CFD MODELING OF A CENTRIFUGE FOR OIL - WATER SEPARATION

H.G.S.M.Thilakarathna

(149268 K)

Degree of Master of Science

Department of Chemical and Process Engineering

University of Moratuwa

Sri Lanka

April 2017

CFD MODELING OF A CENTRIFUGE FOR OIL - WATER SEPARATION

Hirimuthu Godage Sachini Manjula Thilakarathna

(149268 K)

Thesis submitted in partial fulfillment of the requirements for the degree Master of Science

Department of Chemical and Process Engineering

University of Moratuwa

Sri Lanka

April 2017

DECLARATION

I declare that this is my own work and this thesis does not incorporate without acknowledgement any material previously submitted for a Degree or Diploma in any other University or institute of higher learning and to the best of my knowledge and belief it does not contain any material previously published or written by another person except where the acknowledgement is made in the text.

Also, I hereby grant to University of Moratuwa the non-exclusive right to reproduce and distribute my thesis, in whole or in part in print, electronic or other medium. I retain the right to use this content in whole or part in future works (such as articles or books).

Signature: Date:

The above candidate has carried out research for the Masters under my supervision.

Name of the supervisor: Dr. Mahinsasa Narayana

Signature of the supervisor: Date:

ABSTRACT

Centrifugation is an efficient, economical and environmentally friendly method to remove undesirable water content from oil and water mixtures and separate out the desired oil content. Moreover, the disc stack centrifuges are widely used for separating liquids of different densities and applied in industrial coconut oil clarification as well.

The main focus of this research work is to model the fluid flow inside the Westfalia disc stack centrifuge using Ansys Fluent and identify the flow behavior. With the availability of the limited computer hardware facility, the model has been run without the discs to avoid complexities. In the developed 3 dimensional model, the fluid dynamic behavior of the multiphase flow has been considered and modeled using the VOF multiphase model available in fluent.

The step by step procedure of the model development has been discussed such as the very first stage of geometry selection, drawing and importing to the fluent, mesh generation, all solution set ups and even the two stage simulation procedure.

The simulation results of this research work provides an out line of the resulted flow parameters of velocity and pressure profiles, turbulent effects such as turbulence intensity, turbulent kinetic energy, and specific dissipation rate and also the phase volume fractions which have been saved in every critical stage of the simulation process. Despite the phase volume fraction which has been experimentally validated, all other results were theoretically validated.

CFD modeling of flow behavior inside the centrifuges is not a popular topic among the researchers due to the complex flow patterns and the requirement of advanced computer hardware facility. However this research work provides a platform to model the similar flow behaviors and even to model the same case including the discs.

KEY WORDS:

CFD, centrifuge, VOF, multiphase

DEDICATION

I dedicate this thesis to my courageous father, gracious mother and supportive husband for their unconditional love and affection.

ACKNOWLEDGMENTS

This research work would not be a success without the guidance and support of several inspiring personalities whom I am greatly in debt.

First and foremost I would like to express my sincere gratitude to my outstanding research supervisor Dr.Mahinsasa Narayana for his continuous guidance, support, motivation and valuable knowledge sharing sessions throughout the whole period of master program of Sustainable Process Development. Thank you very much sir for being a role model to me.

I extend my immense gratitude to Dr.Sanja Gunawardena and Dr.P.G. Rathnasiri who are the current and former Heads of the Department of Chemical and Process Engineering, University of Moratuwa.

I wish to express my sincere thanks to non-academic staff of the department for their valuable assistance that has been given throughout the research work. My special appreciation goes to Mr.Shantha Peris, Mr.Nihal Perera, Mr.Senarath Epa and Mr.Sunil Dayananda for spending their precious time in facilitating me.

I am profoundly in debt to my friends, Ms. Thamali Jayawickrama, Ms. Bhagya Herath, Ms. Imesha Samarathunga, Ms.Sureshini Warnasooriya and Ms.Gayani Jayathunga for the valuable comments and sharing their research experience.

I would like to express my immense gratitude to my parents, for giving me life in the first place and for educating me with all arts of life and also my two loving brothers for being my life's strength.

Last but not least, I wish to express my sincere thanks to my loving husband for his silent motivation and sustained patience during the many long hours I spent working on this research and thesis at the university.

TABLE OF CONTENTS

Declaration i

Abstract ii

Dedication iii

Acknowledgements iv

Table of contents v

List of figures viii

List of tables xi

List of abbreviations xii

List of appendices xiii

1 : INTRODUCTION 1

1.1 Background 1

1.2 History of Disc Stack Centrifuge 3

1.3 Industrial applications of disc stack centrifuge 3

1.4 CFD modeling of swirling effect 4

1.4.1 Swirling flows inside cyclones and centrifuges 4

1.4.2 Summary of CFD modeling of swirling flows 7

1.5 Research Objectives and Scope 8

1.5.1 Objectives 8

1.5.2 Scope 8

2 : MODELING THEORY AND TECHNIQUE 10

2.1 Theory behind centrifugation of liquid mixtures in a disc centrifuge . 10

2.2 Basic fluid flow equations for simulation 12

2.2.1 Mass Conservation 12
2.2.2 Momentum Conservation 13
2.3 Modeling the moving zone 14
2.3.1 Equations for frame motion with relative velocity formulation 14
2.3.2 Equations for mesh motion with absolute velocity formulation 15
2.4 Turbulence Modeling 15
2.4.1 Model selection for turbulence modeling 17
2.5 Solver Selection 20
2.6 Multi-phase Flow Modeling 21
2.7 Selection of Discretization schemes 23
2.7.1 Finite Volume Method 23
2.7.2 Selected Discretization Schemes in Modeling 24

3 : METHODOLOGY 28
3.1 Geometry and CFD modeling 28
3.1.1 The Geometry Selection 28
3.1.2 Intended Centrifuge Geometry 29
3.1.3 Overview of CFD modeling approach 31
3.2 Mesh Interface 33
3.2.1 Applied features in ANSYS Fluent Meshing 33
3.2.2 Quality of the generated mesh 35
3.3 The solution setup 36
3.3.1 Models 36
3.3.2 Materials and phases 36
3.3.3 Boundary Conditions 36
3.3.4 Solution methods and controls 37
3.3.5 Solution controls and initialization 38
3.4 The Simulation Procedure 38
3.5 Model Validation 39

4 : RESULTS AND DISCUSSION **40**

4.1 Simulation Results and Discussion 40

4.1.1 Velocity Profile 40

4.1.2 Pressure Profile 43

4.1.3 Measures of Turbulence 45

4.1.4 Phase volume fraction 53

4.2 Experiment Procedure, Results and Discussion 72

4.2.1 Procedure 73

4.2.2 Results and Discussion 74

5 : CONCLUSIONS AND RECOMMENDATIONS **78**

5.1 CFD Simulation Results 78

5.2 Experiment Results 79

5.3 Centrifugation as a water removal method from oil-water mixtures . . 79

5.4 Recommendations 79

Appendix A MODIFIED REYNOLDS NUMBER CALCULATION **81**

REFERENCE LIST **83**

LIST OF FIGURES

1.1 The disc stack centrifuge 4

2.1 Two immiscible liquid separation using disc centrifuge 11
2.2 Selected viscous model options in Fluent 18
2.3 Solution procedure of pressure based solver 21
2.4 Selected solution methods and second order time discretization for implicit VOF 27

3.1 The Westfalia Centrifugal Separator 29
3.2 Liquid Flow region of the selected centrifuge geometry 30
3.3 Basic steps of problem analysis in Fluent-CFD 32
3.4 Applied mesh features 34
3.5 The generated tri/tet mesh for centrifuge 34
3.6 Cross section of the generated mesh 35

4.1 Graph of relative velocity magnitude vs 'z' direction in the mid plane 40
4.2 The fitted curve for relative velocity magnitude in positive z direction in the mid plane 41
4.3 Contour diagram of velocity 42
4.4 Vector diagram of velocity 43
4.5 Graph of static pressure(gauge) vs Z in the mid plane of the centrifuge 44
4.6 Graph of static pressure(absolute) vs Z in the mid plane of the centrifuge 44
4.7 Graph of static pressure(guage) vs Z at inlet and two outlets 45
4.8 Graph of Turbulence Intensity vs Z at inlet and two outlets 46
4.9 Graph of Turbulent kinetic energy vs Z at inlet and two outlets 48
4.10 Graph of Turbulent kinetic energy vs Z at the mid plane 48
4.11 Graph of Turbulent kinetic energy vs Turbulence Intensity at the mid plane 50
4.12 Graph of Turbulent dissipation rate vs Z at the mid plane 52

4.13 Graph of Specific dissipation rate vs Z at the mid plane 52
4.14 Oil volume fraction after 1000 iterations($\omega = 102.8 rads^{-1}$) 54
4.15 Oil volume fraction after 4000 iterations($\omega = 102.8 rads^{-1}$) 54
4.16 Oil volume fraction after 1s($\omega = 102.8 rads^{-1}$) 55
4.17 Oil volume fraction after 4s($\omega = 102.8 rads^{-1}$) 55
4.18 Oil volume fraction after 5s($\omega = 308.4 rads^{-1}$) 56
4.19 Oil volume fraction after 8s($\omega = 308.4 rads^{-1}$) 57
4.20 Oil volume fraction after 9s($\omega = 514 rads^{-1}$) 57
4.21 Oil volume fraction after 12s($\omega = 514 rads^{-1}$) 58
4.22 Oil volume fraction after 13s($\omega = 719.6 rads^{-1}$) 58
4.23 Oil volume fraction after 16s($\omega = 719.6 rads^{-1}$) 59
4.24 Oil volume fraction after 17s($\omega = 1028 rads^{-1}$) 60
4.25 Oil volume fraction after 18s($\omega = 1028 rads^{-1}$) 60
4.26 Oil volume fraction after 19s($\omega = 1028 rads^{-1}$) 61
4.27 Oil volume fraction after 20s($\omega = 1028 rads^{-1}$) 61
4.28 Oil volume fraction after 21s($\omega = 1028 rads^{-1}$) 62
4.29 Oil volume fraction after 24s($\omega = 1028 rads^{-1}$) 63
4.30 Oil volume fraction after 27s($\omega = 1028 rads^{-1}$) 63
4.31 Oil volume fraction after 30s($\omega = 1028 rads^{-1}$) 64
4.32 Oil volume fraction after 32s($\omega = 1028 rads^{-1}$) 65
4.33 Oil volume fraction after 32.25s($\omega = 1028 rads^{-1}$) 65
4.34 Oil volume fraction after 32.5s($\omega = 1028 rads^{-1}$) 66
4.35 Oil volume fraction after 32.75s($\omega = 1028 rads^{-1}$) 66
4.36 Oil volume fraction after 33s($\omega = 1028 rads^{-1}$) 67
4.37 Oil volume fraction after 33.25s($\omega = 1028 rads^{-1}$) 68
4.38 Oil volume fraction after 33.5s($\omega = 1028 rads^{-1}$) 68
4.39 Oil volume fraction after 33.75s($\omega = 1028 rads^{-1}$) 69
4.40 Oil volume fraction after 34s($\omega = 1028 rads^{-1}$) 69
4.41 Oil volume fraction after 34.25s($\omega = 1028 rads^{-1}$) 70
4.42 Oil volume fraction after 34.5s($\omega = 1028 rads^{-1}$) 70

4.43 Oil volume fraction after 34.75s($\omega = 1028 rads^{-1}$) 71
4.44 Oil volume fraction after 35s($\omega = 1028 rads^{-1}$) 72
4.45 Oil volume fraction after 36s($\omega = 1028 rads^{-1}$) 72
4.46 Inlet coconut oil-water mixture inside the inlet drum 73
4.47 Liquid out flows after trial 3 74
4.48 Outlet 2 liquid (water) of Trial 3 75
4.49 Water volume fraction in outlet 2 and the mid plane 76
4.50 Oil - water separation in side the separating funnel 76
4.51 Oil volume fraction of a plane below 0.8cm of outlet 1 77

LIST OF TABLES

2.1 Interface scheme comparison for VOF scheme 26

3.1 Quality parameters of the generated mesh 35
3.2 Boundary Conditions 37

4.1 Experiment results without discs 74
4.2 Experiment results of outlet 1 77

LIST OF ABBREVIATIONS

CFD	Compoutational Fluid Dynamics
DPM	Discrete Phase Model
FVM	Finite Volume Method
LES	Large Eddy Simulations
MRF	Moving Reference Frame
PISO	Pressure Implicit with Splitting of Operator
RANS	Reynolds Average Navier Stokes Equation
RNG	Re-Normalization Group
RSM	Reynolds Stress Model
SST	Shear Stress Transport
VOF	Volume Of Fluid
Fr	Froude Number

LIST OF APPENDICES

Appendix - A MODIFIED REYNOLDS NUMBER CALCULATION 81