is the pressure drop over the channel, $f$ the friction coefficient, $L$ the length of the pipe, $D_h$ the hydraulic diameter of the pipe, $\rho_f$ the density of the fluid and $u_a$ the mean axial fluid velocity. For the pressure drop $\Delta P$ holds

$$\frac{dP}{dx} = \frac{\Delta P}{L}$$ \hspace{1cm} (C.2)

Figure C.1. Force balance on an element of a channel.
The hydraulic diameter is defined as

\[ D_h = \frac{4A}{P} \]  

where \( A \) is the cross-sectional area of the channel and \( P \) the wetted perimeter. The friction factor can be derived by applying a force balance on a liquid element in the channel, see figure C.1.

\[ \Delta P A = \tau_w P L \]  

The shear stress at the wall \( \tau_w \) can be written as

\[ \tau_w = \eta \left| \frac{du}{dy} \right|_{wall} \]  

where \( \eta \) is the dynamic viscosity of the fluid, \( u \) the axial velocity in the channel and \( y \) the coordinate in the direction normal to the plane on which the shear stress acts.

Brouwers [14] derived expressions for the parabolic axial velocity profiles in triangularly and sinusoidally shaped channels of the filter element. In these expressions it is assumed that the width of the channels is much larger than the height, as is the case in the filter element. Based on these expressions the friction coefficients for triangularly and sinusoidally shaped channels are derived. First, the friction coefficient in circular channels is deduced.

### C.1 Circular channels

In circular channels, in which a laminar Poiseuille flow is present, the axial velocity profile is given by [58]

\[ u = 2u_a \left( 1 - \frac{r^2}{R^2} \right) \]  

where \( u_a \) is the axial velocity averaged over the entire cross-section, \( r \) is the radial coordinate and \( R \) is the radius of the channel. The wall shear stress can be calculated from equation (C.5) and equation (C.6) as

\[ \tau_w = \eta \left| \frac{du}{dr} \right|_{r=R} = \frac{4 \eta u_a}{R} \]  

The total force over the perimeter of the channel as a result of the wall shear stress is given by

\[ F = \tau_w P = \frac{4 \eta u_a}{R} 2\pi R = 8\pi \eta u_a \]  

Substituting equation (C.8) in equation (C.4) gives the pressure drop over the channel

\[ \Delta P = \frac{8 \eta u_a L}{R^2} \]  

Rewriting equation (C.9) in the form of equation (C.1) gives
\[\Delta P = \frac{64}{Re} \frac{L}{d_c} \frac{1}{2} \rho f u_a^2 \]  
(C.10)
where Re is the Reynolds number based on the diameter of the channel \(d_c\) and the average velocity \(u_a\). In the case of a circular cross-section the hydraulic diameter is equal to the diameter of the channel. It follows that the friction coefficient for circular shaped channels is given by
\[f = \frac{64}{Re} \]  
(C.11)

## C.2 Triangular channels

For triangular channels, with base width \(b\) and maximum height \(d_c\), see figure C.2, the perimeter \(P\) is given by
\[P = b + \frac{b}{\sin(\phi)} \]  
(C.12)
From equation (C.3) it follows that the hydraulic diameter of such channels is given by
\[D_h = \frac{4A}{P} = \frac{4bd_c}{2} \left/ \left( b + \frac{b}{\sin(\phi)} \right) \right. = \frac{2d_c \sin(\phi)}{1 + \sin(\phi)} \]  
(C.13)
For triangular channels for which the base width \(b\) is much larger than the maximum height \(d_c\), the angle \(\phi \to 90^\circ\). From equation (C.12) it follows that the perimeter of such channels is equal to \(2b\). Substituting this value for \(\phi\) in equation (C.13) equals the hydraulic diameter to
\[D_h = d_c \]  
(C.14)
The axial velocity profile for a Hagen-Poiseuille type flow in triangular channels for which the base width \(b\) is much larger than the maximum height \(d_c\) is given by \([14]\)
\[u = \frac{12u_a}{d_c^2} (\mu h - h^2) \quad 0 \leq \mu \leq h \]  
(C.15)
where \(u_a\) is the axial velocity averaged over the entire cross-section, \(\mu\) the coordinate perpendicular to the base and \(h\) the local height of the triangle given by
\[h = d_c - \frac{2|\zeta| d_c}{b} \quad -\frac{b}{2} \leq \zeta \leq \frac{b}{2} \]  
(C.16)
where \(\zeta\) is the coordinate along the base of the triangle and \(b\) the base width. Substituting equation (C.15) in the expression for the shear stress, equation (C.5), gives for the shear stress at the base of the channel
\[\tau_w = \eta \left| \frac{du}{d\mu} \right|_{\mu=0} = \frac{12 h u_a}{d_c^2} \]  
(C.17)
Friction factor in laminar flow for different channel geometries

Figure C.2. Local coordinates and dimensions of a triangularly shaped channel.

and for the two inclined sides of the channel the shear stress is given by

$$\tau_w = \eta \left| \frac{du}{d\mu} \right|_{\mu=h} = \frac{12}{d_c^2} \frac{h u_a}{\eta}$$  \hspace{1cm} (C.18)

For the total force $F$ over the perimeter of the triangle follows

$$F = \sum \tau_w P = 2 \eta \int_{-\frac{b}{2}}^{\frac{b}{2}} \frac{12}{d_c^2} \frac{h u_a}{\eta} \, d\zeta = 12 \eta \frac{b u_a}{d_c}$$  \hspace{1cm} (C.19)

Integration over $\zeta$ is allowed as for $d_c \ll b$ the two inclined sides of the triangle have approximately the same length as the base width of the triangle $b$. For the pressure drop over the channel follows

$$\Delta P = \frac{dP}{dx} = \frac{\tau_w P L}{A} = \frac{12}{b d_c} \frac{\eta u_a b}{\frac{1}{2} \frac{b d_c}{2}} = \frac{24}{d_c^2} \frac{\eta u_a L}{b}$$  \hspace{1cm} (C.20)

Rewriting equation (C.20) in the form of equation (C.1) gives

$$\Delta P = \frac{48}{Re} \frac{L}{d_c} \frac{1}{2} \rho_f u_a^2$$  \hspace{1cm} (C.21)

where $Re$ is the Reynolds number based on the maximum height of the channel $d_c$. It follows that the friction coefficient $f$ for triangularly shaped channels for which the base width is much larger than the maximum height is given by

$$f = \frac{48}{Re}$$  \hspace{1cm} (C.22)

C.3 Sinusoidal channels

For sinusoidally shaped channels for which the base width $b$ is much larger than the maximum height $d_c$ (figure C.3) the two inclined sides of the sinusoid have approximately the same length as the base width $b$. The perimeter is therefore equal to $2b$,
as was also the case for the triangular channels in section C.2. The local height $h$ of such sinusoidal channels, can be expressed as [14]

$$h = \frac{d_c}{2} + \frac{d_c}{2} \cos \frac{2\pi \zeta}{b} \quad \frac{-b}{2} \leq \zeta \leq \frac{b}{2} \tag{C.23}$$

where $\zeta$ is the coordinate along the base of the triangle. The cross-sectional area of such a channel is calculated as

$$A = \int_{-b/2}^{b/2} h \, d\zeta = \int_{-b/2}^{b/2} \frac{d_c}{2} + \frac{d_c}{2} \cos \frac{2\pi \zeta}{b} \, d\zeta = \frac{bd_c}{2} \tag{C.24}$$

Substituting the expression for the cross-sectional area and the perimeter in equation (C.3) gives the hydraulic diameter for sinusoid channels with $b \gg d_c$

$$D_h = \frac{4A}{P} = \frac{4bd_c}{2(2b)} = d_c \tag{C.25}$$

Figure C.3. Local coordinates and dimensions of a sinusoidally shaped channel.

The axial velocity profile for a Hagen-Poiseuille type flow in sinusoidally shaped channels is given by [14]

$$u = \frac{48}{5} \frac{u_a}{d_c^2} \left( \frac{5h}{5d_c^2} - h^2 \right) \quad 0 \leq \mu \leq h \tag{C.26}$$

where $u_a$ is the axial velocity averaged over the entire channel cross-section, $\mu$ the coordinate perpendicular to the width, $d_c$ the maximum channel height and $h$ the local height of the sinusoidal channel, given by equation (C.23). Substituting equation (C.26) in the expression for the shear stress, equation (C.5), gives for the shear stress at the base of the sinusoidally shaped channel

$$\tau_w = \eta \left. \frac{du}{d\mu} \right|_{\mu=0} = \eta \frac{48 h u_a}{5d_c^2} \tag{C.27}$$

and for the two inclined sides of the channel the shear stress is given by

$$\tau_w = \eta \left. \frac{du}{d\mu} \right|_{\mu=h} = \eta \frac{48 h u_a}{5d_c^2} \tag{C.28}$$
For the total force $F$ over the perimeter of the triangle follows

$$F = \sum \tau_w P = 2\eta \int_{\zeta}^{b} \frac{48 h u_a}{5d_c^2} \, d\zeta = \frac{48 \eta b u_a}{5d_c}$$  \hspace{1cm} (C.29)$$

Integration over $\zeta$ is allowed as for $d_c \ll b$ the two inclined sides of the sinusoid have approximately the same length as the base width $b$. For the pressure drop over the channel follows

$$\Delta P = \frac{dP}{dx} \frac{L}{A} = \frac{\tau_w P L}{A} = \frac{48 \eta u_a b}{2 bd_c} L = \frac{96 \eta u_a L}{5 d_c^2}$$  \hspace{1cm} (C.30)$$

Rewriting equation (C.30) in the form of equation (C.1) gives

$$\Delta P = \frac{192}{5 \eta} \frac{L}{d_c^2} \frac{1}{2f} \frac{u_a^2}{\rho}$$  \hspace{1cm} (C.31)$$

where $Re$ is the Reynolds number based on the maximum height of the channel $d_c$. It follows that the friction coefficient $f$ for sinusoidally shaped channels for which the base width $b$ is much larger than the maximum height $d_c$ is given by

$$f = \frac{38.4}{Re}$$  \hspace{1cm} (C.32)$$
Appendix D

Design drawings prototype

In this appendix the construction drawings of the prototype of the natural gas – water separator are presented. Except for the filter element, which was manufactured by Duis V.O.F. in Bladel, the prototype was manufactured by Schinfa Machinerevisie in Emmeloord. The assembly took place at the workshop of the Thermal Fluids Engineering section of the faculty Mechanical Engineering at the Technical University Eindhoven.
Figure D.1. Construction drawing of the shaft.

Figure D.2. Construction drawing of the filter element.
Figure D.3. Construction drawing of the bearing support at the inlet.
Figure D.4. Construction drawing of the bearing support at the outlet.

Overall roughness 1.6, outer diameter (360) 3.2
Tolerance slot : Inner radius R 80 +/- 0.05
Outer radius R 105.79 +/- 0.05
Figure D.5. Construction drawing of the flange in the post-separation area.

Figure D.6. Construction drawing of the swirl generator.
Figure D.7. Construction drawing of the housing.
Appendix E

Mechanically versus condensationally created droplets

In this appendix two mechanisms by which droplets are created, are discussed. First, the mechanical break-up process of droplets is considered. Examples of such processes are the atomization of liquid by nozzles, rotary generators, mechanical vibration etc. It is shown that in practice there is a lower size limit for droplets produced by break-up processes, since the energy necessary for atomization increases quadratically with decreasing particle size.

Second, the generation of droplets by condensation is discussed. The lower size limit of these droplets is mainly predicted by the saturation ratio and is expressed by the critical droplet radius (equation (3.1)).

E.1 Estimation of minimal droplet size for mechanical processes

In section 3.2.2 it was pointed out that droplets break up if the external force on the droplets becomes larger than the surface tension force. The external force may be caused by centrifugal forces, fluctuations of the turbulent flow, etc. To derive a general expression for the droplet size at which break-up occurs, we will regard a spherically shaped droplet with diameter $d_p$. The general expression for the external force on this droplet is

$$F_d = m_p a = \frac{1}{6} \pi \rho_p d_p^3 a$$

(E.1)

where $m_p$ is the mass of the droplet, $a$ the acceleration of the droplet and $\rho_p$ the density of the droplet.
The surface tension force of the droplet is given by

$$F_i = \pi d_p \sigma$$  \hspace{1cm} (E.2)

where $\sigma$ is the surface tension.

The droplet size at which break-up occurs can be found by equalling the external force and interfacial tension force

$$d_p = \sqrt{\frac{6 \sigma}{\rho_p a}}$$  \hspace{1cm} (E.3)

To get idea of the magnitude of the required acceleration to achieve a certain droplet size, the break-up of a water droplet at atmospheric conditions ($T=293$ K, $P=1.013$ bar) is considered. The properties of water are given in appendix A.4. In figure E.1 the droplet diameter versus acceleration is given. It can be seen that droplet size decreases for increasing acceleration. However after an acceleration of about $2 \cdot 10^7$ m s$^{-2}$ a further increase in acceleration hardly reduces the droplet diameter. This means that in order to mechanically create droplet sizes below 10 micron, a very high external force has to be applied.

![Figure E.1. Break-up droplet diameter versus acceleration for a water droplet at atmospheric conditions as calculated by equation (E.3).](image)

This result confirms the fact that atomizers, like nozzle, rotary atomizers etc. are not able to produce liquid droplets smaller than 5 to 10 micron in diameter [26]. Also in high pressure injection systems, like diesel injectors, used nowadays droplet sizes are restricted to ten micron or even larger [31].
E.2 Estimation of minimal droplet size for condensation

In section 3.2.1 it was explained that the formation of condensate droplets occurs by homogeneous or heterogeneous nucleation. In the latter case the vapor condenses on foreign agents, like ions or dust particles. This means that the smallest size of the condensate droplets is determined by the size of those particles. In the case of homogeneous nucleation, the vapor condenses without the assistance of condensation nuclei or ions. It is a statistical process, involving the formation of clusters of molecules of a new phase. As soon as a cluster is significantly larger than a certain critical size, the cluster becomes stable. The critical diameter $d_c$ of such a stable cluster is given by [24]

$$d_c = \frac{4 \sigma M_l}{\rho_l R_{\text{univ}} T \ln S} \quad \text{(E.4)}$$

where $M_l$ is the liquid molar mass, $R_{\text{univ}}$ the universal gas constant ($= 8.314 \, \text{J/mole K}$), $T$ the temperature, $\rho_l$ the liquid mass density and $S$ the saturation ratio. From this equation it follows that a state of supersaturation ($S > 1$) is necessary to form stable droplets.

![Figure E.2. Critical droplet diameter versus saturation ratio for water at 293 K as calculated by equation (E.4).](image)

The relationship between the saturation ratio and the critical droplet diameter for water at 293 K is given in figure E.2. It can be seen that the higher the saturation ratio the smaller the critical droplet size. This also explains the fact that homogeneous nucleation requires saturation ratios of 2-10, whereas heterogeneous nucleation can occur at a supersaturation of only a few percent [24]. For small saturation ratios the critical cluster size is large. This means that for small saturation ratios the chance
that enough molecules collide and stick together to form a cluster of the required critical size is small. The nucleation process will always follow the path along which the smallest amount of energy is necessary to build a stable cluster and therefore at small saturation ratios heterogeneous nucleation dominates.

For high saturation ratios critical droplet sizes are in the order of nanometers. However, once a stable cluster is formed it will immediately grow by condensation and coagulation, as was shown in section 3.2.1 until the saturation ratio falls below 1 or the droplet number concentration has reached its equilibrium value.
Appendix F

Hydrodynamic measurements at elevated pressure

In this appendix the results of the hydrodynamic measurements, which were performed in the test loop at CDS Engineering at average absolute pressures of 1.9 and 2.9 bar, are presented.
Figure F.1. Rotational speed of the filter element as a function of flow rate. Top: at an absolute mean pressure of 1.9 bar. Bottom: at an absolute mean pressure of 2.9 bar.
Figure F.2. Pressure loss over the filter element and the total pressure loss over the prototype as a function of flow rate. Top: at an absolute mean pressure of 1.9 bar. Bottom: at an absolute mean pressure of 2.9 bar.
Dankwoord

Dit onderzoek is uitgevoerd in samenwerking met de bedrijven CDS Engineering en Romico Hold en gesteund door het subsidieprogramma Economie, Ecologie, Technologie (EET).

De samenwerking met CDS Engineering heb ik altijd als zeer prettig ervaren. Vooral de directe link met de praktijk is mij zeer goed bevallen. Het contact met Rob Schook, Toine Hendriks, Philip van Dessel en Victor van Asperen heb ik als heel open ervaren en ik kon altijd terecht met vragen/problemen over proces en praktijk gegevens. Tevens ben ik jullie dank verschuldigd voor het uitvoeren van de metingen.

Ook vanuit de Technische Universiteit Eindhoven heb ik veel steun gehad. In de eerste plaats natuurlijk van mijn directe begeleiders Erik van Kemenade en Bert Brouwers. Hierbij zorgde de mix tussen de experimentele/praktische kennis van Erik en de meer fundamentele/analytische inbreng van Bert, alsmede zijn ideeën aangaande het ontwerp en de berekeningen van de RPS, voor vier prettige en leerzame jaren.

Daarnaast heb ik bij het verkennen van het voor mij nieuwe gebied van nucleatie en druppelgroei van condensaat in aardgas veel hulp gehad van Rinie van Dongen en Paul Peeters. Zij stonden altijd open voor vragen en leverden vele nuttige suggesties.

Voor het realiseren van het prototype en de testopstelling op de TU/e dank ik de medewerkers van de werkplaats en met name Geert-Jan van Hoek. Geert-Jan bedankt voor alle nuttige constructietips, je spierkracht en de fijne samenwerking!

Bij het uitvoeren van de efficiency metingen heb ik veel hulp gehad van Eddy Bramer van de Universiteit Twente. Dankzij zijn hulp heb ik op korte termijn de efficiency metingen met goed resultaat kunnen uitvoeren.

Verder wil ik de volgende studenten bedanken voor hun enthousiaste inzet tijdens de bouw van de lagedruk testopstelling en het uitvoeren van de metingen: Hugo Liebrand, Mart Grooten en Remco Heijmans. I would also like to thanks the Polish students Miroslaw Nosarzewski and Wojciech Zawiejski for their contribution to the experimental work.

In ontspanning is de afgelopen vier jaar ook ruimschoots voorzien. Met name de twee bedrijfsbezoeken aan Zwitserland hebben voor veel lol gezorgd. Ook tijdens de werkuren waren er vaak plezierige momenten met mijn buuraio’s Raymond, Ralph en Carlo en mijn kamergenote Milica. Verder wil ik alle medewerkers van de sectie Procestechnologie bedanken voor de fijne tijd.

Eva

Helmond, september 2005
Curriculum Vitae

20 februari 1978 : geboren te Goes

1990 - 1996 : Voorbereidend Wetenschappelijk Onderwijs, Het Goese Lyceum, Goes


2002 - 2006 : Assistent in Opleiding bij de sectie Proces Technologie, faculteit Werktuigbouwkunde, Technische Universiteit Eindhoven