NUMERICAL SIMULATION OF VORTEX COMBUSTION FOR VARIOUS AIR-FUEL INLET CONFIGURATIONS

JUMARI BIN PORMAN

A project report submitted in partial fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

JANUARY 2015

Specially dedicated to beloved childrens *Aqil Harraz, Akmal Hazim* and wife *Liyana Baharudin*

ACKNOWLEDGEMENT

First and foremost I would like to express my sincere gratitude to my supervisor, Assoc. Prof. Dr Mazlan Abd Wahid, who has supported me throughout my thesis with his excellent guidance, patience and knowledge and providing me with an excellent environment for doing research. I attribute the level of my Masters degree to his encouragement and effort and without him this thesis, would not have been completed or written.

Special thanks also to colleagues of Mechanical Engineering post graduate study at Universiti Technologi Malaysia Kuala Lumpur. Their support and care helped me overcome setbacks and stay focused on my post graduate study. I greatly value their friendship and I deeply appreciate their belief in me.

I would also like to thank to my parents and parents in-law. They were always supporting me and encouraging me with their best wishes.

Finally, I would like to thank to my dearest wife, Liyana Baharudin. She is always there to cheer and understand me during good and bad time.

ABSTRACT

The purpose of this research is to study the effect of various air-fuel inlet configurations to the asymmetric vortex combustor in the non-premixed combustion of methane-air mixture using the standard k-ε turbulent model on Fluent Ansys commercial CFD software. In this study, the investigation is mainly emphasizes the influence of varying the numbers of air inlet of the vortex combustor. The simulation study has been perform in two conditions which are on cold flow (non-reacting flow) using air to define the structure of vortex flow inside the vortex combustor and also on reacting flow with mixture reaction on various equivalence ratio and various configuration numbers of air-inlets. From the isothermal simulation with air, the non-reacting flow field study was found maintain the forced-vortex azimuthal velocity patterns with strongly decaying vortex structure as per previous study. A central recirculation zone (CRZ) and two secondary recirculation zone (SRZ) also found develop in the asymmetric combustor however the size of CRZ and SRZ to be found depend on the velocity inlet magnitude and numbers of air inlet port. The study on reacting flow conditions revealed by increasing the numbers of air inlet, a better chaotic mixing observed at the bottom of the combustor which judged from the temperature distributions contour in the vortex combustor. The local temperature inside the vortex combustor observed proportional to equivalence ratio. The trend of the flame height observed proportional to equivalence ratio and predicted between 10 mm to 50 mm which is comparable to previous investigation.

ABSTRAK

Tujuan utama penyelidikkan ini adalah untuk mengkaji tentang kesan kepelbagaian konfigurasi geometri kemasukan bahan api udara bagi sebuah pembakar vortex bergeometri asimetri yang menggunakan campuran bahan api metana udara secara tidak bercampur dengan menggunakan model gelora k-ε standard yang terdapat di dalam perisian CFD komersial. Dalam kajian ini, penekanan yang lebih diberikan terhadap kesan kepelbagaian bilangan kemasukan udara ke dalam pembakar vortex bergeometri asimetri. Kajian simulasi ini telah dilaksanakan dalam dua keadaan iaitu kajian aliran tanpa reaksi dan kajian yang melibatkan pembakaran. Kajian aliran tanpa reaksi adalah simulasi menggunakan udara bertujuan untuk melihat struktur aliran vortex di dalam ruang pembakar, manakala kajian aliran reaksi adalah simulasi dengan nisbah percampuran bahan api yang berbeza dan juga kepelbagaian konfigurasi kemasukan udara. Dari simulasi aliran tanpa reaksi, didapati struktur aliran yang berlaku di dalam pembakar vortex bergeometri asimetri menunjukkan pola kelajuan azimut vortex paksaan dengan struktur pusaran yang memupus seperti mana hasil kajian yang pernah dilakukan sebelum ini. Satu zon pusaran utama dan zon pusaran terbitan telah dijumpai terbentuk didalam ruang pembakar vortex bergeometri asimetri ini. Walau bagaimanapun saiz zon pusaran utama dan zon pusaran terbitan didapati berkadaran dengan magnitud kelajuan dan juga bilangan kemasukan udara. Kajian dalam keadaan aliran reaksi menunjukkan bahawa dengan peningkatan bilangan kemasukan udara, didapati campuran gelora yang lebih setara terbentuk pada dasar pembakar dan ini diperkukuhkan melalui permerhatian terhadap kontur tebaran suhu di dalam ruang pembakar vortex bergeometri asimetri. Suhu setempat di dalam pembakar didapati berkadaran dengan pertambahan nisbah percampuran bahan api udara. Ketinggian api juga di dapati berkadaran dengan pertambahan nibah percampuran bahan api udara dan nilainya dianggarkan di antara 10 mm hingga 50 mm iaitu setara seperti kajian yang pernah dilakukan sebelum ini.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATION

CFD	Computational Fluid Dynamics
CRZ	Central Recirculation Zone
SRZ	Secondary Recirculation Zone

LIST OF SYMBOLS

AF_{actual}	Air Fuel Ratio Actual
$AF_{theoretical}$	Air Fuel Ratio Theoretical
K	Kelvin
k-ε	k-epsilon turbulence model
P	Pressure (Pa)
Re	Reynolds Number
T	Temperature (K)
V	Velocity (m/s)
V_{in}	Velocity Inlet (m/s)
μ	Dynamic Viscosity ($kg.m^{-1}.s^{-1}$)
ρ	Density (kg/m^{3})
Φ	Equivalence Ratio

REFERENCES

A.M. Steinberg, C.M. Arndt, W. Meier (2012). Parametric study of vortex structures and their dynamics in swirl-stabilized combustion.Proceedings of the Combustion Institute

Amit Gupta, Raganathan Kumar (2007). Three dimensional turbulent swirling flows in a cylinder: Experiments and computations. International Journal of Heat and Fluid Flow

B.E. Launder, D.B. Spalding (1974). The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering.

Consalvi, J. L., Y. Pizzo, B. Porterie and J. L. Torero (2007). On the flame height definition for upward flame spread. Fire Safety Journal

Khalid M. Saqr, Hossam S. Aly, Hassan I. Kareem, Mohsin M. Sies, Mazlan A. Wahid (2010). Computations of shear driven vortex flow in a cylindrical cavity using a modified k-e turbulence model. International Communications in Heat and Mass Transfer

Khalid M. Saqr (2011). Aerodynamics and Thermochemistry of Turbulent Confined Asymmetric Vortex Flames. Thesis Doctor of Philosophy Universiti Teknologi Malaysia.

Khalid M. Saqr, Hossam S. Aly, Mohsin M. Sies, Mazlan A. Wahid (2011). Computational and experimental investigations of turbulent asymmetric vortex flames. International Communications in Heat and Mass Transfer.

Mostafa Khalegi, Mazlan A. Wahid, Mohsin M. Sies, Aminuddin Saat (2013). Investigation of vortex reacting flows in Asymmetric meso scale combustor. Applied Mechanics and Materials.

Mukhopadhyay, A., I. K. Puri, S. Zelepouga and D. M. Rue (2001). Numerical simulation of methane-air nozzle burners for aluminum remelt furnaces. American Society of Mechanical Engineers, Heat Transfer Division, (Publication) HTD, New York, NY

Rajput, R. K. (2005). Internal Combustion Engines, Laxmi Publications.

Richard A. Yetter, Irvin Glassman, H. Clay Gabler (2000). Asymmetric Whirl Combustion : A New Low NOx Approach. Proceedings of the Combustion Institute.

Turns, S. R. (2000). An introduction to combustion: concepts and applications, McGraw-Hill.

Yehia A. Eldrainy, Khalid M. Saqr, Hossam S. Aly, Mohammad Nazri Mohd Jaafar (2009). CFD insight of the flow dynamics in a novel swirler for gas turbine combustors. International Communications in Heat and Mass Transfer.