IOP Conf. Series: Materials Science and Engineering **328** (2018) 012029 doi:10.1088/1757-899X/328/1/012029

Effect of inlet cone pipe angle in catalytic converter

Nurul Amira Zainal[1], Ezzatul Farhain Azmi[2] and Mohd Arifin Samad[3]

[1],[2]Faculty of Engineering Technology,
Universiti Teknikal Malaysia Melaka,Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia.

Email:*[1]nurulamira@utem.edu.my

Abstract: The catalytic converter shows significant consequence to improve the performance of the vehicle start from it launched into production. Nowadays, the geometric design of the catalytic converter has become critical to avoid the behavior of backpressure in the exhaust system. The backpressure essentially reduced the performance of vehicles and increased the fuel consumption gradually. Consequently, this study aims to design various models of catalytic converter and optimize the volume of fluid flow inside the catalytic converter by changing the inlet cone pipe angles. Three different geometry angles of the inlet cone pipe of the catalytic converter were assessed. The model is simulated in Solidworks software to determine the optimum geometric design of the catalytic converter. The result showed that by decreasing the divergence angle of inlet cone pipe will upsurge the performance of the catalytic converter.

1. Introduction

Every vehicle production in automotive industries requires the main component to produce power and make the transport move. Vehicle engine acts as the core of the vehicle body that generate mechanical combustion power of the air-fuel mixture in its system. High pressure through the exhaust valve, the piston pushes the burning smoke out of the chamber as waste products is produced by the combustion of the air-fuel mixture in a combustion chamber and applies force to some part of the engine such as pistons. When the force is applied, piston move to top dead centre (TDC) and to bottom dead centre (BDC). Karuppusamy [1] stated that pressure rises when the piston moves from BDC to TDC and gases are pushed into the exhaust pipe afterwards or unwanted substances which contain chemical material such as Nitroxide (NOx), Carbon Monoxide (COx) and unburned Hydrocarbons (HC). This gas can cause air pollution and affect the environment. However, adding the catalytic converter in the middle of the exhaust pipe can diminish the waste product from emission to environments.

In recent years, much improvement has been made in computer modelling of catalytic converter to assist in design optimization with most of this work focused towards the automotive industry [2]. Major effort has been devoted into the design of a converter which lead to maximum use of catalyst volume. The maximal utilization of the catalyst volume would be attained by having a uniform flow distribution through the monolith substrate [3]. To achieve the uniform flow between sections of different cross-sectional areas, most modern catalytic converters have long tapered inlet and outlet headers. The flow distribution in the monolith catalytic converter have been widely investigated. Lemme [4] conducted an analytical and experimental treatment of flow through catalytic converters. Whereas Lai [5] simulated three-dimensional internal flow of the automotive catalytic converter. Jeong [6] implemented a CFD approach to investigate the three-dimensional unsteady flow in the catalytic converter. It is clearly known that the flow distribution in the monolith catalytic converter is influenced by many factors. Therefore, in order to improve the flow distribution, suggestions concerning baffle design and contoured substrates have been introduced. Howit [7] investigated the flow in round cross-section monolith converters with conical inlet and outlet headers. The study found that the monolith flow field was extremely dispersed. Additionally, Wollin [8] studied the flow performance of ceramic contoured substrates for automotive exhaust catalyst system. It is shown that a countoured substrate can provide improvements in flow uniformity. On another study, Wendland [9] found an insignificant effect of truncating the inlet and outlet diffusers of a monolith converter. Meanwhile, Lee [10] studied the effects of engine operating conditions in catalytic converter temperature in an SI engine. It was found that ignition retard and misfire can result in deactivation of the catalytic converter, which eventually leads the drastic thermal aging of the converter.

An experimental optimization of designing the catalytic converter is extremely expensive and time consuming since the design process involves building several prototypes with different geometries [11]. The flow inside the catalytic converter is very sensitive to geometric deviations thus these models must be exact and precise. Consequently, a computational approach to find an optimum design of the catalytic converter is established. Kumar [12] in his study employed a finite element approach to analyse a multi leaf spring using CAE tools. In the meantime, Vinkel [13] make comparisons between CAE approach and experimental results of leaf spring in the automotive industry. Wu [14] studied the effect of upstream flow distribution on the light of the performance of the catalytic converter by using numerical simulation. Also, Koltsakis[15] compared the cumulative emissions for the uniform and non-uniform distribution flow distributions by using numerical approach.

The design of the catalytic converter is very critical which require deep understanding. For the above reasons, the present work aims to study the effect of three different geometry angles of inlet cone pipe in the catalytic converter to get a better understanding of the catalytic converter processes. The purpose of this study is also to examine the relationship between the flow distribution and the performance of the catalytic converter inside three different models as proposed in order to optimize the catalytic converter design.

2. Design and Geometry

The catalytic converter is ideal based on a straight pipe. Size enlargement and additional angles were modified right after the inlet pipe hole gives some effect of the gas flow in the pipe. Figure 1 shows an example of a catalytic converter.

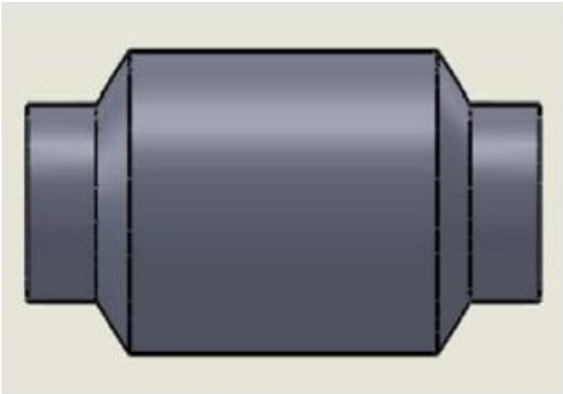

Figure 1: Basic Design of the Catalytic Converter

This study performed a complete drawing, including the inlet pipe, the substrate and the outlet of the catalytic converter. The inlet pipe diameter is set to be 47.6mm. The substrate length is 114.5mm where the diameter is 76.2mm. Total pipe length is measured as 31.5mm. As for the cone pipe angle, the model of the catalytic converter is set to be 30^o degree (model 1) where the other two are 45^o degree (model 2) and 60^o degree (model 3). Figure 2,3 and 4 below show the different designs of the catalytic converter namely Model 1, Model 2 and Model 3.

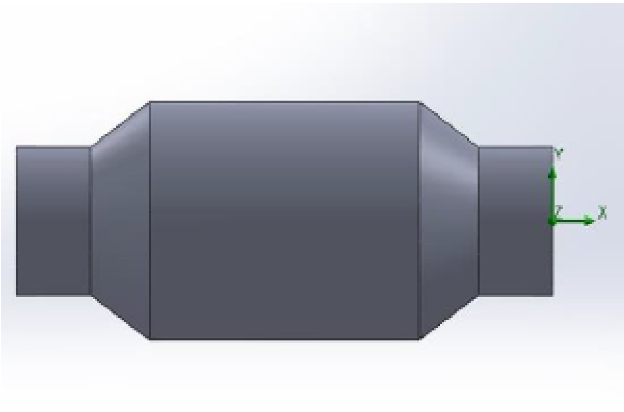

Figure 2: Inlet Cone of the Catalytic Converter 30^o degree (Model 1)

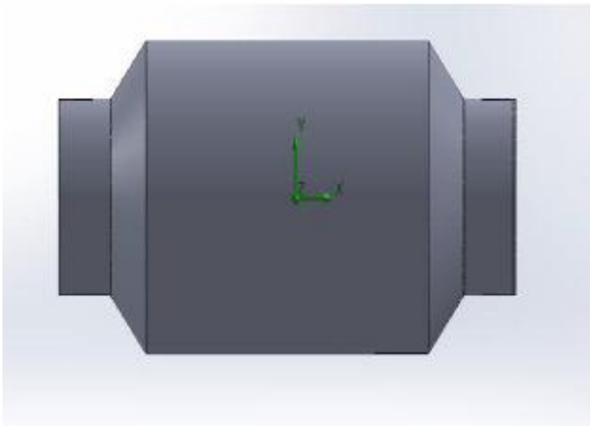

Figure 3: Inlet Cone of the Catalytic Converter 45^o degree (Model 2)

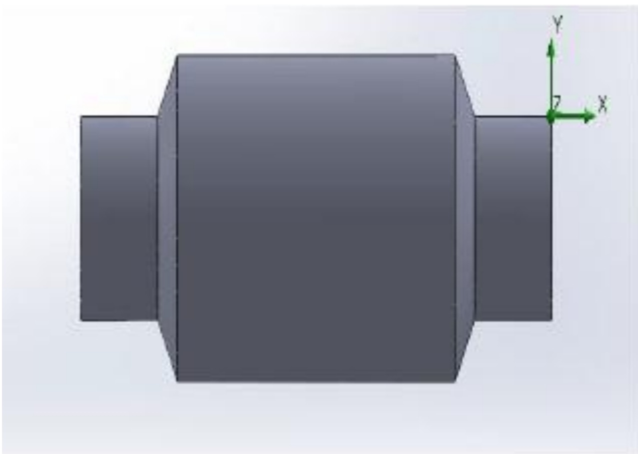

Figure 4: Inlet Cone of the Catalytic Converter 60^o degree (Model 3)

3. Governing Equation and Boundary Conditions

Numerical study and algorithms are used to solve and analyze problem that involve fluid flows. The fundamental basis of almost all fluid problems is the Navier-Stokes equations. Principally, the equations are a set of coupled differential equation and could be solved for a given flow problem mathematically. However, in practice these equations are too difficult to solve analytically. Thus the computational approach was employed in order to solve approximations to the equations using a variety of techniques including finite element method. The Navier-Stokes equation consists of conservation of mass (Eq. 1), conservation of momentum (Eq. 2) and conservation of energy (Eq. 3).

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \qquad (1)$$

Conservation of Momentum:

$$\rho \frac{D\vec{V}}{Dt} = -\nabla P + \rho \vec{g} + \mu \nabla^2 \vec{V} \qquad (2)$$

Conservation of Energy:

$$\rho \left(\frac{\partial h}{\partial t} + \nabla \cdot (h\vec{V}) \right) = -\frac{Dp}{Dt} + \nabla \cdot (k \nabla T) + \phi \quad (3)$$

To solve the above equations, a turbulence model is needed. This study employed Spalart-Allmaras model. Spalart-Allmaras incorporates some of the recent advances in turbulence modelling. It regularly shows that it has equal or superior accuracy for nearly all classes of flows. In addition, the Spalart- Allmaras model is more computationally efficient compared to other model. Javaherchi [16] reviewed the Spalart- Allmaras model and its modification. The model is presented in Eq. (4).

Spalart- Allmaras model :

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \frac{\partial}{\partial x_i}(\rho \vec{v} u_i) = G_\upsilon + \frac{1}{\sigma_{\vec{\upsilon}}} \left[\frac{\partial}{\partial x_j} \left\{ \left(\mu + \rho \vec{\upsilon} \right) \frac{\partial \vec{\upsilon}}{\partial x_j} \right\} + C_{b2\rho} \left(\frac{\partial \vec{\upsilon}}{\partial x_j} \right)^2 \right] - Y_\upsilon + S_{\vec{\upsilon}} \qquad (4)$$

In this study, the fluid is set as air and the temperature is set to 873K assuming environment pressure at the outlet pipe of the catalytic converter. However, heat transfer from the fluid is not considered hence the fluid is reflected to be isothermal. The walls are assumed to have a smooth surface.

4. Results and Discussion

As mentioned earlier, the main objective of the present study is to investigate the effect of three different geometry angles of inlet cone pipe in the catalytic converter to get a better understanding of the catalytic converter processes, as well as to examine the relationship between the flow distribution and the performance of the catalytic converter inside three different models namely model 1, model 2 and model 3 in order to optimize the catalytic converter design.

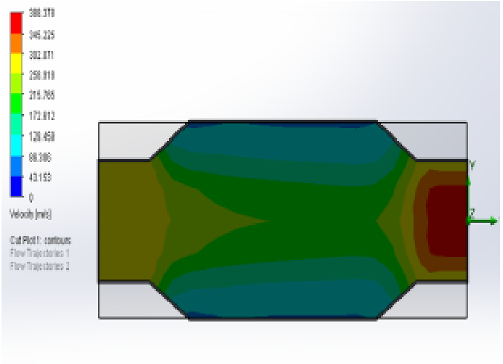

Figure 5: Velocity Distribution in Model 1

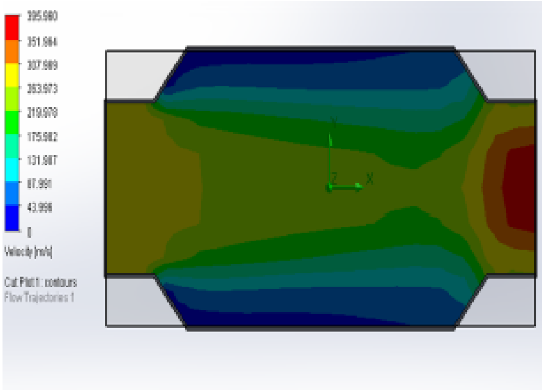

Figure 6: Velocity Distribution in Model 2

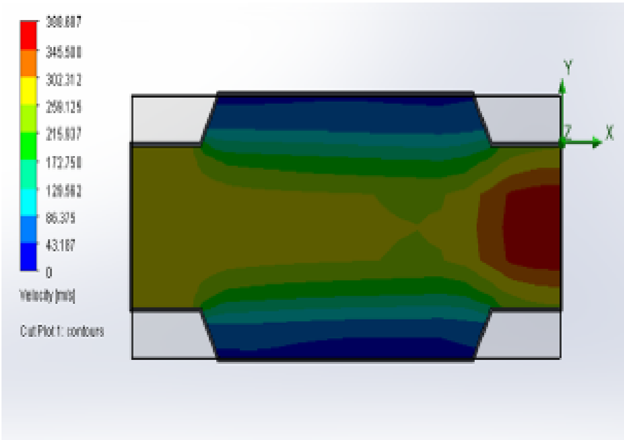

Figure 7: Velocity Distribution in Model 3

Figure 5,6 and 7 show the velocity distribution of the catalytic converter in model 1, model 2 and model 3 accordingly. In model 1, the velocity value of air start to decrease from 258.918m/s to 215.765m/s at the inlet cone of the catalytic converter before it reaches the end of the outlet substrate. However, the velocity increase back and reaches maximum at 388.378m/s after flowing out of the substrate region. In model 2, the mass flow moves roughly and some eddies are formed at the inlet cone of the catalytic converter. The velocity value of air start to decrease from 263.973m/s to 219.978m/s at the inlet pipe. Then the velocity increase back and reaches 395.96m/s. Comparing these two result observations, it is clearly shown that the velocity distribution is uniformly delivered in model 2 meanwhile in model 1, the velocity is highly non-uniform in the substrate inlet. Referring to Figure-6, model 3 shows differ result from model 1 and model 2. In model 3 the velocity flow only distributed in the middle of the catalytic converter mainly at the end of the substrate part. This may be caused from the bigger gradient (angle) of the catalytic converter. The non-uniformity increases in the substrate with the increase of the mass flow. Also, a formed of backpressure is detected in model 3 as the higher angle turns as a flow restriction. The velocity value of the inlet pipe is recorded to be 259.125 and the maximum value is 388.68m/s of the outlet substrate in model 3.

IOP Conf. Series: Materials Science and Engineering **328** (2018) 012029 doi:10.1088/1757-899X/328/1/012029

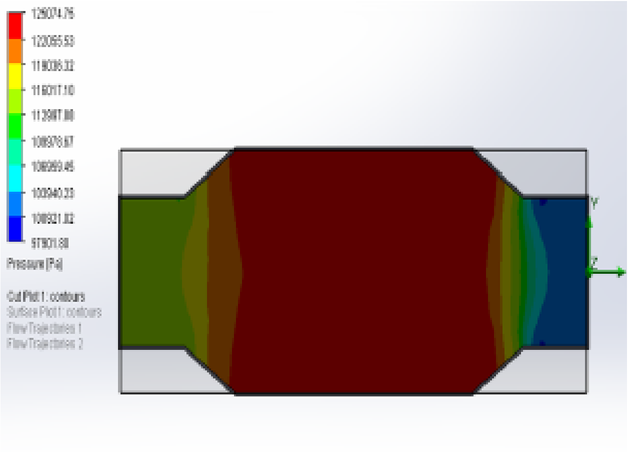

Figure 8: Pressure Distribution in Model 1

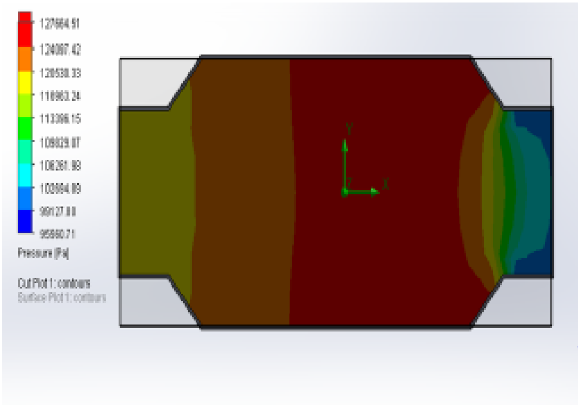

Figure 9: Pressure Distribution in Model 2

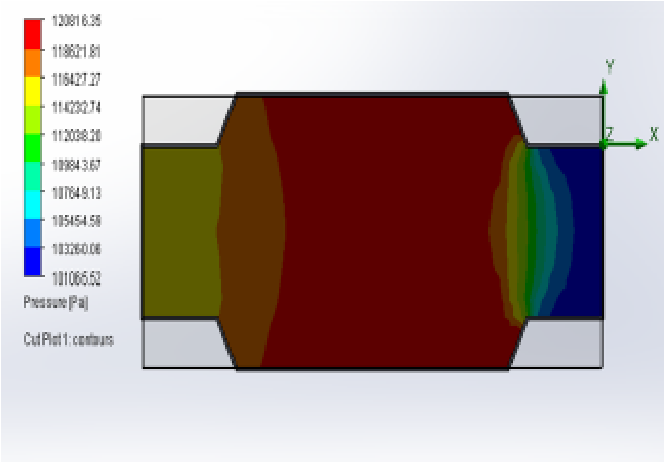

Figure 10: Pressure Distribution in Model 3

As for pressure distribution, the result in all models is shown in Figure 8,9 and 10. In model 1, the pressure starts to increase from 112997.88pa to 122055.53pa at the inlet cone of the catalytic converter and reach the maximum value of 125074.75pa at the substrate region. In model 2, the pressure start of 116963.24pa to 124097.42pa at the inlet cone and the value extent its maximum value of 127664.51pa. In the meantime, model 3 shows the maximum value of pressure distribution can be up to120816.35pa. Overall, the pressure distribution in model 2 is considerable compared to model 1 and model 3. The development of dispersion in pressure distribution in model 2 shows the best distribution flow between other cases.

5. Conclusion

From the present investigation, the results show that different geometries of the catalytic converter have a significant effect on the flow distribution in the monolith substrate. Furthermore, lower angles show less uniformity of flow distribution in the catalytic converter, whereas the flow tends to generate some additional backpressure in higher angles. Additionally, from these three cases it was revealed that if the angles are closer to the inlet cone pipe, the flow distribution is distributed uniformly. As a conclusion, cone inserted in model 2 is very effective and can increase the performance of the catalytic converter. The design of the catalytic converter is the key to provide a converter system to work proficiently as expected. The result is used to optimize and improve the development of new technologies for controlling regulated and non-regulated emissions.

The authors would like to acknowledge Universiti Teknikal Malaysia Melaka (UTeM) for financial support under UTeM short term grant PJP/2015/FTK(1C)/S01394.

References

[1] P. Karuppusamy and R. Senthil, “Design, analysis of flow characteristics of catalytic converter and effects of backpressure on engine performance,” *Int. J. of Research in Engineering & Advanced Technology*, ISSN: 2330-8791, Vol 1, Issue 1, March 2013

[2] R.Wanker, H. Granter, G. Bachler, and G. Rabenstein, “ New physical and chemical models for the CFD simulation of exhaust gas lines: a generic approach,” *SAE Technical Paper*, 2002-01-0066, 2002.

[3] R.R.K. Thundil and R. Ramsai, “Numerical study of fluid flow and effect of inlet pipe angle in catalytic converter using CFD,” *Res.J.RecentSci*, ISSN:2277-2502, Vol 1(7), pp39-44, July 2012.

[4] C. Lemme and W. Givens, "Flow through catalytic converters - an analytical and experimental treatment," *SAE Technical Paper*, 740243, 1974,

[5] M. Lai, J. Kim, C. Cheng, and P. Li, "Three-dimensional simulations of automotive catalytic converter internal flow," *SAE Technical Paper*, 910200, 1991

[6] S. Jeong and T. Kim, "CFD investigation of the 3-dimensional unsteady flow in the catalytic converter," *SAE Technical Paper*, 971025, 1997

[7] J. Howitt and T. Skella, "Flow effects in monolithic honeycomb automotive catalytic converters," *SAE Technical Paper*, 740244, 1974

[8] J. Wollin and S.F. Benjamin, “A study of the flow performance of ceramic contoured substrates for automotive exhaust catalyst systems” *SAE Technical Paper*, 1999-01-3626 , 1999

[9] D. Wendland, W. Matthes, and P. Sorrell, "Effect of header truncation on monolith converter emission-control performance," *SAE Technical Paper*, 922340, 1992

[10] S. Lee, C. Bae, Y. Lee and T. Han, ”Effects of engine operating conditions on catalytic converter temperature in an SI engine,” *Society of Automotive Engineers Inc.*, DOI: 10.4271/2002-01-1677

[11] B. Ramadan, P. Lundberg and R. Richmond, "Characterization of a catalytic converter internal flow," *SAE Technical Paper*, 2007-01-4024, 2007

[12] K. Kumar and M.L. Aggarwal, “A finite element approach for analysis of a multi leaf spring using CAE tools” *Res.J.Recent Sci*., ISSN: 2277-2502,Vol. 1(2), 92-96, Feb. 2012

[13] A. Vinkel, M.L. Aggarwal and G. Bhusnan, “A comparative study of CAE and experimental results of leaf springs in automotive vehicles” *International Journal of Engineering Science and Technology*, ISSN:0975-5462 Vol. 3, No. 9,6856-6866, September 2011

[14] G. Wu and T. Song, “CFD simulation of the effect of upstream flow distribution on the light-off performance of a catalytic converter,” *Energy Conversion and Management*, 46(13-14), 2010-2031, 2005

[15] G.C. Koltsakis, I.P. Kandylas and A.M. Stamatelos, “Three-way catalytic converter modelling and applications” *Chem. Eng. Comm*, Vol. 164:1 , pp. 153-189, 2007
[16] T. Javaherchi, University of Washington, pp 1–13, (2010)