Theoretical and numerical study of swirling flow separation devices for oil-water mixtures

Abdul Motin, Michigan State University

Available at: https://works.bepress.com/abdul-motin/2/
THEORETICAL AND NUMERICAL STUDY OF SWIRLING FLOW SEPARATION DEVICES FOR OIL-WATER MIXTURES

By

Abdul Motin

A DISSERTATION

Submitted to
Michigan State University
in partial fulfillment of the requirements
for the degree of

Mechanical Engineering – Doctor of Philosophy

2015
ABSTRACT

THEORETICAL AND NUMERICAL STUDY OF SWIRLING FLOW SEPARATION DEVICES FOR OIL-WATER MIXTURES

By

Abdul Motin

Oil-water separation is a critical aspect of produced water treatment, oil spill cleanup, and refining of petroleum products. Hydrocyclones are commonly used in these operations. A hydrocyclone is a device that separates two phases based on centrifugal forces acting on the two phases. Conventional hydrocyclones possess a finite turndown ratio and are effective for removing droplets greater than approximately ten microns. The understanding of hydrodynamic phenomena that limit the turndown ratio is crucial for improving hydrocyclone performance and finding a device that is reliable, efficient, and that has the potential to decrease the environmental footprint of oil and gas production. In this work, a quantitative understanding of the turndown ratio of an individual class hydrocyclone has been developed. A computational search is applied for redesigning the geometry of different modules of hydrocyclone. In addition, the desirable attributes of a crossflow filter and a vortex separator are combined into one unit to develop a crossflow filtration hydrocyclone (CFFH) for enhancing separation.

The hydrodynamic characteristics of single and multiphase flows encountered in hydrocyclones, the trajectories of dispersed droplets, interaction of phases that involve breakup and coalescence of dispersed droplets, and the geometry and operating principles that characterize the performances of a hydrocyclone are investigated based on computational fluid dynamic (CFD) simulations using the Eulerian-Lagrangian, the Eulerian-Eulerian, and a coupled CFD-PBM (Population balance method) approaches.
Results show that the finite turndown ratio in conventional hydrocyclones is a hydrodynamic effect that depends on the length of reverse flow core. Tailoring of hydrocyclone geometry with hyperbolic swirl chamber and new underflow outlet geometry significantly increases the separation efficiency and improves the turndown. Based on a parametric study, a novel hydrocyclone design is proposed that is able to achieve desired separation efficiency by a unit operation and possesses a large turndown. CFD studies were also performed on CFFH devices and showed that the swirl can aid in removing droplets from the membrane/filter surface.

The novel hydrocyclone identified provides a stable reverse flow core for an increased range of feed Reynolds numbers and yields less energy loss. With increasing the feed Reynolds number, the novel hydrocyclone gradually decreases the cut size (a size of droplet having 50% separation efficiency); this does not appear in a conventional hydrocyclone. For the feed Reynolds number of 60,000, the cut size in the novel hydrocyclone is less than 10 microns whereas the conventional hydrocyclone has a cut size of 65 microns and is ineffective for droplet less than 10 microns.
This dissertation is lovingly dedicated to my wife, Rezwana Kabir for her support and encouragement. This dissertation also dedicated with due respect to my parents, Mohammad Ali and Sahida Akter for their consistent counsel, support, and encouragement in all endeavors throughout my life.
ACKNOWLEDGMENTS

First of all, I would like to express my deep sense of gratitude, and acknowledge profound indebtedness to my supervisor, Professor André Bénard for his constant guidance, untiring help, invaluable suggestions and unceasing encouragement. He not only supports my academic research but also gives me personal guidance.

In addition, I feel highly grateful to Professors Charles A. Petty, Professor Farhad Jaberi, Professor Volodymyr V. Tarabara and John M. Walsh for serving on my dissertation committee. I would like to express special thanks to Professor Charles A. Petty for his immense support during the summer-2013 and guidance to explore the fundamental principles of hydrocyclone research. Moreover, I like to give special thanks to John M. Wash for his constant guidance toward developing models for droplet shearing and coalescence. I would also like to recognize Professor Merlin L. Bruening for his valuable suggestion and comments in the biweekly meeting of EPA project.

Financial support of this research from the US Environmental Protection Agency (EPA) grant RD-83518301, NSF PIRE project, and Department of Mechanical Engineering of Michigan State University is gratefully acknowledged. Financial support from CETCO Energy Services for my tuition in 2014-2015 academic year and participation in professional conferences is also gratefully acknowledged.

Finally sincere thanks are offered to members of my family and friends for their cooperation and inspiration during the work.
# TABLE OF CONTENTS

LIST OF TABLES .................................................................................................................. xi

LIST OF FIGURES .............................................................................................................. xii

KEY TO SYMBOLS AND ABBREVIATIONS ........................................................................ xxv

CHAPTER 1 .......................................................................................................................... 1

INTRODUCTION .................................................................................................................. 1

1.1. The physical problem ................................................................................................. 1

1.2. Background ................................................................................................................ 5

1.2.1. Numerical study on hydrocyclone ........................................................................ 7

1.2.2. Experimental study on hydrocyclone .................................................................... 13

1.2.3. Analytical study on hydrocyclone ........................................................................ 15

1.3. Objectives and scope of this thesis ............................................................................ 17

1.4. Methodology ............................................................................................................. 18

1.5. Outline of the thesis .................................................................................................. 19

CHAPTER 2 .......................................................................................................................... 21

HYDROCYCLONE DESIGN AND OPERATING PRINCIPLES ........................................... 21

2.1. Introduction ................................................................................................................ 21

2.2. Operating features of a hydrocyclone ....................................................................... 22

2.3. Flow features and separation mechanism .................................................................. 24

2.4. Geometric features of a hydrocyclone ...................................................................... 25

2.4.1. Inlet ...................................................................................................................... 26

2.4.2. Inlet chamber ....................................................................................................... 28

2.4.3. Swirl chamber ...................................................................................................... 29

2.4.4. Tailpipe ................................................................................................................ 30

2.4.5. Overflow and underflow orifice ......................................................................... 30

2.5. Operating and performance metrics .......................................................................... 31

2.5.1. Separation efficiency for dispersed phase mechanics ........................................... 31

2.5.2. Separation efficiency for multiphase phase mechanics ....................................... 32

2.5.3. Reduced separation efficiency ............................................................................. 32

2.5.4. The feed Reynolds number .................................................................................. 33

2.5.5. Overflow ratio ....................................................................................................... 33

2.5.6. Pressure drop ratio .............................................................................................. 34

2.5.7. Euler number ........................................................................................................ 35

CHAPTER 3 .......................................................................................................................... 36

NUMERICAL METHODS AND VALIDATION OF NUMERICAL RESULTS .................. 36

3.1. Introduction ................................................................................................................ 36

3.2. Numerical details and procedure .............................................................................. 37
7.4. Performance matrices of the novel hydrocyclone .................................................. 160
7.5. Velocity profiles in the novel hydrocyclone .......................................................... 161
7.6. Pressure profiles in the novel hydrocyclone .......................................................... 171
7.7. Multiphase fluid mechanics in the novel hydrocyclone ...................................... 176
7.8. Separation performance of the novel hydrocyclone ........................................... 180

CHAPTER 8 .................................................................................................................... 185
ROTATING TUBULAR MEMBRANE .............................................................................. 185
8.1. Introduction ............................................................................................................. 185
8.2. Geometry and test cases ....................................................................................... 186
8.3. Numerical method and boundary condition ......................................................... 189
8.4. Verification of the numerical simulations ............................................................... 190
8.5. Flow patterns ......................................................................................................... 192
8.6. Pressure distribution ............................................................................................. 197
8.7. Wall shear stress components .............................................................................. 198
8.8. Separation performance ....................................................................................... 201
8.8.1. Droplet cut size (d_{D50}) ................................................................................. 201
8.8.2. Grade efficiency ............................................................................................. 203

CHAPTER 9 .................................................................................................................... 209
MODELING OF DISPERSED DROPLETS AND COUPLED MULTIPHASE AND
POPULATION BALANCE SIMULATION ...................................................................... 209
9.1. Introduction ............................................................................................................. 209
9.2. Mechanism of droplets coalescence ..................................................................... 210
9.2.1. Collision frequency (h) ................................................................................... 212
9.2.2. Coalescence efficiency (\lambda) ....................................................................... 215
9.2.2.1. Drainage time (t_d) .................................................................................... 216
9.2.2.2. Compressing force, F ................................................................................. 218
9.2.2.3. Contact time (t_c) ...................................................................................... 219
9.3. Mechanisms of droplet breakage ......................................................................... 220
9.3.1. Breakage frequency .......................................................................................... 221
9.3.2. Daughter particle probability distribution function ........................................ 222
9.4. Effect of flow and material properties on the coalescence rate ......................... 223
9.4.1. Effect of turbulent energy dissipation ............................................................. 223
9.4.2. Effect of reference droplet size ....................................................................... 224
9.5. Population balance method (PBM) ..................................................................... 228
9.5.1. Linear grid discretization .................................................................................. 229
9.5.1.1. Coalescence ............................................................................................... 230
9.5.1.2. Breakage .................................................................................................... 231
9.5.1.3. Difficulties in the linear discretization of the PBE ..................................... 231
9.5.2. Geometric grid discretization of the PBE ......................................................... 232
9.5.2.1. Coalescence ............................................................................................... 234
9.5.2.2. Breakage .................................................................................................... 235
9.5.3. Time discretization .......................................................................................... 237
9.6. Preliminary results using the PBM ..................................................................... 237
LIST OF TABLES

Table 1.1 Common technologies used for produced water treatment and their removal capacity by particle size (Frankiewicz, 2001) ........................................3

Table 3.1: Material properties of the continuous and dispersed phases ......................37

Table 3.2: Geometric groups and dimensions of the single cone hydrocyclone ..........41

Table 3.3: Effect of mesh refinement on the predicted macroscopic properties ..........42

Table 4.1: Test cases and operational parameters used in the simulations ..................54

Table 5.1: Operating conditions for three different pressure drop ratios ..................87

Table 5.2: Operating conditions for the different cases present in Fig. 5.9 ...............96

Table 5.3: Operating conditions for the different angles of conical swirl chamber ....104

Table 6.1: Geometric groups and dimensions of hydrocyclone for the design cases .....116

Table 6.2: Operating conditions for the three designs of swirl chamber (see Fig. 6.1)...123

Table 6.3: Operating conditions for the hydrocyclone with different gadgets ..........138

Table 7.1: Geometric groups and their dimensions ................................................156

Table 7.2: Operating conditions for the numerical simulations in novel hydrocyclone ..158

Table 8.1: Parameters used in simulation test cases. Cases 1-5 correspond to a non-turbulent flow regime (Reₕ < 2300) and Case 6 to a transitional flow regime (2300 < Reₕ < 4000) (White, 1991) .........................................................188

Table 8.2: Flow ratio in studied test cases ..............................................................204

Table 9.1: Numerical values for the empirical constants .........................................221

Table 9.2: Volume averaged turbulent energy dissipation calculated in the novel hydrocyclone (Fig. 7.1) for the different test cases presented in Table 7.2...238

Table 9.3: The surface averaged energy dissipation is calculated at the different planes shown in the contours (Fig. 9.13) ..........................................................247

Table A1: Additional forces that are included in the analyses of dispersed droplets trajectories. The lift force is only applicable for submicron droplet ..........271
LIST OF FIGURES

Figure 1.1: A schematic of a hydrocyclone is illustrating internal flow structures as well as one inflow and the two outflows .................................................................6

Figure 1.2: Typical separation efficiency (defined in Eq. (2.8)) versus feed flow rate relationship for a liquid-liquid separation hydrocyclone. Turndown ratio = Q_{max}/Q_{min}) ........................................................................................................14

Figure 2.1: Shape and dimensions of (a) a reverse flow and (b) a forward flow hydrocyclone used for liquid-liquid separation. .........................................................23

Figure 2.2: Dual cone hydrocyclone used for liquid-liquid separation. .........................26

Figure 3.1: Schematic of a single cone hydrocyclone (Dimensions are given in Table 3.2) ....................................................................................................................40

Figure 3.2: An O-grid structured hexahedral mesh generated by using ANSYS ICEM CFD. Finer grids are created near the wall and at the core.................41

Figure 3.3: Effect of mesh refinement on the normalized tangential velocity at z/D = 1 and 8.38. Relative errors and grid convergence index shown in the table are calculated at a radial location where the differences in the tangential velocities are the maximum. .........................................................44

Figure 3.4: Effect of mesh refinement on the normalized axial velocity at z/D = 1 and 8.38. Relative errors and grid convergence index shown in table are calculated for the maximum difference in the axial velocity at r/R= 0........................................................................................................................................45

Figure 3.5: Effect of mesh refinement on the normalized pressure at z/D = 1 and 8.38. Relative errors and grid convergence index shown in table are calculated for the maximum difference in the normalized pressure at r/R = 0. ........................................................................................................................................46

Figure 3.6: (a) Effect of mesh refinement on the reduced grade efficiency and (b) Extrapolated value of the reduced grade efficiency with GCI as error bar for the finest grid (663812 cells). Extrapolated value of G_{OR} for the finest grid almost matches with that for 663812 cells. The finest mesh (663812 cells) provides a greater GCI for a smaller droplet size. However, the maximum GCI is less than 10%. .........................................................48

Figure 3.7: Qualitative comparison of tangential velocity calculated using the
RANS simulation on a 35 mm single cone hydrocyclone with that obtained from previous research on a similar hydrocyclone based on the LES approach and experimental investigation. Velocity profile is taken at z/D = 2.29. .......................................................... 50

Figure 3.8: Qualitative comparison of reduced grade separation efficiency $G_{OR}(d_D)$ calculated using the RANS simulation on a 35 mm single cone hydrocyclone with that obtained from previous research on a similar hydrocyclone based on the LES approach ............................................. 51

Figure 4.1: Normalized tangential velocity profiles for the different Reynolds numbers (--- $Re_F = 28,600$; ⋯⋯ $Re_F = 23,835$; - - - $Re_F = 17,160$; - - - - $Re_F = 11,440$). A higher normalized peak value represents a greater effective swirling intensity......................................................... 55

Figure 4.2: Contours of normalized tangential velocity for different feed Reynolds numbers.............................................................................................................. 58

Figure 4.3: Normalized axial velocity profiles for different Reynolds number (--- $Re_F = 28,600$; ⋯⋯ $Re_F = 23,835$; - - - $Re_F = 17,160$; - - - - $Re_F = 11,440$). The negative value represents the downward (forward) velocity and the positive value is the upward (reverse) velocity. The $Re_F = 17,160$ provides a greater reverse flow velocity than the other Reynolds numbers .......................................................................................... 59

Figure 4.4: Contours of normalized axial velocity for the different feed Reynolds numbers ............................................................................................................ 60

Figure 4.5: Plane view of velocity vectors in the inlet chamber for the $Re_F = 23,835$. The arrow head indicates the direction of flow. Only a 2D dimensional vector (radial and axial components) is shown in the figure. Short circuit boundary layer flow and some recirculation appear in the flow field ........................................................................................................ 61

Figure 4.6: Two dimensional velocity streamlines for the $Re_F = 23,835$ and three zones divided based on the flow pattern in the upper part of the inlet chamber .................................................................................................................. 62

Figure 4.7: Local radial flows through boundary B1, B2 and B3 as shown in Fig. 4.4. Positive magnitude is the radial inflow and the negative magnitude is the radial outflow ................................................................. 65

Figure 4.8: Effect of the Reynolds number on the contours of reverse (upward) flow velocity. Only the upward axial velocity is mapped and the downward flow field is discarded from the contour map. The length of reverse flow core first increases and then decrease with an increase in the
Reynolds number. .............................................................................................................66

Figure 4.9: Contours of normalized static pressure for the different feed Reynolds numbers...........................................................................................................69

Figure 4.10: Pressure distributions for the different Reynolds number (--- Re_F = 28,600; ....... Re_F = 23,835; - - - Re_F = 17,160; - - - - Re_F = 11,440). A positive axial pressure gradient is observed in the upstream and a negative axial pressure gradient in the downstream of the conical chamber.................................................................70

Figure 4.11: Distribution of centerline (core) pressure for different Reynolds numbers (--- Re_F = 28,600; ....... Re_F = 23,835; - - - Re_F = 17,160; - - - - Re_F = 11,440). z/D = 0 is at the top and z/D = 18 is at the bottom of the hydrocyclone. The location of peak core pressure shifts toward the underflow for the intermediate Reynolds numbers. The low (negative) core pressure at the top and at the end of swirl chamber occurs due to choking of axial flow by, respectively, the vortex finder and the entrance of tail pipe.................................................................71

Figure 4.12: Comparison of normalized length of reverse flow core (l_c/D) with the zero axial gradients of core pressure ((dP/dz)r = 0 = 0) and radial pressure gradient ((d(dP/dr)/dz)r = 0 = 0) for the understanding of the reverse flow core breakdown. The (d(dP/dr)/dz)r = 0 = 0 is a better indicator for both qualitative and quantitative understanding of the reverse flow core breakdown.................................................................73

Figure 4.13: Normalized radial pressure gradient for the different Re_F. With an increase in the Re_F, the location of reverse flow core breakdown (dotted line) moves downstream and then shifts toward overflow.............74

Figure 4.14: Reduced grade efficiency versus droplet size graphs for the different Reynolds numbers (---∆--- Re_F = 28,600; ···· Re_F = 23,835; - - · Re_F = 17,160; - - - - Re_F = 11,440). Droplets are assumed to be spherical. No coalescence and breakup effect is included. Insufficient swirl intensity in the case of Re_F = 11,440 and a shorter reverse flow core in the case of Re_F = 28,600 yields low grade separation efficiency.................................................................76

Figure 4.15: A comparison between the normalized length of reverse flow core (l_c/D) and the normalized cut-size (d_{D50}) for different Reynolds numbers. The cut size represents the size of droplets for which the reduced grade separation efficiency is 50%.................................................................78

Figure 4.16: Effect of the Re_F on the overall reduced separation efficiency (Fig. a) of the flooded hydrocyclone for the given droplet size distribution
The overall separation efficiency is almost the same for the Re_D between 17,000 and 24,000; it decreases for both the lower and higher Re_D. This phenomenon indicates a finite turndown ratio which is dependent on hydrodynamic characteristics.

Figure 4.17: Contours Capillary number for d_D = 130 μm and μ_D/μ_C = 3.3. A higher Capillary number is observed in the vortex finder and at the bottom of the conical chamber. However, the observed Ca is much less than a critical Ca_cr for a droplet to be sheared where Ca_cr > 1 (For a highly confined flow field, Ca_cr > 0.5 for a simple shear flow and Ca > 0.25 for an elongational flow) [Ca = μ_DEdDσ; E = (2E/E), and E = (∇U + ∇UT)/2].

Figure 4.18: Contours of Weber number for d_D = 130 μm and μ_D/μ_C = 3.3. A Weber number of greater than a critical Weber number, We_cr indicates a probability of shearing of dispersed droplets. [We = ρC_D5/3ε2/3/σ].

Figure 4.19: Profiles of the normalized volume fraction at the top (z/D = 1) and bottom (z/D = 8.3) of the swirl chamber for the Re_F = 23,835, φ_F = 0.01 and feed droplet size = 30 microns.

Figure 5.1: Effect of pressure drop ratio on the overflow rate, overflow and underflow Euler numbers for the Re_F = 23,835.

Figure 5.2: Normalized tangential velocity profiles. (—ΔP* = 1.0; •••• ΔP* = 1.45; − − − ΔP* = 2.0). A higher peak value represents a greater effective swirling intensity.

Figure 5.3: Normalized axial velocity profiles. (—ΔP* = 1.0; •••• ΔP* = 1.45; − − − ΔP* = 2.0). The diameter of reverse flow core increases with an increase in the ΔP*.

Figure 5.4: Normalized radial velocity profiles. (—ΔP* = 1.0; •••• ΔP* = 1.45; − − − ΔP* = 2.0). The radial inward velocity increases with an increase in the ΔP*.

Figure 5.5: Normalized pressure profiles. (—ΔP* = 1.0; •••• ΔP* = 1.45; − − − ΔP* = 2.0). The radial pressure gradient increases with a decrease in the ΔP*.

Figure 5.6: 2D streamlines and velocity vectors on a plane at the center of the hydrocyclone for the different ΔP*s. The flow circulation decreases with an increase in the ΔP*. 
Figure 5.7: Local radial flows through boundary B1, B2 and B3 as shown in Fig. 4.6. Positive magnitude is the radial inflow and the negative magnitude is the radial outflow. .................................................................95

Figure 5.8: Grade separation efficiency versus droplet size graphs for the different pressure drop ratio, ΔP*. The ΔP* of 2.0 significantly reduces the separation efficiency. ..............................................................................95

Figure 5.9: Schematic of vortex finder and overflow tube and their dimensions. ..............97

Figure 5.10: 2D streamlines and velocity vectors on a plane at the center of the hydrocyclone for the different design of vortex finder and overflow tube shown in Fig. 5.9. The case 4 provides a higher flow circulation and longer path of flow stream. .................................................................99

Figure 5.11: Local radial flows as a percentage of total overflow rates for different cases presented in Table 5.2. Q1, Q2 and Q3 are the radial flow, respectively, through the boundary B1, B2 and B3 as shown in Fig. 4.6. QT is the sum of radial flow through the three boundaries. ...............100

Figure 5.12: Normalized tangential velocity profiles for the different cases presented in Table 5.2. Vortex finder design does not affect the swirling motion in the inlet chamber. .................................................................102

Figure 5.13: Normalized axial velocity profiles for the different cases presented in Table 5.2. The diameter of reverse flow core directly related to the diameter of overflow tube. ........................................................................102

Figure 5.14: Normalized axial velocity profiles for the different cases presented in Table 5.2. Case 4 provides least amount of short circuit radial inflow in the inlet chamber. ........................................................................103

Figure 5.15: Normalized pressure profiles for the different cases presented in Table 5.2. .................................................................................................................................103

Figure 5.16: Effect of cone angle, θ on the overflow and underflow Euler numbers. .....105

Figure 5.17: Effect of cone angle on the normalized tangential velocity at z/D = 1. An increase in the cone angle reduces the cross-sectional area of the swirl chamber, which thereby increases the centrifugal acceleration. A higher centrifugal acceleration supports a greater migration rate of dispersed droplet toward the center. .................................................................107

Figure 5.18: Effect of cone angle on the normalized axial velocity at z/D = 1. ..........108

Figure 5.19: Effect of cone angle on the normalized radial velocity at z/D = 1.........108
Figure 5.20: Effect of cone angle on the normalized pressure at z/D = 1. A higher cone angle enhances the radial pressure gradient, which thereby support a greater migration rate of dispersed droplets toward the center. ................................................................. 110

Figure 5.21: Effect of the cone angle on the pressure along the center of the hydrocyclone. The positive axial pressure gradient from the location of peak core pressure to the overflow orifice creates reverse flow core. The cone angle of 2 degree provides the location of the peak core pressure far downstream which indicates a longer reverse flow core. ...... 111

Figure 5.22: Contours of reverse flow core for the different cone angles. The axially upward velocity is mapped on the plane. The axially downward and all the other velocity components are discarded from the contour map. The length of the reverse flow core increases with a decrease in the angle of the conical swirl chamber ................................................................. 112

Figure 5.23: Effect of cone angle on the grade separation efficiecy. ......................... 114

Figure 5.24: Relation between the normalize length of reverse flow core and cut-size. .................................................................................................................. 114

Figure 6.1: Schematic profile of three different swirl chambers (a) Design-A (parabolic swirl chamber), (b) Design-B (parabolic swirl chamber), and (c) Design-C (hyperbolic swirl chamber). .................. 117

Figure 6.2: Effect of swirl chamber designs on the tangential velocity profile at z/D = 1 for Re_f = 23835. Design-C (hyperbolic swirl chamber) provides a better swirling intensity, which can enhance the migration rate of dispersed droplets toward the center. ................................................................. 121

Figure 6.3: Effect of swirl chamber designs on the core pressure profiles. A lower core pressure for the Design-C indicates less energy losses in the hydrocyclone chambers. ................................................................. 122

Figure 6.4: Normalized radial pressure differential between the wall and core. P_C and P_W represent the pressure on the wall and at the center of hydrocyclone, respectively. ................................................................. 122

Figure 6.5: Local radial flows as a percentage of total overflow rate for the three different designs of swirl chamber (see Fig. 6.1) having dimension presented in Table 6.1 (Case-2). Q1, Q2 and Q3 are the radial flow, respectively, through the boundary B1, B2 and B3 as shown in Fig. 4.6................................................................. 124

Figure 6.6: Effect of swirl chamber designs on the normalized tangential velocity at
z/D = 1. The Design-C (hyperbolic swirl chamber) yields a higher centrifugal acceleration, which thereby increases the migration rate of dispersed droplets toward the center..............................126

Figure 6.7: Effect of the swirl chamber designs on the normalized axil velocity at z/D = 1. The parabolic swirl chamber (design-B) provides a greater reverse flow velocity than that of other two designs. ..............................126

Figure 6.8: Effect of the swirl chamber design on the normalized radial velocity at z/D = 1. The parabolic swirl chamber (design-B) provides a lower radially inward flow at the interface between the inner and outer vortex. ..............................127

Figure 6.9: Effect of the swirl chamber designs on the normalized static pressure at z/D = 1. The hyperbolic swirl chamber provides a greater radial pressure gradient near the core. ............................................................127

Figure 6.10: Effect of the swirl chamber designs on the normalized static pressure on the wall (dotted lines) and normalized pressure differential between wall and center (solid lines). ............................................................128

Figure 6.11: Effect of the swirl chamber designs on the static pressure along the center of the hydrocyclone. Core pressure profile follows the wall curvature of swirl chamber and a higher wall curvature yields less core pressure. ..................................................................................132

Figure 6.12: Contours are showing the reverse flow core for the three different swirl chamber designs. Only the reverse flow velocity is mapped and the forwarded flow velocity component is discarded from the contours........133

Figure 6.13: Grade separation efficiency graphs for the three designs. Design-B and design-C provides almost same separation efficiency. .........................134

Figure 6.14: Relation between length of reverse flow core and cut size for the three different swirl chamber designs (see Fig. 6.1). .................................134

Figure 6.15: Trajectories of dispersed droplets in a conventional hydrocyclone. The black arrows indicate the direction of motion of migrated dispersed droplets in the core.................................................................136

Figure 6.16: Schematic of a hydrocyclone having a tail gadget. ........................................137

Figure 6.17: Schematic diagram of (a) gadget-A and (b) gadget-B and their dimensions. .........................................................................................138

Figure 6.18: Effect of the design of gadget on the core pressure profile. .......................141
Figure 6.19: Effect of tail gadget on the grade separation efficiency........................................141

Figure 6.20: Schematic diagram of the modified hydrocyclone (design case-1) and its geometric dimensions. The hydrocyclone having hyperbolic swirl chamber and dimensions presented in Table 6.1(Case-2) is cut-off at z/D = 8.7. ..........................................................................................143

Figure 6.21: Normalized static pressure along the center and a contour of normalized static pressure in the modified hydrocyclone (design case-1) for the Re_F = 23,835. ..........................................................................................144

Figure 6.22: Reduced grade separation efficiency and a contour of reverse flow core in the modified hydrocyclone (design case-1) for the Re_F = 23,835. ........145

Figure 6.23: A schematic diagram of modified hydrocyclone (design case-2) and its geometric dimensions. The hydrocyclone having hyperbolic swirl chamber and the dimensions presented in Table 6.1(Case-2) is cut off at z/D = 5 and a tail pipe is connected.................................................................146

Figure 6.24: A normalized static pressure along the center and a contour of normalized static pressure in the modified hydrocyclone (design case-2) for the Re_F = 23,835. ..........................................................................................148

Figure 6.25: The reduced grade separation efficiency plot, and contours of reverse flow core and normalized tangential velocity in the modified hydrocyclone (design case-2) for the Re_F = 23,835.................................149

Figure 6.26: Schematic diagram of the modified hydrocyclone (design case-3) and its geometric dimensions......................................................................................................................150

Figure 6.27: A plot for normalized static pressure along the center, and contours of reverse flow core and normalized tangential velocity in the modified hydrocyclone (design case-2) for the Re_F = 23,835.................................151

Figure 7.1: A schematic of the novel hydrocyclone is showing detail designs. ........155

Figure 7.2: A three dimensional illustration of the novel hydrocyclone with flow streams. .................................................................................................................................159

Figure 7.3: 2D velocity streamlines in the novel hydrocyclone for the different feed Reynolds number. Only axial and radial components are plotted and the tangential component is projected on the plan..............................................................165

Figure 7.4: Effect of the feed Reynolds numbers on the normalized tangential velocity. – – – Re_F = 11,440; -------Re_F = 23,835; – – – Re_F = 34,320; – – Re_F = 45,760 and ——— Re_F = 68,640.................................................................166
Figure 7.5: Contours of normalized tangential velocity for the different feed Reynolds number. .................................................................167

Figure 7.6: Effect of the feed Reynolds number on the normalized axial velocity. – • – Re_F = 11,440; – – – Re_F = 23,835; – – – Re_F = 34,320; – – – Re_F = 45,760 and – – – Re_F = 68,640. .................................................................................168

Figure 7.7: Contours of normalized axial velocity for the different feed Reynolds numbers .................................................................................................................................169

Figure 7.8: Contours of reverse flow core for the different feed Reynolds numbers. The axially upward velocity component is mapped on the plane. The axially downward and all the other velocity components are discarded from the contour map. For the Re_F = 11,440, the swirl intensity is not sufficient to maintain a strong inner vortex and a higher downward velocity (see Fig. 7.6) breaks down the core. For a higher Re_F, the novel hydrocyclone provides a stable and a longer reverse flow core. ......170

Figure 7.9: Effect of the feed Reynolds number on the normalized static pressure. – • – Re_F = 11,440; – – – Re_F = 23,835; – • – Re_F = 34,320; – – – Re_F = 45,760 and – – – Re_F = 68,640. .................................................................................173

Figure 7.10: Contours of normalized static pressure for the different feed Reynolds numbers .................................................................................................................................174

Figure 7.11: Effect of the Re_F on the normalized static pressure along the axis (r/D=0) of the hydrocyclone. – • – Re_F = 11,440; – – – Re_F = 23,835; – • – Re_F = 34,320; – – – Re_F = 45,760 and – – – Re_F = 68,640. For the wide range of Re_F, the similar core pressure profiles and the constant core pressure up to the end of cylindrical chamber support long and stable reverse flow core (see Fig. 7.8). ........................................175

Figure 7.12: Effect of the feed Reynolds number on the normalized static pressure on the wall (solid lines) and the normalized pressure differential between wall and center (dotted lines). .................................................................................176

Figure 7.13: Distribution of volume fraction at different axial locations in the hydrocyclone for the Re_F = 23,835, feed droplet size of 20 microns, feed volume fraction of 0.1. The swirl chamber-2 significantly removes the dispersed droplets from near the wall and migrates them toward the center. .........................................................................................178

Figure 7.14: Effect of feed droplet size on the normalized volume fraction at z/D =10 for the \( \phi_F = 0.01 \) and the Re_F = 23835. .........................................................................................179
Figure 7.15: Effect of feed volume fraction on the normalized volume fraction at z/D = 10 for the dD = 10 microns and the ReF = 23835.

Figure 7.16: Effect of the feed Reynolds numbers on the overflow reduced grade separation efficiency. Solid and dotted lines represent the grade separation efficiency relatives to overflow-1 and overflow-2, respectively. (oReF = 68,640; ■ ReF = 45,760; ● ReF = 34,320; ● ReF = 23,835; ▲ ReF = 11,440)

Figure 7.17: Effect of the feed Reynolds numbers on the total reduced grade separation efficiency. (oReF = 68,640; ■ ReF = 45,760; ● ReF = 34,320; ● ReF = 23,835; ▲ ReF = 11,440)

Figure 7.18: Relation between normalized length of reverse flow core and cut size for the different Reynolds numbers.

Figure 7.19: Overall reduced separation efficiency for the different feed droplet sizes and underflow concentration calculated based on the multiphase flow simulations. φF = 0.01 (10,000 ppm) and ReF = 23,835.

Figure 7.20: Effect of feed concentration on the overall reduced separation efficiency.

Figure 8.1: (a) Illustration of the geometry of a crossflow filtration system wherein the microfiltration membrane is rotating about the vertical axis with an angular velocity of ω; (b) view of the membrane defined by a square box shown in (a). Arrows are used to represent the flow in the axial, radial and azimuthal directions.

Figure 8.2: A longitudinal cross-section of the CFF system simulated. Velocity on Γi, ambient pressure on Γ∞, and 50 kPa on ΓO are specified.

Figure 8.3: Axial and radial velocity profiles at z/Lm = 0.8 for the non-rotating CFF system (Sw = 0). with ReF = 1415. Both axial and radial velocity profiles obtained from numerical simulations match the approximate analytical solution (Eqs. (8.4) and (8.5)) with a maximum deviation of 2%.

Figure 8.4: Normalized axial velocity profiles for the CFF system with ReF = 1415. The axial flow velocity decreases downstream due to the presence of a permeate flux. Axial velocity near the core region increases with an increase of the Swirl number. Uz(0,0) = 0.475 m/s is the velocity at the centerline of the flow within inlet plane.

Figure 8.5: Radial velocity profiles at different axial locations for the CFF system with ReF = 1415. The radial velocity is normalized by the average
permeate velocity \( UR \equiv Qm/(2\pi RmLm) \approx 3.26 \times 10^{-4} \text{ m/s} \). The radial velocity in the bulk flow domain decreases downstream. A sudden rise in the radial velocity near \( r/R \approx 0.35 \) in (b) is explained by the relative importance of centrifugal acceleration compared to the reduction of radial velocity due to the swirling motion.

Figure 8.6: Effect of the Swirl number on the radial velocity profile at \( z/L_m = 0.5 \). There is a balance between the centrifugal acceleration and the increased axial acceleration of the flow for the rotating CFF system at \( r/R_m \approx 0.5 \). A higher axial flux abates the effect of centrifugal acceleration and yields smoother radial velocity profile near \( r/R_m \approx 0.5 \). For this case \( UR \equiv Qm/(2\pi RmLm) \approx 3.26 \times 10^{-4} \text{ m/s} \).

Figure 8.7: Normalized tangential velocity profiles. The tangential velocity is normalized by \( U_{0R} = \omega R_m \approx 0.47 \). Tangential velocity increases downstream due to a decrease in the axial flux. The rise of axial momentum reduces the tangential momentum to satisfy conservation of mass and momentum.

Figure 8.8: A Distribution of normalized pressure \( P^* = (P - P_{ret})/(P_i - P_{ret}) \) at \( z/L_m = 0.5 \) of the CFF system. Higher Swirl number increases axial momentum and pressure drop in the core of the flow resulting in larger radial pressure gradients. A higher Reynolds number with constant rotational speed of the membrane decreases the swirl intensity. As a consequence, the radial pressure gradient decreases with an increase in the Reynolds number.

Figure 8.9: Effect of the Swirl and the Reynolds number on the average wall shear stress. The effect of the Swirl number is illustrated for the CFF system with \( Re = 1415 \). The Reynolds number’s effect is presented for the CFF system with rotation at 1500 rpm. The resultant wall shear stress increases with an increase in both the Swirl and the Reynolds numbers.

Figure 8.10: Effects of the Reynolds and Swirl numbers on the cut size of droplets. The effect of the Reynolds number is illustrated for CFF system with \( \omega = 1500 \text{ rpm} \). The effect of the Swirl number is presented for CFF system with \( Re = 1415 \). An increase of rotational speed in the CFF system greatly decreases the cutoff size.

Figure 8.11: Effect of the Reynolds number on the grade efficiency for the CFF system with rotation at 1500 rpm. Solid and dotted lines represent the total and reduced grade efficiencies, respectively. The reduced grade efficiency is estimated by subtracting the effect of flow ratio from the total grade efficiency. Reduced grade efficiency decreases because of the reduction of swirling strength with an increase in the Reynolds number.
Figure 8.12: Effect of the Swirl number on the grade efficiency for the CFF system with Re = 1415. Higher swirling strength produces larger centrifugal force, enhances the radial pressure gradient, and reduces the radial outward velocity component leading to a significant decrease in grade efficiency.

Figure 9.1: Illustration of coalescence and separation of dispersed droplets. Here, $n_1$, $n_2$, $n_d$, and $n_c$ are the number concentration per unit volume of the monomers, the doublet (coagulation of two droplets), and the coalesced droplet, respectively. $h$ is the collision rate for the formation of the doublet, $k_d$ is the rate of dissociation of the doublet, and $\lambda$ is the rate of coalescence of the doublet. [After Narsimhan (2004)].

Figure 9.2: Collision tube of a bubble moving with a relative speed $u_{rel}$.

Figure 9.3: Four regimes of film drainage during drop-drop coalescence.

Figure 9.4: Mechanism of droplet breakage due to turbulent shear stress. $d$ is the diameter of mother droplet and $d_1$ is the diameter of daughter droplet. (Adopted from Komrakova and Derksen, 2012).

Figure 9.5: Effect of energy dissipation rate on the specific collision frequency, drainage and contact times, coalescence efficiency, and specific coalescence frequency for $d_2 = 100$ microns. $\varepsilon = 1$ m$^2$/s$^3$; $-\varepsilon = 10$ m$^2$/s$^3$; and $-\varepsilon = 100$ m$^2$/s$^3$.

Figure 9.6: Effect of the reference droplet size ($d_2$) on the specific collision frequency, drainage and contact times, coalescence efficiency, and specific coalescence frequency for $\varepsilon = 100$ m$^2$/s$^3$. $d_2 = 100$ $\mu$m; $d_2 = 50$ $\mu$m; and $d_2 = 25$ $\mu$m.

Figure 9.7: A representation of linear grid used for solving the PBE. Here, $V_i = (x_i - 1 + x_i)/2$.

Figure 9.8: The geometric grid used for discretization of the PBE. Here, $V_i = (x_i - 1 + x_i)/2$.

Figure 9.9: Transient distribution of volume fraction for (a) pure coalescence, (b) pure breakage, (c) coalescence and breakage, and (d) profile of Sauter mean diameter. $\varepsilon_{ave} = 425$ m$^2$/s$^3$, $\phi = 0.01$, $\sigma = 0.02$ N/m, $\rho_d = 850$ kg/m$^3$, $\mu_d = 0.0033$ Pa.s.

Figure 9.10: Effect of volume averaged energy dissipation on the time evolution of Sauter mean diameter. $\phi = 0.01$, $\sigma = 0.02$ N/m, $\rho_d = 850$ kg/m$^3$, $\rho_c = 988$
kg/m$^3$, $\mu_d=0.001$ Pa.s, and $\mu_c=0.0033$ Pa.s. ...........................................241

Figure 9.11: Effect of volume fraction on the time evolution of Sauter mean
diameter for $\varepsilon_{ave} = 425$ m$^2$/s$^3$, $\sigma = 0.02$ N/m, $\rho_d=850$ kg/m$^3$, $\rho_c=988$
kg/m$^3$, $\mu_d=0.001$ Pa.s, and $\mu_c=0.0033$ Pa.s. ...........................................241

Figure 9.12: Effect of interfacial tension on the time evolution of Sauter mean
diameter for $\varepsilon_{ave} = 425$ m$^2$/s$^3$, $\phi = 0.01$, $\sigma = 0.02$ N/m, $\rho_d = 850$ kg/m$^3$,
$\rho_c = 988$ kg/m$^3$, $\mu_d = 0.001$ Pa.s, and $\mu_c=0.0033$ Pa.s. ................................242

Figure 9.13: Contours of turbulent energy dissipation for $Re_F = 34320$. .........................246

Figure 9.14: The average distribution of normalized volume fraction at different
cross-sectional planes shown in Fig. 9.13..............................................................247

Figure 9.15: Sauter mean diameter of dispersed droplet at the inlet and the different
axial locations. [$d_{32} = \frac{di3Ni}{di2Ni}$] .................................................................248

Figure D1: A section of a hydrocyclone having parabolic swirl chamber with
asymptote near the tail. .........................................................................................277

Figure D2: A section of a hydrocyclone having parabolic swirl chamber with
asymptote near the top. .........................................................................................279

Figure D3: A section of a hydrocyclone having hyperbolic swirl chamber with
asymptote at the both ends. .................................................................................280
### KEY TO SYMBOLS AND ABBREVIATIONS

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta$</td>
<td>Angle of conical swirl chamber</td>
<td>degree</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Interfacial tension</td>
<td>N/m</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulent energy dissipation</td>
<td>m$^2$/s$^3$</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Volume fraction of dispersed phase</td>
<td></td>
</tr>
<tr>
<td>$\omega$</td>
<td>Angular velocity</td>
<td>rad/s</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>$\dot{m}_{DX}$</td>
<td>Mass flow rate of dispersed phase through x surface</td>
<td>kg/s</td>
</tr>
<tr>
<td>$\dot{E}$</td>
<td>Rate of strain tensor</td>
<td>1/s</td>
</tr>
<tr>
<td>$\dot{m}_C$</td>
<td>Mass flow rate of continuous phase</td>
<td>m$^3$/s</td>
</tr>
<tr>
<td>$\dot{m}_D$</td>
<td>Mass flow rate of dispersed phase</td>
<td>m$^3$/s</td>
</tr>
<tr>
<td>$Eu_x$</td>
<td>Euler number at x surface relative to inlet</td>
<td></td>
</tr>
<tr>
<td>$f_{DX}(d_D) , \partial d_D$</td>
<td>Mass fraction of dispersed droplets having sizes between $d_D$ and $d_D + \Delta d_D$ appears at x surface</td>
<td></td>
</tr>
<tr>
<td>$G_{OR}(d_D)$</td>
<td>Reduced overflow grade separation efficiency</td>
<td></td>
</tr>
<tr>
<td>$G_{OR1}(d_D)$</td>
<td>Reduced grade separation efficiency relative to overflow-1</td>
<td></td>
</tr>
<tr>
<td>$G_{OR2}(d_D)$</td>
<td>Reduced grade separation efficiency relative to overflow-2</td>
<td></td>
</tr>
<tr>
<td>$G_{ORT}(d_D)$</td>
<td>Overall (total) reduced grade separation efficiency</td>
<td></td>
</tr>
<tr>
<td>$N_{DX}(d_D)$</td>
<td>Number of dispersed droplets having size $d_D$ appears at x surface</td>
<td></td>
</tr>
<tr>
<td>$Re_F$</td>
<td>Reynolds number at inlet (feed Reynolds number)</td>
<td></td>
</tr>
<tr>
<td>$\eta_0$</td>
<td>Overall separation efficiency</td>
<td></td>
</tr>
<tr>
<td>$\eta_{OR}$</td>
<td>Reduced overall separation efficiency</td>
<td></td>
</tr>
<tr>
<td>$A_H$</td>
<td>Hamaker constant</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>$C_X$</td>
<td>Concentration of dispersed phase at x surface</td>
<td></td>
</tr>
<tr>
<td>$E_{a_{ext}}$</td>
<td>Extrapolated relative error</td>
<td></td>
</tr>
<tr>
<td>$F_c$</td>
<td>Colloidal force</td>
<td></td>
</tr>
<tr>
<td>$F_t$</td>
<td>Force acting on droplets due to turbulence</td>
<td></td>
</tr>
<tr>
<td>$G_{O(d_D)}$</td>
<td>Overflow grade separation efficiency</td>
<td></td>
</tr>
<tr>
<td>$P_\infty$</td>
<td>Permeate pressure</td>
<td></td>
</tr>
<tr>
<td>$P_X$</td>
<td>Pressure at x surface</td>
<td></td>
</tr>
<tr>
<td>$Q_X$</td>
<td>Volumetric flow rate through x surface</td>
<td></td>
</tr>
<tr>
<td>$n_X$</td>
<td>Number frequency of dispersed droplets of size $d_X$</td>
<td></td>
</tr>
<tr>
<td>$t_c$</td>
<td>Contact time of two droplets</td>
<td></td>
</tr>
<tr>
<td>$t_d$</td>
<td>Drainage time of liquid film between two droplets</td>
<td></td>
</tr>
<tr>
<td>$\mu_r$</td>
<td>Ratio of viscosity between continuous and dispersed phase</td>
<td></td>
</tr>
<tr>
<td>$\phi_{X}$</td>
<td>Volume fraction of dispersed phase at x surface</td>
<td></td>
</tr>
<tr>
<td>$a$</td>
<td>Height of rectangular inlet channel</td>
<td></td>
</tr>
<tr>
<td>$a_c$</td>
<td>Centrifugal acceleration</td>
<td></td>
</tr>
<tr>
<td>$a_g$</td>
<td>Acceleration due to gravity</td>
<td></td>
</tr>
<tr>
<td>$b$</td>
<td>Width of rectangular inlet channel</td>
<td></td>
</tr>
<tr>
<td>$Ca$</td>
<td>Capillary number</td>
<td></td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter of inlet chamber (nominal diameter of hydrocyclone)</td>
<td></td>
</tr>
<tr>
<td>$d_D$</td>
<td>Diameter of dispersed droplet</td>
<td></td>
</tr>
<tr>
<td>$d_{D50}$</td>
<td>Cut size</td>
<td></td>
</tr>
<tr>
<td>$D_F$</td>
<td>Hydraulic diameter of inlet (feed)</td>
<td></td>
</tr>
<tr>
<td>$D_m$</td>
<td>Inner diameter of membrane</td>
<td></td>
</tr>
<tr>
<td>$Do$</td>
<td>Diameter of overflow outlet</td>
<td></td>
</tr>
</tbody>
</table>
\( Du \) \hspace{1cm} \text{Diameter of underflow outlet} \hspace{1cm} m
\( Ea \) \hspace{1cm} \text{Relative error}
\( F \) \hspace{1cm} \text{Net interaction force acting on droplets during collision} \hspace{1cm} N
\( F_C \) \hspace{1cm} \text{Centrifugal force} \hspace{1cm} N
\( F_g \) \hspace{1cm} \text{Force due to gravitation} \hspace{1cm} N
\( h \) \hspace{1cm} \text{Collision frequency} \hspace{1cm} \#/m^3/s
\( h_f \) \hspace{1cm} \text{Final film thickness} \hspace{1cm} m
\( h_i \) \hspace{1cm} \text{Initial film thickness} \hspace{1cm} m
\( I \) \hspace{1cm} \text{Turbulent intensity}
\( k \) \hspace{1cm} \text{Turbulent kinetic energy} \hspace{1cm} m^2/s^2
\( \ell_C \) \hspace{1cm} \text{Length of reverse flow core} \hspace{1cm} m
\( L_i \) \hspace{1cm} \text{Length of inlet chamber} \hspace{1cm} m
\( L_m \) \hspace{1cm} \text{Length of membrane} \hspace{1cm} m
\( L_s \) \hspace{1cm} \text{Length of swirl chamber} \hspace{1cm} m
\( L_t \) \hspace{1cm} \text{Length of tail pipe} \hspace{1cm} m
\( L_v \) \hspace{1cm} \text{Length of vortex finder} \hspace{1cm} m
\( m_D \) \hspace{1cm} \text{Mass of dispersed phase} \hspace{1cm} kg
\( P \) \hspace{1cm} \text{Pressure} \hspace{1cm} Pa
\( \Delta P^*_{1} \) \hspace{1cm} \text{Pressure drop ratio relative to overflow-1}
\( \Delta P^*_{2} \) \hspace{1cm} \text{Pressure drop ratio relative to overflow-2}
\( P_m \) \hspace{1cm} \text{Transmembrane pressure} \hspace{1cm} Pa
\( Q \) \hspace{1cm} \text{Volumetric flow rate} \hspace{1cm} m^3/s
\( Q_m \) \hspace{1cm} \text{Volumetric flow rate through membrane} \hspace{1cm} m^3/s
\( Q_{ret} \) \hspace{1cm} \text{Retentate flow rate} \hspace{1cm} m^3/s
\( R \) \hspace{1cm} \text{Radius of hydrocyclone at any cross-section (it varies over} \hspace{1cm} m
the length of hydrocyclone)

\( R_m \) \hspace{1cm} \text{Inner radius of membrane} \hspace{1cm} M

\( R_O \) \hspace{1cm} \text{Overflow ratio}

\( R_{O1} \) \hspace{1cm} \text{Flow ratio between overflow-1 and inlet}

\( R_{O2} \) \hspace{1cm} \text{Flow ratio between overflow-2 and inlet}

\( S_w \) \hspace{1cm} \text{Swirl number}

\( U \) \hspace{1cm} \text{Instantaneous velocity} \hspace{1cm} m/s

\( u \) \hspace{1cm} \text{Fluctuating velocity} \hspace{1cm} m/s

\( U_\theta \) \hspace{1cm} \text{Tangential velocity} \hspace{1cm} m/s

\( U_r \) \hspace{1cm} \text{Radial velocity} \hspace{1cm} m/s

\( uu \) \hspace{1cm} \text{Kinematic Reynolds momentum flux} \hspace{1cm} m^2/s^2

\( U_z \) \hspace{1cm} \text{Axial velocity} \hspace{1cm} m/s

\( We \) \hspace{1cm} \text{Weber number}

\( We_{cr} \) \hspace{1cm} \text{Critical Weber number}

\( g(d_j) \) \hspace{1cm} \text{Breakage frequency}

\( \Gamma (d_i, d_j) \) \hspace{1cm} \text{Coalescence rate} \hspace{1cm} \#/m^3/s

\( \beta(d_i, d_j) \) \hspace{1cm} \text{Daughter droplets distribution function}

\( \delta_m \) \hspace{1cm} \text{Thickness of membrane} \hspace{1cm} m

\( \lambda \) \hspace{1cm} \text{Coalescence efficiency}

\( \rho \) \hspace{1cm} \text{Density of fluid} \hspace{1cm} kg/m^3
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASM</td>
<td>Algebraic Slip Model</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFF</td>
<td>Crossflow Filtration</td>
</tr>
<tr>
<td>DPM</td>
<td>Discrete Phase Model</td>
</tr>
<tr>
<td>EPA</td>
<td>Environmental Protection Agency</td>
</tr>
<tr>
<td>GCI</td>
<td>Grid Convergence Index</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>PBE</td>
<td>Population Balance Equation</td>
</tr>
<tr>
<td>PBM</td>
<td>Population Balance Method</td>
</tr>
<tr>
<td>PISO</td>
<td>Pressure Implicit with Splitting of Operator</td>
</tr>
<tr>
<td>PRESTO</td>
<td>PREssure STaggering Option</td>
</tr>
<tr>
<td>QUICK</td>
<td>Quadratic Upstream Interpolation for Convective Kinetics</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RCF</td>
<td>Relative Centrifugal Force</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds Stress Model</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Infinite Method for Pressure Linked Equation</td>
</tr>
<tr>
<td>SWF</td>
<td>Standard Wall Function</td>
</tr>
<tr>
<td>UDF</td>
<td>User Defined Function</td>
</tr>
</tbody>
</table>
CHAPTER 1

INTRODUCTION

1.1. The physical problem

A large volume of oily water is generated from the petroleum industry during crude oil production, transportation, storage, and refining processes. Produced water is the largest byproduct stream associated with oil and gas production. The US Department of Energy’s Argonne National Laboratory estimated the volume of produced water in the United States at 21 billion barrels a year (Clark and Veil, 2009). Additional production from the rest of the world was estimated at a volume of more than 50 billion barrels a year (Clark and Veil, 2009). As an oil field becomes depleted, the volume of produced water increases for each barrel of oil production. The typical concentration of oil in this so called produced water ranges from 100 to 5,000 mg/L (Hargreaves and Silvester, 1990). One of the big challenges in the oil industry is the treatment of produced water for reusing it by underground injection or discharging it to surface water. The United States’ Environmental Protection Agency (EPA) issued standards for the underground injection (EPA regulation, July 2012), and the discharges to surface water (EPA regulation, February 2012). According to the environmental regulation, the allowable concentration of oil and grease in the water to be discharged is 29 ppm in weekly average and 42 ppm in daily maximum (Scott Wilson, 2014). The disposal of oily water has increasingly become an expensive challenge for industry and several techniques have been employed
for its treatment. Oil-water separation processes in the oil and gas production are usually divided into three steps:

**Primary treatment:** In the primary treatment, oil and solid particles are removed by gravity separation and skimming. Oil droplets are separated from the water by letting them float to the surface based on their difference in gravity. Skimming process separates oil from water floating on it. It is commonly used for oil spill remediation and in industrial applications such as removing oil from machine tool coolant. Skimming process recover oil-water emulsion comprised of 90% water and 10% (by volume) oil (Hadfield and Riibe, 1991).

**Secondary treatment:** The secondary treatment can be considered as a key stage, where most of the free oily phase is removed. In this stage, the breakdown of emulsion takes place, i.e., the coalescence of oil droplets. Hydrocyclone separation is the most broadly used processes for secondary treatment (Hayes and Arthur, 2004, Sinker, 2007). High swirling motion generated by centrifugal and hydrocyclone separates two phases based on their density difference.

**Tertiary treatment:** Membrane, adsorption, or biological treatment processes are applied to reduce further the oil contamination from the produced water.

The common techniques available for treating oil-water emulsions and the removal capacity of existing technologies are given in Table 1.1.
Table 1.1 Common technologies used for produced water treatment and their removal capacity by particle size (Frankiewicz, 2001).

<table>
<thead>
<tr>
<th>Technology</th>
<th>Removal capacity by particle size (micron)</th>
</tr>
</thead>
<tbody>
<tr>
<td>API gravity separator</td>
<td>150</td>
</tr>
<tr>
<td>Corrugated plate separator</td>
<td>40</td>
</tr>
<tr>
<td>Induced gas flotation without chemical addition</td>
<td>25</td>
</tr>
<tr>
<td>Induced gas flotation with chemical addition</td>
<td>3-5</td>
</tr>
<tr>
<td>Hydrocyclone</td>
<td>10-15</td>
</tr>
<tr>
<td>Mesh coalescer</td>
<td>5</td>
</tr>
<tr>
<td>Media filter</td>
<td>5</td>
</tr>
<tr>
<td>Centrifuge</td>
<td>2</td>
</tr>
<tr>
<td>Membrane filter</td>
<td>Depends on pore size</td>
</tr>
</tbody>
</table>

Gravity separation, plate separator and gas flotation are three-phase separation systems (primary treatments) which are used for the separation of highly concentrated mixtures (Hansen, 2001; Akpan, 2013; Jahangiri and Nouri, 2014). Produced water is dilute (oil concentration by volume is less than 0.5%) and the density difference, in comparison with solid-liquid mixture, between two phases is very small (less than 150 kg/m³). The membrane filtration is capable of separating very fine droplets, but oil droplets foul membrane and flux declines over time. The key challenge in the separation of oil from water is the secondary treatment by means of mechanical systems. Depending on the properties of mixture, targeted cut size, volume of separation, operational feasibility and
applications the most commonly used mechanical devices for secondary treatment involve hydrocyclones (Hayes and Arthur, 2004, Sinker, 2007). The centrifugation process involves the use of the centrifugal force for the two phase mixture with a centrifuge used in industry or laboratory settings. Though the centrifuge can separate about 2 microns droplets, the moving and bulky system, and high operational expenses make it unsuitable for both the onshore and offshore platform.

Hydrocyclones are very suitable for an offshore platform because of compact size, small residence time, no moving parts, insensitivity to orientation, low manufacturing and maintenance cost. However, commonly used hydrocyclones in the oil and gas industries cannot separate droplets less than 10 microns (Kharoua et al., 2010). In other words, the level of oil-water separation using existing hydrocyclones only cannot meet the environmental regulation. Moreover, the hydrocyclones possess finite turndown ratio, i.e. it operates effectively only for a certain range of inlet conditions. An extensive study of hydrodynamics and separation performance of a conventional hydrocyclone are essential for identifying the physics that causes the finite turndown. Clearly, a better understanding of the internal hydrodynamic phenomena that limits the turndown ratio and the cut-size of an individual hydrocyclone separator could lead to an improved compact separator system design that is reliable and simple to operate.
1.2. Background

Hydrocyclones are high-swirl flow devices commonly used for separation of solids from liquids and for the separation of two liquid phases. The first idea of using hydrocyclones for oil-water separation was suggested by Simkin and Olney (1956), but fundamental studies on de-oiling hydrocyclones were started from 1980 by Colman and Thew (Colman, 1981). Research on using hydrocyclones for liquid-liquid separation conducted by Hitchon (1959), Regehr (1962), Kimber and Thew (1974) and Sheng et al. (1974) were not sufficiently promising for commercial applications. A successful hydrocyclone for oil-water separation was developed by Colman et al. (1980). In water de-oiling application, the density difference between the two phases in an oil-water mixture is smaller than a solid-liquid mixture and the separation of oil from water requires much more effort than the solid-liquid separation. Another difference between the two types of hydrocyclones is that the centrifugal force makes solid particles to migrate toward the wall in solid-liquid hydrocyclones while making oil droplets to move toward the center in the liquid-liquid hydrocyclone. So the attention is drawn to the flow field near the center of the liquid-liquid separation hydrocyclone.

Oil-water separation using a hydrocyclone is a challenging task because of the presence of complex flow phenomena and the possibility of shearing of dispersed droplets (Kharoua et al., 2010). The flows within a hydrocyclone consist of two structures: primary and secondary [detail analyses of the internal flow structures can be found in Bradley (1965) and Svarovsky (1992)]. The primary flow consists of two spiral streams moving in the same circular direction (Fig. 1.1). There is a forced-like vortex (inner vortex) in the inner spiral region and a free-like vortex (outer vortex) in the outer
spiral region. The flows in the outer vortex move downward toward the underflow orifice, while the flows in the inner vortex move in a reverse direction toward the overflow orifice. The denser phase (water) is conveyed by the outer vortex and the lighter phase (oil) is transported by the inner vortex.

Figure 1.1: A schematic of a hydrocyclone is illustrating internal flow structures as well as one inflow and the two outflows.

In the secondary flow structure, a boundary layer flow creates a short circuit from the inlet to the vortex finder. Another secondary flow, called eddy or recirculation of flow, appears in the region between the outer and inner vortexes when the reverse flow streams cannot be fully accommodated by the vortex finder (Dlamini et al., 2005;
Noroozi and Hashemabadi, 2011; Motin et al. 2014b). The inward radial flow as well as the migration of dispersed droplets toward the inner vortex is resisted by the presence of eddies in the swirl chamber (Devorak, 1989; Dai et al., 1998). A higher pressure differential between the inlet and overflow outlet than that between the inlet and underflow outlet provides a driving force to create a reverse flow at the core of hydrocyclone (Svarovsky, 1992; Meldrum, 1988; Kharoua et al., 2010; Noroozi and Hashemabadi, 2011; Motin et al., 2014b). A radial pressure gradient generated by swirling motion also helps the lighter phase to migrate toward the center of the hydrocyclone (Noroozi et al., 2009; Saidi et al., 2012). However, the migration rate of dispersed droplets toward the center depends on the density difference between two phases, centrifugal acceleration of dispersion, viscosity of dispersed phase, radial pressure gradient, relaxation time of droplets and the residence time of flow. The oil is much more sensitive to the flow pattern so this leads to the requirement for a low turbulence, reasonably linear vortex core, and low shear flow field to avoid droplet breakup leading to lower oil removal efficiency (Thew, 2000).

1.2.1. Numerical study on hydrocyclone

Flows in hydrocyclones often operate in turbulent flow regime. The Reynolds averaged Navier-Stokes (RANS) equations with appropriate closure models for kinematic Reynolds momentum flux are solved for predicting the flow patterns. Eddy-viscosity closures and Reynolds stress model (RSM) can be employed for closing the RANS equation. The use of the eddy-viscosity models (e.g., $k$-$\varepsilon$) is inappropriate for modeling highly anisotropic turbulence in hydrocyclones since they are developed with the
assumption that the turbulences are isotropic. The eddy-viscosity models over predict turbulent kinetic energy and turbulent viscosity, and gives poor results in the cases of wall bounded internal flows containing a large adverse pressure gradient. The RSM was shown to have good predictive capability for the turbulent flow field in a hydrocyclone (Ma et al., 2000; Jawarneh and Vatistas, 2006; Bhaskar et al., 2007; Motin et al. 2014a and 2014b). The RSM can predict anisotropic turbulence and capture all the fluctuations of highly swirling flow bounded by curved surface (Jawarneh and Vatistas, 2006; Ghadirian, 2012). The stress transport models (e.g., LRR of Launder et al., 1975; SSG of Speziale et al., 1991) in the RSM give acceptable prediction of velocity profiles in a hydrocyclone (Slack et al., 2000; Kharoua et al., 2010). Brennan (2006) compared the linear pressure strain correlation (LRR) with the quadratic pressure strain correlation (SSG) in the RSM and concluded that they are essentially the same. A Large Eddy Simulations (LES) also provides good prediction of turbulence in hydrocyclone (Derksen et al., 2000; Delgadillo and Rajamani, 2005a and 2005b; Gaustad et al., 2009, Saidi et al., 2013), but, however, it requires much higher computational resources when we compare with the RSM.

Saidi et al. (2012, 2013) simulated the flow field of a 35 mm Young (1994) type hydrocyclone based on the LES. They compared the tangential velocity profile with the experimental results of Bai et al. (2009) and observed good qualitative agreement. In their published work they showed that the RSM is a poor model and it predicts both qualitatively and quantitatively inaccurate tangential velocity profiles. In Saidi et al.’s (2012, 2013) numerical investigations, they used the results of the RSM simulation as a precondition for the LES simulation. They probably switched to the LES simulation
before a complete establishment of statistically steady-state flow field in the RSM simulation. However, the axial velocity profile in Saidi et al.’s (2012) calculation using the LES simulation is both qualitatively and quantitatively dissimilar to the Bai et al.’s (2009) experimental results. The LES models also could not accurately capture the near wall fluctuations of both the axial and the tangential components. Reza et al. (2012) performed simulations on a 35 mm single cone Young (1994) type hydrocyclone using the RSM with the LRR and compared their results with the Bai et al.’s (2009) experimental results. From the Reza et al.’s comparison it is evident that both axial and tangential velocity profiles obtained using the RSM yields a better agreement with experimental results. In the region of inlet swirl chamber and the upper half of the conical section, the tangential velocity profile seems both qualitatively and quantitatively similar to the experimental data. In the lower half of the conical section, the diameter of force vortex by using the RSM is greater than the experimental one. Noroozi et al. (2009, 2011) observed the effects of inlet design and inlet chamber body design on the separation efficiency on a Colman and Thew (1980) type 40 mm hydrocyclone based on the multiphase flow Eulerian-Eulerian simulation with the RSM closure. Even though they showed an improved separation efficiency for a helical inlet and an exponential form of inlet chamber, the tangential velocity profiles presented in both of their published works are neither qualitatively nor quantitatively accurate.

Depending on the concentration of the dispersed (secondary) phase in the continuous (primary) phase the multiphase simulations of a hydrocyclone is performed using either Eulerian-Lagrangian approach or Eulerian-Eulerian approach. The Eulerian-Lagrangian approach is utilized for modeling the motion of the primary phase and
dispersed phase motion separately. The volume fraction of dispersed phase is less than 10% for the Eulerian-Lagrangian approach to be appropriate (Huang, 2005; Ansys Fluent documentation: Theory Guide, 2013). In this approach, the velocity and pressure of the primary phase are obtained by solving the continuity and momentum equations. The motion of dispersed phase is then measured by tracking a large number of particles/droplets through the calculated flow field using the Lagrangian approach. The trajectories of dispersed droplets are predicted by integrating the force balance on the droplets. Boysan et al. (1982) used the Lagrangian tracking model with the assumptions of no interaction of droplets with each other and with the continuous phase, no breakup and coalescence of droplets, and no virtual mass effect for the drag-accelerated droplets. These assumptions are suitable for mimicking the solid particles. The fluctuating quantities of instantaneous velocity affect the trajectories of dispersed droplets (Kharoua et al. 2010). The effects of droplets breakup and coalescence, and the interaction of dispersed phase with continuous phase are very essential to be incorporated in the simulation. Saidi et al. (2012) applied the Lagrangian approach to estimate the separation efficiency of a hydrocyclone with the following assumption: no breakup and coalescence of droplets, and no deformation of droplet due to shear, but the pressure gradient and virtual mass effected were taken into account in addition to drag force.

To completely capture the multiphase nature of flows in de-oiling hydrocyclones, the Eulerian-Eulerian approach is applied. In this approach, the different phases are treated as interpenetrating continua and conservation equations (continuity, momentum) are solved for each phase assuming a single pressure field is shared by all phases (Kharoua et al., 2010; Huang, 2005). The Eulerian-Eulerian approach includes as subsets
the full Eulerian multiphase model, the mixture model and the volume of fluid model. The full Eulerian and mixture models are appropriate for flows in which phases mix or separate and the volume of fluid model is appropriate for stratified or free surface flow (Ansys Fluent documentation: Theory guide, 2013). For very low loading, the primary phase influences the droplets via drag and turbulence. In this case, both the mixture and full Euler models are appropriate, but the mixture model is recommended because of the complexity and computationally expensiveness of the full Euler model. The mixture model solves the continuity and momentum equations for the mixture, transport equations for the volume fraction of dispersed phase, and an algebraic equations for velocities relative to continuous phase (slip velocity) and relative to mixture average velocity (drift velocity) (Noroozi and Hashemabadi, 2011; Manninen et al., 1996, Ansys Fluent documentation: Theory guide, 2013). The full Eulerian approach predicts better flows with the suspended dispersed phase. This approach is an alternate of the Langrangian approach and indispensable when simulation with high loading (more than 10%) of dispersed phase is desired (Kharoua et al., 2010, Durst et al., 1984, Huang, 2005). This approach also permits simulating the effects of turbulence on dispersed phase and the interaction of dispersed phase with primary phase. The algebraic slip mixture (ASM) approach for dispersed phase flow allows consideration of the problem at low computational cost in relation to the full Eulerian approach (Noroozi et al., 2009).

Grady et al. (2003) employed multiphase flow simulations of a mini hydrocyclone (cyclone diameter of 10 mm with 3.8° of cone angle) with a single involute inlet. They used the RSM turbulent closure with the LRR and the ASM model for the prediction of flow behavior and separation efficiency with simplifying assumptions that no droplet
breakup or coalescence. Comparing, based on the separation efficiency, the simulation results with experimental data they concluded that the simulation quite accurately predicts for droplet size larger than 20μm. They obtained an unreliable prediction for a smaller droplet because of the fishhook effect, which was observed in the experiment, was not predicted by the simulation. The multiphase flow behavior of a hydrocyclone with two involute inlets for low concentration of oil was analyzed by Paladino et al. (2005). They also observed transient behavior in order to evaluate the time required to stabilize the hydrocyclone operation under variable operational conditions. The RSM with SSG and algebraic slip mixture model were used for the prediction of the effect of turbulence on the separation efficiency. They observed that the mass flow at the overflow and the underflow take more time to be stabilized, whereas the pressure field stabilizes rapidly. Huang (2005) used the RSM with full Eulerian approach to predict the phase behavior for high concentration of oil in the feed of a Colman-Thew type hydrocyclone with two inlets. He verified the simulation results with Colman-Thew’s experimental data in terms of separation efficiency and observed that the full Eulerian approach predicts very well. Noroozi et al. (2009) also used the RSM with the full Eulerian approach to investigate the effect of inlet design on the separation efficiency of a hydrocyclone. They also used a modified drag correlation for liquid-liquid mixture. The simulation result for the separation efficiency, in comparison with Schiller-Naumann correlation, exhibited better agreement with the experimental data. Noroozi and Hashemabadi (2011) employed the algebraic slip mixture model with the RSM for the prediction of multiphase turbulent flow behavior. Their simulation results for the separation efficiency demonstrated a good agreement with experimental data of Belaidi and Thew (2003).
1.2.2. Experimental study on hydrocyclone

Meldrum (1988) outlined a basic construction and operational principle of an oil-water separation hydrocyclone, and discussed system design, operational experiences and test results from the first full-scale commercial application on the Murchison platform. He also discussed results from the Hutton tension leg platform. Meldrum calculated separation efficiency in 4-in-1 (4 hydrocyclones connected in parallel) single stage 35 mm hydrocyclones for a wide range of feed flow rate which is given in Fig. 1.2. Meldrum observed that as flow rate increases, the separation efficiency increases and then levels out over the unit’s operating range. A further increase in the flow rate causes the efficiency to drop sharply (Fig. 1.2). He mentioned that at very high flow rates, shearing of the oil droplets can occur, creating smaller droplets that are harder to separate. Moreover, as the flow rate increases, the core pressure approaches atmospheric pressure, thereby reducing the available pressure to drive. Bennett and Williams (2004) tested a 70 mm hydrocyclone using impedance tomography to investigate the effect of flow rate on the separation efficiency. They obtained qualitatively similar results (Fig. 1.2) as Meldrum (1988) but did not explain the reasons of rise, plateau, and sharp drop of the separation efficiency for the operating range of flow rate. In the literature there is no evidence that the sharp drop in the separation efficiency is caused by shearing of oil droplets in the hydrocyclone flow field. Hence, the actual reasons of the sharp drop in the separation efficiency and the finite turndown ratio (ratio of maximum to minimum feed flow rate for which the separation efficiency is constant, \( Q_{\text{min}}/Q_{\text{max}} \)) are not well understood yet.
Figure 1.2: Typical separation efficiency (defined in Eq. (2.8)) versus feed flow rate relationship for a liquid-liquid separation hydrocyclone. Turndown ratio = $Q_{\text{max}}/Q_{\text{min}}$.

Bai et al. (2009) used a two-component laser Doppler velocimeter to measure flow velocities and turbulent fluctuations in a 35-mm single cone hydrocyclone. They created 5 mm orifices at the wall in the measurement locations to place plane optic glasses that can help to avoid asymmetric refraction of laser beams. Young et al. (1994) did an experimental search for optimum dimensions of a hydrocyclone for oil-water separation. They evaluated the effects of operational variables and geometric dimensions on the separation performance. Meyer and Bohnet (2003) experimentally investigated the influence of entrance droplet size distribution and feed concentration on the separation efficiency. They also observed the occurrence of coalescence or droplet breakup in the flow field. They mentioned that the separation of oil from water occurs in the free like
vortex region and any breakup of oil droplets in the forced vortex region and vortex finder does not affect the overall separation efficiency. The oil-water separation efficiency of a hydrocyclone during transient flow rates was experimentally investigated by Husveg et al. (2007). They concluded that the efficiency of a hydrocyclone is unaffected by variation of feed flow rates. According to the separation performance observed by Meldrum (1988) and Bennett and Williams (2004), Husveg et al.’s (2007) conclusion is true if the variation of feed flow rate is within the minimum and the maximum feed flow rate (Fig. 1.2), where a plateau in the separation efficiency occurs.

1.2.3. Analytical study on hydrocyclone

Few mathematical models for estimation of flow behaviors and performance of hydrocyclones are currently available, but their validity for practical application has still not been established (Kraipec et al., 2000). The selection and design of a hydrocyclone are done based on empirical correlations and experience. Mechanistic models of hydrocyclone can be developed from analytical, empirical, semi-empirical solutions and numerical simulations. Analytical models are developed based on the Navier-Stokes equations with many assumptions that reduce complexity. The analytical models have been abandoned in favor of numerical simulations due to the complexity of the multiphase flow phenomena (Svarovsky, 1992). Empirical and semi-empirical models are developed on the basis of the correlation between the process key parameters and the experimental data (Gomez, 2001a; Gomez et al, 2001b).
Caldentey et al. (2000, 2002) developed an empirical model of swirl intensity based on the experimental data for small cone angle (between 0 and 1.5°) of a hydrocyclone. However, a good prediction has been obtained for angle of 3° (Amini et al. 2012). This mathematical model for the swirl intensity is valid for tapered and tail pipe sections of Colman and Thew’s (1988) type hydrocyclone (Caldentey, 2000). Amini et al. (2012) observed that the swirl intensity predicted by the proposed mechanistic model has a good agreement with the experimental data with a flow ratio (ratio of overflow rate to feed flow rate) of 10%. It under predicts the experimental data by around 15% in the region of \( z/D < 5 \) while the flow ratio is 1%, where \( z \) is the axial distance from the top of the hydrocyclone.

Algifri et al. (1988) proposed an empirical relation for the estimation of tangential velocity using geometric and flow parameters. Gomez et al. (2001) and Amini et al. (2012) compared the velocity field predicted using Algifri’s mechanistic model with the Colman and Thew’s (1980, 1988) experimental data. They found that the model under predicts the experimental data by about 15% in the outer half of the free-like vortex region and over predict by nearly 25% in the inner half of the free-like vortex region. It also over predicts by about 10% in the forced vortex region. Wolbert et al. (1995) developed a dimensionless correlation for axial velocity in the low cone angle section of Colman and Thew’s (1988) hydrocyclone. Gomez (2001a) identified that the axial velocity profile has a polynomial trend with a location of zero axial velocity between the forward and the reverse flow (Caldentey et al., 2002; Amini et al., 2012). Gomez fitted a third-order polynomial equation to estimate the axial velocity in the low cone angle section of Thew’s type hydrocyclone. Wolbert et al. (1995) deduced a radial velocity
profile by using the conservation of mass principle and the wall condition suggested by Kelsall (1952). The trajectories of dispersed droplets are characterized through a relation between the axial and radial coordinates of its position (Wolbert et al. 1995; Amini et al., 2012). The separation efficiency is calculated based on the trajectory analysis of a droplet.

1.3. Objectives and scope of this thesis

Although a large number of analytical, experimental and numerical researches were conducted on the liquid-liquid separation hydrocyclone, the fundamental understanding of the hydrodynamics that causes a finite turndown and the poor separation efficiency for the smaller droplets (less than 10 microns in diameter) are still lacking. The axial gradient of core pressure, length and shape of reverse flow core, toroidal recirculation, short circuit flows have direct influence on the separation performance and the finite turndown ratio which are currently not fully understood. This research addresses fundamental problems related to the oil-water separation using hydrocyclones. Both numerical and analytical investigations are conducted to identify the physics responsible for the finite turndown ratio, toroidal recirculation, short circuit flows, and low separation efficiency in a liquid-liquid separation hydrocyclone. The hydrodynamic behaviors as well as geometry and operating parameters that characterize the performances of a conventional liquid-liquid separation hydrocyclones are investigated. A parametric study of hydrocyclone geometry and operating conditions is also conducted. The parametric study includes: effect of the Reynolds number, pressure drop ratio, vortex finder geometry, overflow ratio, angle of conical swirl chamber, and wall profile of swirl
chamber (conical, parabolic, and hyperbolic) on the internal flow structures, pattern of reverse flow core, core pressure and separation performance. On the basis of the parametric study, the conventional liquid-liquid separation hydrocyclone is modified by tailoring its geometry and redesigning the tail section, which is one of the main objectives of the research. The objective of the redesign is to enhance the turndown ratio and separation efficiency so that the federal regulation for the oil content in the discharged water can be achieved by a single stage operation. The extensive investigations of fundamental flow characteristics lead to design a novel hydrocyclone for oil-water separation which is a breakthrough in the swirling flow separation technology. The breakage and coalescence of dispersed droplets for various operating conditions of the novel hydrocyclone are also quantified through a coupled multiphase and population balance method. The population balance modeling provides detail understanding of the variation of droplet size distribution in the flow field and its effect on the overall performance of the novel hydrocyclone. A guideline for the future development of the novel hydrocyclone is also discussed in the research.

1.4. Methodology

This research is conducted based on the numerical, theoretical and analytical study of the hydrodynamics in the hydrocyclones, and droplets modeling through a coupled multiphase and population balance models. CFD simulations using ANSYS 14.0 to 15.0 are utilized for calculating the velocity and pressure fields and turbulence properties. The Reynolds Averaged Navier-Stokes (RANS) equations with the Reynolds Stress Model (RSM) are solved numerically for the calculation of internal hydrodynamics
of continuous phase (water). Discrete Phase Model (DPM) is used for calculating the trajectories of dispersed oil droplets in the calculated continuous phase flow field. The distribution of dispersed phase in the flow field is calculated using the multiphase mixture theory with the algebraic slip model (ASM). ANSYS ICEM CFD preprocessing tool is used for building geometry and mesh generation, and ANSYS CFD-Post tool is used for post processing of simulation results. The numerical calculations of mathematical models for droplet breakage, coalescence and population balance are performed by writing code in MATLAB. The coupled multiphase and population balance models are solved using the ANSYS FLUENT and the population balance model platform supported by ANSYS FLUENT. The macros for the coalescence and breakage models are written using the user defined function (UDF) interface supported by FLUENT.

1.5. Outline of the thesis

This dissertation is organized in ten chapters. An overview and background of the work, statement of the physical problems, scope and objectives of this work are addressed in this chapter. Chapter 2 addresses the hydrocyclone designs and its operating principles. Flow and geometric features, and performance matrices of hydrocyclones are discussed here. Detail numerical simulation approaches and solution strategies for continuous, dispersed and multiphase flow mechanics are addressed in Chapter 3. The grid independent study and a comparison of result between CFD simulation and experiment, which are used as the validation of the work, are also performed in this chapter. The components of the mean velocity field, the mean pressure
field, the components of the Reynolds stress, and the dissipation of turbulent kinetic energy are examined within a conventional hydrocyclone over a range of Reynolds numbers (Chapter 4). A quantitative understanding of finite turndown ratio is addressed in Chapter 4. Effects of vortex finder design, pressure drop ratio and aspect ratio of swirl chamber on the internal hydrodynamics are examined in Chapter 5. Chapter 6 addresses the new approaches to redesign hydrocyclones by tailoring the swirl chamber and tail section. Possible wall profile of swirl chambers and designs of tail gadgets are discussed and their effectiveness in the oil-water separation is evaluated in this chapter. Based on the parametric study of conventional hydrodynamic and evaluation of modified designs presented in previous chapters a novel efficient hydrocyclone is designed, which is addressed in Chapter 7. Performance of the novel hydrocyclone is evaluated over a range of Reynolds number. Chapter 8 reflects a scope for further modification of hydrocyclone using a rotating membrane. Performances of a rotating membrane over a range of operating conditions are examined in Chapter 8. A detail modeling of droplet breakage and coalescence using population balance method is addressed in Chapter 9. A coupled multiphase and population balance method is applied to examine the variation of droplet size distribution in the flow field of the novel hydrocyclone. A summary of research outcomes are given in Chapter 10.
2.1. Introduction

For numerous separation devices, including hydrocyclone, the size and the shape depend on the operation such as clarification, classification, and thickening. A certain performance criterion and a class of hydrocyclone are available for each type of applications. For example, the design and location of the vortex finder, the geometric aspect ratios, and the flow split ratio, determine whether a hydrocyclone will be used as a clarifier, classifier, or thickener. The design of a thickener maximizes the proportion of solids collected in the underflow. On the other hand, a classifier is designed to split feed flow into two volume fractions based on the above and below a defined cut-off diameter of particles. In contrast, a clarifier is designed to collect clean water from underflow and to maximize the purity coefficient of underflow water.

In the clarification operation, concentrated oil-water phase is collected from overflow outlet, which is usually between 1 to 10 percent of feed flow rate and the cleaner water phase is collected from underflow orifice. An example of a clarifier is the oil-water separation hydrocyclone, in which cleaner water and concentrated oil-water mixture are collected from underflow and overflow orifices, respectively. Since this research is only focused on the clarification operation, further discussion on the classifier
and thickener are ignored here. A detail description of the oil-water separation hydrocyclones (clarifier) is given below.

2.2. Operating features of a hydrocyclone

Hydrocyclones can operate in two distinct modes: a reverse flow mode (Fig. 2.1a) or a forward flow mode (Fig. 2.1b). In the forward flow hydrocyclones, both the lighter and heavier phases are discharged from the bottom. The lighter phase in the reverse flow hydrocyclones is discharged from the top and the heavier phase from the bottom. Hydrocyclones are often tapered and possess two axial outlets (overflow and underflow) and one or two tangential inlets. The main sections of a hydrocyclone are the cylindrical inlet chamber, swirl chamber, tail pipe, and vortex finder. The inlet tube/channel is tangentially attached to the inlet chamber at the top of the hydrocyclone. The shape and angle of swirl chamber characterize the pattern of core flow in the hydrocyclones. In a de-oiling application, the tail pipe is connected to enhance the residence time of flow so that the dispersed droplets get sufficient time to migrate to the center of hydrocyclone. A small vortex finder is installed at the top of the reverse flow hydrocyclone (Fig. 2.1a) to prevent short circuit boundary layer flow. In the forward flow hydrocyclone (Fig. 2.1b), the vortex finder is connected at the bottom and the underflow stream is usually taken radially out from the hydrocyclone. Moreover, the top of the forward flow hydrocyclone is closed and a vortex stabilizer is attached at the middle of top surface to stabilize core flow.
Figure 2.1: Shape and dimensions of (a) a reverse flow and (b) a forward flow hydrocyclone used for liquid-liquid separation.
2.3. Flow features and separation mechanism

Hydrocyclones separate two phases based on their density difference. The liquid mixture (oil-water mixture) is fed tangentially into the hydrocyclone. The liquid mixture spins in a vortex flow pattern (Fig. 1.1) due to the tangential feed and the curvature of hydrocyclone wall. The rotational acceleration increases as the internal diameter reduces over the length of the hydrocyclone. The swirling motion generates centrifugal force on the liquid phases. Due to a difference in the centrifugal force, as a consequence of density differential between two phases, the heavier liquid (water) moves toward the wall and the lighter liquid (oil) migrates toward the axis of the hydrocyclone. The water flows out through the tailpipe creating the underflow stream. An axial pressure gradient generated at the centre (a lower pressure near the overflow orifice and a higher pressure in the downstream of overflow orifice) creates a reverse flow core, which thereby forces the migrated oil to flow in the opposite direction. The oil rich core flows from the hydrocyclone through a vortex finder creating the overflow stream. The centrifugal force varies over the length of the hydrocyclone and is directly proportional to the square of the tangential (rotational) velocity. The migration rate of dispersed droplets or the effectiveness of a separator is generally evaluated in terms of the Relative Centrifugal Force (RCF), which is defined as the ratio between the centrifugal force and the force due to gravity:

\[
\text{RCF} = \frac{F_c}{F_g} = \frac{m_{D}a_c}{m_{D}a_g} = \frac{U_0^2}{R_a g} \tag{2.1}
\]

For a liquid-liquid separation, a 100% separation efficiency of a hydrocyclone is not achievable because of short circuit boundary layer flows. On the wall of a hydrocyclone, the magnitude of tangential velocity component is zero, i.e. the RCF (Eq. (2.1)) is zero.
Adjacent to the wall, the magnitude of tangential velocity component is very small due to the no slip boundary conditions on the wall and the viscous resistance. This slow moving fluid flow adjacent to the wall, which is called side wall boundary layer flow, creates a short circuit flow from inlet to the underflow orifice. If any dispersed droplet appears in the side wall boundary layer, it follows the short circuit flow with experiencing a negligible RCF, which thereby yields the droplet to move directly from inlet to the underflow orifice.

2.4. Geometric features of a hydrocyclone

A wide variation of shapes and dimensions of liquid-liquid separation hydrocyclones are considered. Most of the designs fall into two common categories: single cone (Fig. 2.1) and dual cone (Fig.2.2). The CT-design (see Colman and Thew, 1988) is a dual cone hydrocyclone which has four main sections: a cylindrical inlet chamber, a short conical reducing chamber, a long conical swirl chamber, and a tailpipe (Fig. 2.2). The cone angle of the reducing chamber (β) and the swirl chamber (θ) is usually 20° and 1.5°, respectively. Young et al. (1994) modified the CT-design by eliminating the reducing section. The cone angle was changed from 1.5° to 6°. Young’s design is a single cone hydrocyclone (Fig. 2.1a) which is similar to the Rietema (1961) and Bradley (1965) hydrocyclones used for solid-liquid separation. They also modified the tailpipe section replacing the cylindrical pipe by a minute angle conical section. Belaidi and Thew (2003) revised the CT-design for aiding the removal of free gas in water and enhancing the stability. In their design, the cylindrical inlet chamber was replaced by a tapered inlet chamber and the cone angle is slightly larger, which is of 2°.
The size and shape of different geometric sections of a hydrocyclone varies based on the operations and applications. For each type of application, there is a specific geometric class of hydrocyclones. The geometric features of each section of hydrocyclones used for liquid-liquid are broadly discussed below.

Figure 2.2: Dual cone hydrocyclone used for liquid-liquid separation.

2.4.1. Inlet

The inlet design has noticeable effects on the centrifugal force at the feed of a hydrocyclone (Noroozi et al., 2009). The flow structures and swirl intensity in the hydrocyclones depend on the size and shape of the inlet. Gomez et al. (2001), Caldentey et al. (2000), and Amini et al. (2012) showed in their mathematical models that the swirl intensity can be calculated from the ratio of inlet diameter to the characteristic diameter of a hydrocyclone. So, the inlet configuration is an important parameter in the performance of a hydrocyclone. Rectangular or circular, single or twin inlets have been most frequently used by different researchers. An optimized design of inlet is essential for ensuring the feed of fluid with a higher tangential velocity as well as the reduction of shear stress for avoiding rupture of dispersed droplets. Young et al. (1994) investigated
experimentally the effect of inlet size on the separation efficiency and they found that when the ratio of inlet diameter ($D_F$) to the inlet chamber diameter ($D$) is equal to 0.25 provides a better performance. They mentioned that an inlet size much smaller than the $0.25D$ provides poor separation efficiency. For a rectangular inlet, the thickness of the inlet duct is another influential parameter for the performance of hydrocyclones. A smaller thickness yields a higher pressure drop across the hydrocyclone and vice versa. For liquid-liquid separation, the higher radial pressure gradient across the hydrocyclone is favorable, because the lighter dispersed phase can easily migrate toward the core. However, an excessive radial pressure drop at the inlet chamber may reduce the axial pressure gradient. For ensuring satisfactory separation efficiency in the liquid-liquid separation hydrocyclones, a certain level of axial pressure gradient is required to drive the lighter dispersed phase to the overflow orifice. Yuan (1992) suggested that the $a/D = 0.2$ is suitable for the liquid-liquid separation hydrocyclones. Noroozi et al. (2009) observed the effect of different inlet designs (standard, involute, helical) on the separation efficiency on a 40 mm Colman and Thew (1988) type hydrocyclone. They found that the separation efficiency can be improved, in comparison with standard inlet, by about 7% when using involute inlet and 10% when using a helical form of inlet. The twin inlets make a better symmetry in the swirling motion and maintain a stable reverse flow core.
2.4.2. Inlet chamber

A cylindrical canister is usually used as the inlet chamber for a typical hydrocyclone. The size, shape and length of the inlet chamber affect the separation efficiency since the maximum angular momentum in the hydrocyclone depends on the inlet chamber design. Kharoua et al. (2010) mentioned that the cylindrical inlet chamber is necessary to avoid high shear region downstream of the feed and to reduce the head loss. However, a shorter length produces a better separation (Young et al., 1994). Moreover, an excessively longer inlet chamber leads weakening the angular momentum due to the frictional drag on the wall. At the inlet chamber, the kinetic energy dissipates at faster rate and forms recirculation eddies, which thereby resists dispersed droplets to migrate toward the center. The length of the inlet chamber of a conventional hydrocyclone used for liquid-liquid separation is usually equal to the diameter of that chamber (Thew, 2000). Belaidi and Thew (2003) redesigned the inlet chamber by using a frusto-cone instead of a straight cylinder. They claimed that the tapered inlet chamber enables both the stability and reduction of disturbances in the flow field near the overflow. Noroozi and Hashemabadi (2011) conducted CFD analysis to investigate the effect of inlet chamber body profile on the separation efficiency. They replaced the cylindrical inlet chamber by exponential and conical profiles. They observed that the conical and exponential inlet chamber improves the separation efficiency approximately by 4.5% and 8%, respectively. Moreover, the exponential body profile provides a higher angular momentum, lower kinetic energy dissipation, and eliminates the recirculation eddies in inlet chamber with reference to the cylindrical inlet chamber (Noroozi and Hashemabadi, 2011).
2.4.3. Swirl chamber

The conical section of a hydrocyclone maintains a constant swirl intensity and stronger centrifugal separation (Kharoua et al. 2010). The loss of tangential momentum due to the wall friction is recuperated by the enhancement of momentum in tapered surface. The amount of recovery in the tangential momentum loss depends on the conical angle of swirl chamber. The most important geometric factor that influences the separation efficiency is the cone angle (Saidi et al. 2013). Much of the liquid-liquid separation occurs in the conical swirl chamber of the hydrocyclones. Colman and Thew at the University of Southampton in the late 1970s found that a cone angle as small as 1 – 1 ½ degree produces the necessary stable vortex with a small relatively fast moving core moving to the overflow (Thew, 2000). Young et al. (1994) conducted an experimental search for optimum geometry of a single cone hydrocyclone. They observed that shallow cone angles of 1.5 and 3° provide poor separation because of low tangential velocity. A cone angle of 6° provides good separation over a broad range of feed flow rates (Young et al. 1994). A larger cone angle leads to a more severe radial pressure gradient, which causes a higher migration velocity dispersed droplets toward the center of hydrocyclone. Saidi et al. (2013) investigated the effect of cone angle on the separation performance of Young’s class hydrocyclone for 6°, 10° and 20° degree cone angle. They observed that the hydrocyclone with 6° cone angle has a higher separation efficiency than that of 10° and 20°. Although an increase in the cone angle increases the tangential velocity and pressure gradient, it reduces the flow residence time in the hydrocyclone by increasing the axial velocity.
2.4.4. Tailpipe

A tailpipe is connected at the end of the swirl chamber for increasing the flow residence time. The dispersed droplets are subjected to centrifugal force for much longer time due to an addition of tail pipe. Separation of two phases continues to occur as the fluid spins in the tailpipe section of the hydrocyclone. A very long tailpipe, longer than 18 times of the hydrocyclone diameter (D), does not provide any additional separation (Young et al. 1994). Hargreaves and Silvester (1990) and Thew (2000) described that an optimal/commercial hydrocyclone used for de-oiling application should have a length of tailpipe equal to 10 times of the hydrocyclone diameter (D). A tail pipe longer than 10D does not provide any additional separation, because the tangential momentum in the tailpipe declines in the downstream and eventually vanishes due to the wall friction.

2.4.5. Overflow and underflow orifice

The diameter of overflow orifice is a critical parameter, both in terms of control and separation. The minimum overflow rate to make an effective separation increases with an increase in overflow diameter (Young et al. 1994). Colman and Thew (1988) suggested that the ratio of overflow to inlet chamber diameters for the dual cone hydrocyclone (Fig. 2.2) should be equal or less than 0.07 (Do/D ≤ 0.07). Young et al. (1994) observed a better oil-water separation in a single cone hydrocyclone (Fig. 2.1a) for Do/D = 0.039. However, a too small ratio (Do/D < 0.1) is not be able to accommodate the reverse flow vortex, while a too large ratio (Do/D > 0.2) is not be able to maintain the required axial pressure gradient for an acceptable separation (Belaidi and Thew, 2003).
2.5. Operating and performance metrics

2.5.1. Separation efficiency for dispersed phase mechanics

The separation efficiency of a liquid-liquid separation hydrocyclone is evaluated by estimating the overflow grade separation efficiency \( G_O(d_D) \). The overflow grade separation efficiency \( G_O \) is defined as the fraction of dispersed droplets of certain size \( d_D \) that reports to the overflow outlet. The \( G_O \) is related to the size distributions of dispersed droplets at the overflow and feed as follows:

\[
G_O(d_D) = \frac{\dot{m}_{DO}\phi_{DF}(d_D) \partial d_D}{\dot{m}_{DF}\phi_{DF}(d_D) \partial d_D} \tag{2.2}
\]

where \( \dot{m}_D \) is the total mass flow rate of dispersed phase and \( f_D(d_D) \partial d_D \) represents the mass fraction of dispersed droplets having sizes between \( d_D \) and \( d_D + \Delta d_D \). No interaction of dispersed droplets with each other as well as with the continuous phase, and no coalescence and breakage of dispersed droplets are considered in this definition.

With this assumption, the overflow grade separation efficiency can be calculated based on the number of droplets as

\[
G_O(d_D) = \frac{N_{DO}(d_D) \partial d_D}{N_{DF}(d_D) \partial d_D} \tag{2.3}
\]

where \( \dot{m}_{DF}\phi_{DF}(d_D) \partial d_D = \dot{m}_{DF}(d_D)N_{DF}(d_D) \partial d_D \); \( X \equiv O, F \).

Here, \( \dot{m}_D(d_D) \) is the mass flow rate of a dispersed droplet having the size of \( d_D \), and \( \dot{m}_D \) is the total mass flow rate of dispersed phase. \( N_{DF}(d_D) \partial d_D \) and \( N_{DO}(d_D) \partial d_D \) are, respectively, the number of droplets injected from the inlet and the number of droplets reported at the overflow orifice with sizes between \( d_D \) and \( d_D + \Delta d_D \). If there is no coalescence and breakage, and a group of mono-sized droplets are injected from the inlets then the grade separation efficiency for that certain size of droplets can be calculated as
$G_0(d_D) = \frac{N_{D0}(d_D)}{N_{DF}(d_D)}$ \hspace{1cm} (2.4)

where $N_{DF}(d_D)$ is the number of mono-sized droplets injected (uniformly distributed in the feed) from the inlet and $N_{D0}(d_D)$ is the number of mono-sized droplets appears in the overflow orifice. The overall separation efficiency ($\eta_o$) is calculated by integrating the grade separation efficiency over the entire size range in the distribution, which can be expressed as

$$\eta_o = \int_0^\infty G_0(d_D) f_{DF}(d_D) \, \partial d_D$$ \hspace{1cm} (2.5)

### 2.5.2. Separation efficiency for multiphase phase mechanics

The macroscopic material balance equation for dispersed phase can be given as $Q_F C_F = Q_O C_O + Q_U C_U$ where $C_O$, $C_U$ and $C_F$ are the concentration of dispersed phase at, respectively overflow, underflow and inlet surface. The overall separation efficiency is defined as the ratio of mass flow rate of dispersed phase at the overflow to that at the feed, which can be given as:

$$\eta_o = \frac{\dot{m}_{DO}}{\dot{m}_{DF}} = \frac{Q_O C_O}{Q_F C_F} = 1 - \frac{Q_U C_U}{Q_F C_F}$$ \hspace{1cm} (2.6)

### 2.5.3. Reduced separation efficiency

The effective grade separation efficiency of a hydrocyclone is represented by reduced separation efficiency. According to Stokes’ law, a small droplet corresponding to a very small Stokes number follows the flow stream of continuous phase. For example, if the overflow ratio ($R_O$) (see section 2.5.4 for the definition) of a hydrocyclone is 0.1 then the separation efficiency for a very small Stokes number cannot be less than 0.1. In other words, if the separation efficiency is 0.1 (corresponding to an overflow ratio of 0.1) the
actual separation done by the hydrocyclone is zero. In that case the hydrocyclone acts as a flow splitter. However, the reduced separation efficiency represents the effective separation efficiency of a hydrocyclone which is independent of overflow ratio ($R_O$). The reduced grade efficiency for a certain sized droplet is calculated as:

$$G_{OR}(d_D) = \frac{G_O(d_D) - R_O}{1 - R_O}$$  \hspace{1cm} (2.7)

The overall reduced separation efficiency is calculated as (see APPENDIX B)

$$\eta_{OR} = \frac{\eta_{O} - R_O}{1 - R_O} = 1 - \frac{C_{DU}}{C_{DF}}$$  \hspace{1cm} (2.8)

### 2.5.4. The feed Reynolds number

The feed conditions of hydrocyclones are characterized by the feed Reynolds number ($Re_F = \rho U_F D_F/\mu$). $U_F$ is the feed velocity, $D_F$ is the hydraulic diameter of a rectangular inlet, and $Q_F$ and $Q_O$ are, respectively, the feed and overflow rates. The hydraulic inlet diameter is defined as $D_F = 2ab/(a + b)$ where $a$ and $b$ is, respectively, the width and height of rectangular inlets.

### 2.5.5. Overflow ratio

Overflow ratio ($R_O$) is one of the influential operating and performance parameters of hydrocyclone for liquid-liquid separation. It is defined as the ratio of overflow rate to the feed flow rate. Below a minimum value of the $R_O$, some oil begins to be lost in the underflow and the separation efficiency decreases. An increase in the $R_O$ beyond a certain value implies that more water is present in the overflow (Colman and Thew, 1980; Colman et al., 1984; Meldrum, 1988; Young et al., 1994). As a consequence, a further separation is required to remove the excess water from the
overflow stream. Moreover, a large Ro decreases the reduced separation efficiency, $\eta_{OR}$ (see Eq. (2.8)).

$$R_O = \frac{\text{Overflow rate}}{\text{feed flow rate}} = \frac{(Q_o)}{Q_F} \text{water}$$  \hspace{1cm} (2.9)

2.5.6. Pressure drop ratio

Equations (2.7 and 2.8) indicate that the overall separation efficiency of a hydrocyclone can be enhanced by reducing the $R_O$. However, too small $R_O$ can decrease oil removal. A too large $R_O$ decreases overall separation efficiency and may put unnecessary load on the drains system, which normally handles the oil reject and pumps the oil back into the process. For an optimum operability, hydrocyclones should always be controlled with an acceptable range of $R_O$. The overflow ratio is controlled by adjusting feed, overflow, and underflow pressures together. The combined pressure control parameter is characterized by pressure drop ratio ($\Delta P^*$). It is the ratio of the difference between the feed and overflow pressures to the difference between the feed and underflow pressures.

$$\Delta P^* = \frac{P_F - P_O}{P_F - P_U}$$  \hspace{1cm} (2.10)

where $P_F$, $P_O$ and $P_U$ are the feed, overflow, and underflow pressure, respectively. Usually a minimum underflow pressure is needed to create a back-pressure to force the oil-rich core out from the overflow orifice. Young et al. (1994) have studied the effects of the underflow pressure and the pressure drop between the inlet and the underflow outlet on the separation efficiency for Bumpass crude (0.85 g/cm$^3$) and South China Sea (0.95 g/cm$^3$) in a 35-mm Colman and Thew hydrocyclone. The overflow rate was found to be
dependent significantly on the pressure drop ratio. In general, the effects of feed, underflow, and overflow pressures are controlled by adjusting the $\Delta P^*$ value.

### 2.5.7. Euler number

The Euler number represents a relationship between the pressure drop and the kinetic energy at the feed per unit volume. Two Euler numbers can be defined in the conventional hydrocyclone: overflow Euler number ($Eu_O$) and underflow Euler number ($Eu_U$). The equation for the overflow and underflow Euler numbers can be given as:

$$
Eu_O = \frac{P_F - P_O}{\rho U^2_F} \quad \text{and} \quad Eu_U = \frac{P_F - P_U}{\rho U^2_F}
$$

(2.11)

The ratio between the overflow and underflow Euler number is the same as the $\Delta P^*$.

$$
\frac{Eu_O}{Eu_U} = \frac{P_F - P_O}{\rho U^2_F} / \frac{P_F - P_U}{\rho U^2_F} = \frac{P_F - P_O}{P_F - P_U} = \Delta P^*
$$
3.1. Introduction

Eulerian-Eulerian and Eulerian-Lagrangian approaches are utilized for the numerical solution of multiphase flow problems. In the Eulerian-Eulerian approach, different phases are treated as interpenetrating continua and a set of conservation equations are solved for each phase. The governing equations solved for the multiphase flow simulation using the Eulerian-Eulerian approach are presented in APPENDIX A3. In the Eulerian-Lagrangian approach, the mean velocity and pressure of the continuous phase are calculated by solving the governing equations (APPENDIX A1) using an Eulerian approach. Motions of the dispersed droplets are determined by tracking their trajectories through the calculated flow field using a Lagrangian approach, which is explained in APPENDIX A2. The Eulerian-Lagrangian approach is appropriate for a volume fraction of a dispersed phase less than 0.1 (Ansys document: Theory guide, 2013). A typical concentration of oil in the produced water to be cleaned, which the de-oiling hydrocyclones are typically utilized for, is much less than 1% by volume (Ahmadun et al., 2009; Motin et al., 2014). The properties of continuous (water) phase and dispersed (oil) phase used in this work are shown in Table 3.1. Grid independent study is conducted for finding appropriate mesh size and the numerical simulation is
validated with experimental and published simulation results (see, especially, Celik, 2003; Saidi et al., 2013; Bai et al., 2009)

Table 3.1: Material properties of the continuous and dispersed phases

<table>
<thead>
<tr>
<th>Phase</th>
<th>Density (kg/m$^3$)</th>
<th>Viscosity (Ns/m$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water (continuous)</td>
<td>998</td>
<td>0.001</td>
</tr>
<tr>
<td>Oil (dispersed)</td>
<td>850</td>
<td>0.0032</td>
</tr>
</tbody>
</table>

3.2. Numerical details and procedure

The balance equations for mass and momentum, kinematic Reynolds momentum flux $\langle uu \rangle$, turbulent kinetic energy $k$, and dissipation rate $\varepsilon$ are solved using a pressure-based segregated solver supported by Ansys Fluent 14.5 (see APPENDIX A).

The simulations of the continuous phase (Eulerian-Lagrangian approach) or mixture phase (Eulerian-Eulerian approach) are achieved in three steps to overcome convergence problem with the RSM. First, a preliminary solution is obtained using the standard k-$\varepsilon$ model. The simulation is continued until the scaled residuals for continuity and momentum are about $10^{-4}$. After that, the turbulent model is switched to the RSM and the simulation is continued until the scaled residuals for continuity, momentum and stress components are less than $10^{-4}$. In both steps, simulations are performed in a steady-state manner. The Semi-Infinite Method for Pressure Linked Equation (SIMPLE) algorithm (Ansys Fluent documentation: theory guide, 2013) was used to achieve pressure-velocity coupling between the continuity and momentum equations. Stability of numerical computation was achieved by setting under-relaxation factors for pressure, momentum,
turbulent kinetic energy, turbulent dissipation rate, and Reynolds stresses to 0.3, 0.5, 0.8, 0.8, and 0.5, respectively. The results obtained from the steady-state simulation are then used as an initial condition for the transient simulation. The transient simulation is continued until a statistically steady-state solution is obtained. The simulation time required to achieve the statistically steady-state results for all the cases investigated in this research is observed to be less than 500 milliseconds. At every time step, the residual levels for continuity, momentum and all the stresses were less than $10^{-4}$. Normalized residuals in the range of $10^{-3}$ indicate a practically-converged solution for the RSM (Ansys Fluent documentation: user guide, 2013). A tighter convergence criterion (in the range of $10^{-4}$) is, however, maintained to ensure accuracy.

In the transient simulations, the coupling between pressure and velocity is achieved using Pressure Implicit with Splitting of Operator (PISO) (Andersson et al., 2012). The time step used for the simulation changes from 10-50 microseconds. Pressure interpolation is performed using the PRESSure STaggering Option (PRESTO) scheme (Patankar, 1980). The PRESTO scheme provides a better estimation for high speed swirling flows, and flows in strongly curved domains (Ansys Fluent documentation: user guide, 2013). The equations for momentum, kinetic energy, dissipation and Reynolds stresses are discretized using Quadratic Upstream Interpolation for Convective Kinetics (QUICK) scheme (Leonard, 1979) since it provides a better accuracy and is suitable for a hexahedral mesh (Ansys Fluent documentation: user guide, 2013). The time discretization is achieved by using a second order implicit scheme. All the gradients are discretized base on the least squared cell based approach.
Trajectories of dispersed droplets are calculated by solving the equations presented in APPENDIX A2 using a 5th order Runge-Kutta scheme as derived by Cash and Carp (1990). The integration time step is estimated from the relation \( \Delta t = \Delta t^*/\lambda \) where \( \Delta t^* = V_{\text{cell}}/(A_{\text{face (max)}}\|\langle U_d \rangle\| + \|\langle U_c \rangle\|) \), and \( \lambda \) is the number of integration steps (\( \lambda = 5 \)) for each grid cell (see Motin et al. 2015). \( V_{\text{cell}} \) is the cell volume, and \( A_{\text{face (max)}} \) is the maximum face area of the cell \( \langle U_d \rangle \) and \( \langle U_c \rangle \) represents, respectively, the mean velocity of the dispersed phase and continuous phase. The total number of time steps is set as that there are no droplets with incomplete trajectories in the flow domain.

### 3.3. Boundary conditions and wall function

A plug flow is specified at the inlets. The overflow orifice is in a flooded condition (no air core). The average pressure specified at the overflow and underflow orifices depends on the operating conditions which will be described in the next chapters. A turbulent intensity (I) of 5% is specified at the inlets. The kinematic Reynolds momentum fluxes specified at the inlets are calculated from the turbulent intensity as

\[
\langle u_F u_F \rangle: e_i e_j = (U_F I)^2 \quad \text{for} \quad i = j \quad \text{and} \quad \langle u_F u_F \rangle: e_i e_j = 0 \quad \text{for} \quad i \neq j
\]

where \( i \) and \( j \) are index notations. No-slip boundary conditions are applied at the walls. A scalable wall function is used for near wall treatment. The scalable wall function produces consistent results for grids with arbitrary refinement (Ansys Fluent documentation: theory guide, 2013). Since the hydrocyclone geometry is tapered in shape and a structured hexahedral mesh is employed in it, values of \( y^* \) at the centroid of wall adjacent cells are not constant. The scalable wall function avoids deterioration of accuracy of the Standard Wall Function (SWF) under grid refinement below \( y^* < 11 \) (Ansys Fluent documentation: 39
theory guide, 2013). For $y^*<11$, the scalable wall function introduce a limiter as $\tilde{y}^* = \text{Max}(y^*,11)$. For the region of wall adjacent cells in the hydrocyclone where $y^* > 11$, the scalable wall function is identical to the SWF. The SWF have been most widely used in industrial flows and works with reasonable accuracy for a high Reynolds number and wall bounded flow (Ansys Fluent documentation: user guide, 2013).

3.4. Grid independent study

The grid independent study is conducted in a single cone hydrocyclone which is shown in Fig. 3.1. The geometric dimensions of the hydrocyclone are given in Table 3.2. An O-grid structured hexahedral mesh is generated using the commercial software ANSYS ICEM CFD. A finer mesh is generated at the core region because of the high sensitivity of separation efficiency on the core flow patterns. A finer mesh is also generated near the wall region to capture the near wall flow features accurately. The mesh density near the wall region satisfies the criterion of dimensionless wall distance for the selected wall function. The final mesh density is chosen based on a grid independent study. The dependency of numerical solutions on the grid sizes are examined by comparing the results for the following different grid sizes: 84,600, 202,800 and 663,800 cells (see Fig. 3.2).

Figure 3.1: Schematic of a single cone hydrocyclone (Dimensions are given in Table 3.2)
Table 3.2: Geometric groups and dimensions of the single cone hydrocyclone

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>26</td>
<td>0.07</td>
<td>0.23</td>
<td>0.275</td>
<td>0.4</td>
<td>0.39</td>
<td>2</td>
<td>1</td>
<td>7.37</td>
<td>0.1</td>
<td>9.71</td>
<td>6^0</td>
</tr>
</tbody>
</table>

Figure 3.2: An O-grid structured hexahedral mesh generated by using ANSYS ICEM CFD. Finer grids are created near the wall and at the core.
The effect of grid refinements on the relative numerical errors and grid convergence index for the macroscopic properties is presented in Table 3.3. The relative error \( E_a \) and the grid convergence index (GCI) are calculated based on the procedure described in APPENDIX C. Table 3.3 shows that the approximate relative error \( (E_a) \) is close to 1% between the grid sizes for all the macroscopic properties. Moreover, the extrapolated related error \( (E_{a_{ext}}) \) and grid convergence index (GCI) for all the macroscopic properties are less than 1%. On the basis of the GCI for the macroscopic properties, it can be inferred that the finest grid size (663,812 cells) is sufficiently fine and should provide accurate numerical solution.

### Table 3.3: Effect of mesh refinement on the predicted macroscopic properties

<table>
<thead>
<tr>
<th></th>
<th>( R_0 ) (Eq. (2.9))</th>
<th>( E_{u_0} ) (Eq. (2.11))</th>
<th>( E_{u_1} ) (Eq. (2.11))</th>
<th>PDR (Eq. (2.10))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approximate relative error ( E_a ) (%)</td>
<td>0.97</td>
<td>0.82</td>
<td>1.19</td>
<td>0.37</td>
</tr>
<tr>
<td>Extrapolated relative error ( E_{a_{ext}} ) (%)</td>
<td>0.15</td>
<td>0.22</td>
<td>0.32</td>
<td>0.092</td>
</tr>
<tr>
<td>Grid Convergence Index ( GCI ) (%)</td>
<td>0.19</td>
<td>0.27</td>
<td>0.39</td>
<td>0.12</td>
</tr>
</tbody>
</table>

(see APPENDIX C for the definition of \( E_a, E_{a_{ext}}, \) and GCI)

### 3.4.1. Effect of mesh refinement on the primary variables and derived quantities

The effect of grid refinement on the tangential velocity profile is shown in Fig. 3.3. The normalized peak tangential velocity at both the \( \frac{z}{D} = 1 \) and 8.38 is monotonically increasing with an increase in the mesh density. A similar trend is also observed near the center and near the wall. The relative difference in the normalized
tangential velocity decreases with an increase in the number of grid cells. The $E_a$, $E_{a_{ext}}$ and GCI (see Fig. 3.3) are calculated (see APPENDIX C) at a radial location where the differences in tangential velocities are at a maximum. Figure 3.3 shows that the relative errors and the GCI are higher at $z/D = 1$ than that at the $z/D = 8.38$. The maximum possible numerical error for the finest grid (i.e., 663,812 cells) at $z/D = 1$ is less than 10%. At $z/D = 8.38$, both $E_{a_{ext}}$ and GCI for the finest grid are negligible (i.e., much less than 1%).

On the basis of the relative errors in the normalized tangential velocity the mesh density with 663,812 cells is sufficiently fine for an accurate numerical solution. The effect of the grid refinement on the normalized axial velocity is shown in Fig. 3.4. The relative errors and GCI are calculated at the center of the hydrocyclone ($r/R = 0$), where the differences in axial velocities are maximum. The largest relative errors and the GCI for the axial velocity at $z/D = 1$ and 8.38 are about 5% and less than 1%, respectively. So, the values of GCI, $E_a$ and $E_{a_{ext}}$ in the axial velocity also indicate that the mesh with 663,812 cells is sufficiently fine for an accurate numerical solution.

Figure 3.5 shows the effect of grid refinement on the normalized pressure at $z/D = 1$ and 8.38. At both of these locations, the normalized pressure near the center of the hydrocyclone is monotonically decreasing. The maximum possible extrapolated error for the normalized pressure is found to be less than 5%, which indicates that the grid refinement is adequate for obtaining an accurate numerical solution.
Figure 3.3: Effect of mesh refinement on the normalized tangential velocity at $z/D = 1$ and 8.38. Relative errors and grid convergence index shown in the table are calculated at a radial location where the differences in the tangential velocities are the maximum.
Figure 3.4: Effect of mesh refinement on the normalized axial velocity at $z/D = 1$ and $8.38$. Relative errors and grid convergence index shown in table are calculated for the maximum difference in the axial velocity at $r/R = 0$. 

<table>
<thead>
<tr>
<th>$z/D$</th>
<th>$z/D = 1$</th>
<th>$z/D = 8.38$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_a$</td>
<td>5.8</td>
<td>0.97</td>
</tr>
<tr>
<td>$E_{a_{ext}}$</td>
<td>4.7</td>
<td>0.11</td>
</tr>
<tr>
<td>GCI</td>
<td>5.91</td>
<td>0.14</td>
</tr>
</tbody>
</table>
Figure 3.5: Effect of mesh refinement on the normalized pressure at $z/D = 1$ and 8.38.

Relative errors and grid convergence index shown in table are calculated for the maximum difference in the normalized pressure at $r/R = 0$. 
3.4.2. Effect of mesh refinement on the separation efficiency

The effect of grid size on the numerical accuracy in the grade separation efficiency ($G_R$) is presented in Fig. 3.6. The grade separation efficiency is calculated using the flow properties of continuous and dispersed phases. Numerical accuracy in the grade separation efficiency indicates that the velocity and pressure fields for the continuous phase are predicted accurately. Figure 3.6a shows that the grade separation efficiency is monotonically decreasing with grid refinement. The extrapolated grade separation efficiency, which represents result for an extremely fine mesh, appears almost equal to the result obtained using 663,800 cells. The extrapolated grade separation efficiency and the numerical uncertainty (GCI) for the finest grid (663,812 cells) are presented in Fig. 3.6b. The error bars represents the numerical uncertainty for the result with 663,812 cells. The numerical uncertainty is almost negligible for larger dispersed droplet size. The numerical uncertainty increases with a decrease in the dispersed droplet size. However, the maximum GCI is less than 10%. So, it can be inferred that the mesh with 663,812 cells is sufficiently fine and simulation results obtained in this class of hydrocyclone using more than 663,812 cells are independent of grid size.
Figure 3.6: (a) Effect of mesh refinement on the reduced grade efficiency and (b) Extrapolated value of the reduced grade efficiency with GCI as error bar for the finest grid (663812 cells). Extrapolated value of $G_{OR}$ for the finest grid almost matches with that for 663812 cells. The finest mesh (663812 cells) provides a greater GCI for a smaller droplet size. However, the maximum GCI is less than 10%.
3.5. Validation of numerical results

The tangential velocity profile, which is obtained using the simulation approach presented in APPENDIX A, on a 35 mm (diameter of inlet chamber) Young’s class hydrocyclone is qualitatively compared with experimental and numerical investigations performed on the same hydrocyclone by Bai et al. (2009) and Saidi et al. (2012, 2013), respectively. There is no experimental or numerical investigation of hydrodynamics on a 26 mm single cone hydrocyclone (the nominal diameter that is used in this work) found in the literature. In the experimental investigation of Bai et al., a two-component laser doppler velocimeter was used to measure the mean flow velocities. Saidi et al. numerically investigated the velocity profiles based on an LES approach. The qualitative comparison of tangential velocity is shown in Fig. 3.7. Figure 3.7 shows that the tangential velocity profile obtained in the present work by solving the RANS equations with the RSM is qualitatively and quantitatively more closer to the experimental value when we compare with the Saidi et al.’s result. Near the wall region, the tangential velocity using the LES approach shows a larger gradient. However, the tangential velocity calculated from the RANS simulation provides a similar trend to the experimental data. A considerable difference between the experimental study and the present work appears at the center (r/R ≈ 0) of the hydrocyclone. In the experimental result, the normalized tangential velocity at the center is about 1.5. Although a dual inlet hydrocyclone were used in the Bai et al. study, the tangential velocity at the center shows that the flow is not symmetrical. Nevertheless, the trends in the data and the simulations are similar.
Figure 3.7: Qualitative comparison of tangential velocity calculated using the RANS simulation on a 35 mm single cone hydrocyclone with that obtained from previous research on a similar hydrocyclone based on the LES approach and experimental investigation. Velocity profile is taken at \( z/D = 2.29 \).

The reduced grade separation efficiency \( G_{OR}(d_D) \), which is calculated using Eq. (2.4 and 2.7), on a 35 mm (diameter of inlet chamber) Young’s class hydrocyclone is qualitatively compared with a numerical investigations performed on the same hydrocyclone by Saidi et al. (2013), respectively. In this work, for droplet trajectories calculation, a dispersed phase model with stochastic tracking approach is applied in the flow field obtained using the RANS approach. Saidi et al. calculated the flow field using
a LES approach and applied the same dispersed phase model for droplet trajectory calculation. The qualitative comparison of grade separation efficiency is shown in Fig. 3.8. Figure 3.8 shows that the grade separation efficiency calculated in the present study is both qualitatively and quantitatively similar to that obtained in Saidi et al.’s study.

Figure 3.8: Qualitative comparison of reduced grade separation efficiency $G_{OR}(d_D)$ calculated using the RANS simulation on a 35 mm single cone hydrocyclone with that obtained from previous research on a similar hydrocyclone based on the LES approach.
CHAPTER 4

INTERNAL FLOW STRUCTURES AND SEPARATION PERFORMANCE OF A
CONVENTIONAL HYDROCYCLONE

4.1. Introduction

This chapter includes investigation of internal flow structures and separation performances of a single-cone hydrocyclone (Fig. 3.1) using computational fluid dynamic simulations. The objective of this investigation is to develop a quantitative understanding of finite turndown ratio in a conventional hydrocyclone. The characteristics of the finite turndown ratio and the catastrophic drop in the separation performance are explained from the hydrodynamics points.

Both Eulerian-Lagrangian and Eulerian-Eulerian approach is utilized for understanding the internal hydrodynamics. Velocity and pressure fields are calculated by solving the Reynolds Averaged Navier-Stokes (RANS) equations closed by a Reynolds Stress Model (RSM) using numerical simulations (see APPENDIX A1 and section 2.2). The droplets trajectories and separation performance are calculated using Lagrangian approach (see APPENDIX A2). Effect of the Reynolds number on the mean velocity and pressure field, size and shape of reverse flow core, short circuit flows, toroidal recirculation, dispersed and multiphase mechanics, and the separation performance addressed here. Conditions leading to a finite turndown ratio in a hydrocyclone are discussed from hydrodynamics arguments. The results presented in this chapter identify
fundamental flow characteristics associated with the finite turndown ratio in a hydrocyclone. The simulation provides a hydrodynamic explanation for finite turndown ratio phenomena in a flooded hydrocyclone.

4.2. Hydrocyclone geometry and operating conditions

The geometry of hydrocyclone used for the simulations is shown in Fig. 3.1. The nominal diameter of the inlet chamber is 26 mm. The geometric groups and dimensions are presented in Table 3.2. Two symmetrical rectangular channel inlets (4 mm×10 mm) are fitted tangentially at the top of the cylindrical inlet chamber. The operating conditions for the different Reynolds number are presented in Table 4.1. The range of feed Reynolds number considered here is reasonable because the typical operating feed Reynolds number of an industrial hydrocyclone is within this range. The average gauge pressure at the overflow orifice equals to zero and it is kept same for all the cases. The average gauge pressure at the underflow orifice is adjusted for each feed Reynolds number (ReF) to maintain a constant overflow ratio (RO). The gauge pressures at the underflow orifice are set to 28 kPa, 50 kPa, 120 kPa, 150 kPa and 400 kPa, respectively, for Case 1 to 5. The average feed pressures (PF) for the different test cases are calculated from the simulation results. The pressure drop ratio (ΔP* = (PF − PO)/(PF − PU)) for all the test cases is greater than one.
Table 4.1: Test cases and operational parameters used in the simulations

<table>
<thead>
<tr>
<th>Test Cases</th>
<th>U_F (m/s)</th>
<th>Q_F (m³/hr.)</th>
<th>P_F (kPa)</th>
<th>Re_F</th>
<th>ΔP*</th>
<th>RO</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>2</td>
<td>0.575</td>
<td>83.9</td>
<td>11440</td>
<td>1.48</td>
<td>0.106</td>
</tr>
<tr>
<td>Case 2</td>
<td>3</td>
<td>0.863</td>
<td>187.9</td>
<td>17160</td>
<td>1.34</td>
<td>0.101</td>
</tr>
<tr>
<td>Case 3</td>
<td>4.167</td>
<td>1.20</td>
<td>384.6</td>
<td>23835</td>
<td>1.41</td>
<td>0.108</td>
</tr>
<tr>
<td>Case 4</td>
<td>5</td>
<td>1.44</td>
<td>538.0</td>
<td>28600</td>
<td>1.37</td>
<td>0.104</td>
</tr>
<tr>
<td>Case 5</td>
<td>10</td>
<td>2.88</td>
<td>2049.5</td>
<td>57200</td>
<td>1.22</td>
<td>0.09</td>
</tr>
</tbody>
</table>

4.3. Computed velocity fields

Normalized tangential velocity profiles along the radial locations are shown in Fig. 4.1. The r/R = 0 and 1 indicates the center and wall of the hydrocyclone. The linear rise in the tangential velocity from the center characterizes a forced-like vortex. There is a free-like vortex motion in the outer region. The span of transition region between the forced-like and the free-like vortex increases in the downstream. The diameter of forced vortex region also increases in the downstream. At the inlet chamber, \( z/D = 0.2 \) (Fig. 4.1a) and the top of the swirl chamber (Fig. 4.1b), the \( \text{Re}_F = 23,835 \) reveals the highest peak in the normalized tangential velocity. The peak value declines with a decrease in the Reynolds number. At the \( \text{Re}_F = 28,600 \), the peak value is less than that of the \( \text{Re}_F = 23,835 \) and \( \text{Re}_F = 17,160 \), because a large volumetric flow in the case of \( \text{Re}_F = 28,600 \) provides a less radial pressure gradient (see Fig. 4.7) as well as higher resistance in the swirling motion near the core. However, the location of peak tangential velocity at a certain axial position is almost the same for all the Reynolds numbers.
Figure 4.1: Normalized tangential velocity profiles for the different Reynolds numbers

(— $Re_F = 28,600$; ⋯⋯⋯⋯ $Re_F = 23,835$; − ⋅ − $Re_F = 17,160$; - - - - $Re_F = 11,440$). A higher normalized peak value represents a greater effective swirling intensity.
Figure 4.2 shows the contours of normalized tangential velocity for different Reynolds numbers. In the upper half on the swirl chamber (z/D < 5) the normalized tangential velocity near the wall appears constant in the axial direction. In the downstream, the reduction of swirl intensity by wall friction is recovered by the acceleration of angular velocity in the conical swirl chamber. For this class of de-oiling hydrocyclone (D = 26 mm with 6° cone angle) the rise of swirl intensity due to the acceleration of angular momentum is higher than the frictional loss, which thereby increases the tangential velocity at the end of swirl chamber. The rate of acceleration in the tangential velocity increases with an increase in the Reynolds number. As consequence, the tangential velocity at the bottom of the swirl chamber (z/D = 8.37) for the Re_F = 28,600, which was less than that of the Re_F = 23,835 and 17160, is almost the same as the Re_F = 23,835 and greater than the Re_F = 17,160 (Fig. 4.1d).

The normalized axial velocity profiles at different axial location are shown in Fig. 4.3. The negative value is the forward (downward) flow and the positive value represents the reverse (upward). A considerable difference in the axial velocity for the different Reynolds numbers appears near the center of the hydrocyclone. In the outer flow field (r/R > 0.4) the axial velocity for all the Reynolds numbers are almost the same. Figures 4.3a and 4.3b show that the magnitude of reverse flow velocity is the smallest at the lowest Reynolds number (Re_F = 11,440). The axial velocity at the center increases with an increase in the Reynolds number up to the Re_F = 23,835 (Fig. 4.3a and 4.3b). However, the highest Reynolds number (Re_F = 28,600) provides a lower reverse flow velocity when compared with the Re_F = 23,835 and Re_F = 17,160. Due to a higher acceleration of swirling motion in the reducing conical swirl chamber, the Re_F = 28,600
provides a less reverse flow, which thereby decreases the reverse flow velocity in the inlet chamber (Fig. 4.3a) and in the upper half of the conical swirl chamber (Fig. 4.3b). The values of axial velocity near the center decrease in the downstream and eventually become negative (Fig. 4.4). A flow split occurs in the middle of the swirl chamber; location of the flow split varies with the feed Reynolds number. For all the Reynolds numbers, there is no reverse flow appears in the hydrocyclone at z/D > 5 (Figs. 4.3c and 4.3d). In the middle of the swirl chamber (Fig. 4.3c), the ReF = 17,160 has much less downward velocity than other Reynolds numbers. As a consequence, the ReF = 17,160 should provide a longer reverse flow core, when compare with other Reynolds numbers. A higher reverse flow velocity in the core has a greater potential to carry the migrated droplets to the overflow orifice. At the inlet chamber (Fig. 4.3a) and the top (Fig. 4.3b) of swirl chamber there is a bump in the axial profile at r/R ≈ 0.5. This indicates a recirculation in the flow field between the core and outer flow (see Fig. 4.5 for more discussion).
Figure 4.2: Contours of normalized tangential velocity for different feed Reynolds numbers.

\[
\frac{\langle U_0 \rangle}{\langle U_F \rangle}
\]

Reₜ = 11,440  Reₜ = 17,160  Reₜ = 23,835  Reₜ = 28,600  Reₜ = 57,200
Figure 4.3: Normalized axial velocity profiles for different Reynolds number (— Re_F = 28,600; ……… Re_F = 23,835; - - - Re_F = 17,160; - - - - Re_F = 11,440). The negative value represents the downward (forward) velocity and the positive value is the upward (reverse) velocity. The Re_F = 17,160 provides a greater reverse flow velocity than the other Reynolds numbers.
Figure 4.4: Contours of normalized axial velocity for the different feed Reynolds numbers.
Figure 4.5: Plane view of velocity vectors in the inlet chamber for the $Re_F = 23,835$. The arrow head indicates the direction of flow. Only a 2D dimensional vector (radial and axial components) is shown in the figure. Short circuit boundary layer flow and some recirculation appear in the flow field.

Figure 4.5 shows velocity vectors for the $Re_F = 23,835$. Adjacent to the top wall there is a short circuit flow from the inlet to the vortex finder (region R1). This short circuit flow is called an Ekman boundary layer flow. In the Ekman boundary layer the swirling intensity is negligible. An excessive amount of the short circuit flow can choke the overflow. However, protrude of vortex finder inside the inlet chamber (region R2) restricts the short circuit flows. Figure 4.5 also shows that the reverse flow (upward) streams cannot be fully accommodated by the vortex finder. As a consequence, a portion of the reverse flow streams in the outer periphery of the inner vortex moves outward direction and merges with downward flow streams. This phenomenon creates a toroidal
circulation (region R4) between the reverse flow and forward flow (downward). Another recirculation denoted by region R3 is created near the top wall of the inlet chamber. This recirculation is generated due to an obstruction of short circuit flows by the vortex finder and the outward motion of core flow.

Figure 4.6: Two dimensional velocity streamlines for the $Re_F = 23,835$ and three zones divided based on the flow pattern in the upper part of the inlet chamber.

Two dimensional velocity streamlines on a cross-sectional plane of hydrocyclone is shown in Fig. 4.6. Streamlines from the inlets create two short circuit flows: a short circuit boundary layer flow near the side wall that moves from inlet to the underflow, and a short circuit flow from inlet to the vortex finder. Part of the downward streamlines appears moving in the reverse direction from the middle of the swirl chamber. The streamlines in the outer periphery of the reverse flow core merge with the downward flow at the upper part of the inlet chamber and thus generates a recirculation. This recirculation exists up to the length of the reverse flow core. For the investigation of
effect of feed Reynolds number on the short circuit flow and flow recirculation, the flow pattern at the upper part of the inlet chamber is divided into three different zones (Fig. 4.6b). The total height of the three zones equals to the height of inlet channel. Q1, Q2 and Q3 (shown in Fig. 4.6b) are the radial inflow rate to the inner core through the boundary B1, B2 and B3, respectively. The overflow rate (Q₀) equals to the summation of the three radial inflows and the reverse flow from the downstream (Q4).

The effect of feed Reynolds numbers on the local radial inflow rates, as a percentage of the overflow rate, through the boundaries B1, B2 and B3 are presented in Fig.4.7. The positive and negative values in the local radial flow versus feed Reynolds number graph represent, respectively, the inflow and outflow through the boundaries. The radial inflow rate through the boundary B1 for all the Reynolds numbers is the largest among all the three zones. The inflow through the boundary B1 has a negligible tangential velocity component as well as a short residence time. Consequently, this region will not contribute to the separation of the dispersed phase. It is noteworthy that as the feed Reynolds number increases, Q1 first decreases and then increases; as the feed Reynolds number continues to increase, Q1 decreases again. A high radial inflow through boundary B1 can reduce the reverse flow from downstream and choke off the overflow. For a low feed Reynolds number, the tangential component of the velocity field and the aspect ratio of the reverse flow core are small (see Fig. 4.1 and 4.8). As a consequence, the reverse flow in the core is accommodated by the vortex finder and there is no inward or outward flow through the boundary B2. However, with an increase in the feed Reynolds number, the magnitude of tangential velocity component increases and a larger reverse cannot be accommodated by the effluent nozzle. As a result, part of the reverse
flow from the outer periphery of vortex core moves outward through B2. For the Reynolds number between 15,000 and 20,000, relatively small amount of short circuit flow accumulated to the reverse flow through boundary B1. However, the radial outflows through boundary B2 and B3 merge with core in the downstream, which in turn generates a large recirculation. Moreover, the radial outflow through B2 and B3 resists the short circuit flow from the inlet. For the Reynolds number greater than 20,000, the amount of short circuit flow through B1 increases again. However, the radial outflow through boundary B2 immediately reenter to the vortex core through the boundary B3. Even though there is a recirculation between zone 2 and zone 3 but the residence time is very small. Therefore, the higher inflow through B1 and the smaller recirculation between zone 2 and zone 3 can reduce the reverse flow from downstream.

The shape and length of reverse flow core for the different Reynolds number are shown in Fig. 4.8. At low Reynolds number (Re_F = 11,440), the reverse flow core is very short. With an increase in the Reynolds number from 11,440 to 17,160, the length and strength of reverse flow core significantly increases because of a greater swirling intensity and a higher axial pressure gradient (see Fig. 4.9). Moreover, the radial outflows through boundaries B2 and B3 (see Fig.4.7) restrict short circuit flow and support the reverse flow core to be long. A further increase in the Reynolds number (Re_F > 17,160) yields a reduction in the core length even though the swirling intensity is very high. A strong swirling flow for the Re_F > 23,000 increases the radial pressure gradient (See Fig. 4.9), which thereby decreases the core pressure in the downstream of the conical swirl chamber. As a consequence, the region of positive axial pressure gradient shifts toward upstream of the hydrocyclone (see Fig. 4.10 for detail), which thereby decreases the
length of reverse flow core. Moreover, a higher inflow through B1 and a smaller recirculation between zone 2 and zone 3 (Fig. 4.7) reduces the reverse flow and forces the reverse flow core to be short.

Figure 4.7: Local radial flows through boundary B1, B2 and B3 as shown in Fig. 4.4. Positive magnitude is the radial inflow and the negative magnitude is the radial outflow.
Figure 4.8: Effect of the Reynolds number on the contours of reverse (upward) flow velocity. Only the upward axial velocity is mapped and the downward flow field is discarded from the contour map. The length of reverse flow core first increases and then decrease with an increase in the Reynolds number.

4.4. Computed pressure fields

Contours of normalized static pressure for the different feed Reynolds number is shown in Fig.4.9. Normalized static pressure is high near the wall and the minimum at the center, which develop a radial pressure gradient. In the downstream, the normalized static pressure near the wall decreases because of the acceleration of swirling motion by tapered wall. Figure 4.10 shows normalized static pressure profiles along the radial distance at different axial locations of the hydrocyclone. Near the top wall of hydrocyclone (z/D = 0.2), the radial pressure gradient increases with an increase in the Reynolds number from 11,440 to 17,160 and it is almost the same for the Re\(_F\) = 23,835
and 17,160. The radial pressure gradient decreases again for a higher Reynolds number ($Re_F = 28,600$). At the $Re_F = 11,440$, the smaller swirling motion in the inlet chamber (Fig. 4.1a) yields a lower radial pressure gradient. When the Reynolds number exceeds a certain limit (greater than 23,835) the large volumetric flow has to be accommodated in the inlet chamber, which thereby declines the radial pressure gradient. In the downstream, an acceleration of swirling motion as well as higher axial velocity component (shown in Fig. 4.1 and 4.3) decreases the magnitude of static pressure near the wall. However, for all the Reynolds numbers, the magnitude of static pressure near the center of hydrocyclone at the $z/D = 1$ is higher than that at the $z/D = 0.2$, which yields a positive axial pressure gradient at the core. The core pressure at the middle of the conical chamber (Fig. 4.10c) for the $Re_F = 28,600$ and $Re_F = 11,440$ is less than that at the top of the conical chamber (Fig. 4.10b). Because of the reduction of the core pressure, there is no reverse flow from the middle to the overflow orifice (see Fig. 4.8). On the other hand for the $Re_F = 23,835$ and $Re_F = 17,160$, the core pressure at the $z/D = 4.69$ is slightly higher than that at the $z/D = 1$. This positive axial pressure gradient up to the middle of the conical chamber indicates a longer reverse flow core, which is also observed in the Fig. 4.8. In the downstream ($z/D > 4.69$), the core pressure again decreases for all the Reynolds numbers. As a consequence, there is no reverse flow downstream of the middle of conical chamber.

Figure 4.11 shows the effects of Reynolds number on the static pressure at the core along the longitudinal axis of hydrocyclone. With a distance from the top of hydrocyclone, the core pressure first increases, then decreases and becomes minimum at the bottom of the conical swirl chamber. At the entrance of the tail pipe the core pressure
steeply increases again. The frictional loss on the wall decreases the swirling motion. However, the conical chamber accelerates the swirling motion. Near the top of the conical swirl chamber the frictional loss is dominating over the acceleration of fluid velocity, which thereby decreases the radial pressure gradient (Fig. 4.10). As a consequence, the core pressure increases downstream of the top of hydrocyclone. At the location of peak core pressure (Fig. 4.11), there is a balance in the velocity between the loss due to friction and gain by the flow acceleration. Downstream of the location of peak core pressure, an excessive acceleration of swirling motion increases the radial pressure gradient (Fig. 4.10) which thereby decreases the core pressure. The location of peak core pressure appears depending on the feed Reynolds number. For the Re_F = 11,440, the location of core pressure is close to the inlet chamber. The location of peak core pressure shifts downstream with an increase in the feed Reynolds number from 11,440 to 17,160. Further increase in the Reynolds number shifts the location of peak core pressure in the opposite direction (toward the overflow). A reverse flow at the core occurs due to a positive axial pressure gradient at the upstream of the location of peak core pressure. The downward shift of the location of the peak core pressure, in the case of Re_F = 23,835 and Re_F = 17,160, increases the length of the reverse flow core, which was also observed in Fig. 4.8. Downstream of the location of peak value, the negative axial pressure gradient moves the core flow toward the underflow orifice.
Figure 4.9: Contours of normalized static pressure for the different feed Reynolds numbers.
Figure 4.10: Pressure distributions for the different Reynolds number (—— $Re_F = 28,600$; 
- - - - $Re_F = 23,835$; - - - $Re_F = 17,160$; - - - - $Re_F = 11,440$). A positive axial pressure gradient is observed in the upstream and a negative axial pressure gradient in the downstream of the conical chamber.
Figure 4.11: Distribution of centerline (core) pressure for different Reynolds numbers (--- $Re_F = 28,600$; ····· $Re_F = 23,835$; − − $Re_F = 17,160$; - - - - $Re_F = 11,440$). $z/D = 0$ is at the top and $z/D = 18$ is at the bottom of the hydrocyclone. The location of peak core pressure shifts toward the underflow for the intermediate Reynolds numbers. The low (negative) core pressure at the top and at the end of swirl chamber occurs due to choking of axial flow by, respectively, the vortex finder and the entrance of tail pipe.
4.5. Criteria for the reverse core breakdown

Two criteria are investigated for the understanding of the reverse flow core breakdown: i) the normalized distance of the location peak core pressure from the vortex finder \((\ell/D at (dP/dz)_{r=0} = 0)\), ii) the normalized distance of the zero axial gradient of radial pressure gradient from the vortex finder \((\ell/D at (d(dP/dr)/dz)_{r=0} = 0)\). The normalized length of reverse flow core \((\ell_c/D)\) and the normalized distance of the location of zero axial gradients \((\ell/D at (dP/dz)_{r=0} = 0\) and \(\ell/D at (d(dP/dr)/dz)_{r=0} = 0)\) for the different Re are shown in Fig. 9. The length of reverse flow core \((\ell_c/D)\) is longer than the \(\ell/D at (dP/dz)_{r=0} = 0\). The \((dP/dz)_{r=0} = 0\) occurs at a location where the peak core pressure appears (see Fig. 4.11). Therefore, the location of peak core pressure is not a good indication of reverse flow core breakdown. In the conical swirl chamber, the peak core pressure occurs where the restriction of forward flow by the reverse flow core is the maximum. Because of the elliptical shape in the reverse flow core, the \((dP/dz)_{r=0} = 0\) occurs slightly upstream of the core breakdown. However, the location of \((dP/dz)_{r=0} = 0\) is a good indicator for a qualitative understanding of the reverse flow core breakdown. The magnitude of \(\ell/D at (d(dP/dr)/dz)_{r=0} = 0\) is closer to the values of \(\ell_c/D\) when compared with \(\ell/D at (dP/dz)_{r=0} = 0\). It reveals that the second criterion (location of \((d(dP/dr)/dz)_{r=0} = 0)\) is a better indicator for both the qualitative and quantitative understanding of the reverse flow core breakdown.
Figure 4.12: Comparison of normalized length of reverse flow core ($\ell_c/D$) with the zero axial gradients of core pressure ($\left(\frac{dP}{dz}\right)_{r=0} = 0$) and radial pressure gradient ($\left(\frac{d}{d} \frac{dP}{dr}\right)/dz)_{r=0} = 0$) for the understanding of the reverse flow core breakdown. The $\left(\frac{d}{d} \frac{dP}{dr}\right)/dz)_{r=0} = 0$ is a better indicator for both qualitative and quantitative understanding of the reverse flow core breakdown.

The contours of normalized radial pressure gradient $(dp/dr)D/(\rho U_F^2)$ for the different values of $Re_F$ are shown in Fig. 10. The radial pressure gradient is high near the top; it decreases and then increases again with an increase in the distance from the top. The lower radial pressure gradient at the middle of the swirl chamber generates a higher core pressure which thereby drives the core flow in the reverse direction. The location of the minimum radial pressure gradient at the core varies with the $Re_F$. With increasing the
Re_F, the location of minimum radial pressure gradient \(((d(dP/dr)/dz)_{r=0} = 0\) first moves downstream and then shifts toward the overflow. This phenomenon is similar to the length of reverse flow core shown in Fig. 4.8.

Figure 4.13: Normalized radial pressure gradient for the different Re_F. With an increase in the Re_F, the location of reverse flow core breakdown (dotted line) moves downstream and then shifts toward overflow.
4.6. Separation performance

The separation performance is evaluated according to the process described in Sections 2.5.1 and 2.5.3. Effects of the Reynolds numbers on the reduced grade efficiency for a range of droplet sizes are shown in Fig. 4.14. The grade separation efficiency increases with an increase in the droplet size. A bigger size droplet generates a larger difference in the centrifugal force between the continuous and dispersed phase which thereby enhances the grade separation efficiency. However for produced water application, the hydrocyclone appears ineffective for a droplet size less than 10 microns. The reduced grade efficiency ($G_{OR}$) rises with an increase in the Reynolds number from 11,440 to 23,835. After that, the $G_{OR}$ decreases with a continuous increase in the Reynolds number. For the higher Reynolds number ($Re_F > 23,835$) the grade efficiency decreases significantly because of a reduction in the length of reverse flow core (Fig. 4.8). Though the swirling intensity for the $Re_F = 23,835$ is less than that of $Re_F = 28,600$, the longer length of reverse flow core provides a major contributing to the enhancement of the grade separation efficiency. Between the $Re_F$ of 17,160 and 23,835, though the length of reverse flow core for the $Re_F = 23,835$ is slightly shorter than that of the $Re_F = 17,160$ (see Fig. 4.8), the $Re_F = 23,835$ enhances the centrifugal acceleration, which in turn increases the migration rate of dispersed droplets as well as the separation efficiency. As a consequence, the $Re_F$ of 17,160 and 23,835 provides almost the similar separation efficiency. Several previous studies (Saidi et al., 2012; Meldrum, 1988; Bennett and Williams, 2004) mentioned that a high shearing of droplets in the case of a high feed Reynolds number sharply decreases the separation efficiency. In this study, droplet breakup and coalescence are not considered for the calculation of dispersed phase...
mechanics and the grade separation efficiency. However, Fig. 4.14 provides the evidence that the decrease in the separation efficiency for a higher Reynolds number is a hydrodynamic effect. More specifically, the axial and radial pressure distributions, the length of reverse flow core, and the swirl intensity characterizes the separation efficiency in a de-oiling hydrocyclone.

Figure 4.14: Reduced grade efficiency versus droplet size graphs for the different Reynolds numbers (--- Re_F = 28,600; ····· Re_F = 23,835; - - Re_F = 17,160; - - - Re_F = 11,440). Droplets are assumed to be spherical. No coalescence and breakup effect is included. Insufficient swirl intensity in the case of Re_F = 11,440 and a shorter reverse flow core in the case of Re_F = 28,600 yields low grade separation efficiency.
Figure 4.15 shows the influence of feed Reynolds number (Re_F) on the aspect ratio of reverse flow core, \( \ell_c/D \) and the cut-size, \( d_{D50} \) of the separator. The cut size is defined as the diameter of a dispersed droplet for which the reduced grade efficiency is 50%. The higher the cut size, the lower the separation efficiency becomes. The maximum aspect ratio of the reverse flow core is about 4.5 times the diameter of the swirl chamber. This length decreases with an increase in the Re_F. The \( d_{D50} \) of the hydrocyclone decreases with an increase in the Re_F up to about 23,835. The magnitude of \( d_{D50} \) between the Re_F of 17160 and 23835 is almost the same. A further increase in the Re_F raises the value of \( d_{D50} \) again. Therefore, the separator provides a finite turndown ratio. The \( d_{D50} \) is inversely related to the length of reverse flow core (\( \ell_c/D \)), i.e. the longer reverse flow core yields less cut size which thereby increases the separation efficiency. Therefore, the cut size of the hydrocyclone separator depends on the length of reverse flow core. The effect of Re_F on the overall reduced separation efficiency (\( \eta_{OR} \)) is shown in Fig.4.16a. For the given droplet size distribution (Fig. 4.16b), the overall reduced separation efficiency, \( \eta_{OR} \) is calculates using Eqs. (2.5 and 2.8). At low Re_F, the separation efficiency is small due to an insufficient swirling motion. For the Re_F of between 17160 and 23835, the separation efficiency is almost the same. A further increase in the Re_F results a decreases in the separation efficiency. This phenomena in the separation efficiency is similar to the well know performance characteristics of a de-oiling hydrocyclone. The cut-size, which depends on the length of reverse flow core (see Fig. 4.15), and the overall separation efficiency provides similar and opposite trend, which was expected. Therefore, the finite turndown ratio in a hydrocyclone is a hydrodynamic effect and it depends on the flow characteristics and length of reverse flow core.
Figure 4.15: A comparison between the normalized length of reverse flow core ($\ell_c/D$) and the normalized cut-size ($d_{D50}$) for different Reynolds numbers. The cut size represents the size of droplets for which the reduced grade separation efficiency is 50%.
Figure 4.16: Effect of the Re_F on the overall reduced separation efficiency (Fig. a) of the flooded hydrocyclone for the given droplet size distribution (Fig. b). The overall separation efficiency is almost the same for the Re_F between 17,000 and 24,000; it decreases for both the lower and higher Re_F. This phenomenon indicates a finite turndown ratio which is dependent on hydrodynamic characteristics.

4.7. Macroscopic study of shearing effect on dispersed droplets

A probability of droplet breakup in the flow field of a de-oiling hydrocyclone for the given operating conditions is also discussed based on the analysis of Capillary number, which is illustrated in Fig. 4.17. The Capillary number is defined as Ca = \( \mu_D E_d / \sigma \) where \( E = \sqrt{(2E: E)} \) is the magnitude of the rate of strain tensor \( (E = (\nabla U + \nabla U^T)/2) \) and \( \sigma \) represents the interfacial tension. The contour of the Capillary numbers presented in Fig. 4.17 are calculated for \( \mu_D / \mu_C = 3.3 \), \( d_D = 130 \mu m \) and \( \sigma = 0.027 \text{ N/m} \). Figure 4.17 shows that the Capillary number is higher in the vortex finder and at the
bottom of the conical chamber. The Capillary number is negligible in the outer vortex region where the separation of oil from water occurs. Though near the core of the hydrocyclone has slightly higher Capillary number it does not affect the separation efficiency since the oil droplets already separated from water and migrated to the core. However, for all the cases the Capillary number is below the critical Capillary number for a dispersed droplet to be broken up. For $\mu_D/\mu_C = 3.3$, breakup of a dispersed droplet occurs when $Ca > 1$ (For a highly confined flow field, $Ca > 0.5$ for a simple shear flow (Vananroye et al., 2006) and $Ca > 0.25$ for an elongational flow (Janseen and Meijer, 1993)). Figure 4.17 show that the Capillary number in the flow field is less than 0.1, which is not sufficiently large for breaking up an oil droplet by both the shearing and elongation in the flow field. Besides, it is worth mentioning that the fixtures in the upstream of the feed such as pump, valves, joints etcetera can cause a breakup of dispersed droplets, which are not characterized by the separation performance of a de-oiling hydrocyclone.

Figure 4.18 show the contours of Weber number for different Reynolds numbers. The Weber number is defined as $We = \rho_C d_D^{5/3} \varepsilon^{2/3} / \sigma$ where $\varepsilon$ is the turbulent energy dissipation. The Weber number is calculated for $\mu_D/\mu_C = 3.3$, $d_D = 130 \mu m$ and $\sigma = 0.027$ N/m. The Weber number is another parameter which is used to quantify the condition of droplet shearing. A liquid droplet is assumed to have the potential to be sheared when the Weber number reached to a critical limit, i.e. $We > We_{cr}$. The magnitude of critical Weber number depends on the density of continuous and dispersed phase, size of droplets, and viscosity of dispersed phase. Duan et al. (2003) developed an empirical model based on moving particle semi implicit method to investigate the effect of density
ratio on the \( \text{We}_{cr} \), which is given as \( \text{We}_{cr} = 64.36 \mu_r^{-0.82} \) where \( \mu_r = \mu_D / \mu_C \). For \( \mu_r = 3.3 \), \( d_D = 130 \ \mu\text{m} \) and \( \sigma = 0.027 \ \text{N/m} \) the \( \text{We}_{cr} = 73 \). For this flow condition and fluid properties, the dispersed droplets will break up when the Weber number in the flow field is greater than 73. Figure 4.18 shows that the Weber number for the \( \text{Re}_F < 28,600 \) is much less than the critical Weber number. For the highest Reynolds number (\( \text{Re}_F = 57,200 \)), the Weber number in the upper half of the swirl chamber is much less than the \( \text{We}_{cr} \). However, the Weber number is greater than the \( \text{We}_{cr} \) in the overflow tube and near the end of the swirl chamber, which thereby can yield shearing of droplet. Since there is no reverse flow near the end of swirl chamber, the probability of droplet breakup near the end of swirl chamber does not affect the separation efficiency.
Re_F = 11,440  Re_F = 17,160  Re_F = 23,835  Re_F = 28,600  Re_F = 57,200

Figure 4.17: Contours Capillary number for d_D = 130 μm and μ_D/μ_C = 3.3. A higher Capillary number is observed in the vortex finder and at the bottom of the conical chamber. However, the observed Ca is much less than a critical Ca_cr for a droplet to be sheared where Ca_cr > 1 (For a highly confined flow field, Ca_cr > 0.5 for a simple shear flow and Ca > 0.25 for an elongational flow) [Ca = μ_D Ed_D/σ; E = \sqrt{(2E:E)}, and E = (\nabla U + \nabla U^T)/2]
Re_F = 11,440    Re_F = 17,160    Re_F = 23,835    Re_F = 28,600    Re_F = 57,200

Figure 4.18: Contours of Weber number for d_D = 130 μm and μ_D/μ_C = 3.3. A Weber number of greater than a critical Weber number, We_cr indicates a probability of shearing of dispersed droplets. [We = ρ_C d_D^{5/3} ε^{2/3}/σ]
4.8. Multiphase fluid mechanics

The multiphase fluid mechanics in the hydrocyclone is studied by using a mixture model with algebraic slip model (APPENDIX A-3). The mixture of oil and water contains mono-sized dispersed droplets. The multiphase simulation is performed for the $Re_F = 23,835$, feed volume fraction of 0.01 and feed droplet size of 30 microns. The normalized volume fraction at the top ($z/D = 1$) and the bottom ($z/D = 8.37$) of the conical swirl chamber is shown in Fig. 4.19. At $z/D =1$, the volume fraction decreases in the outer flow field ($0.4 <r/(D/2) <0.9$) by about 15%. This reduced volume of dispersed droplets is collected by the overflow orifice. At $z/D = 8.37$, the normalized volume fraction decreases significantly; near the core the volume fraction is decreased by about 60%. Since there is no reverse flow core at $z/D = 8.37$ (see Fig. 4.3 and 4.4), these separated volume of dispersed droplets cannot be collected by the overflow orifice. The separation efficiency calculated based on the multiphase simulation for the 30 microns droplet is about 25%, which is close to the value calculated using Lagrangian tracking (Dispersed phase model) method for the same size of dispersed droplet (see Fig. 4.14).
Figure 4.19: Profiles of the normalized volume fraction at the top \((z/D = 1)\) and bottom \((z/D = 8.3)\) of the swirl chamber for \(Re_F = 23,835\), \(\phi_F = 0.01\) and feed droplet size = 30 microns.
CHAPTER 5

A PARAMETRIC STUDY OF HYDROCYCLONE OPERATING AND GEOMETRY CONDITIONS

5.1. Introduction

The separation performance of a hydrocyclone is sensitive to internal flow structures, which are influenced by pressure drop ratio ($\Delta P^*$), design of vortex finder and overflow, and the angle of conical swirl chamber. The parametric studies are conducted on a conventional hydrocyclone (see Fig. 3.1 and Table 3.2) to study the effect of these functions. Typically, the overflow ratio ($R_O$) is controlled during the operation of hydrocyclone by adjusting the $\Delta P^*$. Very high and low $\Delta P^*$ may deteriorate the performance of a hydrocyclone. The effects of $\Delta P^*$ on the velocity and pressure profiles, velocity streamlines, velocity vectors and separation efficiency are presented in Section 5.2. The size and shape of vortex finder and overflow tube also influence the internal hydrodynamics, which can lead to the variations of separation performance. The effects of the sizes and shapes of the vortex finder and overflow tube on the internal flow structure are addressed in Section 5.3. The angle of conical swirl chamber is a major geometric parameter; the variation of angle may significantly change the internal hydrodynamics. The effects of angle of conical swirl chamber on the internal flow structure, size and shape of the reverse flow core, and separation performance are addressed in Section 5.4.
5.2. Effect of the pressure drop ratio

The operating conditions for the different pressure drop ratios ($\Delta P^*$) are presented in Table 5.1. The Ro and $\Delta P^*$ are calculated using the Eq. (2.9) and Eq. (2.10), respectively. The pressure at the overflow and underflow outlets is specified as the boundary conditions. The average pressure at the overflow orifice is kept constant for all the cases, which is equal to the atmospheric pressure. A zero pressure gradient boundary condition (plug flow) is applied at the inlets. The effects of $\Delta P^*$ on the Ro, overflow Euler number ($E_{O}$) and underflow Euler numbers ($E_{U}$) are presented in Fig. 5.1. The $E_{O}$ and $E_{U}$ are calculated using the Eq. (2.11). The overflow rate gradually increases with an increase in the $\Delta P^*$. With an increase in the $\Delta P^*$, the $E_{O}$ increases and the $E_{U}$ decreases which supports an increase in the Ro.

<table>
<thead>
<tr>
<th>$Re_F$</th>
<th>$P_F$ (Pa)</th>
<th>$P_O$ (Pa)</th>
<th>$P_U$ (Pa)</th>
<th>$\Delta P^*$</th>
<th>Ro</th>
</tr>
</thead>
<tbody>
<tr>
<td>23,835</td>
<td>276,000</td>
<td>0</td>
<td>0</td>
<td>1.0</td>
<td>0.6</td>
</tr>
<tr>
<td>23,835</td>
<td>384,577</td>
<td>0</td>
<td>120,000</td>
<td>1.45</td>
<td>0.1</td>
</tr>
<tr>
<td>23,835</td>
<td>499,200</td>
<td>0</td>
<td>250,000</td>
<td>2.0</td>
<td>0.13</td>
</tr>
</tbody>
</table>
Figure 5.1: Effect of pressure drop ratio on the overflow rate, overflow and underflow Euler numbers for the $Re_F = 23,835$.

The normalized tangential velocity profiles for the different pressure drop ratios are presented in Fig. 5.2. The variation of $\Delta P^*$ has no significant effect on the normalized tangential velocity except at the region of peak magnitude. The $\Delta P^*$ of 1.0 and 1.45 provides almost the same peak tangential velocity; the $\Delta P^*$ of 2.0 yields less magnitude. However, in the outer vortex region ($r/R > 0.2$), there is no considerable variation in the tangential velocity for different $\Delta P^*$. The normalized axial velocity profiles for the different $\Delta P^*$s are presented in Fig. 5.3. The positive and negative magnitudes indicate, respectively, the flow toward overflow and underflow orifices. The variation $\Delta P^*$ has no considerable effect on the downward flow (negative magnitude). The $\Delta P^*$ of 1.45 provides a greater reverse flow velocity than the other two $\Delta P^*$s. However, the radius of reverse flow core (indicated by $r_1$, $r_2$ and $r_3$ in Fig. 5.3) increases with an increase in the
$\Delta P^*$. The rise of $E_{uO}$, due to an increase in the $\Delta P^*$, yields more pressure differential between the inlet and overflow orifice. As a consequence, short circuit flows from inlets to the overflow orifice increase, which thereby increases the diameter of the reverse flow core. Figure 5.4 shows the effect of $\Delta P^*$ on the radial velocity profiles. The negative value in the radial velocity indicates an inward flow (flow toward the center) and the positive magnitude indicates opposite direction. In the inlet chamber, radially inward velocity increases with an increase in the $\Delta P^*$. This condition supports the upsurge in the radius of reverse flow core (Fig. 5.3). The effect of $\Delta P^*$ on the normalized pressure profile is presented in Fig. 5.5. The radial pressure gradient at the inlet chamber decreases with an increase in the $\Delta P^*$. At the center of hydrocyclone, the magnitude of normalized static pressure is proportional to the $\Delta P^*$. It indicates that a higher $\Delta P^*$ provides a larger axial pressure differential between a location at the reverse flow core and the overflow orifice. As a consequence, a higher $\Delta P^*$ provides a larger overflow ratio and a greater radius of reverse flow core. However, a higher reverse flow rate and a smaller radial pressure gradient may deteriorate the overall performance of a hydrocyclone.
Figure 5.2: Normalized tangential velocity profiles. (––ΔP*= 1.0; ⋯⋯ ΔP*= 1.45; − − − ΔP*= 2.0). A higher peak value represents a greater effective swirling intensity.

Figure 5.3: Normalized axial velocity profiles. (––ΔP*= 1.0; ⋯⋯ ΔP*= 1.45; − − − ΔP*= 2.0). The diameter of reverse flow core increases with an increase in the ΔP*.
Figure 5.4: Normalized radial velocity profiles. (—ΔP* = 1.0; ……. ΔP* = 1.45; ——— ΔP* = 2.0). The radial inward velocity increases with an increase in the ΔP*.

Figure 5.5: Normalized pressure profiles. (—ΔP* = 1.0; ……. ΔP* = 1.45; ——— ΔP* = 2.0). The radial pressure gradient increases with a decrease in the ΔP*.
The velocity streamlines and vectors for different $\Delta P^*$s are presented in Fig. 5.6. The blue lines and arrow head represents the streamlines and vectors, respectively. The axial and radial velocity components are considered for calculating the streamlines and vectors. Tangential velocity component is projected on the plan. Figure 5.6 shows that a small portion of fluid flows from inlet toward the center and creates a short circuit. For a $\Delta P^* = 1.0$, the short circuit flows appears at the upper portion of the inlet chamber. The flow circulations at the middle of the inlet chamber ($z/D \approx 0.5$) breaks up the short circuit flows, which thereby reduces the radial inflow in the inlet chamber. As a consequence, for the case of $\Delta P^* = 1.0$, the major portion of reverse flow is developed from the downstream and yields a smaller radius of reverse flow core. The circulation strength at the middle of the inlet chamber decreases with an increase in the $\Delta P^*$, which thereby increases the short circuit flows from inlets to the center and the diameter of reverse flow core.

Figure 5.7 show the percentage of radial flow through the boundary B1, B2 and B3 shown in Fig. 4.6. Q1, Q2 and Q3 represent the radial flows, respectively, through the boundary B1, B2 and B3. The positive and negative magnitude indicates, respectively, the radial inflow (toward the center) and outflow (from the core). The radial inflow through boundary B1 first increases and then slightly decreases when the $\Delta P^*$ increases. For the PDR = 2.0, the radius of reverse flow core is larger than the other $\Delta P^*$s, which thereby provides a resistance to flow through the boundary B1. The radial flow through the boundary B2 for the $\Delta P^*$ of 1.0 is almost zero. It deceases and becomes negative for the $\Delta P^*$ of 1.45. It indicates that the $\Delta P^*$ of 1.45 provides a radial outflow from the reverse flow core through the boundary B2. The reason is that all the reverse flow cannot
be accommodated by the vortex finder. The $Q_2/Q_0$ increases again for the $\Delta P^*$ of 2.0. Even though the diameter of the reverse flow core for the $\Delta P^*$ of 2.0 is much larger than other $\Delta P^*$s, the higher Euo creates a short circuit from inlets to the reverse flow core through the boundary B2. At the boundary B3, the $\Delta P^*$ of 1.0 provides a radial outflow from the reverse flow core because of less Euo and recirculation in the downstream. The amount of radial outflow through B3 decreases for the $\Delta P^*$ of 1.45 because of less circulation and a higher Euo. For $\Delta P^* = 2.0$, the high Euo and negligible recirculation provides a short circuit flows from inlets to the reverse flow core.

The effect of $\Delta P^*$ on the grade separation efficiency is presented in Fig. 5.8. There is no considerable difference in the grade separation efficiency between the $\Delta P^*$ of 1.0 and 1.45. However, the $\Delta P^*$ of 2.0 provides a significantly reduced separation efficiency. For the $\Delta P^*$ of 1.0 and 1.45, the greater peak tangential velocity (Fig. 5.2), less radial velocity (Fig. 5.4), higher pressure gradient (Fig. 5.5) yields a higher migration rate of dispersed droplets than the $\Delta P^*$ of 2.0. Moreover, the higher strength of circulation (Fig. 5.6) for the $\Delta P^*$ of 1.0 and 1.45 provides a longer flow residence time in the swirl chamber. As a consequence, the $\Delta P^*$ of 1.0 and 1.45 yields greater separation efficiency when we compare with the $\Delta P^*$ of 2.0. Furthermore, the $\Delta P^*$ of 2.0 provides very high Ro, which thereby reduces the $G_{OR}$ (see Eq. (2.7)).
Figure 5.6: 2D streamlines and velocity vectors on a plane at the center of the hydrocyclone for the different $\Delta P^*$s. The flow circulation decreases with an increase in the $\Delta P^*$. 
Figure 5.7: Local radial flows through boundary B1, B2 and B3 as shown in Fig. 4.6. Positive magnitude is the radial inflow and the negative magnitude is the radial outflow.

Figure 5.8: Grade separation efficiency versus droplet size graphs for the different pressure drop ratio, $\Delta P^*$. The $\Delta P^*$ of 2.0 significantly reduces the separation efficiency.
5.3. Effects of the vortex finder and overflow orifice design

Four cases presented in Fig. 5.9 are considered for the investigation of effects of vortex finder design and size of overflow tube on the hydrodynamics and separation performance of a conventional hydrocyclone (see Fig. 3.1). The case 1 has a vortex finder of length equals to 0.1D. The vortex finder in the Case 2 has a flow deviator. The purpose of the use of a deviator is to detour short circuit boundary layer flow. The case 3 and case 4 has no vortex finder. The vortex finder is removed to quantify the effect of vortex finder on overflow choke off. The diameter of the overflow tube for the case 4 is smaller than the other three cases. Dimensions of all the other geometric groups of the hydrocyclone are same as the dimensions presented in Table 3.2. The operating conditions for the different cases are presented in Table 5.2. The Re, F, and overflow and underflow pressures are specified as boundary conditions. The Ro, ΔP*, Euo and Euu are calculated using the Eq. (2.9 - 11). For cases 1, 2 and 3, the variation of design has no effect on the macroscopic properties i.e. Euo, Euu and Ro. A reduction in the Do (case 4) significantly decreases the Ro and slightly increases both the Euo and Euu.

<table>
<thead>
<tr>
<th>Case</th>
<th>ReF</th>
<th>P_F (Pa)</th>
<th>P_O (Pa)</th>
<th>P_U (Pa)</th>
<th>ΔP*</th>
<th>Ro</th>
<th>Euo</th>
<th>Euu</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>23,835</td>
<td>384,577</td>
<td>0</td>
<td>120,000</td>
<td>1.45</td>
<td>0.101</td>
<td>22.15</td>
<td>15.24</td>
</tr>
<tr>
<td>2</td>
<td>23,835</td>
<td>374,442</td>
<td>0</td>
<td>120,000</td>
<td>1.47</td>
<td>0.103</td>
<td>21.56</td>
<td>14.65</td>
</tr>
<tr>
<td>3</td>
<td>23,835</td>
<td>377,975</td>
<td>0</td>
<td>120,000</td>
<td>1.47</td>
<td>0.106</td>
<td>21.77</td>
<td>14.86</td>
</tr>
<tr>
<td>4</td>
<td>23,835</td>
<td>403,058</td>
<td>0</td>
<td>120,000</td>
<td>1.42</td>
<td>0.031</td>
<td>23.21</td>
<td>16.30</td>
</tr>
</tbody>
</table>
Flow streamlines and velocity vectors for different cases are presented in Fig. 5.10. For all the cases there are some short circuit flow streamlines from inlets to the reverse flow core. The Case-1 provides flow circulations at the middle of the inlet chamber (z/D = 0.5). The addition of deviator (Case 2) with the vortex finder reduces the flow circulation. Moreover, the path of flow streams in the case-2 is shorter than that of the case-1. The removal of vortex finder (Case-3) significantly decreases the flow circulation as well as the path of flow streams. However, the reduction in the diameter of
the overflow tube (Case-4) significantly increases flow circulation and provides a longer path of the flow streams. It can be infer that the vortex finder with a smaller diameter of overflow orifice should provide a longer flow path as well as a higher flow circulation, which thereby yields a longer flow residence time.

Local radial flow as a percentage of total overflow rate (Qo) for the different cases (Table 5.2) are presented in Fig. 5.11. Q1, Q2 and Q3 represent the radial flow, respectively, through the boundary B1, B2 and B3 as shown in Fig 4.6. QT is the sum of radial flows through the three boundaries. The positive and negative magnitude indicates, respectively, the radial inflow and outflow through the boundaries. The Case-1 provides a large amount of radial inflow through boundary B1. However, the presence of vortex finder in Case-1 yields radial outflow through B2 and B3, which thereby restricts the short circuit flow from inlets and provides a longer path for flow streams. The addition of deviator with the vortex finder (Case-2) significantly reduces the radial inflow through B1. However, Case-2 provides radial inflow through B2 and B3, which thereby supports a large amount of total short circuit flow (QT/Qo). The removal of vortex finder (Case-3) provides a balanced condition between radial inflow and outflow through the boundaries. The reduction in the diameter of overflow tube (Case-4) provides a large amount of radial outflow through the boundaries B1 and B2 and restricts the short circuit flows. However, there is a large amount of radial inflow thorough B3. This short circuit flow can be reduced by adding a vortex finder which is observed for Case-1. In comparison with other cases, only the Case-4 provides a negative magnitude in the QT. It indicates that the Case-4 offers a better restriction in the overall short circuit flows.
Figure 5.10: 2D streamlines and velocity vectors on a plane at the center of the hydrocyclone for the different design of vortex finder and overflow tube shown in Fig. 5.9. The case 4 provides a higher flow circulation and longer path of flow stream.
Figure 5.11: Local radial flows as a percentage of total overflow rates for different cases presented in Table 5.2. Q1, Q2 and Q3 are the radial flow, respectively, through the boundary B1, B2 and B3 as shown in Fig. 4.6. QT is the sum of radial flow through the three boundaries.

The normalized tangential velocity profiles for the different cases are presented in Fig. 5.12. The Case-1 provides slightly greater tangential velocity at the interface between the forced and free vortex. For all the other cases, there is no considerable difference in the normalized tangential velocity. However, the vortex finder design and diameter of overflow tube yields a significant effect on the axial velocity near the center (Fig. 5.13). The removal of vortex finder (Case-3) provides the largest radius of the reverse flow core ($r_{\text{max}}$). Because of the large diameter of the reverse flow core, the
reverse flow cannot be accommodated by the overflow orifice, which restricts the short circuit flow. The reduction in the diameter of the overflow orifice (Case-4) significantly decreases the radius of the reverse flow core ($r_{\text{min}}$). In the outer flow field there is no considerable difference in the axial velocity among all the cases. The normalized radial velocity profiles for the different cases are presented in Fig. 5.14. The negative and positive magnitude indicates the flow, respectively, toward the center and toward the wall. Due to a tapered geometry in the downstream, all the cases exhibit a radial inward flow. At the interface of forward and reverse flow, the Case-2 and Case-3 provides a higher radial inflow than other cases. However, the Case-4 yields the least amount of radial inflow at the interface. It indicates that a smaller diameter of overflow tube provides less short circuit flows, which thereby increases the flow residence time. The normalized pressure profiles for the different cases are shown in Fig. 5.15. The variation of design in the vortex finder and the diameter of overflow tube have no considerable effect on the radial pressure gradient. However, the Case-1 provides slightly larger pressure gradient inside the reverse flow core.
Figure 5.12: Normalized tangential velocity profiles for the different cases presented in Table 5.2. Vortex finder design does not affect the swirling motion in the inlet chamber.

Figure 5.13: Normalized axial velocity profiles for the different cases presented in Table 5.2. The diameter of reverse flow core directly related to the diameter of overflow tube.
Figure 5.14: Normalized axial velocity profiles for the different cases presented in Table 5.2. Case 4 provides least amount of short circuit radial inflow in the inlet chamber.

Figure 5.15: Normalized pressure profiles for the different cases presented in Table 5.2.
5.4. Effects of angle of the conical swirl chamber

Internal flow structures, pattern of reverse flow core, separation efficiency, and cut-size are evaluated as a function of the angle of conical swirl chamber, \( \theta \). Four angles for the conical swirl chamber are considered, which are 2, 4, 6 and 10 degrees. The lengths of the swirl chamber (Ls) are calculated based on the four different cone angles. All the other geometric groups and dimensions are same as the values presented in Table 3.2. The operating conditions for the different angle of conical swirl chamber are presented in Table 5.3. The \( R_O \) and \( \Delta P^* \) are calculated using the Eq. (2.9) and Eq. (2.10), respectively. The effect of cone angle on the overflow and underflow Euler number is presented in Fig. 5.16. The \( E_{uo} \) and \( E_{uu} \) are calculated using the Eq. (2.11). The both \( E_{uo} \) and \( E_{uu} \) are gradually increases with an increase in the cone angle, which indicates that a higher cone angle yields more energy loss.

<table>
<thead>
<tr>
<th>Cone angle, ( \theta )</th>
<th>Re_F</th>
<th>P_F (Pa)</th>
<th>P_O (Pa)</th>
<th>P_U (Pa)</th>
<th>( \Delta P^* )</th>
<th>R_O</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>23,835</td>
<td>286,767</td>
<td>0</td>
<td>120,000</td>
<td>1.45</td>
<td>0.111</td>
</tr>
<tr>
<td>4</td>
<td>23,835</td>
<td>341,285</td>
<td>0</td>
<td>120,000</td>
<td>1.54</td>
<td>0.105</td>
</tr>
<tr>
<td>6</td>
<td>23,835</td>
<td>384,577</td>
<td>0</td>
<td>120,000</td>
<td>1.45</td>
<td>0.101</td>
</tr>
<tr>
<td>10</td>
<td>23,835</td>
<td>433,409</td>
<td>0</td>
<td>120,000</td>
<td>1.38</td>
<td>0.102</td>
</tr>
</tbody>
</table>
Figure 5.16: Effect of cone angle, $\theta$ on the overflow and underflow Euler numbers.

The normalized tangential velocity profiles for the different cone angles are presented in Fig. 5.17. The normalized tangential velocity significantly increases with an increase in the cone angle. A higher tangential velocity yields a greater centrifugal acceleration ($U_\theta^2/r$) in the flow, which supports the dispersed droplets to be migrated toward the center at a faster rate. An increase in the cone angle reduces the cross sectional area of the swirl chamber. The reduction in the cross sectional area in the downstream accelerates the tangential velocity and enhances the centrifugal acceleration. Moreover, the smaller radius, due to the increase in the cone angle, yields the dispersed droplets to be traveled a shorter path during the migration to the center. On the other hand, the reduction in the length of swirl chamber and the increase in the tangential velocity, due to an increase in the cone angle, significantly reduce the flow residence time, which thereby reduces the overall migration of dispersed droplets.
The normalized axial velocity profiles at $z/D = 1$ for the different angles of conical swirl chamber are presented in Fig. 5.18. The positive and negative magnitudes represent, respectively, the reverse (toward the overflow) and forward (toward the underflow) flows. The normalized axial velocity in the reverse flow core ($r/R < 0.2$) increases with a decrease in the cone angles. A similar trend appears near the wall ($r/R > 0.7$), i.e. the forward flow velocity increases with a decrease in the cone angle. The fast moving flow adjacent to wall region for the case of lower cone angle transport a larger fraction the dispersed droplets toward the underflow, while we compare with the higher cone angle. An interesting trend in the axial velocity profile appears in the intermediate region ($0.2 < r/R < 0.7$). The forward flow velocity decreases with a reduction in the cone angle and; the fluid even flows in the reverse direction for the cone angle of 2 degree. For a lower cone angle the dispersed droplets get a higher residence time to be migrated toward the core due to the slower downward motion or the reverse flow of fluid in the intermediate region ($0.2 < r/R < 0.7$).

The normalized radial velocity for the different cone angles are presented in Fig. 5.19. The negative magnitude indicates a radial flow toward the center. Near the side wall, the radially inward flow velocity decreases with an increase in the cone angle up to 6 degree. A higher cone angle increases the tangential velocity (Fig. 5.17) as well as the centrifugal force on the fluid, which thereby reduces the radially inward flow velocity. The radially inward flow velocity is higher near the wall and decreases toward the center. In the intermediate region ($0.2 < r/R < 0.7$), the increase in the centrifugal acceleration ($U_0^2/r$), due to a decrease in the radius and a rise in the tangential velocity (Fig. 5.17), reduces the radial inward velocity when we compare with a region near the side wall. The
reduction in the inward radial velocity in the intermediate region supports the reduction of downward velocity (Fig. 5.18) in the same region. Near the interface between forward and reverse flows \((r/R \approx 0.2)\), there is no considerable difference in the radial velocity among the cone angles of 2, 4 and 6 degree. For the cone angle of 10 degree, the inward radial velocity decreases as we move toward the center. For the 10 degree cone, the enhancement of radial inflow caused by the high tapered swirl chamber supersedes the reduction of radial inflow caused by the higher centrifugal force. As a consequence, the cone angle of 10 degree provides higher radial inflow near the core.

![Figure 5.17: Effect of cone angle on the normalized tangential velocity at \(z/D = 1\). An increase in the cone angle reduces the cross-sectional area of the swirl chamber, which thereby increases the centrifugal acceleration. A higher centrifugal acceleration supports a greater migration rate of dispersed droplet toward the center.](image)

107
Figure 5.18: Effect of cone angle on the normalized axial velocity at $z/D = 1$.

Figure 5.19: Effect of cone angle on the normalized radial velocity at $z/D = 1$. 

108
The effects of angle of conical swirl chamber on the radial pressure profiles are presented in Fig. 5.20. Due to a swirling motion and centrifugal force acting on the fluid, a higher pressure appears near the wall and a lower pressure at the center. The pressure at the center decreases with an increase in the cone angle, which thereby increases the radial pressure gradient. A higher radial pressure gradient supports a greater rate of droplets migration toward the center. However, there is no considerable difference in the radial pressure profile for $r/R > 0.5$. At a certain location in the conical swirl chamber, the radius of swirl chamber decreases with an increase in the cone angle, which thereby causes an increase in the centrifugal acceleration. As a consequence, the radial pressure gradient increases for a larger cone angle due to a higher centrifugal acceleration.

The effects of the cone angle on the core pressure profiles are presented in Fig. 5.21. The vertical axis represents the normalized pressure at the center of hydrocyclone and the horizontal axis represents the normalized distance along the axis. The core pressure first increases, becomes the maximum and then decreases. The positive axial pressure gradient from the location of maximum pressure toward the overflow creates a reverse flow core. The location of maximum core pressure moves downstream with a decrease in the cone angle. As a consequence, a lower cone angle maintains a positive axial pressure gradient up to a far downstream, which thereby increases the length of reverse flow core (Fig. 5.22). An acceleration of the swirling motion, due to an increase in the cone angle, increases the radial pressure gradient and reduces the core pressure, which in turn reduces the length of reverse flow vortex core. However, length of reverse flow core is an essential parameter for interpreting the dependency of separation
efficiency on the cone angle. A longer of reverse flow core supports the transport of migrated dispersed droplets to the overflow from a far downstream.

Figure 5.20: Effect of cone angle on the normalized pressure at z/D = 1. A higher cone angle enhances the radial pressure gradient, which thereby support a greater migration rate of dispersed droplets toward the center.
Figure 5.21: Effect of the cone angle on the pressure along the center of the hydrocyclone. The positive axial pressure gradient from the location of peak core pressure to the overflow orifice creates reverse flow core. The cone angle of 2 degree provides the location of the peak core pressure far downstream which indicates a longer reverse flow core.
Figure 5.22: Contours of reverse flow core for the different cone angles. The axially upward velocity is mapped on the plane. The axially downward and all the other velocity components are discarded from the contour map. The length of the reverse flow core increases with a decrease in the angle of the conical swirl chamber.
The effects of cone angle on the grade separation efficiency (G_{OR}) are shown in Fig. 2.23. The separation efficiency is calculated based on the process mentioned in section 2.5.1 and 2.5.3. The G_{OR} decreases with an increase in the cone angle. Even though a smaller cone angle decreases the swirl intensity, the longer reverse flow core enhances the separation efficiency by bringing the migrated droplets to the overflow orifice from the far downstream. So, a longer reverse flow core is a key feature for higher separation efficiency. However, a tradeoff between the cone angle, swirling intensity, aspect ratio of reverse flow core (\ell c/D), and the flow residence time is required to achieve an optimum G_{OR}. For all of the cone angles, G_{OR} is almost zero for a droplet size (d_D) less than 10 µm. Small droplets are less sensitive to the swirling motion, which yields a negligible G_{OR}. G_{OR} for the small droplets can be enhanced by increasing the swirl intensity as well as the flow residence time. Figure 5.24 shows a relation between the cut size (d_{D50}) and the normalized length of reverse flow core (\ell c/D). Cut size is a size of dispersed droplet for which the G_{OR} is 50%. The cut-size decreases drastically with an increase in the cone angle. An opposite trend appears for the \ell c/D, i.e. the \ell c/D increases sharply with a decrease the cone angle. So, the d_{D50} is inversely related with the \ell c.
Figure 5.23: Effect of cone angle on the grade separation efficiency.

Figure 5.24: Relation between the normalize length of reverse flow core and cut-size.
CHAPTER 6

TAILORING OF HYDROCYLONE GEOMETRY

6.1. Introduction

The study of internal hydrodynamics conducted on a conventional hydrocyclone, addressed in the previous chapters, reveals that the conventional design is lacking in maintaining a long reverse flow core and for high separation efficiency. Results presented in Chapter 5 indicate that the swirl chamber design significantly changes the performance of a hydrocyclone. For the removal of very small dispersed droplets and the enhancement of overall separation efficiency, the geometry of the swirl chamber should be tailored. Moreover, results presented in the previous chapters show that the reverse flow core breaks down in the swirl chamber. As a result, migrated dispersed droplets cannot be removed from the downstream. This situation guides us to redesign the tail sections so that all the migrated dispersed droplets can be captured and removed from the continuous phase. This chapter addressed the possible designs of swirl and the tail section for enhancing separation efficiency as well as turndown. Three designs of swirl chamber are outlined and their effects on the internal flow structure and separation performance are investigated, which are presented in Section 6.2. In addition, two tail gadgets are designed to capture migrated droplets from underflow; the detail design of the gadgets and their effects on the separation efficiency are addressed in Section 6.3. Design modifications using a hyperbolic swirl chamber is addressed in Section 6.4.
6.2. Tailoring the swirl chamber

6.2.1. Swirl chamber design criteria

The conical swirl chamber of the standard single cone hydrocyclone (see Fig. 3.1) has been modified using parabolic and hyperbolic swirl chambers. The schematic of three different swirl chamber designs are presented in Fig. 6.1. The design-A and design-B have parabolic profile and the design-C has hyperbolic profile. The design-A and design-B have asymptotic profile, respectively, at the bottom and the top end of the swirl chamber. The swirl chamber for the design-C (hyperbolic swirl chamber) is asymptotic at both the ends. There are two design cases considered: Case-1 has a cylindrical tail pipe and Case-2 has no tail pipe. The length of swirl chamber for the Case-2 is about 2.5 times longer than that of the Case-1. The geometric dimensions of different sections of hydrocyclone for the two design cases are presented in Table 6.1.

<table>
<thead>
<tr>
<th>Design cases</th>
<th>D (mm)</th>
<th>Do/D</th>
<th>Du/D</th>
<th>D/D</th>
<th>Li/D</th>
<th>Ls/D</th>
<th>Lv/D</th>
<th>Lt/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case-1</td>
<td>26</td>
<td>0.07</td>
<td>0.23</td>
<td>0.275</td>
<td>1</td>
<td>7.37</td>
<td>0.1</td>
<td>9.71</td>
</tr>
<tr>
<td>Case-2</td>
<td>26</td>
<td>0.07</td>
<td>0.23</td>
<td>0.275</td>
<td>1</td>
<td>19.23</td>
<td>0.1</td>
<td>0</td>
</tr>
</tbody>
</table>
(a) Design – A (parabolic swirl chamber)

The governing equation for the design-A can be expressed as (APPENDIX D1)

\[ r = \begin{cases} 
\frac{D}{2} & \text{for } z < L_i \\
\frac{D}{2} + \left( \frac{D - D_u}{2L_s^2} \right) \left[ (z - L_i)^2 - 2L_s(z - L_i) \right] & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D_u}{2} & \text{for } z > (L_i + L_s)
\end{cases} \]

Figure 6.1: Schematic profile of three different swirl chambers (a) Design-A (parabolic swirl chamber), (b) Design-B (parabolic swirl chamber), and (c) Design-C (hyperbolic swirl chamber).
The governing equation for the design-B can be expressed as (APPENDIX D2)

\[
r = \begin{cases} 
\frac{D}{2} & \text{for } z < L_i \\
\frac{D}{2} - \left( \frac{D}{2} - \frac{D_u}{2} \right) \sqrt{\frac{z - L_i}{L_s}} & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D_u}{2} & \text{for } z > (L_i + L_s)
\end{cases}
\]
(c) Design-C (hyperbolic swirl chamber)

The governing equation for the design-C can be expressed as (APPENDIX D3)

\[
r = \begin{cases} 
\frac{D}{2} & \text{for } z < L_i \\
\frac{D}{2} - \left( \frac{D - D_u}{2} \right) \sqrt{1 - \left( \frac{L_i + L_s - z}{L_s} \right)^2} & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D_u}{2} & \text{for } z > (L_i + L_s)
\end{cases}
\]
6.2.2. Internal flow structure for the different designs of swirl chamber: Case-1

The internal flow structures for the hydrocyclones with the three designs of swirl chamber (Fig. 6.1) having dimensions presented in Table 6.1 (Case-1) are compared with that for a hydrocyclone having conventional swirl chamber (6 degree frustrocone) (Fig. 2.1a). The normalized tangential velocity profiles at $z/D = 1$ for the different designs of swirl chamber are shown in Fig. 6.2. The conventional swirl chamber provides less tangential velocity when compared with all the other designs of the swirl chamber. The Design-C (hyperbolic swirl chamber) yields the greatest tangential velocity at $z/D = 1$ because of the largest wall curvature. However, the location of peak core pressure for the hyperbolic swirl chamber (Design-C) is closer to the overflow orifice when compared with other designs (Fig. 6.3). The conventional conical swirl chamber yields the location of peak core pressure to the farthest point. It indicates that the new designs of swirl chamber (Design-A, Design-B and Design-C) do not increase the length of reverse flow core. In the conventional design, the core pressure significantly reduces at the end of the swirl chamber ($z/D \approx 8.37$). Very high negative core pressure near the end of swirl chamber can initiate cavitation. However, the hyperbolic swirl chamber (Design-C) yield a significant improvement in regards to the minimization of negative core pressure near the end of the swirl chamber. The reasons for the reduction in core pressure near the end of the swirl camber for the different designs can be explained using a plot of radial pressure differential, which is shown in Fig. 6.4. The vertical axis represents the normalized pressure differential between the center and the wall. A higher pressure differential indicates a greater radial pressure gradient. The radial pressure gradient at $z/D = 1$ for the conventional swirl chamber is less than that for the other designs. The
constant angle of conical (conventional) swirl chamber accelerates the swirling motion in the downstream, which thereby significantly increases the radial pressure gradient. Design-A increases the pressure differential at z/D = 1 but does not reduce the pressure differential at a downstream location. As a consequence, the Design-A provides higher negative pressure near the end of swirl chamber.

Figure 6.2: Effect of swirl chamber designs on the tangential velocity profile at z/D = 1 for Re_F = 23835. Design-C (hyperbolic swirl chamber) provides a better swirling intensity, which can enhance the migration rate of dispersed droplets toward the center.
Figure 6.3: Effect of swirl chamber designs on the core pressure profiles. A lower core pressure for the Design-C indicates less energy losses in the hydrocyclone chambers.

Figure 6.4: Normalized radial pressure differential between the wall and core. $P_c$ and $P_w$ represent the pressure on the wall and at the center of hydrocyclone, respectively.
6.2.3. Internal flow structures with modified swirl chamber: Case-2

The operating conditions for the three designs of swirl chamber (Fig. 6.1) having dimensions given in Table 6.1 (Case-2) are presented in Table 6.2. For the same inlet condition, the overflow ratio, $R_O$ for the design-C is less than that of the design-A and design-B. A greater acceleration of swirling flow due to a higher longitudinal wall curvature (hyperbolic profile) of the swirl chamber decreases the $R_O$. However, the $\Delta P^*$ for the design-B is higher than that of the other two designs. The higher $\Delta P^*$ in the case of design-B yields a smaller $E_{UU}$. It indicates that the energy loss in the design-B is less than that of the other two designs. Figure 6.5 shows that the hyperbolic swirl chamber (Design-C) minimizes the Ekman boundary layer flow ($Q_{1}/Q_O$) and radial inflow ($Q_{2}/Q_O$) through boundary B2 (see Fig. 4.6). However, it increases the short circuit flow from inlet to the vortex core through the boundary B3 ($Q_{3}/Q_O$).

Table 6.2: Operating conditions for the three designs of swirl chamber (see Fig. 6.1)

<table>
<thead>
<tr>
<th>Design</th>
<th>$Re_F$</th>
<th>$P_F$ (Pa)</th>
<th>$\Delta P^*$</th>
<th>$R_O$</th>
<th>$Euo$</th>
<th>$Euu$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design-A</td>
<td>23,835</td>
<td>308,107</td>
<td>1.64</td>
<td>0.091</td>
<td>17.7</td>
<td>10.8</td>
</tr>
<tr>
<td>Design-B</td>
<td>23,835</td>
<td>293,516</td>
<td>1.7</td>
<td>0.089</td>
<td>16.9</td>
<td>10.0</td>
</tr>
<tr>
<td>Design-C</td>
<td>23,835</td>
<td>362,470</td>
<td>1.5</td>
<td>0.077</td>
<td>20.9</td>
<td>14.0</td>
</tr>
</tbody>
</table>
Figure 6.5: Local radial flows as a percentage of total overflow rate for the three different designs of swirl chamber (see Fig. 6.1) having dimension presented in Table 6.1 (Case-2). Q1, Q2 and Q3 are the radial flow, respectively, through the boundary B1, B2 and B3 as shown in Fig. 4.6.

The effect of swirl chamber designs on the normalized tangential velocity at $z/D = 1$ is shown in Fig. 6.6. The peak tangential velocity for the design-C is higher than that of both design-A and design-B. The higher longitudinal wall curvature of the hyperbolic swirl chamber (design-C) increases the tangential velocity downstream of the inlet chamber. The higher acceleration of tangential velocity in the case of design-C provides the greater peak value at the interface between inner and outer vortex. The location of peak tangential velocity for all the designs of swirl chambers is almost the same, which indicates that the diameter of inner vortex should be equal. The normalized axial velocity...
at $z/D = 1$ for the three different swirl chamber designs is shown in Fig. 6.7. The positive magnitude represents a reverse (upward) flow and the negative magnitude characterizes a forward (downward) flow. The reverse flow velocity at the core for the design-B is slightly greater than that of other designs. The diameter of the reverse flow core for all the designs is almost the same. The normalized radial velocity at $z/D = 1$ for three different swirl chamber designs is shown in Fig. 6.8. The positive and negative values represent radially outward and inward velocity components, respectively. Near the wall ($r/R \approx 1$), the radial velocity for all the swirl chamber designs is negative because of the reduce diameter of the swirl chamber in the downstream region. However, the hyperbolic swirl chamber provides a greater radially inward velocity near the wall due to a higher longitudinal wall curvature. Near the core ($r/R \approx 0.05$), the negative value in the radial velocity indicates a local radial inflow to the reverse flow core. The design-B yields a reduced local radial inflow to the reverse flow core at the inlet chamber. The reduced local radial inflow to the reverse flow core at the inlet chamber mitigates the short circuit flow, which thereby increases residence time for the dispersed droplets to migrate toward the core.
Figure 6.6: Effect of swirl chamber designs on the normalized tangential velocity at \( \frac{z}{D} = 1 \). The Design-C (hyperbolic swirl chamber) yields a higher centrifugal acceleration, which thereby increases the migration rate of dispersed droplets toward the center.

Figure 6.7: Effect of the swirl chamber designs on the normalized axil velocity at \( \frac{z}{D} = 1 \). The parabolic swirl chamber (design-B) provides a greater reverse flow velocity than that of other two designs.
Figure 6.8: Effect of the swirl chamber design on the normalized radial velocity at \( z/D = 1 \). The parabolic swirl chamber (design-B) provides a lower radially inward flow at the interface between the inner and outer vortex.

Figure 6.9: Effect of the swirl chamber designs on the normalized static pressure at \( z/D = 1 \). The hyperbolic swirl chamber provides a greater radial pressure gradient near the core.
The effect of swirl chamber designs on the normalized static pressure along the radial positions at $z/D = 1$ is shown in Fig. 6.9. Near the wall ($r/R \approx 1$) the normalized pressure for all the swirl chamber designs is the same, which is almost equal to one. The static pressure decreases toward the core of the hydrocyclone and it is the minimum at the center. The radial pressure gradient for the hyperbolic swirl chamber (design-C) is greater than that of the design-B and it is the minimum for the design-A. Downstream of $z/D = 1$, a higher acceleration of the swirling motion due to a greater longitudinal wall curvature for the hyperbolic swirl chamber (design-C) yields a larger radial pressure gradient and a
smaller static pressure near the wall (Fig. 6.10). As we move downstream, the radial pressure gradient decreases for all the designs (Fig. 6.10) due to the reduction of swirling motion caused by the frictional losses on the wall. In comparison with other designs, the design-C yields a greater pressure differential up to $z/D \approx 15$. It indicates that the hyperbolic swirl chamber provides a higher swirling motion almost all the way downstream which thereby yields a greater centrifugal acceleration.

Figure 6.11 shows that the core pressure profiles for all the designs with longer swirl chamber (see data in Table 6.1-Case-2) are greatly improved when compared with the profiles shown in Fig. 6.3. The longer swirl chambers reduce the wall curvature and gradually decreases the diameter of the chamber, which thereby makes a good balance between the acceleration and deceleration of swirling motion, respectively, by reduced diameter and wall friction. As a consequence, the core pressure profile becomes smoother. Unlike the profiles shown in Fig. 6.3, there is no sharp drop and negative magnitude in the core pressure in the swirl chamber. Among the three designs, the design-C yields the smoothest pressure profile all the way downstream. An interesting phenomenon appears in Fig. 6.11 that the core pressure profile exactly follows the wall profiles of swirl chamber (radius versus $z/D$ chart in Fig. 6.11). The higher wall curvature (i.e., lower radius) for the design-C yields a greater centrifugal acceleration, which thereby supports the lower core pressure in the entire swirl chamber. In comparison with design-B, the design-A provides higher core pressure up to $z/D < 8$ due to a lower wall curvature. The core pressure profile for the design-A crosses the profile for design-B exactly at the same location where the wall profiles of these two designs intersect each other. At $z/D > 8$, the core pressure for the design-A reduces quickly due to a higher wall
curvature (smaller radius) in comparison with the design-B. Therefore, it can be inferred that the core pressure is directly proportional to the radius of swirl chamber. The design-B maintains a positive axial pressure gradient up to far downstream \((z/D \approx 7)\), where it is \(z/D \approx 5.5\) for design-C and \(z/D \approx 6\) for design-A. At the upstream, the higher wall curvature for the design-C enhances the swirling motion, which quickly reduces the core pressure. Even though the wall curvature in the upstream for the design-B is greater than that for the design-A, it is less than design-A at \(z/D > 5\). As a consequence, the design-B provides positive axial pressure gradient up to far downstream.

The positive axial pressure gradient provides the driving force for the core to be moved toward the overflow. The closer the location of peak value of core pressure to the overflow orifice the shorter the reverse flow core is. The core pressure profiles shown in Fig. 6.1 supports the length of reverse flow cores which are presented in Fig. 6.12. The length of the reverse flow core for the design-A and design-C are almost the same because of their locations of peak core pressure are close to each other. The location of peak core pressure for the design-B is farther downstream, which supports the reverse flow core to be longer than other designs. The grade separation efficiency for the three designs is shown in Fig. 6.13. Due to the longer reverse flow core, the design-B supports the transport of migrated dispersed droplets to the overflow from a far downstream and yields better separation efficiency. However, the separation efficiency for the design-C is very close to the design-B and much higher than that of design-A. The relation between the length of reverse flow and the cut size presented in Fig. 6.14 shows that the cut size for the design-B and design-C is almost the same and it is much less than the design-A. Though the length of reverse flow core for the design-C is much shorter than that of
design-B, the higher g-force for the design-C, which is caused by a larger wall curvature as well as a smaller radius of swirl chamber, enhances the migration rate of droplets. Besides, length of reverse flow core for the design-A and design-C is the same but their separations efficiency and cut size is significantly different. Because of lower wall curvature, the design-A provides less g-force as well as less migration rate of dispersed droplet when we compare with design-C. Therefore, it can be inferred that design-C has a better potential for a higher separation. Moreover, following the design-C we can make a short hydrocyclone because of its ability to provide high separation efficiency with a shorter reverse flow core.
Figure 6.11: Effect of the swirl chamber designs on the static pressure along the center of the hydrocyclone. Core pressure profile follows the wall curvature of swirl chamber and a higher wall curvature yields less core pressure.
Figure 6.12: Contours are showing the reverse flow core for the three different swirl chamber designs. Only the reverse flow velocity is mapped and the forwarded flow velocity component is discarded from the contours.
Figure 6.13: Grade separation efficiency graphs for the three designs. Design-B and design-C provides almost same separation efficiency.

Figure 6.14: Relation between length of reverse flow core and cut size for the three different swirl chamber designs (see Fig. 6.1).
6.3. Tailoring the tail section

The simulation results for the hydrocyclones with conventional and modified geometries, which are presented in this Chapter and the previous Chapter, reflect that the reverse flow core breaks down in the swirl chamber and never reaches close to the underflow. Even though dispersed droplets in the tail pipe still migrates to the center they cannot move to the overflow orifice due to the absence of reverse flow. Figure 6.15 shows that the fraction of migrated droplets moves toward the overflow orifice and the other fraction moves toward the underflow. This split-up occurs in the core at the region of reverse flow breakdown. We have seen in the previous sections and chapters that the length of the reverse flow core varies depending on the operating and geometric conditions. So, the fate of migrated droplets depends on the location of reverse flow core breakdown. Figure 6.15 also shows that all the dispersed droplets migrate to the core before reaching the underflow. The separation efficiency of a hydrocyclone would be greatly improved if we could develop a mechanism for capturing the migrated droplets at the underflow. Moreover, the mechanism for capturing migrated droplets from both the overflow and underflow should support the performance of a hydrocyclone to be independent of length of reverse flow core. As a consequence, hydrocyclone will provide high separation efficiency at high fluctuation in the feed flow, which thereby will increase the turndown. The mechanisms for capturing migrated droplets from the underflow have developed in this work, which are discussed in the following Sections.
6.3.1. Geometry of tail “gadget”

Two tail gadgets, which are attached to the tail pipe, are designed to capture the migrated dispersed from the tail end. The schematic of a hydrocyclone with a tail gadget is shown in Fig. 6.16. Two types of gadget are designed and used to evaluate the overall performance. The gadget-A has a vortex finder, which has the diameter equals to overflow-2. A rectangular slot is tangentially connected for underflow outlets. The gadget-B is a reducer having two slots for underflow outlet. It has now vortex finder. The dimensions of these slots are such that the area of total opening is equal to the area of underflow outlet of a conventional hydrocyclone. The schematic diagrams of these two gadgets with appropriate dimensions are given in Fig. 6.17. The operating conditions for the numerical simulations are presented in Table 6.3. The hydrocyclone with a gadget has two overflow (effluent) outlets (Overflow-1 and Overflow-2), which yields two pressure drop ratios ($\Delta P^*_{1}$ and $\Delta P^*_{2}$) and two flow ratios ($R_{O1}$ and $R_{O2}$). The $\Delta P^*_{1} = (P_F-P_{O1})/(P_F-P_{O2})$.
\( P_U \) and \( R_{O1} = \left( \frac{Q_{O1}}{Q_F} \right) \) are relative to overflow-1. The \( \Delta P^*_{2} = \left( \frac{P_F - P_{O2}}{P_F - P_U} \right) \) and \( R_{O2} = \left( \frac{Q_{O2}}{Q_F} \right) \) are relative to overflow-2.

Figure 6.16: Schematic of a hydrocyclone having a tail gadget.
Figure 6.17: Schematic diagram of (a) gadget-A and (b) gadget-B and their dimensions.

Table 6.3: Operating conditions for the hydrocyclone with different gadgets

<table>
<thead>
<tr>
<th>Design</th>
<th>$Re_F$</th>
<th>$P_F$ (Pa)</th>
<th>$\Delta P^*_1$</th>
<th>$\Delta P^*_2$</th>
<th>$Ro_1$</th>
<th>$Ro_2$</th>
<th>$Ro$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gadget-A</td>
<td>23835</td>
<td>329576</td>
<td>1</td>
<td>0.88</td>
<td>0.08</td>
<td>0.056</td>
<td>0.137</td>
</tr>
<tr>
<td>Gadget-B</td>
<td>23835</td>
<td>575444</td>
<td>1</td>
<td>1</td>
<td>0.042</td>
<td>0.052</td>
<td>0.094</td>
</tr>
</tbody>
</table>

$[\Delta P^*_X = (P_F - P_{OX})/(P_F - P_U); \, R_{OX} = Q_{OX}/Q_F ; \, \text{and} \, R_O = R_{O1} + R_{O2}; \, X \geq 1, 2]$
6.3.2. Effect of tail gadget on the core pressure and separation performance

The effect of tail gadget designs on the core pressure profile is shown in Fig. 6.18. For both cases the location of peak core pressure is almost the same. As a consequence, the length of reverse flow core does not affected by the pattern of tail gadget. However, the core pressure profiles are significantly different near the end of the swirl chamber \((z/D \approx 8)\) and in the tail pipe \((z/D > 8)\). For Gadget-A, the core pressure drastically reduces at the end of the swirl chamber and it becomes negative in the tail pipe. On the other hand, the Gadget-B provides less negative pressure gradient near the end of swirl chamber. Moreover, it maintains positive pressure in the tail pipe. The conical shaped gadget (Gadget-A) supports the flow field to have high pressure in the upstream for as to adjust the high reduction of pressure in the reducing conical tail gadget. This hydrodynamic feature indicates that a conical shaped geometry at the end of tail pipe is beneficiary in relatives to the reduction of probability of cavitation.

The performance of a hydrocyclone having the gadgets is evaluated for a given operating conditions shown in Table-6.3, which is presented in Fig. 6.19. The vertical and horizontal axis represents the grade separation efficiency and droplets size, respectively. The solid and dotted line represents the grade separation efficiency for hydrocyclone having, respectively, Gadget-A and Gadget-B. The \(\text{GOR}_1\) and \(\text{GOR}_2\) are the grade separation efficiency relatives to the overflow-1 and overflow-2, respectively; the \(\text{GOR}_{\text{RT}} = (\text{GOR}_1 + \text{GOR}_2)\) is the total grade separation efficiency. The \(\text{GOR}_1\) indicates the grade separation efficiency of a convention hydrocyclone (without a gadget). The grade efficiency relative to overflow-1 \((\text{GOR}_1)\) gradually increases with an increase in the droplet sizes and there is no considerable difference in the \(\text{GOR}_{\text{ROI}}\) between Gadget-A and
Gadget-B. The reason is that the location of peak core pressure and reverse flow breakdown for both the tail gadgets is at the same position. The grade efficiency relative to overflow-2 \((G_{OR2})\) first increases and then decreases with an increase in the droplet size. A larger droplet increases the migration rate, which thereby increases the \(G_{RO2}\). Meanwhile, the \(G_{OR1}\) also increases, which thereby reduces the number of available droplets in the overflow-2. As a consequence, the \(G_{OR2}\) decreases for the larger droplet sizes. In comparison with Gadget-A, the Gadget-B provides less \(G_{OR2}\). The reason is that the radial flow through the slots in the conical gadget (Gadget-B) remixes the migrated droplets with continuous phase, which thereby reduces the \(G_{OR2}\). Figure 6.19 also shows that the overall grade efficiency \((G_{ORT})\) is greatly improved due the addition of tail gadget when we compare with the grade separation efficiency of conventional hydrocyclone \((G_{OR1})\). The \(G_{OR2}\) is the additional grade separation efficiency that has been added by a tail gadget to the overall grade separation efficiency \((G_{ORT})\).
Figure 6.18: Effect of the design of gadget on the core pressure profile.

Figure 6.19: Effect of tail gadget on the grade separation efficiency.
6.4. Design of hydrocyclone geometry based on hyperbolic wall profile

The analyses addressed in section 6.2.3 shows that the hyperbolic swirl chamber (design-C in section 6.2.3) has a better potential for higher separation efficiency. Because, the hyperbolic swirl chamber provides a greater g-force, which thereby enhances the migration rate of dispersed droplets toward the center. So, by following the hyperbolic wall profile (design-C) we can make a short hydrocyclone because of its ability to provide high separation efficiency with a shorter reverse flow core (see Fig. 6.14). This section addresses a search for an optimized geometry of hydrocyclone based on the hyperbolic wall profile. The step-by-step approaches and corresponding design cases are discussed below.

6.4.1. Modified hydrocyclone: design case-1

Figure 6.14 shows that the length of reverse flow core for the hyperbolic swirl chamber (design-C) is about 7.0 times of the diameter of inlet chamber (i.e., \( \ell_c/D \approx 7 \)). So, if a hydrocyclone having the hyperbolic swirl chamber (design-C) and the dimensions presented in Table 6.1 (Case-2) is longer than the core length (\( \ell_c/D \approx 7 \)) then there will be no advantage in terms of overflow grade separation efficiency. In this regard, it is reasonable to reduce the overall length of hydrocyclone, which will increase compactness and reduce manufacturing cost. In this design case, the hydrocyclone having the hyperbolic swirl chamber is cut off at \( z/D = 8.7 \). The schematic diagram and geometric dimensions are presented in Fig.6.20.
Hyperbolic

<table>
<thead>
<tr>
<th>Wall profile</th>
<th>D (mm)</th>
<th>Do/D</th>
<th>Du/D</th>
<th>a/D</th>
<th>b/D</th>
<th>Li/D</th>
<th>Ls/D</th>
<th>Lv/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hyperbolic</td>
<td>26</td>
<td>0.07</td>
<td>0.385</td>
<td>0.154</td>
<td>0.385</td>
<td>1</td>
<td>7.7</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Figure 6.20: Schematic diagram of the modified hydrocyclone (design case-1) and its geometric dimensions. The hydrocyclone having hyperbolic swirl chamber and dimensions presented in Table 6.1(Case-2) is cut-off at z/D = 8.7.

Single phase simulation using the RSM model has been performed in this design case. The feed Reynolds number and feed pressure is 23,835 and 232,297 Pa, respectively. The gauge pressure at the overflow and underflow orifice is, respectively, 0 Pa and 120,000 Pa, which provides a flow ratio (R₀) of 0.054 and the pressure drop ratio (ΔP*) of 2.1. The normalized static pressure along the center and a contour of normalized static pressure are presented in Fig. 6.21. The contour shows a lower pressure in the core and a higher pressure near the wall. Moreover, the static pressure increases toward the underflow. A positive pressure gradient appears all the way from underflow to the
overflow. A contour of reverse flow core and the grade separation efficiency are presented in Fig. 6.22. The reverse flow core appears up to the underflow orifice, which support the transport of all migrated droplets to the overflow orifice. The separation efficiency slightly increases when we compare with that for the Design-C (presented in Fig. 6.13). The reason is that the length of reverse flow for the Design-C was about 7D, which is about 8.7D in this design. These results indicate that with maintaining the same separation efficiency we can reduce the length of a hydrocyclone with hyperbolic swirl chamber to the half of conventional one.

Figure 6.21: Normalized static pressure along the center and a contour of normalized static pressure in the modified hydrocyclone (design case-1) for the $Re_F = 23,835$. 
Figure 6.22: Reduced grade separation efficiency and a contour of reverse flow core in the modified hydrocyclone (design case-1) for the Re_F = 23,835.

6.4.2. Modified hydrocyclone: design case-2

Results presented in section 6.4.1 shows that in the short hydrocyclone we can maintain a positive pressure gradient up to the underflow by patching high pressure at the underflow orifice. This phenomenon leads us to increase the length of hydrocyclone so that the residence time of flow as well as the reverse flow core can be increased. But, Fig. 6.11 (Design-C) shows that the continuous reduction in the diameter of swirl chamber destroys the positive pressure gradient at z/D ≈ 5. For avoiding the breakdown of positive pressure gradient in the middle of swirl chamber, it would be wise not to reduce the diameter of swirl chamber after z/D = 5. Based on this concept, the design of hydrocyclone having hyperbolic swirl chamber (Design-C) is modified. The hydrocyclone with hyperbolic swirl chamber having the dimensions presented in Table
6.1 (Case-2) is cut off at z/D = 5 and a long cylindrical tail pipe is connected. The schematic of modified hydrocyclone (design case-2) and its geometric dimensions are shown in Fig. 6.23.

<table>
<thead>
<tr>
<th>Wall profile</th>
<th>D (mm)</th>
<th>Do/D</th>
<th>Du/D</th>
<th>a/D</th>
<th>b/D</th>
<th>Li/D</th>
<th>Ls/D</th>
<th>Lt/D</th>
<th>Lv/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hyperbolic</td>
<td>26</td>
<td>0.07</td>
<td>0.53</td>
<td>0.154</td>
<td>0.385</td>
<td>1</td>
<td>4</td>
<td>15.23</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Figure 6.23: A schematic diagram of modified hydrocyclone (design case-2) and its geometric dimensions. The hydrocyclone having hyperbolic swirl chamber and the dimensions presented in Table 6.1(Case-2) is cut off at z/D = 5 and a tail pipe is connected.

The simulation for this design of hydrocyclone is performed for the $Re_F = 23,835$. The gauge pressure at the feed, overflow and underflow is, respectively, 187,247 Pa, 0 Pa and 120,000 Pa. The overflow ratio ($R_O$) and the pressure drop ratio ($\Delta P^*$) is about 0.069 and 2.8, respectively. The normalized static pressure along the center and a contour of normalized tangential velocity are shown in Fig. 6.24. The core pressure increases
gradually toward the underflow and provides a positive pressure gradient all the way up to the underflow orifice. The contour shows that the static pressure near the wall decreases toward the underflow. The reason is that the frictional losses on the wall reduce the swirl intensity (see Fig. 6.25), which thereby decreases the radial pressure gradient. The contours of reverse flow core and normalized tangential velocity, and the plot of reduced grade separation efficiency are shown in Fig. 6.25. The reverse flow core appears up to the underflow. It indicates that this class of hydrocyclone (design case-2) is able to transport all the migrated droplets to the overflow orifice. However, the separation efficiency is almost the same when compared with the design case-1 (see Fig. 6.22). Even though the length of reverse flow core for this design is more than double of the design case-1 (see Section 6.4.1), there is no considerable improvement in the separation efficiency. The reason is that the larger diameter of the tail pipe (in comparison with the diameter of swirl chamber at $z/D > 5$ for the design case-1) significantly reduces the centrifugal acceleration. At $z/D > 10$, the tangential velocity in the hydrocyclone becomes less than the feed velocity. This lower swirl motion reduces the migration rate of dispersed droplets toward the center.
Figure 6.24: A normalized static pressure along the center and a contour of normalized static pressure in the modified hydrocyclone (design case-2) for the Re_F = 23,835.
Figure 6.25: The reduced grade separation efficiency plot, and contours of reverse flow core and normalized tangential velocity in the modified hydrocyclone (design case-2) for the $Re_F = 23,835$.

6.4.3. Modified hydrocyclone: design case-3

Even though the modified hydrocyclone (design case-2) does not provide better separation efficiency than that of the design case-1, it is possible to enhance separation efficiency for the design case-2 by increases the feed Reynolds number. The fact is that, because of very long reverse flow core and longer residence time, the design case-2 should provide a better separation efficiency at a larger feed Reynolds number; which may not be possible in the design case-1 because of shorter flow residence time. In this perspective, the design case-1 is more suitable. In manufacturing point of view, the hyperbolic wall profile is complex for machining. For this reason, the swirl chamber of design case-2 is reshaped using three frustocones of different cone angle in such a way
that the wall curvature of swirl chamber almost follows the hyperbolic profile. The schematic diagram of modified hydrocyclone (design case-3) and its geometric dimensions are presented in Fig. 6.26. The contours of normalized tangential velocity and reverse flow core, and a plot for the static pressure along the center are presented in Fig. 6.27. In comparison with the results shown in Fig. 6.25 for the design case-2, there is no variation in the contours of normalized tangential velocity and reverse flow core for the design case-3. Moreover, the static pressure profiles for the design case-2 and the design case-3 are almost the same (Fig. 6.27). These results indicate that the reshaping of swirl chamber of the design case-2 using the three frustocones of conical angles presented in Fig. 6.26 is appropriate. So, the modified hydrocyclone (design case-3) is suitable from both high separation efficiency and simplicity in manufacturing points of view.

| $\text{Do/D}$ | $\text{Du/D}$ | $\text{a/D}$ | $\text{b/D}$ | $\text{Li/D}$ | $\text{Lr1/D}$ | $\text{Lr2/D}$ | $\text{Ls/D}$ | $\text{Lt/D}$ | $\text{Lv/D}$ | $\alpha_1$ | $\alpha_1$ | $\alpha_1$
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.07</td>
<td>0.55</td>
<td>0.154</td>
<td>0.385</td>
<td>1</td>
<td>0.25</td>
<td>0.75</td>
<td>3</td>
<td>15.23</td>
<td>0.1</td>
<td>14°</td>
<td>4.5°</td>
<td>2°</td>
</tr>
</tbody>
</table>

Figure 6.26: Schematic diagram of the modified hydrocyclone (design case-3) and its geometric dimensions.
Figure 6.27: A plot for normalized static pressure along the center, and contours of reverse flow core and normalized tangential velocity in the modified hydrocyclone (design case-2) for the Re_F = 23,835.
CHAPTER 7

NOVEL EFFICIENT LIQUID-LIQUID SEPARATION HYDROCYCLONE

7.1. Introduction

From the detailed analyses of hydrodynamics and separation performance on conventional and modified hydrocyclones, which are presented in the previous chapters, the following key points can be summarized:

I) A vortex finder of length equals to 0.1D and the diameter of overflow tube less than 0.07D provides a better restriction to the short circuit flows in the inlet chamber (see Section 5.3).

II) A hyperbolic shaped swirl chamber provides higher swirl intensity and it is suitable for a shorter hydrocyclone (see Section 6.2.3).

III) A hyperbolic shaped swirl chamber has to be cut-off at z/D = 5 to avoid breakdown of reverse flow core; the wall curvature can be reshaped by three frustocones having half angles of 14°, 4.5° and 2° (see Section 6.4.2 and 6.2.3).

IV) A cylindrical tail pipe longer than 5.0D (z/D = 10) significantly reduces the swirl intensity (see Section 6.4.3)

V) A tail gadget is essential for capturing migrated droplets from the underflow; the gadget-A performs better than gadget-B (see Section 6.3).
Considering the design conditions mentioned above, a high efficiency hydrocyclone is developed by tailoring the geometry of a currently available commercial hydrocyclone. The invented hydrocyclone consists of two overflow outlets positioned at the top and the bottom of the hydrocyclone for collecting separated dispersed oil, two tangential inlets, and an underflow outlet for collecting cleaner water (Fig. 7.2). The said hydrocyclone will be able to operate on a very wide range of inlet conditions without deteriorating the separation efficiency. Moreover, the said hydrocyclone will be a very efficient device for separation of very small sized dispersed droplets and will be able to meet the federal regulations for discharge water. The invented hydrocyclone has two separation zones which response to a large variation of inlet conditions in such a way that a decrease in efficiency in one separation zone is balanced by an increase in efficiency in another separation zone.

7.2. Geometric features of the novel hydrocyclone

Referring to Fig. 7.1a, the invented hydrocyclone comprises of two separation zones: \( \text{SZ1} \) and \( \text{SZ2} \). The first separation zone \( \text{SZ1} \) consists a cylindrical inlet chamber \( 1 \), two reducers \( 2 \) and \( 3 \), a frusto-conical swirl chamber \( 4 \), a cylindrical chamber \( 5 \), and vortex finder \( 11 \). The second separation zone \( \text{SZ2} \) includes a frusto-conical swirl chamber \( 6 \), a tail section \( 14 \), and a vortex finder \( 13 \). All the components mentioned here are coaxial. The diameter and length of the inlet chamber \( 1 \) is \( D \) and \( L_{i} \), respectively. Feed is supplied to the inlet chamber by two inlet channels \( 7 \) and \( 8 \). A vortex finder \( 11 \) of diameter \( D_{01} \) is positioned at the center of the top wall \( 12 \) of the inlet chamber \( 1 \). Protrude length of the vortex finder \( 11 \) inside the inlet chamber \( 1 \) is \( L_{V1} \). Inlet channels \( 7 \)
and 8 are tangentially connected to the inlet chamber 1 (Fig. 7.1b) and positioned at the top end of the inlet chamber 1. The cross-section of the inlet channels having the hydraulic diameter of $D_F$ can be either rectangular or circular (Fig. 7.1c). Two reducers 2 and 3 are adjoined with inlet chamber 1. The first reducer 2 is of length $L_{r1}$ and has a taper angle of $\alpha_1$. The second reducer 3 is of length $L_{r2}$ and has a taper angle of $\alpha_2$. The second reducer 3 adjoins three separation portions which are a frusto-conical swirl chamber 4 of length $L_{s1}$ and taper angle of $\alpha_3$, a cylindrical swirl chamber 5 of length $L_c$, and a frusto-conical swirl chamber 6 of length $L_{s2}$ and taper angle of $\alpha_4$. The second frusto-conical swirl chamber 6 connects a tail section 14 of length $L_t$ which comprises a vortex finder 13 of diameter $D_{O2}$ positioned at the center and two underflow outlets 16 and 17 (Fig. 7.1d). The protrude length of vortex finder 13 inside the hydrocyclone is $L_{V2}$. The underflow outlets are two rectangular opening of height $d$ and width $c$ (Fig. 7.1e). All the dimensions of the hydrocyclone are expressed by $D$ which is the nominal diameter of the inlet chamber 1. The value of $D$ for this hydrocyclone is 26 mm or higher. All the dimensions of the invented hydrocyclone are given in Table 7.1:
Figure 7.1: A schematic of the novel hydrocyclone is showing detail designs.
Table 7.1: Geometric groups and their dimensions

<table>
<thead>
<tr>
<th>Geometric groups</th>
<th>Dimension</th>
<th>Geometric groups</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Li/D</td>
<td>1.0</td>
<td>D_α1/D</td>
<td>0.05</td>
</tr>
<tr>
<td>Lr1/D</td>
<td>0.25</td>
<td>D_α2/D</td>
<td>0.05</td>
</tr>
<tr>
<td>Lr2/D</td>
<td>0.75</td>
<td>α1</td>
<td>14°</td>
</tr>
<tr>
<td>Ls1/D</td>
<td>3.0</td>
<td>α2</td>
<td>4.5°</td>
</tr>
<tr>
<td>Lc/D</td>
<td>5.0</td>
<td>α3</td>
<td>2°</td>
</tr>
<tr>
<td>Ls2/D</td>
<td>5.0</td>
<td>α4</td>
<td>2.25°</td>
</tr>
<tr>
<td>Lt/D</td>
<td>0.5</td>
<td>a/D</td>
<td>0.154</td>
</tr>
<tr>
<td>Lv1/D</td>
<td>0.1</td>
<td>b/D</td>
<td>0.385</td>
</tr>
<tr>
<td>Lv2/D</td>
<td>0.75</td>
<td>c/D</td>
<td>0.077</td>
</tr>
</tbody>
</table>

D_F = 2ab/(a+b) is the hydraulic diameter of inlet channels 9 and 10 = 0.22D. For a circular cross-section the diameter of inlet channels 9 and 10 is D_F.

7.3. Operating principles of the novel hydrocyclone

An overview of flow streams in the novel hydrocyclone can be found in Fig. 7.2. Feeds from the inlets generate a strong swirling motion in the inlet chamber. Due to a difference in centrifugal forces between the continuous and dispersed phases, the dispersed phase migrates toward the center of the hydrocyclone. A greater swirling motion develops a larger difference in centrifugal forces. However, frictional resistances on the wall dissipate the kinetic energy of the swirling motion which thereby decreases
the migration rate of dispersed droplets. The tapered wall of the reducer-1 accelerates the swirling motion and recovers the kinetic energy loss occurred by wall friction. An excessive acceleration by the tapered wall yields the radial pressure gradient at a downstream location larger than that at the inlet chamber, which thereby decreases the core pressure at the downstream location. This negative axial pressure gradient at the center kills off the reverse low core. A long reverse flow core is able to drive the migrated dispersed droplets to the overflow-1 outlet from a far downstream. The gradual reduction of taper angles in using reducers-1, reducer-2 and swirl chamber-1 maintains a positive axial pressure gradient at the center up to a far downstream and recovers the kinetic energy loss by wall friction. The swirl chamber-1 is the main separation chamber for the first separation zone SZ1 (Fig. 7.1a). The reverse flow core continues to the cylindrical chamber. However, length of the penetrated reverse flow core in the cylindrical chamber depends on the feed Reynolds number. At very higher feed Reynolds number, the length of reverse flow core can get shorten (see Fig. 4.8). As a consequence, all the migrated dispersed droplets cannot move to the overflow-1 outlet. In this case, the vortex finder-2 collects the remaining migrated dispersed droplets and enhances separation efficiency. The tapered swirl chamber-2, which is the main separation area for the second separation zone SZ2 (Fig. 7.1a), accelerates the swirling motion and enhances the migration rate of dispersed droplets. The migrated droplets in the swirl chamber-2 as well as in the lower part of the cylindrical chamber are separated by the vortex finder-2. The cleaner continuous phase is radially withdrawn from the underflow outlets.
This invented hydrocyclone provides high separation efficiency for a very large range of feed Reynolds number. At very low feed Reynolds number, the reverse flow core is small. In this case, a large amount of dispersed droplets are collected from the vortex finder-2, which thereby enhances the overall separation efficiency. The length of reverse flow core increases with an increase in the feed Reynolds number. In this case, the longer reverse flow core supports significantly high separation through the overflow-1 and the separation of smaller droplets through overflow-2 enhances the overall separation efficiency of the hydrocyclone. At a very high Reynolds number, the reverse flow core becomes shorter again which yield a lower separation through the overflow-1 but separation through the overflow-2 increases significantly because of the collection of all the remaining migrated droplets by the vortex finder-2. Therefore, for a very wide range of operating conditions this invented hydrocyclone provides high overall separation efficiency. For evaluating separation performance of the novel hydrocyclone, numerical simulations are performed for a wide range of feed Reynolds number. The operating condition of these simulation cases are presented in Table 7.2.

<table>
<thead>
<tr>
<th>U_F (m/s)</th>
<th>Re_F</th>
<th>P_F (pa)</th>
<th>ΔP*_1</th>
<th>ΔP*_2</th>
<th>R_O1</th>
<th>R_O1</th>
<th>R_OT</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>11,440</td>
<td>133,052</td>
<td>1.0</td>
<td>1.0</td>
<td>0.081</td>
<td>0.08</td>
<td>0.16</td>
</tr>
<tr>
<td>4.167</td>
<td>23,835</td>
<td>578,385</td>
<td>1.0</td>
<td>1.0</td>
<td>0.079</td>
<td>0.079</td>
<td>0.158</td>
</tr>
<tr>
<td>6</td>
<td>34,320</td>
<td>1,204,499</td>
<td>1.0</td>
<td>1.0</td>
<td>0.079</td>
<td>0.078</td>
<td>0.157</td>
</tr>
<tr>
<td>8</td>
<td>45,760</td>
<td>2,142,317</td>
<td>1.0</td>
<td>1.0</td>
<td>0.078</td>
<td>0.078</td>
<td>0.156</td>
</tr>
<tr>
<td>12</td>
<td>68,640</td>
<td>4,887,276</td>
<td>1.0</td>
<td>1.0</td>
<td>0.078</td>
<td>0.077</td>
<td>0.155</td>
</tr>
</tbody>
</table>
Figure 7.2: A three dimensional illustration of the novel hydrocyclone with flow streams.
7.4. Performance matrices of the novel hydrocyclone

The performance matrices presented in Section 2.5 are not appropriate for the novel hydrocyclone because of the two overflow outlets. The appropriate performance matrices are given below

**Overflow ratio:** Two overflow flow ratios: the overflow ratio relatives to Overflow-1 \((R_{O1})\) and Overflow-2 \((R_{O2})\), which are defined as

\[
R_{O1} = \left( \frac{\text{flow rate at Overflow-1}}{\text{Feed flow rate}} \right)_{\text{Water}} = \left( \frac{Q_{O1}}{Q_F} \right)_{\text{Water}} \quad (7.1a)
\]

\[
R_{O2} = \left( \frac{\text{flow rate at Overflow-2}}{\text{Feed flow rate}} \right)_{\text{Water}} = \left( \frac{Q_{O2}}{Q_F} \right)_{\text{Water}} \quad (7.1b)
\]

The total overflow ratio is calculated as: \(R_{OT} = R_{O1} + R_{O2}\)

**Pressure drop ratio:** For controlling two overflow ratios, the pressures at both overflow outlets are needed to be controlled. So, two pressure drop ratios are needed to be adjusted during the operation of this novel hydrocyclone. The definitions of these two pressure drop ratios (pressure drop ratio relative to overflow-1 \(\Delta P^*_1\) and pressure drop ratio relative to overflow-2 \(\Delta P^*_2\)) are given below:

\[
\Delta P^*_1 = \frac{(P_F-P_{O1})}{(P_F-P_U)} \quad \text{and} \quad \Delta P^*_2 = \frac{(P_F-P_{O2})}{(P_F-P_U)} \quad (7.2)
\]

**Separation efficiency:** The overall (total) reduced grade separation efficiency \((G_{ROT})\) for this novel hydrocyclone can be given as

\[
G_{ROT}(d_D) = \frac{\left( G_{OT}(d_D)-R_{OT} \right)}{(1-R_{OT})} = \frac{G_{O1}(d_D)+G_{O2}(d_D)-(R_{O1}+R_{O2})}{(1-R_{OT})}
\]

\[
\Rightarrow G_{ROT}(d_D) = \frac{G_{O1}(d_D)-R_{O1}}{(1-R_{OT})} + \frac{G_{O2}(d_D)-R_{O2}}{(1-R_{OT})} = G_{OR1}(d_D) + G_{OR2}(d_D) \quad (7.3)
\]
where $G_{OR1}(d_D)$ and $G_{OR2}(d_D)$ are reduced grade efficiencies relative to the Overflow-1 and Overflow-2 outlets, respectively. Here, $G_{OR1}(d_D)$ and $G_{OR2}(d_D)$ are the grade separation efficiencies relative to the Overflow-1 and Overflow-2 outlets, respectively, which are calculated based on the approach discussed in Section 2.5.1. The overall reduced separation efficiency ($\eta_R$) is calculated based on the feed and underflow concentration of dispersed phase using the Eq. (2.8).

### 7.5. Velocity profiles in the novel hydrocyclone

The velocity streamlines for the different Reynolds number are shown in Fig. 7.3. For all the Reynolds numbers, some short circuit flows from inlets to the vortex core appear in the inlet chamber. In addition, flow circulations appear in the inlet chamber. The area of the circulations in the inlet chamber increases with an increase in the feed Reynolds number. The positions of the circulations shift downstream with an increase in the feed Reynolds number. For a small Reynolds number ($Re_F = 11,440$), the near wall flow streams near the end of inlet chamber move toward the center due to the tapered wall of the reducer-1. In the swirl chamber-1, the flow streams merge to the core flow due to a reduction of centrifugal acceleration by wall frictional loss (see Fig. 7.6). In the downstream, the reducer-1 and reducer-2 increases centrifugal acceleration of continuous phase flow, which thereby moves the flow streams back toward the wall. A higher feed Reynolds number also enhances the centrifugal acceleration and keeps the flow streams near the wall, which is reflected in Fig. 7.3 for the Reynolds number of 23,835. For a higher feed Reynolds ($Re_F > 23,835$), no deviation of near wall flow streams toward the
vortex core appears in the downstream of first circulation. So, a higher feed Reynolds number reduces the secondary flows in the swirl chamber.

Effects of the feed Reynolds number on the normalized tangential velocity at different axial locations are shown in Fig. 7.4. The normalized tangential velocity increases with an increase in the feed Reynolds number. A higher Reynolds number provides greater swirl intensity in the cyclone chambers relative to the swirl intensity at the feed. The novel hydrocyclone maintains high tangential velocity (almost equal to the peak value) in the hydrocyclone chambers up to \( r/R \approx 0.6 \) and then gradually decreases. In the conventional hydrocyclone, the peak tangential velocity appears at the interface between inner and outer vortex \( (r/R \approx 0.15) \) and significantly reduces in the outer flow field \( (r/R > 0.15) \); the \( <U_\theta>/<U_F> \) is less than 1.0 near the wall (see Fig. 4.1). In comparison with the conventional hydrocyclone, the novel hydrocyclone provides a much higher \( <U_\theta>/<U_F> \) near the wall. A higher tangential velocity in the outer flow field is more important than having high tangential velocity near the inner vortex. Because, a higher tangential velocity in the outer flow field provides a larger g-force and accelerate the rate of droplet migration toward the core. Moreover, because of the high tangential velocity near the wall the side wall boundary layer in the novel hydrocyclone will be thinner than that of a conventional one. As a consequence, the number of droplets that follow the side wall boundary layer of the novel hydrocyclone will be less than that would appear in a conventional hydrocyclone. So, the novel hydrocyclone provides higher g-force in the outer vortex, greater migration rate of dispersed droplets, and a thinner side wall boundary when compared with a conventional hydrocyclone.
Contours of normalized tangential velocity for the different feed Reynolds numbers are shown in Fig. 7.5. In the inlet chamber, reducers and swirl chamber-1, there is no considerable reduction in tangential velocity. Because, an acceleration of swirling motion by the frustocones and a reduction of swirling motion by the wall friction are well balanced in these sections. In the cylindrical chamber (5 < z/D < 10), tangential velocity near the wall slightly decreases because of wall friction. This reduction of tangential velocity can be recovered by replacing the straight cylindrical chamber with a minute tapered chamber (a frustocone of 1° cone angle) of same length. In the swirl chamber-2, the tangential velocity increases again due to the tapered wall. The acceleration of tangential velocity and the smaller radius in the swirl chamber-2 provides a high g-force.

Normalized axial velocity profiles for the different feed Reynolds numbers at various axial positions are shown in Fig. 7.6. The positive and negative magnitudes represent, respectively, the reverse (upward) and forward (downward) flow. The forward flow velocity near the wall increases with an increase in the axial distance from the top; it reduces at the end of the cylindrical chamber (z/D ≈ 10) because of shear stress on the side wall as well as forward flow near the core. The forward flow near the side wall of this novel hydrocyclone is much greater when compared with a conventional hydrocyclone. At z/D = 5, the axial velocity in the core for the Re_F = 11440 is negative. For the higher Reynolds numbers, reverse flow velocity appears almost up to the end of the cylindrical chamber (z/D ≈ 10), which is shown in Fig. 7.7. The size and shape of the reverse flow core for all the feed Reynolds numbers are shown in Fig. 7.8. The axially upward velocity component is mapped on the plane. The axially downward and all the other velocity components are discarded from the contour map. For the Re_F = 11,440, the
reverse flow core breaks down at the end of the swirl chamber-1. The swirling motion for the feed Reynolds number of 11,440 is not sufficient to maintain a reverse flow in the cylindrical chamber. However, the length of reverse flow core significantly increases for the higher feed Reynolds numbers. However, the lengths of reverse flow core for all the higher feed Reynolds numbers are almost the same. Because of the constant length of reverse flow core, the separation efficiency relative to the overflow-1 should continuously increase with an increase in the feed Reynolds number.
Figure 7.3: 2D velocity streamlines in the novel hydrocyclone for the different feed Reynolds number. Only axial and radial components are plotted and the tangential component is projected on the plan.
Figure 7.4: Effect of the feed Reynolds numbers on the normalized tangential velocity. – – – $Re_F = 11,440$; …… $Re_F = 23,835$; – – – $Re_F = 34,320$; – – – $Re_F = 45,760$ and —— $Re_F = 68,640$. 
Figure 7.5: Contours of normalized tangential velocity for the different feed Reynolds number.
Figure 7.6: Effect of the feed Reynolds number on the normalized axial velocity. 
- - - $Re_F = 11,440$; 
- - - - $Re_F = 23,835$; 
- - - - - $Re_F = 34,320$; 
- - - - - - $Re_F = 45,760$ and 
- - - - - - - $Re_F = 68,640$. 

168
Figure 7.7: Contours of normalized axial velocity for the different feed Reynolds numbers.
Ref = 11,440  Ref = 23,835  Ref = 34,320  Ref = 45,760  Ref = 68,640

Figure 7.8: Contours of reverse flow core for the different feed Reynolds numbers. The axially upward velocity component is mapped on the plane. The axially downward and all the other velocity components are discarded from the contour map. For the Ref = 11,440, the swirl intensity is not sufficient to maintain a strong inner vortex and a higher downward velocity (see Fig. 7.6) breaks down the core. For a higher Ref, the novel hydrocyclone provides a stable and a longer reverse flow core.
7.6. Pressure profiles in the novel hydrocyclone

Normalized static pressure profiles at four different axial locations for the different feed Reynolds numbers are presented in Fig. 7.9. The static pressure gradually decreases from the side wall toward the center of the hydrocyclone. The static pressure at the center decreases with an increase in the feed Reynolds number. A higher feed Reynolds number generates a greater g-force in the hydrocyclone, which thereby decreases the center pressure. A better understanding of the variation of static pressure in the flow field can be developed from Fig. 7.10. Near the side wall, the static pressure gradually decreases with an increase in distance from the top toward the bottom of the hydrocyclone. In the swirl chamber-2, a high acceleration of swirling motion dramatically decreases the static pressure; it eventually becomes negative near the tail end. In the core region, the normalized static pressure for all the feed Reynolds numbers (except $Re_F = 11,440$) is almost unchanging up to the end of cylindrical chamber ($z/D \approx 10$), which can be seen in Fig. 7.11. The core pressure dramatically reduces in the swirl chamber-2. For all the feed Reynolds numbers, the long reverse flow core (see Fig. 7.8) appears in the hydrocyclone due to the positive axial pressure gradient up to the end of cylindrical chamber ($z/D \approx 10$). In the vortex finder-1, the core pressure drastically decreases because of the orifice effect by the vortex finder. This catastrophic reduction in the core pressure can be minimized by pressure patching on the overflow-1 outlet. A similar effect appears in the vortex finder-2. A great advantage of the novel hydrocyclone is that, even though a higher feed Reynolds number significantly increases the g-force, it always maintains positive axial pressure gradient up to the end of cylindrical chamber. The reduction of normalized static pressures on the side wall for all the feed Reynolds
numbers are the same (Fig. 7.12). Figure 7.12 also shows that the normalized pressure differential between the side wall and center ((<P_W> - <P_C>)/<P_F>) first decreases in the swirl chamber-1 (z/D < 5) and then increases in the cylindrical chamber (5 < z/D < 10) (Fig. 7.12). As a consequence, the reduction of core pressure in the swirl chamber-1 is balanced by the enhancement of core pressure in the cylindrical swirl chamber.
Figure 7.9: Effect of the feed Reynolds number on the normalized static pressure. – – – Re$_F$ = 11,440; · · · · Re$_F$ = 23,835; – – – Re$_F$ = 34,320; – – – Re$_F$ = 45,760 and — — Re$_F$ = 68,640.
Figure 7.10: Contours of normalized static pressure for the different feed Reynolds numbers.
Figure 7.11: Effect of the Re\textsubscript{F} on the normalized static pressure along the axis (r/D=0) of the hydrocyclone. – – – Re\textsubscript{F} = 11,440; – – – Re\textsubscript{F} = 23,835; – – Re\textsubscript{F} = 34,320; – – – – Re\textsubscript{F} = 45,760 and – – – – Re\textsubscript{F} = 68,640. For the wide range of Re\textsubscript{F}, the similar core pressure profiles and the constant core pressure up to the end of cylindrical chamber support long and stable reverse flow core (see Fig. 7.8).
Figure 7.12: Effect of the feed Reynolds number on the normalized static pressure on the wall (solid lines) and the normalized pressure differential between wall and center (dotted lines).

7.7. **Multiphase fluid mechanics in the novel hydrocyclone**

Performance of the novel hydrocyclone has also been evaluated based on the multiphase flow simulations using a mixture theory (see APPENDIX A3). The effects of concentration and size of dispersed droplets at the feed on the separation performance of the novel hydrocyclone are presented in this section. The normalized volume fraction profiles of dispersed droplets at different axial locations in the flow field for the Re_F of
23,835, feed volume fraction ($\phi_F$) of 0.01 and the feed droplet size of 20 microns are shown in Fig. 7.13. For this operating condition, the volume fraction near the wall of the swirl chamber-1 is reduced by about 80%. In the middle of the flow field ($r/(D/2) \approx 0.4$), the volume fraction reduction is about 20%. The separation of dispersed droplets in the swirl chamber-1 is not high because of a lees residence time. At the end of the cylindrical chamber, the volume fraction near the wall decreases by about 140%; this reduction is about 40% in the middle of the flow field ($r/(D/2) \approx 0.15$). Since, reverse flow core was observed almost up to the end of the cylindrical chamber (see Fig. 7.8), these reduced amount of dispersed droplets migrates to the core and are transported to the overflow-1 by the reverse flow. Near the wall of the swirl chamber-2, a significant amount of reduction in the volume fraction appears. Moreover, the volume fraction near the core is less than 50% of the feed fraction at $z/D = 14$. This oil rich core is collected by the overflow-2. However, the volume fraction can further be reduced by increasing the flow residence time using a longer swirl chamber-2. An effect of the feed droplet sizes on the normalized volume fraction at the end of cylindrical chamber ($z/D = 10$) is shown in Fig. 7.14. For the feed droplet size of 1 micron, the normalized volume fraction is 1.0. So, there is no separation of dispersed droplets occurs for the 1 micron droplets. The normalized volume fraction decreases with an increase in the droplet size. A significant amount of dispersed droplets are separated for the droplet size of 20 microns. An effect of the feed concentrations on the normalized volume fraction at $z/D = 10$ for the $d_D$ of 10 microns and $Re_F$ of 23,835 is shown in Fig. 7.15. There is no considerable variation in the volume fraction for the different feed concentration appears.
Figure 7.13: Distribution of volume fraction at different axial locations in the hydrocyclone for the $Re_F = 23,835$, feed droplet size of 20 microns, feed volume fraction of 0.1. The swirl chamber-2 significantly removes the dispersed droplets from near the wall and migrates them toward the center.
Figure 7.14: Effect of feed droplet size on the normalized volume fraction at \( z/D = 10 \) for the \( \phi_F = 0.01 \) and the \( \text{Re}_F = 23835 \).

Figure 7.15: Effect of feed volume fraction on the normalized volume fraction at \( z/D = 10 \) for the \( d_D = 10 \) microns and the \( \text{Re}_F = 23835 \).
7.8. Separation performance of the novel hydrocyclone

An effect of the feed Reynolds numbers on the grade separation efficiency of the novel hydrocyclone is shown in Fig. 4.16. The GO\(_{R1}\) (solid line) and GO\(_{R2}\) (dotted line) represent the reduced grade separation efficiency relative to overflow-1 and overflow-2, respectively. The grade separation efficiencies are calculated using the Eq. (7.3) (see Section 7.4). The GO\(_{R1}\) significantly increases with an increase in the Re\(_F\) from 11,440 to 23,835. For the Re\(_F\) > 23,835, the GO\(_{R1}\) gradually increases with an increase in the Re\(_F\).

For the Re\(_F\) = 11,440, the swirling motion is not sufficiently strong to create a long reverse flow core (see Fig. 7.8). In comparison with the Re\(_F\) = 11,440, the Re\(_F\) = 23,835 provides a longer reverse flow core and a higher g-force; these two effects significantly enhance the GO\(_{R1}\) for the Re\(_F\) = 23,835. For the Re\(_F\) > 23835, the length of reverse flow core is almost constant (see Fig. 7.8). However, a higher Reynolds number enhances the g-force quadratically and reduces the residence time of flow. As a consequence, a higher Reynolds number increases the GO\(_{R1}\) gradually. In the conventional hydrocyclone (see Chapter 4), the grade separation efficiency first increases and then decreases for the higher Reynolds numbers and possesses a finite turndown ratio. However in the novel hydrocyclone, the GO\(_{R1}\) continuously increases with an increase in the feed Reynolds number. So, the novel hydrocyclone should possess a wide turndown ratio. Moreover, the GO\(_{R2}\) (a reduced grade efficiency relative to overflow-2) first increases and then decreases, when the feed droplet size increases. For a larger dispersed droplets, the GO\(_{R2}\) decreases with an increase in the feed Reynolds number. Since the GO\(_{R1}\) increases with an increase in the Re\(_F\), there is less number of large dispersed droplets available to be separated in the swirl chamber-2. For a smaller droplets (d\(_D\) < 15 microns), the GO\(_{R2}\)
increases with an increase in the feed Reynolds number. The smaller droplets slowly migrate toward the core and need high residence time to be separated. Since, the $G_{OR1}$ is very small for the smaller droplets, a large amount of smaller droplets are available to be separated in the swirl chamber-2. A higher feed Reynolds number increases the migration rate of these smaller droplets in the swirl chamber-2 and thereby increase the $G_{OR2}$. The summation of the $G_{OR1}$ and $G_{OR2}$ is the total reduced grade separation efficiency ($G_{ORT}$) which is shown in Fig. 7.17. A relation between the length of reverse flow core and the cut-size is shown in Fig. 7.18. The left and right vertical axes represent the normalized length of reverse flow core and cut size, respectively. The length of reverse flow core is almost constant for the $Re_F > 30,000$. The cut size gradually decreases with an increase in the feed Reynolds number. The overall separation efficiency and underflow concentration calculated based on the multiphase flow simulations (mixture theory) are presented in Fig. 7.19. The overall separation efficiency calculated from the multiphase flow simulation is higher than that calculated based on the Lagrangian tracking. For 20 microns droplets, the overall separation efficiency in the multiphase flow simulation is about 75%; it is about 65% in the Lagrangian tracking. In the discrete phase model (Lagrangian tracking), only one way coupling (only continuous phase affects the droplet motion) and no buoyancy force are considered. The simplified assumptions in the Lagrangian tracking can cause this difference in the separation efficiency. The effect of feed concentration the on the overall separation efficiency is shown in Fig. 7.20. The efficiency relative to overflow-1 slightly increases for the higher feed volume fraction. However, the total efficiency is constant for all the feed concentration.
Figure 7.16: Effect of the feed Reynolds numbers on the overflow reduced grade separation efficiency. Solid and dotted lines represent the grade separation efficiency relatives to overflow-1 and overflow-2, respectively. ($\text{oRe}_F = 68,640$; $\text{Re}_F = 45,760$; $\text{Re}_F = 34,320$; $\text{Re}_F = 23,835$; $\text{Re}_F = 11,440$)
Figure 7.17: Effect of the feed Reynolds numbers on the total reduced grade separation efficiency. (\(0 \text{Re}_F = 68,640\); \(\text{Re}_F = 45,760\); \(\text{Re}_F = 34,320\); \(\text{Re}_F = 23,835\); \(\text{Re}_F = 11,440\))

Figure 7.18: Relation between normalized length of reverse flow core and cut size for the different Reynolds numbers.
Figure 7.19: Overall reduced separation efficiency for the different feed droplet sizes and underflow concentration calculated based on the multiphase flow simulations. $\phi_F = 0.01$ (10,000 ppm) and $Re_F = 23,835$.

Figure 7.20: Effect of feed concentration on the overall reduced separation efficiency.
CHAPTER 8

ROTATING TUBULAR MEMBRANE

8.1. Introduction

The performance of a hydrocyclone can be further improved by replacing the cylindrical swirl chamber by a tubular membrane. The thought behind this recommendation is that fraction of underflow water will be radial withdrawn through the tubular membrane. The membrane will remove all the fine droplets from the radial withdrawn. The rotating flow in the swirl chamber of will continue to swirl in the tubular membrane. This swirling motion creates a crossflow on the membrane surface; it keeps the dispersed droplets away from the wall as well as reduces the fouling. For mimicking the rotating flow in the porous tail pipe, a rotating tubular membrane is modeled. In this chapter a numerical study of an axially rotating tubular ceramic membrane operated in a crossflow regime is performed with oil-water dispersions used as a model mixture. Internal hydrodynamics are explored using computational fluid dynamics simulations to obtain the velocity field in the continuous phase and predict the separation efficiency with respect to the dispersed phase. A discrete phase model is used to estimate trajectories of dispersed oil droplets within the membrane channel. The separation performance of the process is evaluated in terms of the droplet cutoff size. Effects of the Reynolds and Swirl numbers on velocity and pressure fields, shear stress, droplet cutoff size, and separation efficiency are investigated.
8.2. Geometry and test cases

The rotating CFF system studied in the following simulation corresponds to a tubular microfiltration membrane. The geometry of the CFF system is shown in Fig. 8.1. The CFF system has a membrane that rotates about the longitudinal z-axis with an angular velocity $\omega$. The membrane has a thickness ($\delta_m$) of 2 mm and a length ($L_m$) of 250 mm mimicking the physical dimension of ceramic membranes manufactured by TAMI Industries. The inner diameter ($D_m$) and the permeability ($\alpha$) of the membrane are 6 mm and $1 \times 10^{-14}$ m² (10 millidarcy) respectively. The permeability is the proportionality constant in Darcy’s law ($\bar{U} = \alpha \Delta P / \mu \delta_m$) which relates discharge and fluid viscosity ($\mu$) to a pressure differential ($\Delta P$) across the membrane. The value of the permeability in the simulation is selected to match the value experimentally measured in our laboratory (Ji, 2013). A fully developed Hagen-Poiseuille laminar flow is defined as the inlet condition. Simulations are performed for three different swirl numbers ($S_w$) and three different feed Reynolds numbers ($Re_F$) as shown in Table 8.1. The inlet Reynolds number is defined, based on the cylindrical geometry of the membrane, as $Re_F = 4 \dot{m} / (\pi \mu D_m)$ where $\dot{m}_C$ is the mass flow rate at the inlet and $\mu_C$ represents the viscosity of water.
Figure 8.1: (a) Illustration of the geometry of a crossflow filtration system wherein the microfiltration membrane is rotating about the vertical axis with an angular velocity of $\omega$; (b) view of the membrane defined by a square box shown in (a). Arrows are used to represent the flow in the axial, radial and azimuthal directions.
Table 8.1: Parameters used in simulation test cases. Cases 1-5 correspond to a non-turbulent flow regime ($Re_F < 2300$) and Case 6 to a transitional flow regime ($2300 < Re_F < 4000$) (White, 1991).

<table>
<thead>
<tr>
<th>Cases</th>
<th>N Angular velocity of the membrane (RPM)</th>
<th>$Q_i$ Flow rate at inlet (m$^3$/s)</th>
<th>$S_w$ Swirl number $S_w = \omega R/\bar{U}_z$</th>
<th>Re Reynolds number $Re = 4\dot{m}_w/(\pi \mu_w D)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>6.67x10^{-6}</td>
<td>0</td>
<td>1415</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>6.67x10^{-6}</td>
<td>0.75</td>
<td>1415</td>
</tr>
<tr>
<td>3</td>
<td>1000</td>
<td>6.67x10^{-6}</td>
<td>1.50</td>
<td>1415</td>
</tr>
<tr>
<td>4</td>
<td>1500</td>
<td>6.67x10^{-6}</td>
<td>2.25</td>
<td>1415</td>
</tr>
<tr>
<td>5</td>
<td>1500</td>
<td>1.00x10^{-5}</td>
<td>1.50</td>
<td>2122</td>
</tr>
<tr>
<td>6</td>
<td>1500</td>
<td>1.33x10^{-3}</td>
<td>1.10</td>
<td>2822</td>
</tr>
</tbody>
</table>

The swirl intensity of a flow field can be characterized by the Swirl number ($S_w$), which is defined as $S_w = M_\theta/(RM_z)$, where $M_\theta$ is the axial flux of the angular momentum, $M_z$ is the axial flux of the axial momentum, and $R_m$ is the inner radius of the membrane. On the basis of this relation the integral form of the Swirl number ($S_w$) is given by (Sheen et al., 1996; Chang and Dhir, 1994; Jawarneh, Vatistas, 2006; Mantilla, 1998)

$$S_w = \int_0^{R_m} U_z U_\theta r^2 dr / (R \int_0^{R_m} U_z^2 r dr) \cong \omega R_m/\bar{U}_z,$$

(8.1)

where $U_z$ and $U_\theta$ are the axial and the tangential velocity components respectively. Eq. (8.1) is simplified using an average axial velocity ($\bar{U}_z$) and assuming that a free vortex exists inside tube, i.e. $U_z = \bar{U}_z$ and $U_\theta = \omega R_m^2/r$ where $\bar{U}_z = 2(Q_i + Q_o)/(\pi D_m^2)$ and $\omega = 2\pi N/60$. $Q_F$ and $Q_{ret}$ is the inlet and outlet flow rates respectively.
8.3. Numerical method and boundary condition

The velocity specified at the inlet of the CFF system is that of a fully developed Hagen-Poiseuille flow. This velocity profile is defined by writing a subroutine through user-defined function of Ansys Fluent 14.5. The transmembrane pressure is applied by specifying a pressure at the outlet boundary ($\Gamma_O$) of 50 kPa for all the simulations. The gauge pressure at the permeate side is zero. The feed flow rates at the inlet boundary ($\Gamma_i$) are set to be one of the following three values: $6.67 \times 10^{-6}$ m$^3$/s, $1.0 \times 10^{-5}$ m$^3$/s, and $1.33 \times 10^{-5}$ m$^3$/s, which correspond to the Reynolds numbers 1415, 2122, and 2822 respectively. Detailed operating conditions are presented in Table 8.1. The membrane rotates about the longitudinal axis at an angular velocity $\omega$. The boundary conditions are shown in Fig. 8.2. $P = P_i$ and $U_i = U \cdot \hat{e}_z$ on $\Gamma_i$, $P = P_O$ on $\Gamma_O$, and $P = 0$ on $\Gamma_\infty$ are specified. No-slip condition is applied on all the solid internal walls.

![Diagram of CFF system](image)

Figure 8.2: A longitudinal cross-section of the CFF system simulated. Velocity on $\Gamma_i$, ambient pressure on $\Gamma_\infty$, and 50 kPa on $\Gamma_O$ are specified.
8.4. Verification of the numerical simulations

The numerical solutions are verified by comparing them against an approximate analytical solution presented by Mellis et al. (1993) for a flow through a pipe with porous non-rotating walls. Mellis et al. (1993) estimated the pressure with respect to axial position, \( z \), as

\[
P(z) = P_\infty - 2\bar{U}_F \left[ \frac{\delta_m \mu^2}{R_m \alpha} \sinh \left( \frac{4}{R_m} \sqrt{\frac{\alpha}{R_m \delta_m}} z \right) \right] + \left( P_F - P_\infty \right) \cosh \left( \frac{4}{R_m} \sqrt{\frac{\alpha}{R_m \delta_m}} z \right),
\]

where \( P_\infty \) and \( P_F \) represent the ambient and feed pressures, respectively, and \( \delta_m \) is the thickness of the porous wall. From Darcy’s law \( U_R(z) = (\alpha/\delta_m \mu)(P(z) - P_\infty) \), the volumetric crossflow rate through the membrane \( (Q_m) \) at any axial position can be written as

\[
Q_m(z) = 2\pi R_m \int_0^z U_R(z) \, dz =
\]

\[
\frac{\pi}{2} (P_F - P_\infty) \sqrt{\frac{R_m^2 \alpha}{\delta_m \mu^2}} \sinh \left( \frac{4}{R_m} \sqrt{\frac{\alpha}{R_m \delta_m}} z \right) - \pi R_m^2 \bar{U}_F \left( \cosh \left( \frac{4}{R_m} \sqrt{\frac{\alpha}{R_m \delta_m}} z \right) - 1 \right).
\]

Assuming that a Hagen-Poiseuille velocity distribution is valid in any cross-section of the CFF system, the axial velocity in the bulk flow domain can be approximated as

\[
U_z(r, z) = \frac{2Q_z(z)}{\pi R_m^3} \left[ 1 - \left( \frac{r}{R_m} \right)^2 \right],
\]

where \( Q_z(z) \) is the volumetric flow rate in the longitudinal direction and is calculated from the mass balance as \( Q_z(z) = Q_F - Q_m(z) \). From the continuity equation, the radial velocity \( (U_r(r,z)) \) can be expressed as

\[
U_r(r, z) = \frac{1}{r} \int r \frac{\partial U_z(r, z)}{\partial z} \, dr \quad ; \quad U_r(0, z) = 0.
\]
Figure 8.3 shows a comparison of axial and radial velocity profiles obtained (a) from the analytical solution (Eqs. (8.4) and (8.5)) and (b) from the numerical simulations. The axial velocity ($U_z$) for a specific Reynolds number is normalized by the maximum axial velocity ($U_z(0, 0)$) in the inlet plane of the CFF system. The radial velocity ($U_r$) is normalized by the average permeate velocity defined as $\bar{U}_R \equiv Q_m/(2\pi R_m L_m)$. It is observed that both the axial and the radial velocity profiles obtained from numerical simulation using Ansys Fluent 14.5 are in agreement with the analytical solution.

![Figure 8.3: Axial and radial velocity profiles at $z/L_m = 0.8$ for the non-rotating CFF system ($S_w = 0$). with $Re_F = 1415$. Both axial and radial velocity profiles obtained from numerical simulations match the approximate analytical solution (Eqs. (8.4) and (8.5)) with a maximum deviation of 2%.](image-url)
8.5. Flow patterns

The axial, radial, and tangential velocity components and their dependencies on the Reynolds and Swirl numbers are presented in Figs. 8.4-8.7. Figure 8.4a shows the variation of the dimensionless axial velocity profile at different cross-sections in the axial direction for the non-rotating CFF system. It reveals that the axial velocity drops linearly which results in a decrease in the shear stress at the membrane surface along the axial direction. The effect of the Swirl number on the axial velocity profile is shown in Fig. 8.4b. The axial velocity is observed to increase slightly with an increase in the Swirl number.

Figure 8.4: Normalized axial velocity profiles for the CFF system with $Re_F = 1415$. The axial flow velocity decreases downstream due to the presence of a permeate flux. Axial velocity near the core region increases with an increase of the Swirl number. $U_z(0,0) = 0.475$ m/s is the velocity at the centerline of the flow within inlet plane.
Figure 8.5: Radial velocity profiles at different axial locations for the CFF system with $Re_F = 1415$. The radial velocity is normalized by the average permeate velocity $\bar{U}_R \equiv Q_m/(2\pi R_m L_m) \approx 3.26 \times 10^{-4}$ m/s. The radial velocity in the bulk flow domain decreases downstream. A sudden rise in the radial velocity near $r/ R \approx 0.35$ in (b) is explained by the relative importance of centrifugal acceleration compared to the reduction of radial velocity due to the swirling motion.

In Figure 8.5, the normalized radial velocity is presented for different operating conditions. Figure 6a shows that the radial velocity increases almost linearly from the centerline of the flow ($r = 0$) to $r/R_m \approx 0.8$. All profiles peak around $r/R_m = 0.8$ and then decrease gradually because of the viscous resistance at the membrane surface. Since the axial velocity profile for the stationary CFF is parabolic, the radial velocity profile shown in Fig. 8.5 (a) satisfies the continuity equation. The radial velocity of a rotating CFF system, as shown in Fig. 8.5b, gradually increases then peaks near the membrane surface. At $r/R_m \approx 0.5$, the radial velocity increases steeply. For the rotating CFF system, a centrifugal force applied on the fluid increases the radial velocity. The effect of
centrifugal force on the radial velocity increases with the distance away from the center of the CFF system. However, near the membrane surface this effect is suppressed by a high radial pressure gradient (see Fig. 8.8). Between \( r/R_m = 0 \) and 0.8, the radial velocity for both non-rotating and rotating membranes increases with the distance along the membrane channel. The increased radial velocity in the downstream region (higher \( z \)) results in additional drag on the droplets and can reduce grade efficiency (see section 8.8.2). Figures 8.5a and 8.5b also show that the radial velocity at the membrane surface is constant along the axial direction. This is because the pressure drop between inlet and outlet on the feed side is in the 15 Pa to 40 Pa range (depending on the test case) and does not have a significant influence on the transmembrane pressure (\( \sim 50 \) kPa) and its gradient along the axial direction. Therefore the permeate flux is effectively constant along the membrane length. The radially outward drag force on the droplets situated near the membrane surface is almost independent of the axial location.

Figure 8.6a shows that the radial velocity, except at \( r/R_m \approx 0.5 \), decreases with an increase in the Swirl number for a given inlet Reynolds number. This may reduce the drag force on the droplets and can increases their migration rate towards the centerline of the flow. To understand what happens at \( r/R_m < 0.5 \) two effects need to be considered. An increase of the axial velocity towards the core (i.e. with \( r \) approaching zero) decreases the radial velocity (see Fig. 8.4b). In addition, the magnitude of the centrifugal force increases with distance away from the center of the CFF system; the higher centrifugal force again increases the magnitude of radial velocity. At \( r/R_m \approx 0.5 \), it appears that a balance between the above two effects is achieved. As a consequence, the radial velocity at \( r/R_m \approx 0.5 \) for both the non-rotating membrane is the same as for the rotating
membrane. In the outer region \((r/R_m > 0.5)\), a rise in the radial pressure gradient near the membrane surface with an increase in the Swirl number (see Fig. 8.8a) decreases the radial velocity. In comparison with the non-rotating membrane, relatively smaller drag may be exerted on the droplets as they move towards the center of the flow field due to the reduced radial flow. The effect of the Reynolds number on radial velocity profile is shown in Fig. 8.6b. The radial velocity decreases with an increase in the Reynolds number. Thus, a CFF system operating with a higher Reynolds number as well as a higher Swirl number can provide greater separation efficiency (see section 8.8.2).

![Graphs showing the effect of Swirl number and Reynolds number on radial velocity profile.](image)

Figure 8.6: Effect of the Swirl number on the radial velocity profile at \(z/L_m = 0.5\). There is a balance between the centrifugal acceleration and the increased axial acceleration of the flow for the rotating CFF system at \(r/R_m \approx 0.5\). A higher axial flux abates the effect of centrifugal acceleration and yields smoother radial velocity profile near \(r/R_m \approx 0.5\). For this case \(\bar{U}_R \equiv Q_m/(2\pi R_m L_m) \approx 3.26 \times 10^{-4} \text{ m/s.} \)
Figure 8.7: Normalized tangential velocity profiles. The tangential velocity is normalized by $U_{0R} = \omega R_m \approx 0.47$. Tangential velocity increases downstream due to a decrease in the axial flux. The rise of axial momentum reduces the tangential momentum to satisfy conservation of mass and momentum.

Figure 8.7a shows the normalized tangential velocity at different axial locations of the CFF system. The normalized tangential velocity increases with distance along the longitudinal direction. This is because the reduction of axial velocity downstream (see Fig. 8.4a) increases the swirl intensity (see Eq. (8.1)) indicating that the swirling flow develops in the CFF system. The increased tangential velocity downstream yields an increase in the difference of centrifugal force between two phases. However, the probability of the lighter dispersed phase (droplets) to migrate toward the core increases downstream. The effect of the Reynolds number on the tangential velocity is shown in Fig. 8.7b. For a constant rotational speed of membrane the tangential velocity decreases with an increase in the Reynolds number, because the higher axial momentum due to the larger Reynolds number yields a decrease in the swirling intensity in the flow field (See Eq. (8.1))
8.6. Pressure distribution

The effect of the Swirl number on the normalized pressure distribution at \( z/L_m = 0.5 \) of the CFF system with the Reynolds number of 1415 is shown in Fig. 8.8a. The pressure across the non-rotating CFF system (\( S_w = 0 \)) is found to be constant; whereas, for the rotating CFF system, the pressure profile is parabolic. The pressure gradient near the membrane surface increases with an increase of the Swirl number. According to dispersed phase mechanics (APPENDIX A2), the high pressure near the membrane and low pressure at the center facilitates the movement of droplets toward the center and thus reduces fouling of the membrane. The higher pressure gradient, corresponding to the higher Swirl number, reduces the radially outward velocity in the bulk flow field (Fig. 8.6a). Near the membrane surface the difference between the velocities of continuous and dispersed phases is very small because of no slip condition. In that case, the droplet movement should be highly affected by the Swirl number due to the variation of the pressure gradient (Fig. 8.8a). The effect of the Reynolds number on the normalized pressure distribution at \( z/L_m = 0.5 \) of the CFF system is shown in Fig. 8.8b. For \( r/R_m < 0.4 \), the pressure is equal for all the Reynolds numbers and decreases with an increase in the Reynolds number for \( r/R_m > 0.4 \). The swirling effect reduces with an increase in the Reynolds number (Table 8.1, Fig. 8.7b), which causes the reduction of the pressure gradient near the membrane surface. The increased radially outward velocity of the flow, due to a decrease of pressure gradient, can provide an additional drag on the droplet while moving toward the center of the CFF system. The magnitude of pressure at \( r/R_m < 0.4 \) is constant, which reveals that the droplet motion near the core is not influenced by radial pressure force.
Figure 8.8: A Distribution of normalized pressure $P^* = (P - P_{ret})/(P_i - P_{ret})$ at $z/L_m = 0.5$ of the CFF system. Higher Swirl number increases axial momentum and pressure drop in the core of the flow resulting in larger radial pressure gradients. A higher Reynolds number with constant rotational speed of the membrane decreases the swirl intensity. As a consequence, the radial pressure gradient decreases with an increase in the Reynolds number.

8.7. Wall shear stress components

The shear or traction forces acting on the membrane surface are calculated as $\tau \cdot \hat{e}_r = \tau_{rr} \hat{e}_r + \tau_{rz} \hat{e}_z + \tau_{r\theta} \hat{e}_\theta$. $\tau_{rr}$ is the normal component which is ignored since it is normal to the membrane surface. $\tau_{rz} = \mu \left[ \frac{\partial U_z}{\partial r} + \frac{\partial U_r}{\partial z} \right]_{r=R_m}$ is the average shear component in the longitudinal direction and $\tau_{r\theta} = \mu \left[ r \frac{\partial}{\partial r} \left( \frac{U_\theta}{r} \right) + \frac{1}{r} \frac{\partial U_r}{\partial \theta} \right]_{r=R_m}$ is the shear component in the tangential direction. The average longitudinal wall shear stress ($\tau_{rz}$) for a pressure-driven flow is given by $\tau_{rz} = \Delta P D_m/(4L_m)$, based on the total axial pressure drop ($\Delta P$) for a flow through a solid wall tube. For flow through a tubular membrane,
using Eqs. (8.4) and (8.5), an improved result can be obtained and is given by \( \tau_{rz} \approx \frac{\mu}{2\pi R_m} \left[ \frac{\partial^2 Q_z(z)}{\partial z^2} - 8 \frac{Q_z(z)}{R_m^2} \right] \). For flow in a channel of a rotating tubular membrane, there are no analytical expressions available but numerical results can be employed to estimate the shear at the wall. The average angular component of the wall shear stress is calculated by integrating \( \tau_{r\theta} \) over the length of the membrane. \( \tau_{r\theta} \) is given by

\[
\tau_{r\theta} \approx \frac{1}{L_m} \int_0^{L_m} \left( \frac{\partial U_\theta}{\partial r} \right)_{r=R_m} \, dz - \omega, \tag{8.6}
\]

where \( \omega \) is the angular velocity. Similarly, the average longitudinal component of the wall shear stress is computed as

\[
\tau_{rz} = \frac{1}{L_m} \int_0^{L_m} \left( \frac{\partial U_z}{\partial r} \right)_{r=R_m} + \frac{\partial U_r}{\partial z} \right|_{r=R_m} \, dz. \tag{8.7}
\]

The resultant of the components of average wall shear stress and its direction are calculated as \( \tau_R = \sqrt{\tau_{rz}^2 + \tau_{r\theta}^2} \) and \( \varphi = \tan^{-1}(\tau_{r\theta}/\tau_{rz}) \) respectively. The variation of wall shear stress as a function of the Swirl number and the Reynolds number is shown in Fig. 8.9.

The average axial component of wall shear stress gradually increases with an increase in the Swirl number (Fig. 8.9a). The axial wall shear stress also linearly increases with an increase in the Reynolds number. Moreover, the tangential component of the wall shear stress linearly increases with a rise in the Swirl number and an increase in the Reynolds number (Fig. 8.9b). However, the tangential component is more significant than the axial component. The increased resultant wall shear stress (Fig. 8.9c) due to the higher swirling flow as well as the higher Reynolds number helps to sweep oil droplets from the membrane surface. The higher swirl number increases the tangential
component while the higher Reynolds number increases the axial component of the wall shear stress (Fig. 8.9d). Thus, the increased wall shear stress in case of the rotating CFF system with the higher Reynolds number can enhance the grade efficiency and reduce the accumulation of droplets on the membrane surface.

Figure 8.9: Effect of the Swirl and the Reynolds number on the average wall shear stress. The effect of the Swirl number is illustrated for the CFF system with Re = 1415. The Reynolds number’s effect is presented for the CFF system with rotation at 1500 rpm. The resultant wall shear stress increases with an increase in both the Swirl and the Reynolds numbers.
8.8. Separation performance

The separation performance of the CFF system is evaluated in terms of the grade efficiency \(G\) and droplet cut size \(d_{d50}\). The effects of the Reynolds number, droplet size, the Swirl number, and the Stokes number on the grade efficiency and droplet cutoff size are discussed in this section.

8.8.1. Droplet cut size \(d_{d50}\)

The droplet cut size \(d_{d50}\) provides a simple and clear indication of the separation performance making it easy to compare the performance of the CFF system under different operating conditions. The cut size is defined as the diameter of the droplet for which the separation efficiency due to the centrifugal action is 50\%. In other words, the droplet of size \(d_{d50}\) has a 50\% probability to contact the membrane surface. We note that the separation performance of a microfiltration membrane is quantified using a different metric – the nominal pore size defined as the diameter of the smallest particle completely removed by the membrane. Thus, the rotating membrane system can be viewed as providing two consecutive separation barriers – hydrodynamic removal of oil droplets away from the membrane and membrane filtration of the droplets that do contact the membrane surface – with the two separation steps described by two distinct metrics.

The cut size \(d_{d50}\) can be estimated using the assumption that this 50\% probability occurs when the radially inward velocity of a droplet is equal to the radially outward velocity of the continuous phase (water). The radial velocity of a droplet in a rotating system can be determined from the balance between the centrifugal force and the drag force on the droplet Rushton et al. (1996):
\[ U_{RD} = -\frac{dr_D}{dt} = \frac{d_D^2}{18 \mu} (\rho_C - \rho_D) R m \omega^2, \]  

(8.8)

where \( U_{Rd} \) is the maximum radially inward velocity of a droplet of size \( d_d \). If \( U_{RD} = U_R \), the corresponding droplet size, \( d_D \), is equal to \( d_{D50} \).

Figure 8.10: Effects of the Reynolds and Swirl numbers on the cut size of droplets. The effect of the Reynolds number is illustrated for CFF system with \( \omega = 1500 \) rpm. The effect of the Swirl number is presented for CFF system with \( \text{Re} = 1415 \). An increase of rotational speed in the CFF system greatly decreases the cutoff size.
The effects of the Reynolds and Swirl numbers on \( d_{50} \) for the rotating CFF system are shown in Fig. 8.10. The cut size slightly decreases with an increase in the Reynolds number. This is due to the increase of the wall shear stress with the Reynolds number. Though the effect of the axial component of wall shear stress is not considered in the analytical solution, almost no difference between the analytical and the numerical cut size for the high Reynolds number (\( \text{Re} = 2822 \)) is observed. The cut size greatly decreases with an increase of the Swirl number. Therefore, the swirl can significantly increase the separation efficiency.

### 8.8.2. Grade efficiency

The total grade efficiency is the fraction of the total mass of droplets that leaves the membrane channel with the retentate stream: 

\[
G_T = \frac{m_{D,\text{ret}}}{m_{D,F}} = \frac{N_{D,\text{ret}}}{N_{D,F}},
\]

where \( m_{D,\text{ret}} \) is the mass flow rate of droplets at the outlet and \( m_{D,F} \) is the mass flow rate of droplets at the inlet. Since the droplets are assumed not to interact with each other or with the continuous phase, and not to break up, the mass of droplets can be represented as a number of droplets. \( N_{D,\text{ret}} \) is the number of droplets that leave the membrane channel with the retentate stream and \( N_{D,F} \) is the total number of droplets entering the membrane channel at the inlet. The reduced grade efficiency can then be used to quantify the performance of centrifugal separation. The reduced grade efficiency is given by

\[
G_R = G_T - R_{\text{ret}},
\]

where \( R_f \) is the flow ratio, which is defined as the ratio of the flow rate at the outlet \( (Q_{ret}) \) to the feed flow rate \( (Q_F) \): 

\[
R_{\text{ret}} = Q_{ret}/Q_F = (1 - Q_m)/Q_F.
\]

Table 8.2 shows the flow ratio for different operating conditions. The mass flow rate through the membrane
depends on the transmembrane pressure \( P_m = (P_F + P_{ret})/2 \) and the permeate flow rate for all the cases are constant since the transmembrane pressure is kept constant for all operating conditions.

Table 8.2: Flow ratio in studied test cases

<table>
<thead>
<tr>
<th>Cases</th>
<th>( Q_F ) Flow rate at inlet (m(^3)/s)</th>
<th>( Q_m ) Permeate flow rate (m(^3)/s)</th>
<th>( R_{ret} ) Flow ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6.67×10(^{-6})</td>
<td>1.6×10(^{-6})</td>
<td>0.76</td>
</tr>
<tr>
<td>2</td>
<td>6.67×10(^{-6})</td>
<td>1.6×10(^{-6})</td>
<td>0.76</td>
</tr>
<tr>
<td>3</td>
<td>6.67×10(^{-6})</td>
<td>1.6×10(^{-6})</td>
<td>0.76</td>
</tr>
<tr>
<td>4</td>
<td>6.67×10(^{-6})</td>
<td>1.6×10(^{-6})</td>
<td>0.76</td>
</tr>
<tr>
<td>5</td>
<td>1.00×10(^{-5})</td>
<td>1.6×10(^{-6})</td>
<td>0.84</td>
</tr>
<tr>
<td>6</td>
<td>1.33×10(^{-5})</td>
<td>1.6×10(^{-6})</td>
<td>0.88</td>
</tr>
</tbody>
</table>

The Stokes number is a measure of the response time of droplets to a flow. Droplets with the low Stokes number follow fluid streamlines whereas for large Stokes number, the droplet’s inertia dominates so that the droplet will continue along its initial trajectory. The Stokes number is defined as the ratio of droplet response time \( \tau_d \) to the system response time \( \tau_s \), i.e.

\[
\text{Stk} = \frac{\tau_d}{\tau_s} = \frac{d^2(\rho_C - \rho_D)U_z}{18\mu_D D_m}.
\]  

(8.10)

Figure 8.11 illustrates the effect of the Stokes number on the grade efficiency for different values of the Reynolds number. The solid and dotted lines represent the total grade efficiency \( G_T \) and the reduced grade efficiency \( G_R \) respectively. The Stokes number is normalized by \( \text{Stk}_{50} \) defined as

\[
\text{Stk}_{50} = \frac{d_{50}^2(\rho_C - \rho_D)U_z}{18\mu_D D_m}.
\]  

(8.11)
The Stokes number is a function of the droplet size and the Reynolds number. The total grade efficiency ($G_T$) for the Reynolds numbers of 1415, 2122, and 2822 decreases to values of 0.76, 0.84, and 0.88, respectively. It means that for very small Stokes numbers the vortex separation is ineffective, i.e., the separation is done by the membrane only. The reduced grade efficiency ($G_R$) graphs represent the efficiency of the vortex separation. Figure 8.11 also shows that the efficiency of vortex separation (i.e. reduced grade efficiency) exponentially increases with an increase in the Stokes number. The reduced grade efficiency (as a function of the Stokes number) is directly related to the Reynolds number for $\frac{Stk}{Stk_{50}} < 1$ and the opposite trend is observed for $\frac{Stk}{Stk_{50}} > 1$. For $\frac{Stk}{Stk_{50}} < 1$, the higher shear stress due to an increase in the Reynolds number may sweep the droplets from the membrane surface and increase the reduced grade efficiency. The swirling strength reduces with an increase in the Reynolds number (Fig. 8.7b). This reduced swirling effect on droplets with $\frac{Stk}{Stk_{50}} > 1$ can lessen the reduced grade efficiency.
Figure 8.11: Effect of the Reynolds number on the grade efficiency for the CFF system with rotation at 1500 rpm. Solid and dotted lines represent the total and reduced grade efficiencies, respectively. The reduced grade efficiency is estimated by subtracting the effect of flow ratio from the total grade efficiency. Reduced grade efficiency decreases because of the reduction of swirling strength with an increase in the Reynolds number.
Figure 8.12: Effect of the Swirl number on the grade efficiency for the CFF system with \( \text{Re} = 1415 \). Higher swirling strength produces larger centrifugal force, enhances the radial pressure gradient, and reduces the radial outward velocity component leading to a significant decrease in grade efficiency.

The effect of the Swirl number on the grade efficiency of droplets is shown in Fig. 8.12. The total grade efficiency for all the Swirl numbers of 0.75, 1.5, and 2.25 approached 0.76 for very small Stokes numbers. It indicates that the separation of droplets corresponding to very small Stokes number is not influenced by the rotation of the membrane. However, the rotation greatly affects the grade efficiency of droplets with larger Stokes number. Fig. 8.12 also shows that the reduced grade efficiency increased exponentially with an increase in the Stokes number. The reduced grade efficiency is directly related to the Swirl number, i.e., \( G_R \) increases with an increase of the Swirl number. The higher centrifugal force as well as the increase wall shear stress (Fig. 8.9)
due to higher swirling effect decreases the accumulated of droplets on the membrane surface and increases the grade efficiency. The 100% grade efficiency, or complete separation, for $S = 0.75$, $1.5$ and $2.25$ (for $Re = 1415$) is achieved for droplet size of $58 \mu m$, $27 \mu m$ and $18 \mu m$, respectively.
CHAPTER 9

MODELING OF DISPERSED DROPLETS AND COUPLED MULTIPHASE AND POPULATION BALANCE SIMULATION

9.1. Introduction

Velocity fields encountered in a hydrocyclone are very complex and highly turbulent. Depending on the feed flow rate, and the geometry of the hydrocyclone, the feed Reynolds number can reach values higher than 70,000. A natural question is to ask what extent the high feed Reynolds number causes shearing of the oil droplets. Droplets breakup and coalescence are processes that are strongly dependent on collision rates in the fluid which are dependent on turbulence intensity. Both processes typically occur simultaneously although one will be dominant over the other, sometimes by several orders of magnitude. If a droplet does not break apart, as in a localized low shear zone, the net effect of turbulence is to increase collisions between droplets which may lead to coalescence, depending on the trajectories and collision rate. The net effect of this shear rate distribution is to cause some droplets to break and others to coalesce with one another. Thus, an emulsion that undergoes shear experiences both breaking and coalescing processes simultaneously, albeit to different degrees depending on the shear intensity and flow configuration. Because of the breakage and coalescence of dispersed droplets, the distribution of droplet sizes and their relative volume fraction changes in the flow field; this is required to be incorporated in the numerical simulation of multiphase.
The simultaneous events of droplets coalescence and breakage are quantified using population balance method. This chapter addresses a mathematical model for the breakage and coalescence of dispersed droplets, a solution method of population balance model, and the coupling of population balance model with the multiphase flow phase flow theory.

9.2. Mechanism of droplets coalescence

A collision between two dispersed droplets is usually caused by their relative motion which may occur due to a motion induced by

i) turbulent fluctuations in the continuous phase;

ii) mean velocity gradient in the flow; and

iii) buoyancy arising from rise velocities, wake interactions or helical/zigzag trajectories.

In produced water systems the typical range of dispersed droplet size is 1-100 microns. On the basis of the turbulent length scales for this flow field, the relative motion of dispersed droplets having the typical sizes 1-100 microns mostly induced by turbulent fluctuations. In this case, the effects (ii) and (iii) are insignificant and neglected in this study. In a turbulent flow field, the local velocity fluctuations enhance the rate of collisions and thereby increase the coalescence frequency (Lee et al., 1987). The coalescence frequency, \( \Gamma \) depends on the collision frequency, \( h \) and the efficiency of coalescence, \( \lambda \) (Coulaloglou and Tavlarides, 1977). For a binary collision (two droplets), the coalescence rate per unit volume (#coalescence/m\(^3\)/s) of dispersion between droplets of diameters \( d_1 \) and \( d_2 \) is given by
\[ \Gamma(d_1, d_2) = h(d_1, d_2)\lambda(d_1, d_2) \] (9.1)

The coalescence of dispersed droplets is modeled as occurring in three steps (Lee et al., 1987):

i) Two droplets approach to within a distance of \( h_i \) (initial film thickness). In this step the formation of a new doublet, which can either coalesce or separate into the original single droplets under the influence of fluctuating turbulent force, occurs (Fig. 9.1).

ii) The liquid film between the droplets drains out to a critical thickness of \( h_f \)

iii) The thin liquid film ruptures and results in a successful coalescence.

![Figure 9.1: Illustration of coalescence and separation of dispersed droplets. Here, n1, n2, nd, and nc are the number concentration per unit volume of the monomers, the doublet (coagulation of two droplets), and the coalesced droplet, respectively. h is the collision rete for the formation doublet, \( k_d \) is the rate of dissociation of the doublet, and \( \lambda \) is the rate of coalescence of the doublet. [After Narsimhan (2004)]

The value of initial film thickness \( h_i \) is not well known. Factors affecting the initial film thickness are: viscosity and interfacial tension of liquid, sizes of colliding droplets and their approach velocity. The initial film thickness Lee et al. (1987) mentioned that the initial film thickness, \( h_i \) is within the range of 10-100 \( \mu \)m. Marrucci et al. (1972) estimated \( h_i \) as 10 \( \mu \)m and Kirkpatrick and Lockett (1974) used \( h_i = 100 \mu \)m.
However, Tsouris and Tavlarides (1994) considered the initial thickness as $h_i \approx 0.1r_{eq}$ where $r_{eq} = d_1d_2/(d_1 + d_2)$ is the equivalent radius of two colliding droplets. Chesters (1975) given an estimate for $h_i$, assuming a substantial deformation of droplets caused by a high pressure generated between approaching spherical droplets, as $h_i = \frac{\rho_cV^2r_{eq}^2}{4\sigma}$ where $V$ is the approach velocity and $\sigma$ is the interfacial tension of droplets. Venneker et al. (2002) considered the approach velocity, $V$ equal to the relative velocity $u_{rel}$ which is given in Eq. (9.6). Tsouris and Tavlarides (1994) mentioned that the film rupture occurs when the final thickness, $h_f$ becomes smaller than 0.05 µm. However, $h_f$ has generally been agreed upon as 0.01-0.1µm (Lee et al., 1987). Venneker et al. (2002) adopted Chesters (1991) approach to calculate the $h_f$ that leads to the expression $h_f = \left(\frac{A_{Hr_{eq}}}{8\pi\sigma}\right)^{1/3}$ where $A_H$ is the Hamaker constant which ranges between $10^{-20}$ and $10^{-19}$ joules (Lee et al., 1987).

9.2.1. Collision frequency (h)

Collision frequency can be represented as the effective volume swept by a moving dispersed droplet per unit time (Venneker et al., 2002). If a droplet moving with a relative velocity $u_{rel}$ sweeps the collision tube (Fig. 9.2) per unit time then the rate of effective volume swept by the moving bubble is $h = Su_{rel}$ where $S$ is the cross-sectional area of the collision tube and $h$ is called a specific collision frequency and its unit is $(m^3/s)$. Instead of the representation by swept volume Kuboi et al. (1972b) and Lee et al. (1987) expressed the collision frequency as a number frequency ($\#$collisions/m$^3$/s) by multiplying the number density of dispersed droplet with $h$. For a binary collision the collision frequency can be defined as $h = S_{12}u_{rel}n_1n_2$ where $n_1$ and $n_2$ are the number
densities (number of droplets per unit dispersed phase volume) of dispersed droplets of diameters $d_1$ and $d_2$. $S_{12}$ is the mutual collision cross sectional area and can be expressed as $S_{12} = \frac{\pi}{4} (d_1 + d_2)^2$.

Figure 9.2: Collision tube of a bubble moving with a relative speed $u_{rel}$.

The relative velocity $u_{rel}$ between the dispersed droplets is calculated usually assuming that the velocity of colliding droplets is same as equal sized eddy (Tsouris and Tavlarides, 1994; Coulaloglou and Tavlarides, 1977; Lee et al., 1987; Luo, 1993). The relative velocity is defined as $u_{rel} = \sqrt{u_{t1}^2 + u_{t2}^2}$ where $u_t$ is the velocity of eddy with characteristic size $d$ which is defined by Batchelor (1951) and Kuboi et al. (1972a) as $u_t^2 = C_1(\varepsilon_c d)^{2/3}$ where $\varepsilon_c$ is the turbulent energy dissipation of continuous phase and is $C_1$ is a constant. So, the collision frequency for a binary collision can be calculated as

$$h(d_1, d_2) = C_2 (d_1 + d_2)^2 \left( \frac{d_1^3 + d_2^3}{d_1^3 + d_2^3} \right)^{1/2} \varepsilon_c^{1/3} n_1 n_2$$
Bapat and Tavlarides (1985) and Coulaloglou and Tavlarides (1977) incorporated the effect of volume fraction of dispersed phase and suggested the relation for collision frequency as

\[ h(d_1, d_2) = C_3 \frac{1}{(1+\phi)} (d_1 + d_2)^2 \left( d_1^2 + d_2^2 \right)^{1/2} \varepsilon_c^{1/3} n_1 n_2 \]  
(9.2)

The value of imperial constant \( C_3 \) is \( 1.9 \times 10^{-3} \) (Bapat and Tavlarides, 1985). In the Eq. (9.6), the length scales of colliding droplets are considered to be in the inertial subrange, i.e. \( \eta < d_1, d_2 < l_T \) and the velocities of droplets are same as the velocity of equal-sized eddies. For droplets larger than the inertial subrange and/or the relative velocity of droplets is not same as the eddy velocity, the collision frequency can be calculated using the Kamp et al. (2001) approach as

\[ h(d_1, d_2) = C_t \frac{\pi}{(d_1 + d_2)^2} (\varepsilon_c d_1)^{1/3} n_1 n_2 \text{ for } d_1 < l_T; \text{ } d_2 > l_T \]  
(9.3)

where \( C_t \) represents the ratio between the dispersed phase velocity fluctuations and the continuous phase velocity fluctuations and it is given by Kamp et al., (2001) as

\[ C_t^2 = \frac{9 + 72 \beta \nu_c l_T / (d^2 u')} {1 + 72 \beta \nu_c l_T / (d^2 u')} \]  
where \( \beta \) equals to 0.6, \( \nu_c \) is the kinematic viscosity of continuous phase, and \( u' \) is the r.m.s value of fluctuating continuous phase velocity which can be expressed as

\[ u' = (\varepsilon l_T)^{1/3} \].

For the inertial subrange the turbulent energy is sufficiently high and the drop sizes lie within the range of local isotropy. For viscous subrange, the relative motion of droplets and the rate of collisions depends on the viscosity, since the turbulent energy in the small scale eddies is insignificant. Narsimhan (2004) developed a model for the viscous subrange of dispersed droplets as
\[ h(d_1, d_2) = \frac{4\pi}{5 \nu_c^{1/2} e^{-1/2} \left( \frac{8}{(d_1 + d_2)^2} \right) ^{1/2} \frac{3}{7 \nu_c^{1/2} \eta^{-1/3}}} n_1 n_2 \quad \text{for } R < \eta \quad (9.4) \]

where \( R = (d_1 + d_2)/2 \).

9.2.2. Coalescence efficiency (\( \lambda \))

Three different models are usually used for the calculation of coalescence efficiency. These are energy model, critical approach velocity model, and the film drainage model (Liao and Lucas, 2010). The film drainage model is widely used for the case where the Weber number based on the equivalent diameter of dispersed droplets and the relative velocity between droplets is much less than unity (Kamp et al., 2001) i.e.,

\[ \frac{\rho c u^2_{rel}}{\sigma} \ll 1 \quad \text{and} \quad u_{rel} \ll 1. \]

The colliding droplets either coalesce or bounce away depending on the approach velocity and the time required for the liquid film between the two droplets to be drained out until a critical film thickness, \( h_f \) is reached (Coulaloglou and Tavlarides, 1977). By assuming the contact time is a random variable and the drainage time is not randomly distributed Coulaloglou (1975) expressed the coalescence efficiency as

\[ \lambda(d_1, d_2) = \exp \left( -\frac{t_d}{t_c} \right) \quad (9.5) \]

where \( t_c \) and \( t_d \) are called, respectively, the contact time and film drainage time. Once droplets collide they stay together for a certain time, called contact time, \( t_c \). The time needed for the film drainage is called drainage time, \( t_d \). Coalescence occurs if \( t_d < t_c \) (Tsouris and Tavlarides, 1994).
9.2.2.1. Drainage time ($t_d$)

On the basis of the rigidity and mobility of the contact interfaces of dispersed droplets there are four regimes of film drainage found in the literatures (Fig. 9.3).

**Non-deformable rigid spheres:** When dispersed droplets are highly viscous in comparison with the continuous phase and very small, they behave like rigid spherical particles. The drainage time for non-deformable rigid spherical droplets is derived by using Poiseuille relation as (Jeffreys and Davies, 1971; Davis et al. 1989; Chesters, 1991; Tsouris and Tavlarides 1994)

$$t_d = \frac{3\pi \mu_c}{2F} r_{eq}^2 \ln\left(\frac{h_i}{h_f}\right) \quad (9.6)$$

**Deformable droplets with immobile interfaces:** The immobility of the interface is applicable only to a system with extremely high viscosity of dispersed phase. The mobility of the interface of a dispersed droplet can be characterized by the mobility factor, $m \equiv \mu_r^{-1} \sqrt{r_{eq}/(2h_i)}$ (Davis et al., 1989). For an immobile interface the $m << 1$ which yields $\mu_r = \frac{\mu_D}{\mu_c} \gg \frac{1}{\sqrt{r_{eq}/(2h_i)}}$ (Davis et al., 1989; Tsouris and Tavlarides, 1994). As initially spherical droplets having immobile interface approach each other under the action of constant compressing force, the pressure at the center of the intervening film rises. The droplets interfaces become flatten when this pressure become the order of $2\sigma/r$ (Chasters, 1991). In this classification of drainage regimes, the film drainage is controlled by a viscous thinning and the velocity profile in the film is parabolic (Liao and
The drainage time for deformable droplets with immobile interfaces is given by (Chappelear, 1961; Coulaloglou and Tavlarides, 1977)

\[
t_d = \frac{3\mu_c F}{16\pi\sigma^2 r_{eq} \left( \frac{1}{h_i^2} - \frac{1}{h_f^2} \right)} \tag{9.7}
\]

Figure 9.3: Four regimes of film drainage during drop-drop coalescence

**Non-deformable droplets with partially mobile interfaces:** When \( \mu_r = O\left(\sqrt{r_{eq}/2h_i}\right) \), droplets offer significant resistance to the radial flow of the intervening film, but the resistance is not so high that they behave like rigid spheres (Davis et al., 1989). In this case the mobility factor is the order of one (m=O(1)) and the droplet interface is called partially mobile interface. The regime of non-deformable droplets with partially mobile interface is applied for \( 10^{-2} < \mu_r < 10^2 \) (Cosijnse, 1994). By using the approach applied by Davis et al. (1989), Tsouris and Tavlarides (1994) derived the following expression for the non-deformable droplets with partially mobile interface

\[
t_d = \frac{3\pi \mu_c r_{eq}^2}{2F} \xi \tag{9.8}
\]
where \( \xi = 1.872 \ln \left( \frac{\sqrt{h_i} + 1.378q}{\sqrt{h_f} + 1.378q} \right) + 0.127 \ln \left( \frac{\sqrt{h_i} + 0.312q}{\sqrt{h_f} + 0.312q} \right) \) and \( q = \mu_r^{-1} \sqrt{r_{eq}/2} \)

**Non-deformable droplets with fully mobile interfaces:** If the viscosity of dispersed droplets is much smaller than the continuous phase \( (\mu_r \ll \sqrt{r_{eq}/(2h_i)}) \) then the resistance offered by the droplets is insignificant to resist the radial flow of intervening film (Chasters, 1991, Davis et al., 1989). In this case the mobility factor is much higher than one \( (m >> 1) \). On the basis of the resisting hydrodynamic force given by Davis et al. (1989), Tsouris and Tavlarides (1994) derived the following expression for the non-deformable droplets with fully mobile interface

\[
t_d = \frac{16.5\mu_d}{2\sqrt{2F}} r_{eq}^{3/2} (h_i^{1/2} - h_f^{1/2})
\]  

(9.9)

**9.2.2.2. Compressing force, F**

The compressing force, \( F \) used in all the above equations for drainage time calculation (Eq. (9.6-9.9)) is assumed constant during the film drainage process. The force \( F \) is usually assumed to be proportional to the mean-square velocity difference at either ends of the eddy with a size of equivalent diameter of dispersed droplets (Coulaloglou and Tavlarides, 1977; Tsouris and Tavlarides, 1994). For local isotropy, the turbulent force given by Narsimhan (2004) is

For inertial subrange

\[
F_t = 2\pi \rho_c \overline{e^2}^{3/2} r_{eq}^2 \left( \frac{d_1 + d_2}{2} \right)^{2/3} \quad \text{for } R > \eta
\]  

(9.10a)
and for viscous subrange

\[ F_t = \frac{\pi \rho_c^2 \epsilon^* r_{eq}^2 (d_1 + d_2)^2}{\mu_c} \quad \text{for } R < \eta \]  

(9.10b)

where, \( \epsilon^* \approx \epsilon \left( \frac{\rho_c \mu_c}{\rho_s \mu_s} \right)^3 \), \( \mu^* = \mu_c \left[ 1 + 2.5 \phi \left( \frac{\mu_d + 0.4 \mu_c}{\mu_d + \mu_c} \right) \right] \), \( \rho^* = \rho_d \phi + (1 - \phi) \rho_c \). \( \epsilon^* \), \( \mu^* \) and \( \rho^* \) are the energy dissipation, dynamic viscosity and density of a mixture, respectively and \( \phi \) is the volume fraction of dispersed phase. The net force of interaction experienced by the droplet pair is the sum of the turbulent force and colloidal force. The colloidal force has two effects: van der Waals force and electrostatic double-layer repulsive force. The electrostatic force depends on the charge of droplets which is ignored here. So, the colloidal force given by Narsimhan (2004) is

\[ F_c = \frac{A H r_{eq}}{12 h_i} \]  

(9.11)

Therefore, the net interaction force is \( F = F_t + F_c \).

### 9.2.2.3. Contact time (t<sub>c</sub>)

Kamp et al. (2001) defined the contact time as the interval between the onset of film formation and the moment at which the bubbles begin to rebound. By assuming a balance between the increasing surface free energy and the corresponding reduction in the kinetic energy of the system, they derived the contact time as

\[ t_c = \frac{\pi}{\sqrt{6}} \left( \frac{\rho_c C_{VM} r_{eq}^3}{\sigma} \right)^{1/2} \]  

(9.12)

where the value of \( C_{VM} \) varies from 0.5 to 0.803 depending on the droplets’ ratio and their distance. On the basis of the isotropic turbulence, the contact is given by Levich (1962), Coulaloglou and Tavlarides (1977) and Lee et al. (1987) as

\[ t_c = (d_1 + d_2)^{2/3} \epsilon_c^{-1/3} \]  

(9.13)
9.3. Mechanisms of droplet breakage

Breakup of a droplet in a turbulent dispersion depends on the balance between two stresses: external stress to the droplet due to turbulence in the continuous phase and internal stress of the fluid inside the droplet. The external stress deforms the droplet. The internal stress acts to restore the spherical shape of the drop and minimize the interfacial area. If the external stress due to turbulence overcomes the restorative stress, the drop will break apart. The balance between the stresses leads to the prediction of a maximum stable particle diameter (Kocamustafaogullari and Ishii, 1995; Revankar, 2001). The mechanism of droplet breakup can be classified into following categories (Liao and Lucas, 2009):

i) Breakup due to turbulent fluctuation and collision
ii) Breakup due to viscous shear stress
iii) Breakup due to shearing-off process, and
iv) Breakup due to interfacial instability

Since the sizes of dispersed droplets in the produced water are very small and the flow field in the hydrocyclone is highly turbulent, the breakup of dispersed droplets is dominated by the turbulent fluctuation. The other three mechanisms (ii to iv), which are related to large droplets and to presence of velocity gradient across/around the interface of droplets, are neglected in this study. The breakage rate of a droplet due to the turbulent shearing stress can be defined as Breakage rate = \( g(d_j) \times \beta(d_i, d_j) \) where \( g(d_j) \) is the breakage frequency and \( \beta(d_i, d_j) \) is the daughter droplet distribution function. Here, \( d_j \) and \( d_i \) are the diameter of mother and daughter droplets, respectively.
9.3.1. Breakage frequency

A spherical droplet deforms due to the action of external flow caused by turbulent eddies. When the turbulent shear stress is dominating over the restorative stress, the spherical droplet continuously deforms and its shape becomes slender. After a critical limit of deformation, the deformed droplet breaks up into smaller droplets. The fraction of droplets that have turbulent shear stress greater than restorative stress is characterized by breakage probability. The breakage frequency is calculated by normalizing the breakage probability with breakage time. The breakage time is calculated by assuming that the size of daughter droplet is equal to the size of an eddy of isotropic turbulence. The mathematical model for the breakage frequency is given by (Coulaloglou and Tavlarides, 1977) as

\[
g(d_j) = C_4 \frac{\varepsilon^{1/3}}{(1+\phi)d_j^{2/3}} \exp \left( -C_5 \frac{\sigma(1+\phi)^2}{\rho_d \varepsilon d_j^2} \right)
\]

(9.14)

where the \( C_4 \) and \( C_5 \) are empirical coefficients and their values are given in Table 9.1.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_4 )</td>
<td>0.00487</td>
<td>0.00487</td>
<td>0.01031</td>
<td>0.00481</td>
</tr>
<tr>
<td>( C_5 )</td>
<td>0.0552</td>
<td>0.08</td>
<td>0.06354</td>
<td>0.08</td>
</tr>
</tbody>
</table>
9.3.2. Daughter particle probability distribution function

It can be seen from Fig. 9.4 that the size of the droplets that are formed during the breakage process, the so-called daughter size distribution, has to be known. There are several daughter particle probability distribution functions are available in the literature. The beta function used here is given by Bapat and Tavarides (1985) as

$$\beta(x_i, x_j) = 30 \left( \frac{d_i^3}{d_j^3} \right)^2 \left( 1 - \frac{d_i^3}{d_j^3} \right)^2$$  \hspace{1cm} (9.15)

where $d_j$ is the mother particle diameter that undergoes a binary breakage into daughter particle of diameters $d_i$ and $(d_j^3 - d_i^3)^{1/3}$. 

Figure 9.4: Mechanism of droplet breakage due to turbulent shear stress. $d$ is the diameter of mother droplet and $d_i$ is the diameter of daughter droplet. (Adopted from Komrakova and Derksen, 2012).
9.4. Effect of flow and material properties on the coalescence rate

Effect of the energy dissipation, size of reference droplet and volume fraction of dispersed phase, interfacial tension of dispersed phase on the rate of collision, contact and drainage time, critical film thickness, coalescence efficiency and rate of coalescence are investigated here. In the numerical calculation, the size of droplet $d_1$ ranges from 1 to 100 microns which are considered to be collided with a reference droplet of size $d_2$. The properties of dispersed phase are: $\mu_d = 0.0033$ Pa.s, $\rho_d = 850$ kg/m$^3$ and interfacial tension $\sigma = 0.02$ N/m$^2$. The initial thickness of the film between droplets to be drain out is considered constant which is 10 microns. The viscosity ratio satisfies the condition of $10^{-2} < \mu_r = \frac{\mu_d}{\mu_c} = 18 < 10^2$ and Eq. (9.8) is used for the film drainage. The contact time is calculated using Eq. (9.13).

9.4.1. Effect of turbulent energy dissipation

The effect of turbulent energy dissipation on the specific collision frequency, drainage and contact time, coalescence efficiency, and the specific coalescence frequency are shown in Fig. 9.5. The size of reference droplet ($d_2$) is 100 microns and the colliding droplet ranges from 1 to 100 microns. The volume fraction of dispersed phase is 0.01. Three different turbulent energy dissipations are considered: 1, 10 and 100 m$^2$/s$^3$. The critical film thickness profile is not presented here because it is independent of the turbulent energy dissipation. Figure 9.5 shows that the specific collision frequency increases with an increase in the energy dissipation rate. Higher energy dissipation rate contribute more on the relative motion of dispersed droplets which thereby increases the chance of collision. Both the drainage time and contact time are inversely related with the
energy dissipation rate. Higher energy dissipation increases the compressing force between two droplets which thereby force the thin film to be drained out quickly. Moreover, a higher viscous resistance due to a higher dissipation rate rebound the droplets and reduces the contact time. Both the coalescence efficiency and the specific coalescence frequency increases with an increase in the energy dissipation rate.

9.4.2. Effect of reference droplet size

The effect of reference droplet size $d_2$ on the specific collision frequency, final film thickness, drainage and contact time, coalescence efficiency and specific coalescence frequency are shown in Fig. 9.6. The turbulent energy dissipation is $100 \text{ m}^2/\text{s}^3$ and the volume fraction of dispersed phase is 0.01. Three different diameters of reference droplet are considered which are 10, 50 and 100 microns. The colliding droplets $d_1$ ranges from 1-100 microns. Figure 9.6 shows that the specific collision frequency gradually increases with an increase in the colliding droplet size and the reference size as well. A larger droplet has a higher kinetic energy which thereby increases the collision frequency. The critical film thickness dramatically increases with droplet size for smaller droplets and rete becomes flatten for larger droplet size. The same trend appears for the size of reference droplets. An opposite trend is observed between the drainage and contact time. A larger droplet yields a smaller drainage time and a greater contact time. Based on the Eq. (9.5) the coalescence efficiency increases inverse-exponentially with a decrease in the ratio of drainage time to contact time which is observed in the coalescence efficiency versus droplet size graph. The specific
coalescence efficiency gradually increases with an increase in the size of colliding droplets and as well as the size of reference droplets.
Figure 9.5: Effect of energy dissipation rate on the specific collision frequency, drainage and contact times, coalescence efficiency, and specific coalescence frequency for $d_2 = 100$ microns. ($\varepsilon = 1 \text{ m}^2/\text{s}^3$; $\varepsilon = 10 \text{ m}^2/\text{s}^3$; and $\varepsilon = 100 \text{ m}^2/\text{s}^3$).
Figure 9.6: Effect of the reference droplet size ($d_2$) on the specific collision frequency, drainage and contact times, coalescence efficiency, and specific coalescence frequency for $\varepsilon = 100 \text{ m}^2/\text{s}^3$. (--- $d_2=100 \text{ \mu m}$; -- $d_2=50 \text{ \mu m}$; and - - $d_2=25 \text{ \mu m}$).
9.5. Population balance method (PBM)

Consider a population of droplets distributed according to their size $x$ which we shall take to be the volume of the droplets and allow it to vary between 0 and $\infty$. The droplets are uniformly distributed in space so that the number density probability ($n$) is independent of external coordinates. Further, we assume for the present that the environment does not play any explicit role in droplet behavior. The population balance equation (PBE) represents the dynamics of the droplet size distribution due to a process involving continuous interactions between individual droplets (such as coalescence and breakup). PBE is a balance equation of the number density probability ($n$) of some practical property. In practical situations, one is generally interested in the number density $N_i$ (number of $i^{th}$ sized particle per unit volume of continuous phase) instead of number density probability ($n$). The number density $N_i$ is the number of particles of a class $i$ per unit volume itself. The general form of the PBE in terms of number density $N_i$ of particles is given as

\[
\frac{\partial N_i}{\partial t} + C_i + G_i = Bc_i - Dc_i + Bb_i - Db_i
\]  

(9.16)

where $C$ is the rate of change of droplet population in a given domain due to convection. The $C$ is zero if the rate of inlet and outlet of droplets to and from a domain is equal. $G$ is the growth of droplets in the domain (such as nucleation of bubble). $Bc$ and $Dc$ is the birth and death of particle due to coalescence. $Bb$ and $Db$ is the birth and death of particle due to breakage. There are different methods available to solve the PBE. In some particular cases, it is possible to solve spatially homogeneous PBE analytically but in
general the numerical solution is required because of the nature of PBE as it is an Integra-

partial differential equation. Computationally speaking, this procedure is more expensive

than solving the PBE by discretization techniques. The most commonly used techniques

are Monte Carlo method, the Methods of Classes (MC) and the method of moments.

Hereafter, in particular interest and because of its relevance in CFD applications, the

method of classes is used for this calculation. Two common discretization techniques for

the method of classes that are used for the solution of PBE are: (1) linear grid
discretization, and (2) geometric grid discretization.

9.5.1. Linear grid discretization

Hidy and Brock (1970) used a linear grid for solving the PBE. They considered

particle population in a size range \((V_i, V_{i+1})\) to be represented by a characteristic size \(x_i\).

A schematic of linear grid discretization is shown in Fig. 9.7. Here, \(V_1\) and \(V_{nc}\) are the

minimum and maximum volume of droplets in the computational domain; \(nc\) is the

number of class; \(V_i = i^*V_1\) and \(V_i = (x_{i-1} + x_i)/2\).

![Figure 9.7: A representation of linear grid used for solving the PBE. Here, \(V_i = (x_{i-1} + x_i)/2\).](image)
For the linear grid as shown in Fig. 9.7, the discretized equations for the birth and death due to coalescence and breakage are given by

\[ Bc_i = \frac{1}{2} \sum_{j=1}^{i-1} \Gamma_{j,i-j} N_j N_{i-1} \]  
\[ (9.17\text{a}) \]
\[ Dc_i = N_i \sum_{j=1}^{nc} \Gamma_{ij} N_j \]  
\[ (9.17\text{b}) \]
\[ Bb_i = \sum_{j=i+1}^{nc} \nu_j g_j \beta(x_i, x_j) N_j \]  
\[ (9.17\text{c}) \]
\[ Db_i = g_i N_i \]  
\[ (9.17\text{d}) \]

**9.5.1.1. Coalescence**

The term \( \Gamma_{j,i-j} \) in Eq. (9.17a) is the specific coalescence frequency due to collision between the droplets of size \( x_j \) and \( x_{i-j} \) which is calculated using the Eq. (9.1). The unit of \( \Gamma_{j,i-j} \) is \( m^3/s \). \( N_j \) and \( N_{i-j} \) is the number density of droplets in the \( j \)th and \( (i-j) \)th class, respectively. Because of linear grid spacing, a new droplet of size \( x_i \) is formed by the coalescence of droplet of size \( x_j \) and \( x_{i-j} \); the droplet of size \( x_{i-j} \) must be represented by \( (i-j) \)th characteristic size. In this process, the formation new droplet of size \( x_i \) is considered as the birth of a droplet by coalescence in the \( i \)th class. The Eq. (9.17a) is multiplied by \( \frac{1}{2} \) because the collision between \( j \)th and \( (i-j) \)th droplets counted twice when the index \( j \) varies from 1 to \( (i-1) \). The Eq. (9.17b) is the death rate of droplet in the \( i \)th class due to coalescence of droplet of size \( x_i \) with any other droplet. When a particle of size \( x_i \) coalescence with any droplet of size \( x_j \), then the resulting size is distributed to the \( (i+j) \)th class. It indicates that the droplet of size \( x_i \) left the \( i \)th class by coalescence process which thereby represented as a death of droplet in the \( i \)th class. The index of \( j \) in \( Dc_i \)
varies from 1 to nc, because the $i^{th}$ droplet can coalescence with any droplet in the range (1, nc) which thereby yields a death droplet from the $i^{th}$ class.

9.5.1.2. Breakage

The birth rate of droplet in the $i^{th}$ class due to a breakage of droplet in the $j^{th}$ class is given by Eq. (9.17c). The $g_j$ is the breakage rate of a droplet in the $j^{th}$ class. The breakage is assumed to be binary, so that $v_j$ is always 2. The $\beta(x_i, x_j)$ in the Eq. (9.17c) is the probability density function of daughter size, which is given in Eq. (9.15). The index of $j$ in the Eq. (9.17c) varies from $i+1$ to nc, because the breakage of any droplet of size less that $(i+1)^{th}$ class does not result a daughter droplet of size that can be equal to $x_i$. The Eq. (9.17d) represents the death rate of droplet from the $i^{th}$ class due to breakage process.

9.5.1.3. Difficulties in the linear discretization of the PBE

The linear discretization technique maintains internal consistency, provides good resolution at the small size, and yields extremely accurate solution for the complete size distribution as well as its moments. The major disadvantage of this technique is that it requires a large number of size ranges to cover the entire size range with acceptable resolution, and therefore incurs very high computational costs. For example, let us consider two possible size distributions. One size distribution ranges from 5 to 100 microns and other one range from 1 to 100 microns.
For 5-100 micros droplets:

\[ d_1 = 5 \text{ micron and} \]
\[ d_{nc} = 100 \text{ microns} \]

Total number of grid points required is

\[ n_c = \frac{v_{nc}}{v_1} = \left( \frac{d_{nc}}{d_1} \right)^3 = \left( \frac{100}{5} \right)^3 = 8000 \]

For 1-100 micros droplets:

\[ d_1 = 1 \text{ micron and} \]
\[ d_{nc} = 100 \text{ microns} \]

Total number of grid points required is

\[ n_c = \frac{v_{nc}}{v_1} = \left( \frac{d_{nc}}{d_1} \right)^3 = \left( \frac{100}{1} \right)^3 = 10^6 \]

These two simple calculations indicate that the number of grid points required for the linear grid discretization method varies significantly when the minimum size of droplet getting smaller. For a wide range of droplet size, discretization of the PBE using the linear grid is not a good choice because of very high computational cost.

### 9.5.2. Geometric grid discretization of the PBE

The discretization of PBE using the geometric grid (Fig. 9.8) is very robust because it requires less number of grid points and an arbitrary grid refinement is possible. The grid size in the geometric grid can be defined as 

\[ x_{i+1} = 2^{1/q} x_i \text{ where } q = 1, 2, 3 \ldots \]

Droplets having sizes between \( V_i \) and \( V_{i+1} \) are denoted as the \( i^{\text{th}} \) class droplet. The droplet population in this size range is represented by the characteristic size \( x_i \) (also called grid point), such that \( V_i < x_i < V_{i+1} \). The upper and lower bounds of the \( i^{\text{th}} \) class can be defined as 

\[ V_{i+1} = (x_{i+1} + x_i)/2 \text{ and } V_i = (x_{i-1} + x_i)/2, \] respectively. For the droplet size ranges
from 1 to 100 microns, the number of geometric grid point required for the q = 2 in 
\( x_{i+1} = 2^{1/q} x_i \) is:

\[
nc = \frac{1}{\ln(2^{1/2})} \ln \left( \frac{V_{nc}}{V_1} \right) = \frac{1}{\ln(2^{1/2})} \ln \left( \frac{d_{nc}}{d_1} \right)^3 = \frac{1}{\ln(2^{1/2})} \ln \left( \frac{100}{1} \right)^3 = 40
\]

This simple calculation shows that the number of grid point required in the geometric 
grid system is only 40, which was 1 million in the linear grid system. So the geometric 
grid is very efficient. However, the solution procedure in the geometric grid system is 
much more complex than that of linear grid system.

![Diagram](image)

Figure 9.8: The geometric grid used for discretization of the PBE. Here, \( V_i = (x_{i-1} + 
\]
x_i)/2.

For the geometric grid as shown in Fig. 9.8, the discretized equations for the birth and 
death due to coalescence and breakage processes are given by

\[
Bc_i = \sum_{k=1}^{nc} \sum_{j=k}^{nc} G_{k,j} N_k N_j, \quad (9.18a)
\]

\[
Dc_i = N_i \sum_{j=1}^{nc} I_{i,j} N_j, \quad (9.18b)
\]

\[
Bb_i = \sum_{k=i}^{nc} \gamma_j g_{k,j} \alpha_{k,j} N_k, \text{ and} \quad (9.18c)
\]

\[
Db_i = g_k N_k \quad (9.18d)
\]
9.5.2.1. Coalescence

The term $\Gamma_{kj}$ in Eq. (9.18a) is the specific coalescence frequency due to a collision between the droplets of size $x_k$ and $x_j$ which is calculated using the Eq. (9.1). The unit of $\Gamma_{kj}$ is m$^3$/s. $N_k$ and $N_j$ is the number density of droplets in the $k^{th}$ and $j^{th}$ classes, respectively. Due to the coalescence process, the formation of droplet of size $x_i$ occurs when a droplet of size $x_k$ coalescence with a droplet of size $x_i-x_k$. The volume of droplets in the $i^{th}$ and $k^{th}$ class ranges from $V_i$ to $V_{i+1}$ and $V_k$ to $V_{k+1}$, respectively. In the geometric grid spacing, if a droplet of size $x_k$ coalescence with a droplet of size $k_j$, then the resultant droplet of size $x_k+x_j$ may not be in the $(k+j)^{th}$ class. However, coalescence between any two droplets is considered as a birth of droplet in the $i^{th}$ class when the aggregated volume is in between $V_k$, and $V_{i+1}$. This criteria is satisfied by introducing the function $\xi_{kj}$, which is defined as

$$\xi_{kj} = \begin{cases} 
1 & \text{for } x_{i-1} < V_{ag} < x_{i+1} \text{ where } 2 \leq i \leq nc - 1 \text{ and } V_{ag} = V_k + V_j \\
0 & \text{otherwise}
\end{cases}$$

This function indicates that if the aggregated volume ($V_{ag}$) resulting from the coalescence of droplets of size $x_k$ and $x_j$ is within the range $(x_{i-1}, x_{i+1})$ then the fraction of the new droplet is stored in the $i^{th}$ class.

The function $f_{kj}$ in Eq. (9.18a) represents the fraction of aggregated droplet distributed in the $i^{th}$ class. In the case of linear grid ($x_i = ix_1$), the size of new aggregated droplet always exactly matches with one of the $x_i$’s. Thus, for the linear grid discretization, changes in any property corresponding to the aggregation of two droplets are exactly preserved. In the geometric grid, the new droplet does not match with any of
the characteristics sizes \((x_i)\). In that case, the new droplet is assigned to the nearby characteristic sizes such that the changes in properties (such as number, mass and momentum) due to aggregation events are exactly preserved in the PBE. The fraction of aggregated droplet \((f_{kj})\) distributed in the \(i^{th}\) class is calculated using the following function.

\[
f_{kj} = \begin{cases} 
\frac{x_{i+1} - V_{ag}}{x_{i+1} - x_i} & \text{for } x_i < V_{ag} < x_{i+1} \\
\frac{V_{ag} - x_{i-1}}{x_i - x_{i-1}} & \text{for } x_{i-1} < V_{ag} < x_i 
\end{cases}
\]

The calculation of death of droplets from the \(i^{th}\) class is the same as the linear grid technique which is described in the section 9.5.1.1.

**9.5.2.2. Breakage**

For the geometric grid, the birth rate of droplet in the \(i^{th}\) class due to a breakage of droplet of size \(x_k\) is given in Eq. (9.18c). Unlike the linear grid, the index of \(k\) starts from \(i\). The reason is that a binary breakage of a droplet from the \(i^{th}\) class can generate a new droplet with the size that belongs to the range \((V_i, V_{i+1})\). Similar to the coalescence process, the new droplet generated by the breakage process may not match with any characteristic size, \(x_i\)'s. In that case, the new droplet is distributed to the nearby characteristic classes. The term \(\alpha_{ik}\) in Eq. (9.18c) is used for the fractional distribution of new droplets to the nearby characteristic sizes. The \(\alpha_{ik}\) exactly preserve the number density and mass of droplets in the breakage kernel. The \(\alpha_{ik}\) is defined as
\[ \alpha_{ik} = \int_{x_{i-1}}^{x_i} \frac{v - x_{i-1}}{x_i - x_{i-1}} \beta_2(v, x_k) dv + \int_{x_i}^{x_{i+1}} \frac{x_{i+1} - v}{x_{i+1} - x_i} \beta_2(v, x_k) dv \]

The integral term can be evaluated analytically for a distribution function \( \beta_2(v, x_k) \). In this work, the one-dimensional integration for the \( \alpha_{ik} \) is performed numerically using the trapezoidal rule. For the numerical integration, the equation for the \( \alpha_{ik} \) can be expressed as

\[
\alpha_{ik} = \Delta x_1 \sum_{j=1}^{m} \frac{A(v_j) + A(v_{j+1})}{2} + \Delta x_2 \sum_{p=1}^{m} \frac{B(v_p) + B(v_{p+1})}{2}
\]

where

\[
\Delta x_1 = \frac{x_i - x_{i-1}}{m}
\]

\[
v_j = x_{i-1} + (j - 1)\Delta x_1
\]

\[
A(v_j) = \frac{v_j - x_{i-1}}{x_i - x_{i-1}} \beta_2(v_j, x_k)
\]

\[
\Delta x_2 = \frac{x_{i+1} - x_i}{m}
\]

\[
v_p = x_i + (j - 1)\Delta x_2
\]

\[
B(v_p) = \frac{x_{i+1} - v_p}{x_{i+1} - x_i} \beta_2(v_p, x_k)
\]

and

\[
\beta_2(x, x_k) = \frac{30}{x_k^2} \left( \frac{x}{x_k} \right)^2 \left( 1 - \frac{x}{x_k} \right)^2
\]

Here, \( m \) is the number of integration steps. For \( i = 1 \), the first part of the \( \alpha_{ik} \) is zero because \( x_1 \) is the minimum size of the droplets and there would be no droplet of size less
than $x_1$ generated due to breakage. The second part of the $\alpha_{ik}$ is reduced to zero for $i=k$, because a breakage of kth droplet cannot create a droplet of same size. Moreover for $i = nc$ (number of class), the second part of the $\alpha_{ik}$ is also zero because there is no droplet larger than $x_{nc}$. The rate of breakage $g_k$ is calculated using Eq. (9.14). The Eq. (9.17d) is the same as Eq. (9.18d).

9.5.3. Time discretization

In Eq. (9.16), the $dN_i/dt$ provides the rate of change of the number density $N_i$ at the $i^{th}$ class. To calculate both the transient and steady state number density the Eq. (9.16) needs to be integrated over the prescribed time period. This integration can be evaluated using Eulerian integration method as

$$N(i, t + \Delta t) = \int_t^{t+\Delta t} \frac{dN_i}{dt} \, dt' = N(i, t) + \frac{dN_i}{dt} \times \Delta t$$

9.6. Preliminary results using the PBM

The preliminary results are obtained using the PBM for the volume averaged turbulent energy dissipations calculated for the different cases presented in Table 7.2. The volume averaged turbulent energy dissipations for the different cases are given in Table 9.2. The flow residence time for the different Reynolds number are also shown in Table 9.2. The time evolution of volume fraction and Sauter mean diameter are calculated with the assumption that there is no convection ($C_i$ in Eq. (9.16)) and growth ($G_i$ in Eq. (9.16)) dispersed droplets in the computational domain. The geometric grid explained in the section 9.5.2 is used for this calculation because it requires significantly less number of
grid points. The span of droplet size for the grid generation is considered to be 1-1024 microns and the total number of grid point used is 61.

Table 9.2: Volume averaged turbulent energy dissipation calculated in the novel hydrocyclone (Fig. 7.1) for the different test cases presented in Table 7.2

<table>
<thead>
<tr>
<th>Feed Reynolds number Re_F</th>
<th>Volume averaged turbulent energy dissipation ( \varepsilon_{\text{ave}} ) (m(^2)/s(^3))</th>
<th>Flow residence time ( \tau ) (millisecond)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11440</td>
<td>46</td>
<td>458</td>
</tr>
<tr>
<td>23835</td>
<td>425</td>
<td>220</td>
</tr>
<tr>
<td>34320</td>
<td>1277</td>
<td>153</td>
</tr>
<tr>
<td>45760</td>
<td>2795</td>
<td>112</td>
</tr>
<tr>
<td>68640</td>
<td>10018</td>
<td>76</td>
</tr>
</tbody>
</table>

The time evolution of volume fraction and Sauter mean diameter for the feed Reynolds number of 34320 and the corresponding volume averaged turbulent energy dissipation of 425 m\(^2\)/s\(^3\) are shown in Fig. 9.9. The simulation using the PBM is run for 250 milliseconds, which is slightly longer than the flow residence time for the given feed Reynolds number (Table 9.2). The simulations are conducted in three different modes: pure coalescence, pure breakage, and breakage and coalescence together. For the pure coalescence, the birth and death of droplets due to breakage \( \text{Bb}_i \) and \( \text{Db}_i \) in Eq. (7.16) are set to zero. Similarly for the pure breakage, the birth and death of droplets due to coalescence \( \text{Bc}_i \) and \( \text{Dc}_i \) in Eq. (7.16) are set to zero. For pure coalescence (Fig. 9.9a) the distribution of volume fraction shifts toward the larger droplet. In comparison with
the pure coalescence, the pure breakage process shifts the distribution of volume fraction significantly toward the smaller droplets (Fig. 9.9b). The distribution of volume fraction for the coalescence and breakage processes together is very close to that of pure breakage (Fig. 9.9c). It means that, for this magnitude of energy dissipation (425 m$^2$/s$^3$), the breakage is dominating over the coalescence. The time evolutions of Sauter mean diameter for the different mechanisms are shown in Fig. 9.9d. The initial $d_{32}$ of 25 microns reduces to 19 microns (for coalescence and breakage together) at the end of flow residence time of 250 milliseconds. The effect of volume averaged turbulent energy dissipation (Table 9.2) on the time evolution of Sauter mean diameter is shown in Fig. 9.10. The horizontal axis represents the simulation time normalized by the flow residence time in the hydrocyclone. Due to the dominating droplets breakage over coalescence the Sauter mean diameter decreases gradually. The rate of reduction in the Sauter mean diameter increases with an increase in the energy dissipation. For the highest energy dissipation, the Sauter mean size of droplets reduces to about 8 microns at the end of flow residence time. The effects of total volume fraction and interfacial tension on the time evolution of Sauter mean diameter are shown in Fig. 9.11 and 9.12 respectively. The variation of volume fraction in this small range (Fig. 9.11) does not provide considerable change in the time evolution of Sauter mean diameter. The Sauter mean diameter of droplets increases with an increase in the interfacial tension. Higher interfacial tension increases the restorative force which thereby decreases the breakage rate.
Figure 9.9: Transient distribution of volume fraction for (a) pure coalescence, (b) pure breakage, (c) coalescence and breakage, and (d) profile of Sauter mean diameter. $\varepsilon_{\text{ave}}=425 \text{ m}^2/\text{s}^3$, $\phi=0.01$, $\sigma=0.02 \text{ N/m}$, $\rho_d=850 \text{ kg/m}^3$, $\mu_d=0.0033 \text{ Pa.s}$.
Figure 9.10: Effect of volume averaged energy dissipation on the time evolution of Sauter mean diameter. $\phi=0.01$, $\sigma = 0.02$ N/m, $\rho_d=850$ kg/m$^3$, $\rho_c=988$ kg/m$^3$, $\mu_d=0.001$ Pa.s, and $\mu_c=0.0033$ Pa.s.

Figure 9.11: Effect of volume fraction on the time evolution of Sauter mean diameter for $\epsilon_{ave} = 425$ m$^2$/s$^3$, $\sigma = 0.02$ N/m, $\rho_d=850$ kg/m$^3$, $\rho_c=988$ kg/m$^3$, $\mu_d=0.001$ Pa.s, and $\mu_c=0.0033$ Pa.s.
Figure 9.12: Effect of interfacial tension on the time evolution of Sauter mean diameter for $\varepsilon_{\text{ave}} = 425 \text{ m}^2/\text{s}^3$, $\phi = 0.01$, $\sigma = 0.02 \text{ N/m}$, $\rho_d = 850 \text{ kg/m}^3$, $\rho_c = 988 \text{ kg/m}^3$, $\mu_d = 0.001 \text{ Pa.s}$, and $\mu_c = 0.0033 \text{ Pa.s}$.

9.7. Coupled multiphase and population balance modeling

Coupling between the multiphase simulation and the PBM provides a detailed understanding of hydrodynamics, phenomena of droplet shearing and coalescence affected by the hydrodynamics, and the mechanics of dispersed phase affected by droplet shearing and coalescence. The coupled CFD and PBM accounts the convection of droplets in a local grid cells which is normally not considered in the stand alone PBM. The PBM needs information of flow and fluid properties which are normally incorporated by simplified correlations in stand-alone PBM codes. Usually, the multiphase CFD simulations (two fluid models such as mixture theory) uses a constant Sauter mean diameter of dispersed droplets in the equations of relative velocity and volume fraction (APPENDIX A3). A change in the droplet size distribution due to coalescence and
breakage is also not considered in the stand alone CFD simulation. In reality, the distribution of dispersed droplets varies in the flow field of hydrocyclone due to breakage and coalescence. These differences in the local distribution influence the hydrodynamics and account for changes in the relative velocities and volume fraction of the dispersed phase. A constant diameter of dispersed droplets in the CFD simulations cannot reflect these influences. The coupling between the CFD and PBM can benefit from each other. Coupled CFD-PBM is a two way coupling, which is conducted by an add-on of Population Balance Model platform to the ANSYS FLUENT. In the FLUENT platform, the input for the diameter of dispersed droplet is linked to the Sauter mean size of droplet calculated in the PBM. In the PBM platform, user defined functions (UDF) are written for the coalescence frequency, breakage frequency and daughter droplet distribution models discussed above. At every time step, the PBM platform receives information of fluid velocities, turbulent energy dissipation and fluid properties for every computational grid cells sequentially. After receiving this information, the PBM platform applies all the models to calculate Sauter mean diameter for the corresponding computational grid cell. The PBM then transfer the information of Sauter mean sizes to the CFD platform. Meanwhile, the PBM platform store the information droplet size distribution (number density) for that given time step. The CFD platform then uses the information of Sauter mean size of dispersed droplet to solve the equation of relative velocities, and volume fraction which are coupled with the Navier-Stokes equations for the mixture. The coupled CFD-PBM is computationally very expensive. A larger number of bins (grid points) in the PBM increase the accuracy in the calculation of Sauter mean diameter. However, the computational time drastically increases with an increase in the number of bins. So, there
is a tradeoff between the number of bins, numerical accuracy and computation time depending on the flow conditions and purpose.

9.8. Simulation results using coupled CFD-PBM

The coupled CFD-PBM simulation is performed in the novel hydrocyclone (Fig. 7.1 and Table 7.1) for the feed Reynolds number of 34320. The total volume fraction at the inlet is 0.01. The droplet sizes at the inlet ranges from 2 to 107 microns. The distribution of volume fraction in the flow field depends on the local turbulent energy dissipation. The contours of turbulent energy dissipation in the flow field are shown in Fig. 9.1. The legend for the energy dissipation shows the level ranges from 0 to 1000 m²/s³. But the actual magnitude in the contour can be higher than 1000 m²/s³. The turbulent energy dissipation increases up to z/D = 5 and the decreases in the cylindrical chamber (5 < z/D < 10). The conical swirl chamber (z/D > 10) accelerates the swirling motion and increases the pressure drop, which thereby increases the turbulent energy dissipation. Moreover, higher energy dissipation appears near the wall. The magnitudes of surface averaged turbulent energy dissipation (shown in Table 9.3) are calculated in the different planes shown in the contours (Fig. 9.13). The droplet breakage and coalescence rate should vary along the axial and radial direction because of the variation of turbulent energy dissipation. The average distributions of volume fraction at different cross-sectional planes are shown in Fig. 9.14. A significant shearing of larger droplets occurs in the inlet duct and upper part of the inlet chamber, which thereby increase the volume fraction of smaller droplets. Because of the shift of volume fraction distribution toward the smaller droplets, the rate of droplet shearing decreases in the downstream. At
$z/D = 2$, the minimum volume fraction appears for the larger droplets and the largest amount of volume observed for 2 micros droplets. In the simulation, for the sake of less computation effort, the smallest bin size was chosen to be of 2 microns. Since there are no bins available for smaller sizes, the volume fraction lost from the larger bins cumulated the bin size of 2 microns. However, additional bins for the droplet size less than 2 microns would provide smoother distribution for the smaller sizes. Even though the surface averaged energy dissipation at $z/D = 5$ is much higher than that at $z/D = 2$, the volume fraction slightly increases for the larger droplets sizes at $z/D = 5$. At $z/D = 5$, due to centripetal force, the droplets already moved away of the wall and are not being experienced high energy dissipation observed near the wall. Moreover, the smaller size of droplets generated due to shear in the upstream yields less breakage rate. As a consequence, the coalescence of droplets starts dominating. In the cylindrical chamber ($z/D > 5$), both the surface averaged value and local magnitude of energy dissipation decrease, which thereby increases the coalescence rate. Higher energy dissipation in the conical swirl chamber ($z/D > 14$) again increases the droplet shearing rate. The Sauter mean diameter of droplets at different axial locations is shown in Fig. 9.15. The Sauter mean diameter gradually decreases from inlet up to the $z/D = 2$; it is then increases in the swirl chamber ($2 < z/D < 5$) and cylindrical chamber ($5 < z/D < 10$).
Figure 9.13: Contours of turbulent energy dissipation for $\text{Re}_F = 34320$. 

$\varepsilon [\text{m}^2/\text{s}^3]$
Table 9.3: The surface averaged energy dissipation is calculated at the different planes shown in the contours (Fig. 9.13)

<table>
<thead>
<tr>
<th>z/D</th>
<th>0.385</th>
<th>1</th>
<th>2</th>
<th>5</th>
<th>10</th>
<th>14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface averaged energy dissipation (m²/s³)</td>
<td>67</td>
<td>66</td>
<td>176</td>
<td>335</td>
<td>92</td>
<td>2184</td>
</tr>
</tbody>
</table>

Figure 9.14: The average distribution of normalized volume fraction at different cross-sectional planes shown in Fig. 9.13.
Figure 9.15: Sauter mean diameter of dispersed droplet at the inlet and the different axial locations. \[ d_{32} = \frac{\sum d_i^3 N_i}{\sum d_i^2 N_i} \]
10.1. Simulation results summary

Theoretical and numerical studies of swirling flow separators (hydrocyclones) are addressed in this work. The hydrodynamic features in the flow field inside of hydrocyclones are studied for a range of operating and geometric conditions. The numerical simulations resulted in a novel understanding of critical hydrodynamic features responsible for the finite turndown ratio observed in hydrocyclones as well as a novel design for an efficient hydrocyclone.

The Reynolds Averaged Navier-Stokes equations closed-off with the Reynolds Stress Models (see APPENDIX A1) are numerically solved to calculate the flow and pressure fields inside the hydrocyclones. The uncertainty of numerical solutions and the accuracy of numerical procedures are evaluated based on a conventional hydrocyclone (see Fig. 3.1). The GCI approach (see APPENDIX C) is used for the numerical uncertainty analysis (see Section 3.4). The accuracy of numerical results of this study is confirmed by validating the results with previous experimental and numerical results obtained by other researches (see Section 3.5). A guideline for achieving a converged and accurate numerical solution for the complex flow field in the hydrocyclone is also addressed (see Section 3.2). The maximum GCI for the mesh size of 663,812 cells if found to be about 5% for the microscopic properties (see Figs. 3.3-3.5) and less than 1%
for the macroscopic properties (see Table 3.3). The uncertainty in the droplet trajectories calculation, which depends on the hydrodynamic characteristics, also observed to be negligible (see Fig. 3.6). The tangential velocity calculated based on the numerical methods mentioned in this work is found to be very close to the experimental value (see Fig. 3.7). Based on these observations, the mesh size and the numerical methods applied in this work are appropriate.

10.2. Effect of the Reynolds number

The results presented previously (Chapter 4) for a wide range of feed Reynolds number in a conventional hydrocyclone (see Fig. 3.1) reflect the fundamental hydrodynamic characteristics that are directly related to the separation performance of a de-oiling hydrocyclone.

Numerical simulations indicate that a boundary layer flow near the top wall creates a short circuit from the inlet to the vortex finder (see Fig. 4.5). A toroidal recirculation appears between the inner and outer vortices (see Fig. 4.6). At low and high values of the Reynolds number ($12,000 > \text{Re}_F > 25,000$), the large volume of that short circuits in the inlet chamber reduces the reverse flow from downstream. For intermediate Reynolds number values, a smaller amount of short circuit flow and large toroidal recirculation support the length of reverse flow core to be long (see Figs. 4.7 and 4.8). The location of peak core pressure shifts towards the underflow for the intermediate Reynolds numbers, which thereby yields a positive axial pressure gradient up to a far downstream (see Fig. 4.11). A reverse flow core appears in the region of positive axial pressure gradient.
Two criteria are investigated for the understanding of the reverse flow core breakdown: i) the normalized distance of the location peak core pressure from the vortex finder ($\ell/D$ at $(dP/dz)_{r=0} = 0$), ii) the normalized distance of the zero axial gradient of radial pressure gradient from the vortex finder ($\ell/D$ at $(d(dP/dr)/dz)_{r=0} = 0$). In the conical swirl chamber, the peak core pressure occurs where the restriction of forward flow by the reverse flow core is at maximum. Because of the elliptical shape near the bottom of the reverse flow core, the $(dP/dz)_{r=0} = 0$ occurs slightly upstream of the core breakdown (see Fig. 4.12). So, the location of $(dP/dz)_{r=0} = 0$ is a good indicator for a qualitative understanding of the reverse flow core breakdown. The magnitude of $\ell/D$ at $(d(dP/dr)/dz)_{r=0} = 0$ is closer to the values of $\ell_C/D$ when compared with $\ell/D$ at $(dP/dz)_{r=0} = 0$ (see Fig. 4.12). The second criterion (location of $(d(dP/dr)/dz)_{r=0} = 0$) appears to be a better indicator for both the qualitative and quantitative understanding of the reverse flow core breakdown.

The location of zero axial gradient of radial pressure gradient from the vortex finder ($\ell/D$ at $(d(dP/dr)/dz)_{r=0} = 0$) at the core varies with the feed Reynolds number ($Re_F$). With increasing the $Re_F$, the location of $(d(dP/dr)/dz)_{r=0} = 0$ first moves downstream and then shifts toward the overflow (see Fig. 4.13). This phenomenon is similar to the behavior observed for the length of reverse flow core as shown in Fig. 4.8. With a constant overflow ratio ($R_0$), the length of reverse flow core increases with an increase in the Reynolds number up to the $Re_F$ of 17,160. It then decreases with a further increase in the $Re_F$ (see Fig. 4.8). The cut-size ($d_{50}$) of the hydrocyclone is inversely related to the length of the reverse flow core ($\ell_C$) i.e., longer the reverse flow core the smaller the cut-size becomes (see Fig. 4.15). A smaller cut-size yields higher separation efficiency. So,
the separation efficiency increases with an increase in the length of reverse flow core and vice versa.

The Capillary number \( (Ca) \) in regions of flow field where the separation of oil and water occurs is found to be less than 0.1 (see Fig. 4.17). This value is insignificant for an oil droplet to be broken up. Moreover, the Weber number in regions of oil-water separation is found to be much less than the critical Weber number (see Fig. 4.18), which also indicates a negligible probability of droplet shearing. Without considering droplet coalescence and shearing, the separation efficiency of a conventional hydrocyclone first increases with the feed Reynolds number, becomes plateau for the intermediate Reynolds numbers, and drops for the higher feed Reynolds numbers (Fig. 4.16). This finite turndown ratio for a range of feed Reynolds numbers follows the similar trend of the variation of length of reverse flow core with the feed Reynolds number (see Fig. 4.8). This phenomenon suggests that the finite turndown ratio and the drop of separation efficiency for the high Reynolds number is a hydrodynamic effect. This aspect of the flow physics led us to design a better hydrocyclone that would maintain a long reverse flow core for a wide range of feed Reynolds number.

### 10.3. Effect of pressure drop ratio

The higher pressure drop ratio \( (\Delta P^*) \) increases the overflow Euler number \( (Eu_O) \), which thereby increases the overflow ratio \( (R_O) \) (see Fig. 5.1). The tangential velocity at the interface between the inner and outer vortexes first increases for the \( \Delta P^* \) of 1.0 to 1.45 and then decreases for the \( \Delta P^* \) of 2.0 (see Fig. 5.2). In the outer vortex region \( (\tau/R > 0.2) \), there is no considerable variation in the tangential and axial velocity for the different
\[ \Delta P^* \]. However, the radially inward velocity increases with an increase in the \( \Delta P^* \) (see Fig. 5.4), which thereby increases the short circuit flows in the inlet chamber. As a consequence, the radius of reverse flow core increases with an increase in the \( \Delta P^* \) (see Fig. 5.3). In the inlet chamber, the flow circulation strength and radial pressure gradient decreases with an increase in the \( \Delta P^* \) (see Fig. 5.5). There is no considerable difference that appears in the grade separation efficiency between the \( \Delta P^* \) of 1.0 and 1.45. Due to a high overflow rate, a \( \Delta P^* \) of 2.0 provides a reduced separation efficiency (see Fig. 5.8).

### 10.4. Effect of design of vortex finder and overflow tube

The designs of overflow tube and vortex finder are found to affect the short circuit flows and flow circulation in the inlet chamber. A vortex finder with a smaller diameter of overflow tube provides a longer flow path as well as a higher flow circulation, which thereby yields a longer flow residence time (see Fig. 5.10). Moreover, it decreases the radial inflow and offers a better restriction in the overall short circuit flows (see Fig. 5.11). In the outer vortex region, there is no considerable difference in the tangential velocity, axial velocity, and radial pressure that appears due to the variation in the design of vortex finder and diameter of overflow tube (see Figs. 5.12-5.14). However, the reduction in the diameter of the overflow orifice decreases the radius of the reverse flow core and radial inflow (see Figs. 5.13 and 5.14).
10.5. Effect of angle of conical swirl chamber

The angle of conical swirl chamber significantly influences the internal hydrodynamics and separation performance. With a constant overflow ratio ($R_o$), both overflow and underflow Euler numbers ($E_{uo}$ and $E_{uu}$, respectively) gradually increase with an increase in the cone angle (see Fig. 5.16). This indicates that a higher cone angle yields more energy loss when compared with a smaller cone angle. A higher cone angle enhances the tangential velocity as well as the centrifugal acceleration (see Fig. 5.17). However, a shorter length of the swirl chamber and a higher centrifugal acceleration, due to an increase in the cone angle, decreases the flow residence time. In addition, a higher cone angle reduces the radially inward flow velocity (see Fig. 5.19). The reverse flow velocity in the inner vortex ($r/R < 0.2$) and downward flow velocity near the wall ($r/R > 0.7$) increases with a decrease in the cone angle (see Fig. 5.18). In the intermediate region ($0.2 < r/R < 0.7$), the trend is opposite when compared with the region of $r/R > 0.7$. For the different cone angles, the radial pressure is almost the same at $r/R > 0.5$. However, the radial pressure gradient at $r/R < 0.5$ increases with an increase in the cone angle (see Fig. 5.20). The location of maximum core pressure shifts downstream with a decrease in the cone angle (see Fig. 5.21). So, a lower cone angle maintains a positive axial pressure gradient up to a far downstream, which thereby increases the length of reverse flow core (see Fig. 5.22). Even though a smaller cone angle decreases the swirl intensity, the longer reverse flow core enhances the separation efficiency by bringing the migrated droplets to the overflow orifice from the far downstream (see Fig. 5.23). The cut-size decreases drastically with an increase in the cone angle. Moreover, the cut size is inversely related with the length of reverse flow core (see Fig. 5.24).
10.6. Effect of swirl chamber design

The various designs for an improved swirl chamber considered in this study are: Design-A (parabolic wall profile with asymptote at the bottom end), Design-B (parabolic wall profile with asymptote at the top end) and Design-C (hyperbolic wall profile) (see Section 6.2.1 and APPENDIX D.

In the first step, the conical swirl chamber of a conventional hydrocyclone is replaced by parabolic and hyperbolic swirl chambers. In comparison with the conical and parabolic profile, the hyperbolic swirl chamber yields a greater tangential velocity because of the largest wall curvature (see Fig. 6.2). In comparison with the conical and parabolic walls, Design-C yields the shortest reverse flow core because of the location of peak core pressure closer to the overflow (see Fig. 6.3). At the end of swirl chamber (beginning of tail pipe), a very high negative core pressure for the conical wall can initiate cavitation (see Fig. 6.3). However, the Design-C yields a significant improvement in regards to the minimization of negative core pressure.

In the next step, to improve the length of reverse flow core, the tail pipe has been removed and the parabolic and hyperbolic profiles are extended up to the total length of original hydrocyclone (see Table 6.1, Case-2). Due to a higher longitudinal wall curvature, the Design-C provides a greater tangential velocity at r/R < 0.7 (see Fig. 6.6) and radial inward velocity at r/R > 0.7 (see Fig. 6.8) when compared with the parabolic swirl chambers (Design-A and Design-B). In comparison with other designs, the Design-C yields a greater pressure differential between the side wall and the center up to z/D ≈ 15; it indicates a higher swirling motion almost all the way downstream, which thereby yields a greater centrifugal acceleration (see Fig. 6.10).
A longer swirl chamber reduces the wall curvature and gradually decreases the diameter of the chamber, which thereby makes a good balance between the acceleration and deceleration of swirling motion by reduced diameter and wall friction, respectively. However, the core pressure profile exactly follows the wall profiles of swirl chamber, i.e., higher the wall curvature (lower radius) lower the core pressure becomes (see Fig. 6.11). The Design-B maintains reverse flow core up to z/D \approx 11, where it is z/D \approx 7 for the Design-C and the Design-A (see Figs. 6.12 and 6.14). Due to a longer reverse flow core, the Design-B yields better separation efficiency. However, the separation efficiency for the Design-C is very close to the Design-B and much higher than that of Design-A (see Fig. 6.13). Though the length of reverse flow core for the Design-C is much shorter than that of Design-B, the higher g-force for the Design-C, which is caused by a larger wall curvature as well as a smaller radius of swirl chamber, enhances the migration rate of droplets. So, the Design-C has a better potential for a higher separation. Moreover, following the Design-C a shorter hydrocyclone can be made because of its ability to provide higher separation efficiency with a shorter reverse flow core.

10.7. Effect of tail gadget

The simulation results for the hydrocyclone with conical and modified swirl chamber geometries reflect that the reverse flow core breaks down in the swirl chamber and never reaches close to the underflow. Even though the dispersed droplets near the tail end still migrate to the center they cannot move to the overflow orifice due to the absence of reverse flow core. For capturing the migrated droplet from the underflow, two gadgets: Gadget-A (cylindrical with a vortex finder) and Gadget-B (conical without a vortex
finder) are designed and connected at the underflow orifice of a conventional hydrocyclone (see Figs. 6.16 and 6.17). The length of reverse flow core is unaffected by the tail gadgets. The radial flow through the outlet slots in Gadget-B remixes the migrated droplets with continuous phase and provides less grade efficiency relative to the overflow-2 \( (G_{OR2}) \), when compared with Gadget-A. The total grade separation efficiency \( (G_{ORT} = G_{OR1} + G_{OR2}) \) is greatly improved due the addition of tail gadgets when compared with the \( G_{OR1} \) of the conventional hydrocyclone (see Fig. 6.19). Moreover, the mechanism for capturing migrated droplets from both the overflow and underflow supports the performance of a hydrocyclone to be independent of the length of reverse flow core. So with a tail gadget, hydrocyclone will provide high separation efficiency at high fluctuation in the feed flow, which thereby will increase turndown ratio.

10.8. Tailoring of swirl chamber based on hyperbolic wall profile

The previous discussion addresses that the hyperbolic swirl chamber (Design-C) provides a greater g-force, enhances the migration rate of dispersed droplets toward the center as well has a better potential for higher separation efficiency. This results leads to tailor the hydrocyclone swirl chamber based on the hyperbolic wall profile, which resulted in development of a novel hydrocyclone.

Case-1: To increase the compactness, the hydrocyclone having the Design-C is cut off at \( z/D = 8.7 \) (see Fig. 6.20) since there is no reverse flow at \( z/D > 7 \) (see Fig. 6.14). In the modified design, the positive axial pressure gradient and the reverse flow core appear up to the underflow orifice (see Fig. 6.21), which support the transport of all migrated droplets to the overflow orifice. The separation efficiency slightly increases
when compared with a long hydrocyclone (L/D ≈ 20) having the Design-C (see Figs. 6.13 and 6.22).

**Case-2:** The long hydrocyclone (L/D ≈ 20) with the Design-C breaks down the positive axial pressure gradient at z/D ≈ 5 (see Fig. 6.11). For avoiding the breakdown of the positive axial pressure gradient, the swirl chamber is cut off at z/D = 5 and a long cylindrical tail pipe is connected (see Fig. 6.23). This modification allows the core pressure to be increased gradually toward the underflow; the positive axial gradient appears all the way up to the underflow orifice (see Fig. 6.24). In addition, the reverse flow core appears up to the underflow (see Fig. 6.25). However, the separation efficiency in the Case-2 (see Fig. 6.25) is almost the same when compared with that of the Design-C (see Figs. 6.13). The larger diameter of the tail pipe (in comparison with the diameter of swirl chamber at z/D > 5 for the Design-C) significantly reduces the swirl intensity. At z/D > 10, the lower swirl intensity reduces the migration rate of dispersed droplets toward the center.

Even though the modified hydrocyclone (case-2) does not provide better separation efficiency than that of design case-1, it is possible to enhance separation efficiency for the design case-2 by increases the feed Reynolds number. The fact is that, because of very long reverse flow core and longer residence time, the design case-2 should provide much better separation efficiency at a larger feed Reynolds number; which may not be possible in the design case-1 because of shorter flow residence time.
**Case-3:** In manufacturing point of view, the hyperbolic wall profile is complex for machining. For this reason, the swirl chamber of design case-2 ($z/D \leq 5$) is reshaped using three frustocones of half-cone angles 14, 4.5 and 2 degrees in such a way that the wall curvature almost follows the hyperbolic profile (see Fig. 6.26). For the hydrocyclone with the three frustocones, no variation in the velocity components and reverse flow core appears when compared with the pure hyperbolic wall profile (see Fig. 6.27). Moreover, the core pressure profiles for both the pure hyperbolic and frustocones are almost the same (see Fig. 6.27), which was expected. So, the reshaping of hyperbolic swirl chamber using three frustocones of half-angle 14, 4.5 and 2 degrees is appropriate. This modification in the hydrocyclone geometry is suitable from both the high separation efficiency and simplicity in manufacturing point of views.

**10.9. Novel efficient hydrocyclone**

From the detailed analyses of hydrodynamics and separation performance on conventional and modified hydrocyclones, the follow key points are summarized from the design point of views:

I) A vortex finder of length equals to 0.1D and the diameter of overflow tube less than 0.04D provides a better restriction to the short circuit flows in the inlet chamber.

II) A hyperbolic shaped swirl chamber provides higher swirl intensity and it is suitable for shorter hydrocyclone.
III) The hyperbolic shaped swirl chamber has to be cut off at z/D = 5 to avoid breakdown of reverse flow core; the wall curvature can be reshaped by three frustocones having half angles of 14°, 4.5° and 2°.

IV) A cylindrical tail pipe longer than 5.0D (z/D > 10) significantly reduces the swirl intensity; a frustocone is essential at z/D > 10.

V) A tail gadget is essential for capturing the migrated droplets from the underflow; the gadget-A performs better than gadget-B.

Considering the design conditions mentioned above, a very high efficiency hydrocyclone is developed. The invented hydrocyclone consists of two overflow outlets positioned at the top and the bottom of the hydrocyclone for collecting separated dispersed oil, two tangential inlets, and underflow outlets for collecting cleaner water (see Fig. 7.1). The invented hydrocyclone has two separation zones which response to a large variation of inlet conditions in such a way that a decrease in separation efficiency in one separation zone is balanced by an increase in separation efficiency in another separation zone. The hydrodynamics and separation performance of the invented hydrocyclone are investigated for a range of feed Reynolds number.

The invented hydrocyclone maintains high tangential velocity (almost equal to the peak value) in the hydrocyclone chambers up to r/R ≈ 0.6 (see Fig. 7.4) whereas the peak value appears at r/R ≈ 0.15 for conventional hydrocyclone (see Fig. 4.1). The novel hydrocyclone provides a higher g-force in the outer vortex region, a greater migration rate of dispersed droplets, and thinner side wall boundary when we compare with a conventional hydrocyclone. In addition, at z/D < 5, the reduction of swirling motion by the wall friction is well balanced by the acceleration of swirling motion by the
frustocones. The cylindrical chamber \((5 < z/D < 10)\) reduces swirling motion by wall friction; it can be recovered by replacing the straight cylindrical chamber with a minute tapered chamber (a frustocone of \(1^0\) cone angle) of same length. The second swirl chamber \((z/D > 10)\) again accelerates swirling motion and provides a very high g-force (see Figs. 7.4 and 7.5).

Though a higher Reynolds number significantly increases the g-force, the invented hydrocyclone always maintains positive axial pressure gradient up to the end of cylindrical chamber \((z/D \approx 10)\) (see Fig. 7.11). The normalized pressure differential between the side wall and the center first decrease in the swirl chamber-1\((z/D < 5)\) and then increases in the cylindrical chamber \((5 < z/D < 10)\) (see Fig. 7.12). So, the reduction of core pressure in the swirl chamber-1 is balanced by the enhancement of core pressure in the cylindrical swirl chamber. In the swirl chamber-2 \((z/D > 10)\), the high acceleration of swirling motion dramatically decreases the static pressure (see Fig. 7.11); it eventually becomes negative near the tail end. The length of reverse flow core increases with the Reynolds number and become stable for the \(\text{Re}_F > 23,835\) (see Fig. 7.8). For the \(\text{Re}_F > 23,835\), the reverse flow core appears up to the end of cylindrical chamber \((z/D \approx 10)\). Because of the constant length of reverse flow core, the separation efficiency relative to the overflow-1 \((G_{\text{OR1}})\) continuously increases with an increase in the feed Reynolds number (see Fig. 7.16).

The \(G_{\text{OR1}}\) gradually increases with an increase in the feed Reynolds number, which was not observed in the conventional hydrocyclone. This continuous increase in the separation efficiency possesses a wide turndown. For smaller droplets \((d_D < 15\) microns) the \(G_{\text{OR2}}\) increases with an increase in the feed Reynolds number (see Fig. 7.16).
The smaller droplets slowly migrate toward the core and need high residence time to be separated. Since, the \( G_{OR1} \) is very small for the smaller droplets, a large amount of smaller droplets are available to be separated in the swirl chamber-2. A higher feed Reynolds number increases the migration rate of these smaller droplets in the swirl chamber-2 and thereby increase the \( G_{OR2} \). With an increase in the feed Reynolds number the cut-size of the novel hydrocyclone gradually decreases, whereas a conventional hydrocyclone first decreases and then increases the cut-size as well as possesses a finite turndown ratio.

The total reduced separation efficiency (\( G_{ORT} \)) calculated from the multiphase simulation is higher than that calculated based on the Lagrangian tracking (see Figs. 7.17 and 7.19). For 20 microns droplets, the total separation efficiency in the multiphase simulation is about 75%; it is about 65% in the Lagrangian tracking. In the discrete phase model (Lagrangian tracking), only one way coupling (only continuous phase affects the droplet motion) and no buoyancy force are considered. The simplified assumptions in the Lagrangian tracking can cause this difference in the separation efficiency. However, the total reduced separation efficiency in the novel hydrocyclone is found to be almost constant for all the feed concentration of dispersed droplets (see Fig. 7.20).

10.10. Droplet modeling and coupled CFD-PBM

High turbulent energy dissipation in the hydrocyclone causes droplets breakup and coalescence simultaneously. A higher turbulent energy dissipation increases relative velocity and thereby collision frequency between droplets, which thereby increases coalescence rate. The coalescence rate also increases with an increase in the size of
colliding droplets. In the novel hydrocyclone, for the ReF of 34320 and the corresponding volume averaged turbulent energy dissipation of 425 m$^2$/s$^3$, the droplets breakage is dominating over the coalescence; the Sauter mean size decreases from 25 to 19 microns by the flow residence time of 250 milliseconds (see Fig. 9.9d). The rate of reduction in the Sauter mean diameter increases with an increase in the energy dissipation (see Fig. 9.10).

The droplet breakage and coalescence rate varies along the axial and radial direction of the novel hydrocyclone because of the variation of turbulent energy dissipation in the flow field (see Fig. 9.13 and Table 9.3). A significant shearing of larger droplets occurs in the inlet duct and the inlet chamber. Because of shifting of volume fraction distribution toward the smaller droplets, the rate of droplet shearing decreases in the downstream (see Fig. 9.14). In the downstream, due to migration of droplets toward the center, a large number of droplets are not subjected to high shearing near the side wall. At z/D > 2, the coalescence starts dominating. In the cylindrical chamber (z/D > 5), both the surface averaged value and local magnitude of energy dissipation decrease, which thereby increases the coalescence rate. In the second swirl chamber (z/D > 10), the breakage rate is again dominating over the coalescence due to a significant rise in the turbulent energy dissipation. The Sauter mean diameter gradually decreases from inlet up to the z/D = 2; it then increases in the swirl chamber (2 < z/D < 5) and cylindrical chamber (5 < z/D < 10); it decrease again at z/D > 10 (see Fig. 9.15).
APPENDICES
APPENDIX A

Governing equations

A1. Continuous phase mechanics

Conservation of mass

\[ \nabla \cdot \underline{U}_C = 0, \quad \rho_c = \text{Constant} \]  \hspace{1cm} (a1)

Conservation of momentum

\[ \rho_c \frac{\partial \underline{U}_C}{\partial t} + \rho_c \underline{U}_C \cdot \nabla \underline{U}_C = -\nabla p_c + \rho_c \mathbf{g} + \mu_c \nabla^2 \underline{U}_C, \quad \mu_c = \text{Constant} \]  \hspace{1cm} (a2)

The subscript ‘c’ denotes the continuous phase and the under bar represents the vector quantity. In the turbulent flow field, the instantaneous velocity can be decomposed to mean and fluctuating components as \( \underline{U}_c = \langle \underline{U}_c \rangle + \mathbf{u}_c \). The quantity within \( \langle \rangle \) represents the ensemble average of that quantity. The ensemble average of fluctuating quantity is zero i.e. \( \langle \mathbf{u}_c \rangle = 0 \). Averaging Eq. (a1) yields

\[ \nabla \langle \underline{U}_c \rangle = 0 \]  \hspace{1cm} (a3)

Applying the Reynolds decomposition to Eq. (a1) and using Eq. (a3), the microscopic balance of fluctuating velocity filed can be written as

\[ \nabla \cdot \mathbf{u}_c = 0 \]  \hspace{1cm} (a4)

Applying the Reynolds decomposition and averaging technique, the exact (unclosed) Reynolds average microscopic balance equation for linear momentum equation can be written as (see Pope, 2000; ANSYS Fluent documentation: theory guide, 2013 (page 42))

\[ \rho_c \frac{\partial \langle \underline{U}_c \rangle}{\partial t} + \rho_c \langle \underline{U}_c \rangle \cdot \nabla \langle \underline{U}_c \rangle = -\nabla \langle p_c \rangle + \mu_c \nabla^2 \langle \underline{U}_c \rangle - \rho_c \nabla \cdot \langle \mathbf{u}_c \mathbf{u}_c \rangle \]  \hspace{1cm} (a5)
The force due to gravitational acceleration has been neglected since it is insignificant when compared with the centrifugal acceleration \( (U_0^2/r) \) in a hydrocyclone (Ma et al., 2000). For example, if the feed velocity \( (U_F) \) equals to 4 m/s \((\text{Re}_F = \rho_c U_F D_F/\mu_c = 22,880)\) the centrifugal acceleration in the inlet chamber is \( U_0^2/r = 1230 \gg g \) (\( U_0 \) near the wall in the inlet chamber is close to the \( U_F \)). The above RANS equation, Eq. (a5) is unclosed and the kinematic Reynolds momentum flux \( \langle u_c u_c \rangle \) needs a closure model.

A transport equation for the kinematic Reynolds momentum flux can be written as (see ANSYS Fluent documentation: theory guide, 2013 (page 86))

\[
\frac{\rho_c}{\text{Local time derivative}} \frac{\partial \langle u_c u_c \rangle}{\partial t} + \rho_c \langle U_c \rangle \cdot \nabla \langle u_c u_c \rangle = \mu_c \nabla^2 \langle u_c u_c \rangle + D + \Pi - \varepsilon \tag{a6}
\]

where \( D \) is the turbulent mixing of Reynolds stress tensor by turbulent diffusion and \( \Pi \) is the redistribution of Reynolds stress tensor by pressure-strain and \( \varepsilon \) is the tensor of turbulent energy dissipation. \( D, \Pi \) and \( \varepsilon \) are unclosed terms and need closure models.

The \( D \) given in Eq. (a6) is defined as

\[
D = \rho_c \nabla \cdot \left( \langle u_c u_c u_c \rangle \right) + \left( \nabla \cdot \langle p_c I u_c \rangle \right)^T + \nabla \cdot \langle p_c I^T u_c \rangle \tag{a7}
\]

The \( D \) is closed according to the model proposed in ANSYS Fluent documentation: theory guide, 2013 (page 87) for a scalar turbulent diffusivity (see Lien and Leschziner (1994)) which is given below:
\[ D = \nabla \cdot \left( \frac{\mu_t}{\sigma_k} \nabla < u_C u_C > \right) \] where \( \mu_t = 0.09 \rho C k^2 / \varepsilon_c \) and \( \sigma_k = 0.82 \) \hspace{1cm} (a8)

The pressure strain term \( \Pi \) is defined as

\[ \Pi = < p_C \left( \nabla u_C + (\nabla u_C)^T \right) > \] \hspace{1cm} (a9)

The pressure strain term \( \Pi \) is decomposed into three components

\[ \Pi = \Pi^S + \Pi^R + \Pi^{WR} \] \hspace{1cm} (a10)

where \( \Pi^S \), \( \Pi^R \) and \( \Pi^{WR} \) are slow, rapid and wall reflection terms, respectively. The slow term reflects the turbulent-turbulent interaction and the rapid term represent the influence of mean flow in turbulent field. The wall reflection term is responsible for redistribution of normal stress near the wall. It tends to damp the normal stress perpendicular to wall, while enhancing the stress parallel to the wall. These three components of the pressure strain term are modeled according to the proposal given by Gibson and Launder (1978), Fu et al. (1987), and Launder (1989) (see ANSYS Fluent documentation: theory guide, 2013 (page 87-88)).

\[ \Pi^S = -1.8 \rho C k^2 \varepsilon_c \left( < u_C u_C > - 2/3 \right) \] \hspace{1cm} (a11)

\[ \Pi^R = -0.6 \left( \left( P - C \right) - \frac{1}{3} \left( \text{tr} \left( P \right) - \text{tr} \left( C \right) \right) \right) \] \hspace{1cm} (a12)

\[ \Pi^{WR} = \frac{0.5 \varepsilon_c}{k C} \left( < u_C u_C > : \left( nn \right) I - \frac{3}{2} < u_C u_C > : \left( nn \right) - \frac{3}{2} \left( nn \right) : < u_C u_C > \right) \right] \right] C_l k^3 \varepsilon_c^2 \delta \] \hspace{1cm} (a13)

where \( \delta \) is the normal distance to the wall, \( n \) is the unit normal vector to the wall, and \( C_l = C_\mu^{3/4} / \kappa \) where \( C_\mu = 0.09 \) and \( \kappa = 0.42 \).
The tensor for turbulent energy dissipation, $\varepsilon$ is defined as

$$
\varepsilon = 2\mu_C \left[ (\nabla \mathbf{u}_C)^T \cdot (\nabla \mathbf{u}_C) \right]
$$

(a14)

The energy dissipation tensor, $\varepsilon$ is modeled as (see ANSYS Fluent documentation: theory guide, 2013 (page 93))

$$
\varepsilon = \frac{2}{3} \rho_c \varepsilon_c I
$$

(a15)

where $\varepsilon_c$ is the scalar energy dissipation rate of isotropic turbulence. The energy dissipation rate $\varepsilon_c$ is calculated from the following equation:

$$
\rho_c \frac{\partial \varepsilon_c}{\partial t} + \rho_c \langle \mathbf{u}_c \mathbf{u}_c \rangle \cdot \nabla \varepsilon_c = \mu_C \nabla \cdot \left[ \left(1 + \frac{C_1}{\sigma_\varepsilon} \text{Re}_t \right) \nabla \varepsilon_c \right] + 1.44 \left( \frac{1}{2} \text{tr} \left( \frac{P}{\varepsilon_c} \right) \right) \varepsilon_c \frac{\varepsilon_c}{k_c} - 1.92 \rho_c \frac{\varepsilon_c^2}{k_c}
$$

(a16)

where $\sigma_\varepsilon = 1$ and Re$_t$ is the turbulent Reynolds number, which is defined as $\rho_c k_c^2 / (\mu_c \varepsilon_c)$. The turbulent kinetic energy, $k_c$ is calculated from the trace of kinematic Reynolds momentum flux as $k_c = \frac{1}{2} \text{tr} \langle \mathbf{u}_c \mathbf{u}_c \rangle$. At wall (solid/fluid interface), all the velocity components (mean and fluctuating) are zero which yield the turbulent kinetic energy, $k_c$ is zero. As a consequence, the last two terms in the equation of turbulent energy dissipation rate (Eq. (a16)) is infinity ($\infty$). Therefore, boundary conditions are required for the individual Reynolds stresses ($\langle \mathbf{u}_c \mathbf{u}_c \rangle$) and turbulent energy dissipation rate, $\varepsilon_c$. The Reynolds stresses at the wall-adjacent cells are fixed which are computed as:

the diagonal components:

$$
\frac{\langle (u_c)^2 \rangle}{k_c} = 1.098, \quad \frac{\langle (u_c)^2 \rangle}{k_c} = 0.247, \quad \frac{\langle (u_c)^2 \rangle}{k_c} = 0.655
$$
and the off diagonal components: 
\[
\frac{\langle \mathbf{u} \cdot \mathbf{c} \rangle}{k_c} = 0.255
\]

where \(\theta\), \(r\) and \(z\) are local tangential, normal and bi-normal coordinates, respectively. The kinetic energy, \(k_c\) at the wall adjacent cells are calculated from the transport equation as
\[
\rho_c \frac{\partial k_c}{\partial t} + \rho_c \langle U_c \rangle \cdot \nabla k_c = \mu_c \nabla \cdot \left[ \left( 1 + \frac{C_u}{\sigma_k} \text{Re}_t \right) \nabla k_c \right] + \frac{1}{2} \text{tr} \left( P \right) - \mu_c \varepsilon_c
\]

For the wall-adjacent cells, instead of using Eq. (a16), the \(\varepsilon_c\) in Eq. (a17) is calculated as
\[
\varepsilon_c = \frac{C_\mu^{3/4} k_c^{3/2}}{\kappa y_p}
\]
where \(y_p\) is the distance of the centroid of wall-adjacent cells from the wall.

A2. Dispersed phase mechanics

The motions of dispersed droplets are estimated in the calculated flow field of the continuous phase by injecting a large number of droplets from the inlet. The trajectories of dispersed droplets denoted as \(\chi_D(t)\) is predicted using the kinematic equation shown in Eq. (a13), which is written in Lagrangian frame of reference as
\[
\frac{\partial \chi_D(t)}{\partial t} = U_D(t)
\]

where subscript ‘\(D\)’ represents the dispersed phase. At any time instant the position of dispersed droplet can be estimated by integrating the kinematic Eq. (a13) over the time period as
\[
\chi_D(t + \Delta t) = \chi_D(t) + \int_t^{t+\Delta t} U_D(t) dt
\]

The dispersed phase velocity in the right hand side of Eq. (a14) can be calculated by solving the following equation of motion:
\[
\frac{\partial U_D(t)}{\partial t} = \frac{\rho_c}{\rho_D} \frac{U_{\text{drift}}}{\tau_D} + \sum_{n=1}^{N} a_n
\]
where, \( \tau_D = \frac{4d_D}{3C_D\|U_{\text{drift}}\|} \) and \( U_{\text{drift}} = U_C - U_D \)

\[ C_D = C_1 + \frac{C_2}{\text{Re}_D} + \frac{C_3}{\text{Re}_D^2} \quad , \quad \text{Re}_D = \frac{\rho C \|U_{\text{drift}}\| d_D}{\mu_C} \]

The integral form of the Eq. (a14) is given below

\[
U_D(t + \Delta t) = U_D(t) + \frac{\rho C}{\rho_D} \int_t^{t+\Delta t} \left[ \frac{U_{\text{drift}}}{\tau_D} + \sum_{n=1}^{N} a_n \right] \, dt
\]

(a15)

where \( a_n \) represents the acceleration of droplets due to the effect pressure gradient, and virtual mass force as shown in Table A1. The dispersed phase droplets are assumed to be spherical and the coefficient of drag force, \( C_D \) is calculated by applying a spherical drag law proposed by Mosi and Alexander (1972). The coefficients \( C_1, C_2 \) and \( C_3 \) in the equation of \( C_D \) depends on the ranges of Reynolds number which are given in Mosi and Alexander (1972). The stochastic tracking model has been used to predict the dispersion of droplets due to turbulence in the fluid phase. The fluctuations in the instantaneous velocity of the fluid phase affect the droplets trajectories and their momentum. The fluctuating velocity components are discrete piecewise constant and a function of time Zhang et al. (2012). The instantaneous velocity, \( U_C \) in the drift velocity includes the mean and fluctuating quantity. The RANS equation associated with RSM closure can only predict the mean field and the kinematic Reynolds momentum flux. The fluctuating quantities in the calculated flow field is approximated from the kinematic Reynolds momentum flux as

\[
u_c \cdot \xi = \xi \sqrt{<u_c u_c> : (ee)} \]

(a16)

where \( \xi \) is the permutation and \( \xi \) represents a random variable. The random values are kept constant over an interval of time, which is estimated by characteristics lifetime of eddies (\( \tau_e \)) or cross time (\( \tau_{\text{cross}} \)). When a droplet crossing an eddy the situation could be
such that this eddy can be vanished before the droplet moves out of the eddy or the droplet can cross that eddy before it is being vanished. The random variable \( \zeta \) is kept constant up to \( \min [\tau_e, \tau_{cross}] \). A new random variable is set once the \( \min [\tau_e, \tau_{cross}] \) exceeds which yields a new fluctuating velocity component. The characteristic life time and cross time can be defined as

\[
\tau_e \equiv -0.3 \frac{k_c}{\varepsilon_c} \ln(\gamma) \quad \tau_{cross} \equiv -\tau_D \ln \left[ 1 - \frac{C_{L_1} [k_c^{3/2}/\varepsilon_c] \ln(\gamma)}{\tau_D \|U_{drift}\|} \right]
\]

(a17)

where \( \gamma \) is a uniformly distributed random number that varies between 0 and 1.

Table A1: Additional forces that are included in the analyses of dispersed droplets trajectories. The lift force is only applicable for submicron droplet

<table>
<thead>
<tr>
<th>Forces</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acceleration due to pressure gradient force</td>
<td>( F_{n1} = (U_D \cdot \nabla U_C) )</td>
</tr>
<tr>
<td>Acceleration due to virtual mass force</td>
<td>( F_{n2} = \frac{1}{2} \frac{DU_{drift}}{Dt} )</td>
</tr>
<tr>
<td>Acceleration due to lift force</td>
<td>( F_{n3} = \frac{5.188 v^{1/2} E U_{drift}}{d_D \left( E : E \right)^{1/4}} )</td>
</tr>
</tbody>
</table>

where \( E = \frac{1}{2} \left[ \nabla U + (\nabla U)^T \right] \) is deformation tensor.
A3. Multiphase phase mechanics

The mixture model has been applied for the multiphase phase simulation discussed in the previous chapters. The mixture model solves the continuity and momentum equations for mixture and volume fraction equation for dispersed phase.

The continuity equation for the mixture is

$$\frac{\partial}{\partial t} \rho_m + \nabla \cdot (\rho_m \mathbf{U}_m) = 0$$

where $\mathbf{U}_m = (\sum_{k=1}^{2} \alpha_k \rho_k \mathbf{U}_k)/\rho_m$ is the mass averaged mixture velocity and $\rho_m = \sum_{k=1}^{2} \alpha_k \rho_k$ is the mixture density. Here $\alpha_k$ is the volume fraction of phase $k$; $k = 1$ for continuous phase and 2 for dispersed phase.

The momentum equation for the mixture is

$$\frac{\partial}{\partial t} (\rho_m \mathbf{U}_m) + \nabla \cdot (\rho_m \mathbf{U}_m \mathbf{U}_m) = -\nabla p_m + \nabla \cdot [\mu_m (\nabla \mathbf{U}_m + \nabla \mathbf{U}_m^T)] + \nabla \cdot (\sum_{k=1}^{2} \alpha_k \rho_k \mathbf{U}_{dr,k} \mathbf{U}_{dr,k}) + \mathbf{F}_m$$

where $\mu_m = \sum_{k=1}^{2} \alpha_k \mu_k$ is the mixture viscosity and $\mathbf{U}_{dr,k} = \mathbf{U}_k - \mathbf{U}_m$ is the drift velocity (velocity of phase $k$ relative to the center of the mixture mass). The drift velocity is calculated using an Algebraic Slip Model (ASM). The ASM provides an algebraic relation for the relative velocities and a local equilibrium between the phases. The slip velocity (velocity of dispersed phase relative to the velocity of continuous phase) is $\mathbf{U}_{DC} = \mathbf{U}_D - \mathbf{U}_C$. So, the drift velocity of dispersed phase is calculated can be expressed as

$$\mathbf{U}_{dr,D} = \mathbf{U}_D - \mathbf{U}_m = \left(1 - \frac{\alpha_D \rho_D}{\rho_m}\right) \mathbf{U}_{DC}$$

where the slip velocity $\mathbf{U}_{DC}$ is calculated using the Manninen et al. (1996) model as

$$\mathbf{U}_{DC} = \frac{\tau_D}{f_{drag}} \left(\frac{\rho_D - \rho_m}{\rho_D}\right) \mathbf{a} - \frac{\eta_t}{\alpha_t} \left(\frac{\mathbf{v}_{\alpha_D}}{\alpha_D} - \frac{\mathbf{v}_{\alpha_C}}{\alpha_C}\right)$$
where $\tau_D = \frac{\rho_D d_D^2}{18 \mu_c}$ is the droplet relaxation time, $d_D$ is the diameter of dispersed droplet and $a = -(U_m \cdot \nabla)U_m - \partial U_m / \partial t$ is the acceleration of dispersed droplets, $\eta_t$ is the turbulent diffusivity and $\sigma_t$ is the Prandtl-Schmidt number (see Ansys Fluent documentation: theory guide, 2013). The drag force is calculated using the Schiller and Naumann (1935) correlation which is given as

$$f_{drag} = \begin{cases} 
1 + 0.15 Re^{0.687} & Re \leq 1000 \\
0.0183 Re & Re > 1000 
\end{cases}$$

The equation of volume fraction for the secondary phase is

$$\frac{\partial}{\partial t} (\alpha_D \rho_D) + \nabla \cdot (\alpha_D \rho_D U_m) = -\nabla \cdot (\alpha_D \rho_D U_{dr,D})$$

All the equations shown in this section are instantaneous. The ensemble averaging is performed in these equations and the Reynolds averaged mixture model is solved using the Reynolds stress model described in APPENDIX A1.
APPENDIX B

Separation efficiency

In case of oil-water separations using a hydrocyclone, the dispersed oil is the lighter phase and is, therefore, predominantly found in the overflow stream. Separation efficiency of a liquid-liquid separation hydrocyclone can be calculated using the mass balance equation for the dispersed, which is given as

\[
\dot{m}_{DF} = \dot{m}_{DO} + \dot{m}_{DU} \rightarrow Q_F C_{DF} = Q_O C_{DO} + Q_U C_{DU}
\]  

(b1)

where \( \dot{m}_D \) is the mass flow rate of dispersed phase, \( Q \) is the volume flow rate of oil-water mixture and \( C_D \) is the concentration of dispersed phase. The overall separation efficiency can be calculated as

\[
\eta_O = \frac{\dot{m}_{DO}}{\dot{m}_{DF}} = \frac{Q_O C_{DO}}{Q_F C_{DF}}
\]

The overall reduced separation efficiency can be calculated as

\[
\eta_{OR} = \eta_O - R_O = \frac{Q_O C_{DO} - Q_O}{Q_F C_{DF} - Q_F} \frac{Q_F}{Q_F}
\]

\[
\Rightarrow \eta_{OR} = \frac{Q_F C_{DF} - Q_U C_{DU} - Q_F - Q_U}{Q_F C_{DF}} \frac{Q_F - Q_O}{Q_F}
\]

\[
\Rightarrow \eta_{OR} = \frac{Q_F C_{DF} - Q_U C_{DU} - Q_F C_{DF} + Q_U C_{DF}}{Q_F}
\]

\[
\Rightarrow \eta_{OR} = \frac{-Q_U C_{DU} + Q_U C_{DF}}{Q_U C_{DF}} = 1 - \frac{C_{DU}}{C_{DF}}
\]
APPENDIX C

Grid independent study

The grid independent study is performed to examine whether the grid resolution is sufficiently fine to provide solution with acceptable accuracy. Coarse mesh deteriorates the numerical accuracy and finer mesh reduces the numerical uncertainty. However, from the computation point of view, it is not desirable to create extremely fine mesh. So, the grid independent study allows us to identify the minimum grid resolution that can provide solution with acceptable accuracy. The grid convergence index (GCI) method proposed by Celik (2008) is implemented in this study. The steps for the GCI methods are given below:

Step 1: Create three different set of grids and run the numerical simulation for each set of grids. The grid resolution should be such that the grid refinement factor \( r = x_{coarse}/x_{fine} \) is greater than 1.3. Here, \( x \) is the grid size which is calculated as \( x = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i) \right]^{1/3} \) where \( N \) is the number of grid cells and \( \Delta V_i \) is the volume of \( i^{th} \) cell.

Step 2: Let \( x_1 < x_2 < x_3 \) and \( r_{21} = x_2/x_1, r_{32} = x_3/x_2 \). Calculate the apparent order \( p \) by solving the Eq. (c1) numerically. Fixed-point iteration with initial guess value can be used for the numerical solution.

\[
p = \frac{1}{\ln(r_{21})} |\ln|\varepsilon_{32}/\varepsilon_{21}| + f(p)| \quad \text{where} \quad f(p) = \ln \left( \frac{r_{21}^p - s}{r_{32}^p - s} \right) \text{ and } s = 1 \times \text{sign} \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \quad (c1)
\]

Here, \( s = \begin{cases} 
-1 & \text{if } \varepsilon_{32}/\varepsilon_{21} < 0 \\
0 & \text{if } \varepsilon_{32}/\varepsilon_{21} = 0 \\
1 & \text{if } \varepsilon_{32}/\varepsilon_{21} > 0 
\end{cases} \)
where \( \epsilon_{32} = \phi_3 - \phi_2, \epsilon_{21} = \phi_2 - \phi_1 \) and \( \phi_k \) is the value of solution variable for the \( k^{th} \) set of grid.

**Step 3:** Calculate the extrapolated values of the variable from

\[
\phi_{\text{ext}}^{21} = \left( r_{21}^p \phi_1 - \phi_2 \right) / \left( r_{21}^p - 1 \right)
\]

and

**Step 4:** Calculate the errors and GCI from the follow relations

Approximate relative error: \( e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right| \)

Extrapolated relative error: \( e_{\text{ext}}^{21} = \left| \frac{\phi_{\text{ext}}^{21} - \phi_1}{\phi_{\text{ext}}^{21}} \right| \)

Grid convergence index (GCI): \( GCI_{\text{fine}}^{21} = \frac{1.25 e_a^{21}}{r_{21}^p - 1} \)
APPENDIX D

Swirl chamber design profile

D1. Parabolic profile with asymptote near the tail:

Figure D1: A section of a hydrocyclone having parabolic swirl chamber with asymptote near the tail.

Governing equation: \( r = a(z - L_t)^2 + b(z - L_t) + c \)  \hspace{1cm} (D1)

BC-1: at \( z = L_t, r = \frac{D}{2} \)

\[ \Rightarrow c = \frac{D}{2} \]

BC-2: at \( z = L_t + L_s, r = \frac{D_u}{2} \)

\[ \Rightarrow \frac{D_u}{2} = al_s^2 + bL_s + \frac{D}{2} \]

\[ \Rightarrow al_s^2 + bL_s = -\left(\frac{D - D_u}{2}\right) \]
BC-3: at $z = L_i + L_s$, $\frac{dr}{dz} = 0$

$$\Rightarrow \frac{dr}{dz} = 2a(z - L_i) + b$$

$$0 = 2aL_s + b$$

$$b = -2aL_s$$

Eq. (D1) $\Rightarrow$

$$\Rightarrow aL_s^2 - 2aL_s^2 = -\left(\frac{D - D_u}{2}\right)$$

$$\Rightarrow a = \left(\frac{D - D_u}{2L_s^2}\right)$$

and

$$b = -\left(\frac{D - D_u}{L_s}\right)$$

$$r = \frac{D}{2} + \left(\frac{D - D_u}{2L_s^2}\right)\left[(z - L_i)^2 - 2L_s(z - L_i)\right]$$

$$r = \begin{cases} 
\frac{D}{2} + \left(\frac{D - D_u}{2L_s^2}\right)\left[(z - L_i)^2 - 2L_s(z - L_i)\right] & \text{for } z < L_i \\
\frac{D}{2} + \left(\frac{D - D_u}{2L_s^2}\right)\left[(z - L_i)^2 - 2L_s(z - L_i)\right] & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D_u}{2} & \text{for } z > (L_i + L_s) 
\end{cases}$$
D2. Parabolic profile with asymptote near the top:

Figure D2: A section of a hydrocyclone having parabolic swirl chamber with asymptote near the top.

Governing equation: \( z - L_i = a \left( \frac{D}{2} - r \right)^2 + b \left( \frac{D}{2} - r \right) + c \)  \hspace{1cm} (D2)

BC-1: at \( z = L_i, r = \frac{D}{2} \)

\[ \Rightarrow c = 0 \]

BC-2: at \( z = L_i \) and \( r = \frac{D}{2}, \frac{dz}{dr} = 0 \)

\[ \Rightarrow \frac{dz}{dr} = 2a \left( \frac{D}{2} - r \right) + b \]

\[ b = 0 \]

BC-3: at \( z = L_i + L_s, r = \frac{D_u}{2} \)

\[ \Rightarrow L_s = a \left( \frac{D}{2} - \frac{D_u}{2} \right)^2 \]

\[ a = \frac{L_s}{\left( \frac{D}{2} - \frac{D_u}{2} \right)^2} \]
Eq. (D2) ⇒

\[ z - L_i = \frac{L_s}{\left(\frac{D}{2} - \frac{D_u}{2}\right)^2} \left(\frac{D}{2} - r\right)^2 \]

⇒ \[ r = \frac{D}{2} - \left(\frac{D}{2} - \frac{D_u}{2}\right) \sqrt{\frac{z - L_i}{L_s}} \]

\[ r = \begin{cases} 
\frac{D}{2} - \left(\frac{D}{2} - \frac{D_u}{2}\right) \sqrt{\frac{z - L_i}{L_s}} & \text{for } z < L_i \\
\frac{D_u}{2} & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D}{2} - \left(\frac{D}{2} - \frac{D_u}{2}\right) \sqrt{\frac{z - L_i}{L_s}} & \text{for } z > (L_i + L_s) 
\end{cases} \]

D3: Hyperbolic profile with asymptote at the both ends:

Figure D3: A section of a hydrocyclone having hyperbolic swirl chamber with asymptote at the both ends.

Governing equation:

\[ \left(\frac{D}{2} - r\right)^2 + \frac{(L_i + L_s - z)^2}{b} = 1 \quad (D3) \]
BC-1: at \( z = L_i, r = \frac{D}{2} \)

\[ \Rightarrow b = L_s^2 \]

BC-2: at \( z = L_i + L_s, r = \frac{D_u}{2} \)

\[ \Rightarrow a = \left( \frac{D - D_u}{2} \right)^2 \]

Eq. (D3) \Rightarrow

\[ \frac{(\frac{D}{2} - r)^2}{(\frac{D - D_u}{2})^2} + \frac{(L_i + L_s - z)^2}{L_s^2} = 1 \]

\[ r = \frac{D}{2} - \left( \frac{D - D_u}{2} \right) \sqrt{1 - \left( \frac{L_i + L_s - z}{L_s} \right)^2} \]

\[ r = \begin{cases} 
\frac{D}{2} - \left( \frac{D - D_u}{2} \right) \sqrt{1 - \left( \frac{L_i + L_s - z}{L_s} \right)^2} & \text{for } z < L_i \\
\frac{D_u}{2} & \text{for } L_i \leq z \leq (L_i + L_s) \\
\frac{D}{2} & \text{for } z > (L_i + L_s) 
\end{cases} \]
REFERENCES


Komrakova, A. E., Derksen, J., 2012. Lattice Boltzmann simulation of drop deformation and breakup in shear flow. AIChE annual meeting, Pittsburgh, PA


Wilson, S., 2014. U.S. environmental protection agency regulatory and environmental framework for oil and gas extraction. PWS meeting, Houston, TX.


