SWIRLING STABILITY OF FLUID AT HIGH VELOCITY FLOW IN PIPE

NOR AMIRAH BINTI CHE MOHAMAD NOR

This report is submitted in partial fulfillment of the requirements needed for the award of Bachelor in Chemical Engineering (Hons.)

FACULTY OF CHEMICAL ENGINEERING UNIVERSITI TEKNOLOGI MARA SHAH ALAM

JULY 2017

ACKNOWLEDGEMENT

In preparing this research project report, many people including academicians and lecturers had involved. They had contributed on my understanding to perform this project. On top of that, I would like to give a special thank you to my supervisor, Sir Hanafiah bin Zainal Abidin for all the guidance and time spent from the beginning of the project until completion. Because of his expertise in this scope of research, I am able to gain as much knowledge as I can. In fact, it gave me plenty of ideas to write this research paper. Other than that, a huge gratitude goes to Universiti Teknologi Mara for giving me the opportunity to involve and establish this research paper successfully. Last but not least, I wish to express my appreciation to my parents and colleagues for the support and encouragement along the journey.

ABSTRACT

The purpose of this study is to investigate the performance and flow of fluid inside a cyclone. A cyclone present a complex flow structure which gave different prediction depends on geometry and flow velocity. A numerical simulation is performed to simulate the flow of fluid. A three-dimension simulation of single phase fluid within cyclone has been carried out using Computational Fluid Dynamics (CFD) with FLUENT software. The solver predictions using k- ε model to predict the turbulent flow inside the cyclone. Patterns of flow in every parts of cyclone are observed in order to identify the performance of separation process. Comparisons between sweep surfaces are made based on vectors and streamlines. Velocity profile and swirling stability at the vortex core is observed. High velocity fluid as an inlet stream which is 10 m/s caused the forced vortex at the center of cyclone bigger in shape. It can be shown that high velocity region mostly occurred at the wall of cyclone and cone chamber experienced low velocity region.

TABLE OF CONTENTS

PAGE

DECLARATION	ii
CERTIFICATION	iii
ACKNOWLEDGEMENT	iv
ABSTRACT	v
TABLE OF CONTENTS	vi
LIST OF TABLES	vii
LIST OF FIGURES	viii
LIST OF ABBREVIATIONS	X
LIST OF SYMBOLS	xi

CHAPTER 1 INTRODUCTION

1.1	Research Background	1
1.2	2 Problem Statement	3
1.3	3 Objectives of Research	5
1.4	Scope of Research	6

CHAPTER 2 LITERATURE REVIEW

2.1 Experimental Studies	7
2.2 Design of Cyclone	10
2.3 Theories of CFD	12
2.3.1 Method	12
2.3.2 Applications	14
2.4 Numerical Simulation	
2.4.1 Continuity Equation	14
2.4.2 Conservation Of Mass	15

CHAPTER 1

INTRODUCTION

1.1 Research Background

In the past decade, there are a lot of problem on how to improve the performance of separation equipment and also equipment that involve flow. This problem mostly occurred in aerospace industries and chemical industries. One of the reason is people cannot find the suitable solution because of the limited knowledge and source. Previously, the progress in IT is slow not many researchers are doing research on this major. At 1960s, researchers start to introduce the first program to analyze the flow and performance of process equipment. The program called computational fluid dynamic and become the first one that was developed. Unfortunately, the restriction of computers was severely limited in that era. Then, improvement had made during 1980s that commercial codes became available (Bakker et al, 2001).

Computational fluid dynamic is firstly used in aerospace, automotive and nuclear industries where the most of physics term involved. It is much easier to manipulate the flow using CFD compare to experimental method and manual calculation. Nevertheless, because of complicated system which involves many unit operations, multiphase flow and chemical reaction, application of CFD became