UNIVERSITI TEKNIKAL MALAYSIA MELAKA

A STUDY OF THE EFFECT OF INLET PIPE ANGLE IN CATALYTIC CONVERTER

This report submitted in accordance with requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Mechanical Engineering Technology
(Automotive Technology) (Hons.)

by

MOHD ARIFIN BIN SAMAD
B071310509
930617-01-5087

FACULTY OF ENGINEERING TECHNOLOGY
2016

DECLARATION

I hereby, declared this report entitled The study of the effect of inlet pipe angle in Catalytic Converter is the results of my own research except as cited in references.

Signature :………………………

Name : Mohd Arifin Bin Samad

Date : ………………………

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfilment of the requirements for the degree of Bachelor of Engineering Technology (Automotive). The member of the supervisory is as follow:

……………………………….

(Pn. Nurul Amira Binti Zainal)

ABSTRACT

This study focuses on investigation of the effect of inlet pipe angle in catalytic converter. The catalytic converter is used to catalyst the emission gases from engine combustion, converting the harmful gases into less toxic gases. Until the present day, the catalytic converter has improved the performance of the vehicle start from it launched into production. In order to give the best performance of catalytic converters, the geometry of the catalytic converter must be exact to avoid the behaviour of backpressure in the exhaust system. The backpressure actually reduced the vehicle performances and increase in fuel consumption. Thus, this study aims to design various models of catalytic converter and optimized the volume of fluid flow inside the catalytic converter by changing the inlet cone pipe angles. There are three different angles of inlet pipe that being considered which are 30°, 45°, and 60° degrees. Additionally, two different types of fluid flow are also studied which are air and gas (Carbon Monoxide, CO). The model is simulated in SolidWorks software to achieve the result of this study. From the result, the design of catalytic converter is a critical issue to determine the optimum geometry design of catalytic converter. From the result obtained, it is clearly showed that by decreasing the divergence angle can increase the performance of catalytic converter. The pressure at inlet is directly proportional to inlet angle. Thus, in order to increase the flow uniformity (lower pressure) the inlet divergence angle of catalytic converter should be decreased.

ABSTRAK

Kajian ini memfokuskan kepada penyiasatan terhadap kesan sudut paip masuk dalam penukar bermangkin. Penukar pemangkin ini digunakan untuk pemangkin gas pelepasan daripada pembakaran enjin, menukar gas berbahaya ke dalam gas kurang toksik. Sehingga hari ini, penukar pemangkin telah meningkatkan prestasi permulaan kenderaan sejak ia dipasarkan. Dalam usaha untuk memberikan prestasi terbaik daripada penukar bermangkin, geometri penukar pemangkin mestilah tepat untuk mengelakkan tingkah laku tekanan belakang dalam sistem ekzos. Tekanan belakang boleh mengurangkan prestasi kenderaan dan peningkatan dalam penggunaan bahan api. Oleh itu, kajian ini bertujuan untuk mereka bentuk pelbagai model penukar pemangkin dan mengoptimumkan jumlah aliran cecair di dalam penukar bermangkin dengan menukar masuk sudut paip kon. Terdapat tiga sudut yang berbeza paip masuk yang sedang dipertimbangkan iaitu 30 °, 45 °, dan 60 ° darjah. Selain itu, dua jenis aliran bendalir juga dikaji iaitu udara dan gas (karbon monoksida, CO). Model ini di simulasi di dalam perisian SolidWorks untuk mencapai hasil daripada kajian ini. Dari keputusan itu, reka bentuk penukar pemangkin merupakan isu kritikal untuk menentukan reka bentuk geometri yang optimum bagi penukar pemangkin. Dari keputusan yang diperolehi, ia jelas menunjukkan bahawa dengan mengurangkan sudut perbezaan yang boleh meningkatkan prestasi penukar pemangkin. Tekanan di salur masuk adalah berkadar terus dengan sudut masuk. Oleh itu, dalam usaha untuk meningkatkan keseragaman aliran (tekanan rendah) perbezaan sudut salur masuk penukar pemangkin perlu dikurangkan.

DEDICATIONS

I would like to dedicate this to my father, Samad Bin Hasan and my mother, Asiah Binti Paimin, my supervisor, Mrs. Nurul Amira Binti Zainal, Sir Suffian and my friends for supporting me from the beginning until the end of this study.

ACKNOWLEDGEMENTS

I am grateful to Allah for giving me the good health and wellbeing that were necessary to complete this thesis. I owe my deepest gratitude to my research supervisor Puan Nurul Amira Binti Zainal and my co supervisor En Mohd Suffian Ab. Razak. Without their assistance and dedicated involvement in every step throughout the process, this paper would have never been accomplished. I am indebted to my supervisor for providing me with all the necessary facilities for the research. I take this opportunity to express gratitude to all of my group members for their help and support. They have been unwavering in their personal and professional support during the time I spent to finish up my thesis. Words cannot be expressed how grateful I am to my parents for their fully encouragement, support and attention that they have made on my behalf.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE

TDC	-	Top Dead Centre
BDC	-	Bottom Dead Centre
NOx	-	Nitroxide
Cox	-	Carbon monoxide
HC	-	Hydrocarbon
H_2O	-	Water
CO_2	-	Carbon dioxide
CFD	-	Computational Fluid Dynamic
CAD	-	Coronary artery diseases
O_3	-	Ozone
TWC	-	Three-way Catalytic converter
A/F	-	Air-fuel ratio
DNS	-	Direct numerical simulation
CATIA	-	Computer Aided Three-dimensional Interactive Application
CAE	-	Computer-Aided Engineering
K-ε	-	K-epsilon
K-ω	-	K-omega
V	-	velocity
D	-	Diameter of pipe
N	-	Kinematic viscosity of fluid

ρ	-	Density of fluid
µ	-	Dynamic viscosity of fluid
f	-	Friction factor
Re	-	Reynolds number
K	-	Kelvin
Mm	-	Millimeter
FTK	-	Fakulti Teknologi Kejuruteraan
UTeM	-	Universiti Teknikal Malaysia Melaka

CHAPTER 1
INTRODUCTION

1.0 Introduction

This chapter introduced the background of the study about the fluid flow inside the catalytic converter. Next, the problem statements of this study is discussed. After that, the objectives and the scopes of the study is explained. Finally, outlines of the report is presented.

1.1 Background of the Study

In every vehicles production in automotive industries have the main component to produce power and make the transport move. Engine as the main component will produce power to make the vehicles complete. Combustion by the air-fuel mixture in the engine system are generates mechanical power. On the other hand, high pressure produces by the combustion of air-fuel mixture in combustion chamber is apply force to some part in engine such as piston. Piston move to top dead centre(TDC) and to bottom dead centre(BDC) when the force is applied. The piston is push the burning smoke to flow out from the chamber through the exhaust valve as the waste products. Based on the previous study, during the exhaust stroke when the piston moves from BDC to TDC, pressure rises and gases are pushed into exhaust pipe. Thus the power required to drive exhaust gases is called exhaust stroke loss and increase in speed increases the exhaust stroke loss (Karuppusamy et al. 2013).

Meanwhile, waste product is an unwanted substances or material produced during the combustion process. The waste product from the combustion can called as fluid in form of gases. The gases contain chemical material such as nitroxide(NOx), carbon monoxide(COx) and unburned hydrocarbon(HC). This gas can cause air pollution and effect to environments. The quantity of the waste product produces depends on the ratio of air-fuel mixture. There are three type of ratio of air-fuel mixture such as 14.7:1 stoichiometric (good mixture of air and fuel), 15:1 when more oxygen than fuel and 13:1 when the fuel is more than the air during mixture. When the ratio of air-fuel mixture is higher and lower than 14.7 it will cause unperfect combustion in engine. Unperfect combustion usually produce more quantity of waste products and will increase the air pollution level and bad to the environments. From this situation, the automotive engineers have tried to solve the problem by checking exhaust gas product and think how it can control for not harmful to environments.

By adding the catalytic converter in the middle of exhaust pipe it can reduce the waste product from emission to environments. Catalytic converter can be defining as an automobile exhaust-system component containing a catalyst that causes conversion of harmful gases into mostly harmless products as water (H2O) and carbon dioxide (CO2). They provide a filter inside the body or substrate known as honeycomb. The honeycomb that react with the waste product as catalyst can convert to less harmful gases.

Further, the design of catalytic converter will be tested to make sure the maximum use of catalytic volume. The maximum catalyst volume would be achieved by a uniform flow distribution through the monolith substrate. From the previous study, some papers have studied the flow in round cross-section monolith converter with conical inlet and outlet headers. The monolith flow field to be extremely maldistributed, (Howitt and Sekella, 1974). The effect of truncating the inlet and outlet diffusers of a monolith catalytic converter was found in to be insignificant. Others has tested through water-visualization on full scale transparent model of a double-brick converter with tapered inlet and outlet headers. The result show the characteristic dynamic flow was different from those under steady flow condition in catalytic converter, (Lee et al. 2002).

This study aims to design and model different angles inlet pipe in catalytic converter. Then the fluid flow behaviour in the catalytic converter will be investigated. Previous study has shown that, the harmful emissions gases from engines such as nitrogen oxides, hydrocarbons, and carbon monoxide are because of incomplete combustion, (Naveenkumar et al. 2015). Recent catalytic converters are substrate coated with platinum, rhodium, or palladium, which are nobel metals and expensive. Due to non-uniform flow inside the catalytic converter, the outer most region of substrate are less reactant to the emission by utilizing these regions we may able to increase the efficiency and life span of the converter, (Bahrami, 1948). The study about inlet flow of catalytic converter is to ensure the suitable design for product in automotive field. The design of catalytic converter is critical which require a deep understanding of fluid flow inside the catalytic converter in order to determine the optimum or maximum use set of the catalytic converter.

1.2 Problem Statement

The previous study has shown that the catalytic converters body or substrate and shape of the inlet cone contribute the backpressure. This increase in backpressure causes increase in fuel consumption, (Karuppusamy et al. 2013). Meanwhile, an experimental optimization of the design parameters of a catalytic converter is extremely expensive and time consuming. This is because the design process involves building several prototypes with different geometries for experimental testing, (Bassem et al. 2007). Hence, a computational approach to design optimization of catalytic converters is more feasible, (Kumar and Aggarwal, 2012). These models must be absolutely exact, since the flow inside a catalytic converter is extremely sensitive to geometric deviations. Stereo-lithographic manufacturing of plastic models from CAD data has proved to be an exact method and a useful tool for experimental investigation of internal flow devices. (R & Ramsai, 2012). Thus, this study aims to determine the optimum geometry design of catalytic converter by using numerical approach which are cost effective and time-saving.

1.3 Objectives of the Study

The objectives of this study are:

1. To design and model three different angles of inlet pipe in catalytic converter by using CATIA software.
2. To simulate numerically the fluid flow interaction in three different angles of inlet pipe in catalytic converter by using SolidWorks.
3. To determine the optimum geometry between three different angles of inlet pipe in catalytic converter that give most significant effect on flow distribution.

1.4 Scope of the Study

The scope of this study is first to design and investigate numerically the simulation of fluid flow in catalytic converter by using SolidWorks. Three different geometry angles of inlet pipe of catalytic converter will be considered, which are 30°, 45°, and 60° degrees. Type of fluid that will be taken into account are air and gas (carbon monoxide), assuming environment pressure at the outlet pipe of catalytic converter and finally the type of flow is considered to be turbulent in the catalytic converter. Since turbulent flow is analysed, this study focuses on Spallart-Allmaras model in order to have significant results. The Spalart–Allmaras model is a one-equation model that solves a modelled transport equation for the kinematic eddy turbulent viscosity.

1.5 Outline of the Report

This report is divided into five chapter including this introductory chapter. Chapter 1 briefly discusses some geometric introduction and highlight the objectives of this study, also the problem statement and scope of the study.

Chapter 2 makes review about the literature review of this study, meanwhile some mathematical formulation which describes on model descriptions is discussed in Chapter 3.

Followed by Chapter 4, complete pre-processing results of the problem is obtained in details and will be discussed briefly.

Chapter 5 finally provides the conclusions of this study as well as some suggestions and recommendations for future study.

CHAPTER 2
LITERATURE REVIEW

2.0 Introduction

This paper deals with the study of flow inside the catalytic converter. The flow inside the converter is non-uniform due to the geometry of the catalytic converter. From the previous study, analysis of real time catalytic converter is more expensive and if there is any error we have to change the whole model (Naveenkumar et al. 2015). Thus, by using SolidWorks Software, it is easy to identified the errors and rectifying it in an easy way. The catalytic converters are consisting of substrate at the centre through which the conversion of pollutant gases will take place. At the point when the smoldered gasses go into the exhaust system of certain speed it will specifically hit the substrate at the inside and synthetic responses will occur. The separation between the substrate and the way out of the bay cone is less so the gas hit the middle segment of the substrate. By this only the middle part of the substrate is responded to the delta gasses, so the stream partition at the gulf is not adequate to extend the stream in all the surfaces of the substrate so that there will be uniform stream inside the converter.

The toxic substances are harmful to the environments and give effect on health. The waste product from the combustion can called as fluid in the form of gases. The gases contain chemical material such as nitrogen oxide(NOx), carbon monoxide (COx) and unburned hydrocarbon(HC) that can be combined with others substance and produce toxic. Besides, by installing the catalytic converter in exhaust systems will form the backpressure in the system. The backpressure is directly proportional to the catalytic converter design. The catalytic converters body or substrate and shape of the inlet cone forming the backpressure. This increase in backpressure causes increase in fuel consumption, (Karuppusamy et al. 2013). It is very difficult to achieved a good flow distribution at the inlet cross section of the catalyst substrate. Therefore, it is important to study the effect of the geometry of the catalytic converter on flow uniformity in the substrate.

2.1 Study on Catalytic Converter

First of all, the catalytic converter is employed in a wide variety of industrial engines, including engines used for power generation, co-generation, gas compression and other stationary and industrial applications (Chen et al. 2004). Based on previous paper, in recent year advances have been made in computer modelling of catalytic converters to assist in design optimization, with most of this work focused toward the automotive industry, (Wanker et al. 2002). According to (Karuppusamy et al. 2013), the catalytic converter can be defined as vehicle emission control device which is to convert the toxic gases from the combustion engine to harmless substances by way of catalysed chemical reactions. Next, the catalytic converters are used with internal combustion engines fuel by either petrol or diesel including as well as kerosene heaters and stoves. Inside the converter body contains some materials called honeycomb that function as catalyst for neutralize the toxic pollutants to become harmless substances. The honeycomb monolith is also available in different cell densities and shapes offering potential flexibility. (Karuppusamy et al. 2013).

According to (Pankaj and Manish, 2012), United Stated market is the first country that introduced the catalytic converters, where 1975 automotive industry were so equipped to comply with tightening U.S. Environmental Protection Agency regulations on automobile exhaust emissions. Besides, catalytic converters are most regularly connected to fumes frameworks in automotive field, they are additionally utilized on electrical generators, forklifts, trucks, transportations, trains and bikes.

They are additionally utilized on some wood stoves to control outflows. This is as a rule in light of government control, either through direct natural direction or through wellbeing and security directions. Figure 2.1 below shows an example of catalytic converter.

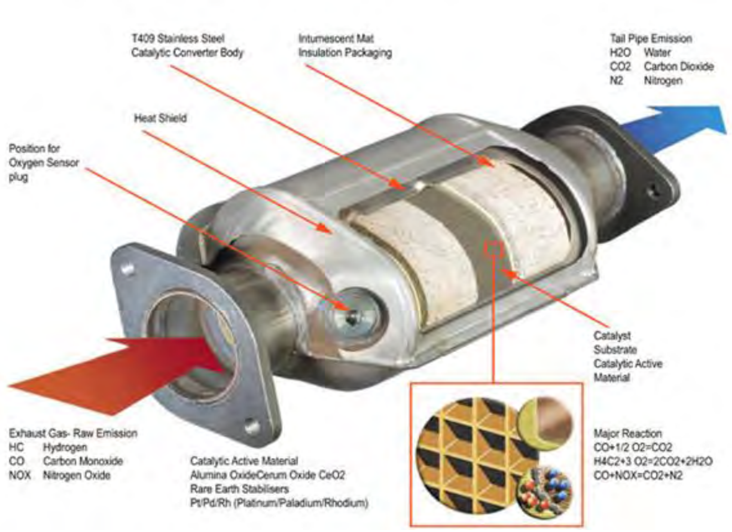

Figure 2. 1: Catalytic Converter.

2.2 Hydrocarbon(HCs), Carbon Monoxide(CO) and Nitrogen Oxides(NOx) Formation Mechanism from Combustion Engine

The majority of vehicles use the technology of combustion of fuels derived from crude oil at a refinery to run the engine. Petrol and diesel are mixtures of hydrocarbons (HC) which are consist of carbon and hydrogen atoms. Based on the previous study, the perfect combustion of hydrocarbons (HC) in a perfect engine ideally leads to the formation of water (H2O) and carbon dioxide (CO2) only depends on combustion with oxygen and nitrogen from air acts as non-reactive species (Islam 2015) followed by equation (2.1) below. However, since the vehicles engines do not act as an ideal engine in reality, imperfect combustion and high temperatures in the combustion chamber lead to the formation of significant amounts of pollutants as represented in equation (2.2):