# FLOW CHARACTERISTIC OF A TWO-DIMENSIONAL DIFFUSER OF CATALYTIC CONVERTERS SITUATED CLOSE TO ENGINES USING SIMULATION

MASTURA BINTI MUTAFA

MASTER OF SCIENCE
IN MECHANICAL ENGINEERING

2019

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# Faculty of Mechanical Engineering

## FLOW CHARACTERISTICS OF A TWO-DIMENSIONAL DIFFUSER OF CATALYTIC CONVERTERS SITUATED CLOSE TO ENGINES USING SIMULATION

**Mastura binti Mutafa**

**Master of Science in Mechanical Engineering**

**2019**

# FLOW CHARACTERISTICS OF A TWO-DIMENSIONAL DIFFUSER OF CATALYTIC CONVERTERS SITUATED CLOSE TO ENGINES USING SIMULATION

**MASTURA BINTI MUTAFA**

**A thesis submitted**
**in fulfillment of the requirements for the degree of Master of Science**
**in Mechanical Engineering**

**Faculty of Mechanical Engineering**

**UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

**2019**

**DECLARATION**

I declare that this thesis entitled “Flow Characteristics of a Two-Dimensional Diffuser of Catalytic Converters Situated Close to Engines Using Simulation” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

| | |
|---|---|
| Signature | : …………………….. |
| Name | : Mastura Binti Mutafa |
| Date | : …………………….. |

**APPROVAL**

I hereby declare that I have read this thesis and in my own opinion this thesis is sufficient in terms of scope and quality for the award of Master of Science in Mechanical Engineering.

Signature : …………………………………...

Supervisor Name : Dr. Ahmad Kamal Bin Mat Yamin

Date : ……………………………………

**DEDICATION**

To my beloved mother and father

## ABSTRACT

A close-coupled catalytic converter is placed close to the engine to efficiently treat harmful exhaust gases during warm up. The engine exhaust obliquely enters the diffuser upstream of the catalyst monolith through an inclined inlet pipe in which affects the degree of flow separation in the diffuser as well as the conversion efficiency of the after-treatment system. With reliable mathematical models, the effect of the inlet pipe geometry can be evaluated prior to expensive manufacture and testing. The main objective of this research is to predict the steady-state oblique entry flow in the diffuser upstream of the monolith using computational fluid dynamics (CFD) alongside the flow maldistibution across the monolith. The CFD domains were simplified as planar systems to capture the internal flow through the converter. The monolith was represented as a porous medium whereas the resistance was measured with uniform air flow being presented at the front face of the monolith. The predictions were performed at steady state using Reynolds Averaged Navier-Stokes (RANS) equations at a range of Reynolds numbers (provide the value range). The results show a formation of an oblique entry jet traversing the diffuser along with a large recirculation. With respect to the velocity profile across the monolith, the maximum velocities can be observed at the region where the monolith being hit by the jet.

***ABSTRAK***

*Penukar bermangkin gandingan dekat dipasang hampir kepada enjin untuk merawat gas ekzos dengan lebih cekap semasa enjin mula dihidupkan. Gas ekzos enjin memasuki pemancar yang berada di hadapan sarang pemangkin secara menyerong melalui paip masuk di mana ia mempengaruhi tahap pemisahan aliran dalam pemancar serta kecekapan sistem rawatan ekzos. Dengan menggunakan model matematik yang tepat, kesan geometri paip masuk dapat dinilai sebelum pembuatan dan pengujian yang mahal. Objektif utama kajian ini adalah untuk mengira medan aliran berkeadaan stabil di dalam pemancar menggunakan pengkomputeran dinamik bendalir (CFD) bersama dengan ketidakseragaman aliran merentangi sarang pemangkin. Domain CFD telah diwakilkan sebagai sistem satah untuk mendapatkan aliran dalaman melalui sarang pemangkin. Sarang pemangkin telah diwakilkan sebagai medium porus di mana rintangan diukur dengan aliran udara seragam disalurkan di bahagian hadapan sarang pemangkin. Pengiraan telah dibuat pada satu julat nombor Reynolds menggunakan persamaan Reynolds Averaged Navier-Stokes (RANS). Keputusan menunjukkan pembentukan satu jet serong merentasi pemancar bersama-sama dengan satu pusaran besar. Dari segi profil halaju merentangi sarang pemangkin, halaju maksimum dapat diperhatikan pada kawasan di mana jet melanggar permukaan sarang pemangkin.*

# ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my supervisor Dr. Ahmad Kamal bin Mat Yamin for his essential supervision, support and encouragement towards the completion of this thesis.

I also would like to express my greatest gratitude to Professor Ts. Dr. Noreffendy bin Tamaldin as co-supervisor of this project, for his advices and suggestions. Special thanks to Ministry of Higher Education grant funding for the financial support through this project.

Particularly, I would also like to express my deepest gratitude to Mr. Ridzuan, the technician from automotive laboratory of Faculty of Mechanical Engineering, Mr. Firdaus, the technician from the CAE laboratory, Faculty of Mechanical Engineering and Mr. Kamaruddin, the technician from Rapid Prototyping Laboratory of Mechanical Engineering for their assistance and effort in installing ANSYS software and analysis work.

Special thanks to all my peers, my beloved parents and siblings for their moral support in completing this Master degree. Lastly, thank you to everyone who had been to the crucial parts of realization of this project.

## TABLE OF CONTENTS

## LIST OF TABLES

# LIST OF FIGURES

## LIST OF APPENDICES

## LIST OF ABBREVIATIONS AND SYMBOLS

| | | |
|---|---|---|
| 2-D | - | Two Dimensional |
| CFD | - | Computational fluid dynamic |
| CCC | - | Closed-couple catalyst |
| CO | - | Carbon monoxide |
| $CO_2$ | - | Carbon oxide |
| cpsi | - | Cell per square inch |
| HC | - | Hydrocarbon |
| $H_2O$ | - | Water |
| H-P | - | Hagen-Poiseuille relationship |
| HWA | - | Hot-wire anemometry |
| K-W | - | Kuchemann and Weber |
| $Re_c$ | - | Channel Reynolds number |
| RKE | - | Realizable k-epsilon |

## LIST OF PUBLICATIONS

JOURNAL

1. Mastura Mutafa, Ahmad Kamal Mat Yamin, Noreffendy Tamaldin. 2019. CFD Investigation of the flow characteristics of an automotive catalyst monoliths. *CFD Letters*, 11(6), pp. 39-46.

# CHAPTER 1

# INTRODUCTION

## 1.1 Introduction

A close-coupled catalytic converter (CCC) is used in modern passenger vehicles to meet the exhaust emission limits by accelerating the activation of the catalyst during warm-up. As its name implies, the CCC is directly coupled to the exhaust manifold at the expense of oblique entry flow to the diffuser due to the packaging constraint. Another type of converter is underbody catalytic converter (UBC) that is not capable of reducing emission during warm-up. Usually, the UBC were located underneath the vehicle which is far from the engine to reduce the emission post-light-off as stated in Liu et al., (2019).

Figure 1.1 illustrates the CCC design featuring three major components, i.e. inclined inlet pipe, diffuser, monolith and outlet pipe. Catalysts can be made from ceramic or metallic comprises of thousands of parallel channels with a small hydraulic diameter (~1mm) to provide large surface area for exhaust gas treatment. The channels are washcoated with noble metals such as, platinum (Pt), Rhodium (Rh) and Ruthenium (Ru) are to reduce the activation energy of the chemical reactions. The nobel metal was used from platinum group metal catalyst supported on ceramic monoliths for control of gaseous emission from automobiles.

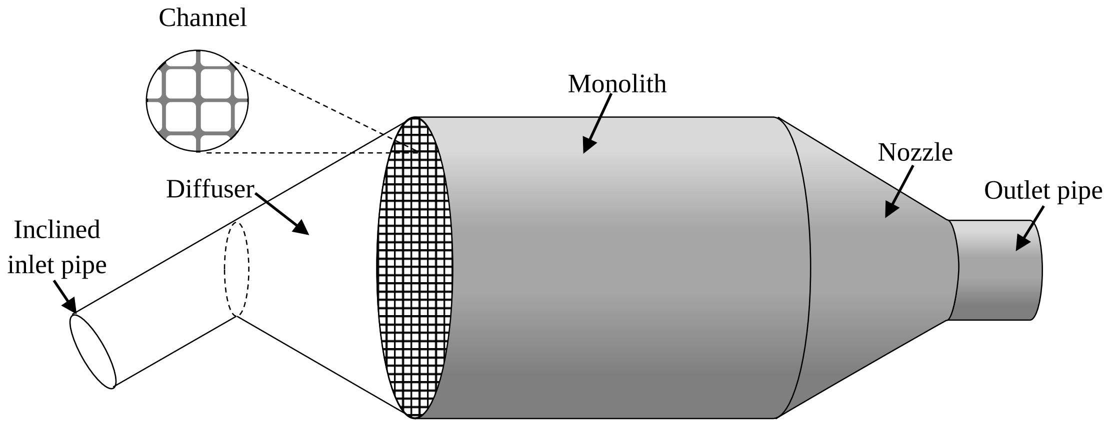


Figure 1.1: Schematic of the CCC components

Theoretically, the flow separates at the entrance to form an oblique entry jet along with a large air recirculation. The jet traverses the diffuser when air entering the diffuser with the incline inlet pipe will quickly spreading closed the front of monolith. Then, the air will enter the channels while the air circulating occurs at bottom of the inclined diffuser. During air flow entering the monolith channel it causes the air to recirculate between jet and the diffuser wall before entering the monolith channel. After the flows reach the monolith, the jet will decelerate and separated radially. The net effect of air flow entering the catalyst is the conversion efficiency was specified from the flow distribution across the monolith.

The flow maldistribution across the monolith has been long associated with sudden expansion upstream of the monolith. Porter et al., (2014) was mentioned that the introduction of ultra-thin substrate, the degree of the flow separation is anticipated due to the lower downstream resistance.  Testing and simulation work are often carried out to optimise the exhaust flow distribution and improve the efficiency of the catalyst systems. The latter is based on computational fluid dynamics (CFD) techniques where virtual prototypes of designs can be analysed.

CFD modelling were chosen to use in this research based on the cost effective and time saving on modelling the automotive catalytic converter. Modelling the automotive catalytic converter were need more on manufacturing cost and when collecting the sample of test will be more complicated if the tools of measurements were not complete. The computational domain of CFD was the most compatible to uses and if used judiciously, the accurate result with lesser computational resources will obtained. Majority, the new CFD tools were used to solve basic equations that model flow movement. Besides, CFD modelling also used to reduce time constrain because no manufacturing needed.

### 1.2 Research background

The UBC design has been thoroughly investigated from steady state condition to transient. As stated in previous research, Om Ariara Guhan et al., (2015) investigated the prediction of uniformity index and also the pressure drop of the difference types of converter which is CCC, UBC and muffler using the CFD simulations. The research analysed the pressure contours of inlet pipe, closed inlet of catalytic converter, under body catalytic converter and muffler inlet of using CFD modelling. The studies focused on the porous inlet and outlet of under body and close-coupled catalytic converter. As the result reported the total pressure drop was obtained in the close-couple catalytic converter is 72 mbar while the flow is more uniform under body of catalytic converter.

Others method to measure the upstream diffuser for steady flow using Particle Image Velocimetry (PIV) was reported. Quadri et al., (2015) examined the measurement flow over a monolith of automotive catalytic converter placed at the end (downstream) of planar wide-angled diffuser. It was reported that the PIV was obtained at the diffuser and Hot Wire Anemometry (HWA) technique was picked to predict and measurement the flow distribution at the end of monolith exits. As resulted in this research, the jet formation traversed the planar wide-angled diffuser will spread out before entering the monolith channel. It is because formations of jet will feature a potential core and saddle-type of velocity profile when free shear layer was separated. During the formation of jet core, the two large recirculation was developed narrowing from the inlet of planar wide-angled diffuser.

With the introduction of CCC design of catalyst, the steady flow analysis is in appropriate as the flow is highly pulsating. The studies that have been performed for these systems have been made using either experimental of simulation. For example, Zhao et al., (1995) and Park et al., (1998) were studying on the flow structure of catalytic converter and focusing on the internal of catalytic converter of a gasoline engine through the applications of the laser Doppler velocimetry (LDV) technique. Using the CFD numerical calculation shows that the result of overall flow characteristics of exhaust system was governed largely by the primary flow was agreed to the LDV technique.

Iaccarino et al., (2001) and Benjamin et al., (2001) measured the effect of flow pulsations on the flow distribution within ceramic contoured monoliths. From two difference shapes of substrates by observing the cycle-averaged flow distribution at the exit to the monoliths using HWA. A standard substrate was remains as the reference for comparing with the two difference substrates which were two cone shaped substrates and to one dome shape substrates. CFD predictions were obtained to predict the entrance effect within the central region of the substrates. For the result, contoured of monoliths were shown to be less sensitive to changes in flow rate and pulsation frequency than compared to a standard monolith.

In summary, whilst these studies were able to drive useful correlations between flow maldistribution and system parameters it is often difficult to interpret the findings in terms of processed within the diffuser itself. Some other studies were reported on the mechanism of flow separation through simple diffusers with pulsating flows. Mat Yamin et al., (2013) shows that the pulsating flow in a planar diffuser upstream of automotive catalyst monoliths was investigate suing the PIV and HWA technique. The PIV and HWA measurements was placed at the downstream of the monolith at the steady flow that producing the highest maldistribution even at the same Reynolds number, Re. The catalyst monolith was design in 2D and that was conducted under isothermal conditions. The result shows that, increasing monolith resistance flattens the flow field just upstream and within the monolith for both steady and pulsating flow.

## 1.3 Problem statement

The study of the automotive after-treatment system of the catalytic converter is to fit the emission regulations has been subjected of considerable research over several decades. The functions of catalytic converter at the automotive industry is used to reduce the emission of carbon monoxide (CO), hydrocarbons (HC), and nitrogen oxide ($NO_x$) will form during the combustion process occurs. The design of automotive catalytic converter of UBC and CCC varies considerably. The UBC was design underneath the vehicles which is a meter or two downstream of the engine. To reduce the pressure drop of the monolith, the CCC was chosen cause of the design of diffuser is short as possible. The functions of CCC was used for rapid warm-up at the engine system with the underbody catalyst by reducing the emission during post-light-off.

The catalysts comprise between ceramic and metallic of monoliths featuring thousands of parallel channels through which exhaust gas flows with hydraulic diameter of ~1 mm to provide ample surface area for high conversion efficiency. The volume of the monolith depends on many factors such as wall thickness, hydraulic diameter, frontal area and superficial velocity. To reduce the monolith resistance and region of flow field inside diffuser dictates the maldistribution flow across the monolith. The flow maldistribution of the upstream of the monolith affected by the monolith, which is pressure drop, geometry diffuser and Reynolds number.

Thus far, the studies of CFD predictions of flow characterization have only emphasised on the UBC catalyst. The UBC catalyst in the exhaust system and the velocity profile was predicted. Velocity profile of the UBC catalyst feature the maximum velocity at the central region. The maximum velocity of the UBC catalyst usually lead to the catalyst aging compared to the CCC catalyst. However, the CCC catalyst was replacing the UBC catalyst to reducing the maximum velocity profile. The degree of velocity peak for CCC catalyst was improve by the automotive technology improves. Besides, the upstream flow field of the CCC catalyst was chosen to be improved to reduce the maximum velocity profile.

In this study, the CCC is simplified as a two-dimensional system to provide insight of the flow field comprising the oblique entry jet and the large recirculation, inside the planar diffuser. The corresponding velocity profiles across the monolith are investigated with the monolith being represented as a porous medium. Hence, the flow field inside the diffuser can be the sole parameter affecting the flow maldistribution across the monolith of CCC.

## 1.4 Aim and objectives

In this research, the goal is to provide the insight into flow structure inside the diffuser of CCC by integrating porous medium and individual channel approaches. This project targets on the following specific objectives:

a) To assess the pressure drop across the monolith channel using the steady-state CFD prediction.
b) To investigate the steady-state flow field inside the diffuser of close-coupled catalytic converter and the flow distribution across the monolith using the CFD prediction.

## 1.5 Scope

The scope for this study is focusing on prediction of flow distribution at the planar wide-angle diffusers of automotive catalyst monolith using the new method of CFD modelling. The predictions of CFD modelling were used by integrating the porous medium and individual channels approaches. There were the specific scopes of these projects:

a. The measurement and predictions of the monolith resistance of automotive catalyst monolith was modelled as single channels and simulate using CFD simulations. The CFD predictions along single channel of monolith will get the velocity of the air flow against the pressure drop per length while the predictions data was plotted against channel Reynolds number, $Re_{c.}$
b. Automotive catalytic converter was model as the two-dimensional to predict the air flow inside the straight and incline inlet pipe of diffuser. The CFD prediction was used to investigate the flow characteristic of planar wide angled diffuser from different medium of CFD studies which is coarse, medium and fine.

## 1.6 Significant of study

As the nation takes a detailed look at NAP2014's, for the fourth annual report card since the policy was announced in January 2014 about the Malaysian Automotive Industries because the continuously holistic growth into 2018. Energy Efficient Vehicles (EEV) was reaching 52 percent of vehicles sold in Malaysia from 50 percent target of EEV penetration that was set by the government in 2017. The Malaysian Automotive Institute (MAI) and MITI was defines the EEV as the vehicles which meet the definition of specification in terms of carbon emission level and other fuel consumption. The EEV was includes the fuel-