CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

CFD Investigation of the Flow Characteristics of an Automotive Catalyst Monolith

Open Access

Mastura Mutafa[1], Ahmad Kamal Mat Yamin[1,*], Noreffendy Tamaldin[1]

[1] Department of Automotive Engineering, Fakulti Kejuruteraan Mekanikal, Universiti Teknikal Malaysia Melaka, 75350 Ayer Keroh, Melaka, Malaysia

ARTICLE INFO

Article history:
Received 3 April 2019
Received in revised form 20 June 2019
Accepted 29 June 2019
Available online 3 July 2019

ABSTRACT

The boundary layer development inside the automotive catalyst monoliths has a significant effect on their pressure losses. Typically, the monolith is made of ceramic, comprises of several thousand channels of small hydraulic diameter. This paper investigated the pressure drop of the single channel and to predict the boundary layer development along the monolith single channel using the Computational Fluid Dynamics (CFD) simulation. The modelling of the flow through the monolith single channel using CFD was represented with a single channel to reduce computational effort. The monolith single channel was using the k-epsilon from the turbulence model to predict the pressure drop using CFD simulation. The result of the CFD prediction of pressure drop data was compared with the measurement and formulation data while the development of boundary layer was observed along the monolith channel. The outcome of this paper are shows that the CFD prediction of pressure drop along the monolith channel corresponds to the measurement and formulation data. For the development of boundary layer along the monolith channel was observed that the higher the Re number the higher the hydrodynamic entry length.

Keywords:
Ceramic cordierite monolith; computational fluid dynamic; pressure drop

1. Introduction

The automotive catalytic converter is a device used to reduce the harmful exhaust emissions from an internal combustion engine. The impact of automotive exhaust emission would be control by platinum (Nobel) metal from the group metal based catalyst in catalytic converter [1-2]. The catalytic converter was available to be a better way for establishing an efficient combustion in the controller engine and reducing the emission gas of passenger car [3-4]. Figure 1 shows a catalytic converter comprising three major parts, i.e. an inlet pipe, a diffuser, a washcoated monolith, and an outlet pipe. The monolith can be ceramic or metallic featuring thousands of parallel channels with a small hydraulic diameter (~1mm), hence the flow through the channel is laminar. The monolith can be predict using the CFD simulation. By using the CFD simulations, the effect of some parameters used that would affect the cell density and the wall thickness at the constant washcoat loading of the

[*] *Corresponding author.*
E-mail address: ahmadkamal@utem.edu.my (Ahmad Kamal Mat Yamin)

catalytic converter was illustrated [5]. Based on the previous research shows that the catalyst monolith can be predict using CFD simulations and also by using the measurement setup.

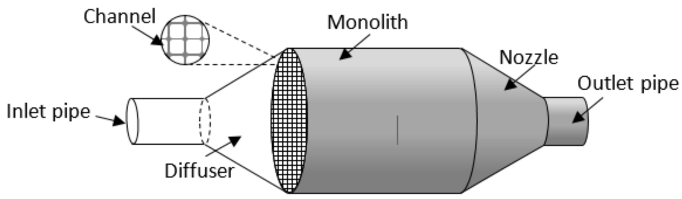


Fig. 1. Schematic of catalytic converter

During measurement, a uniform flow of air is presented at the front face of the monolith to obtain the monolith resistance. As the air enters the monolith channels the flow begins to develop the velocity boundary layer or know as boundary layer due to non-slip condition and shear stress. The non-slip condition occurs when the air particles in the layer in contact with the wall come to a complete stop while the air viscosity will affect the viscous shearing forces. Based on the effect of the flow resistance during the air entering the monolith channel, a pressure drop of laminar flow along the monolith channel will occur. The H-P equation below shows that the pressure drop is proportional to the pressure losses for the fully developed laminar flow by eliminating the pressure loss along the monolith channel. H-P equations will represent the laminar flow along the monolith channel.

$$\frac{\Delta P_m}{L} = 2fRe_c\frac{\mu U_c}{d_h^2} \tag{1}$$

Besides, the pressure drop of laminar flow in the monolith channel can be predicted using the Shah's formulation. The Shah's formulation below shows the flow is developing until the fully developed flow is achieved along the monolith length. The pressure losses $K(x)$ along the monolith channel will be included as stated in the Shah Eq. (2) until Eq. (4). It shows that the shah equations will represent the equations for developing flow of the monolith catalyst.

$$\frac{\Delta P_d}{(\frac{1}{2})\rho U_c^2} = (f_{app}Re_c)(4X^+) = (fRe_c)(4X^+) + K(x) \tag{2}$$

$$X^+ = \frac{X}{d_h Re_c} \tag{3}$$

$$f_{app}Re_c = \frac{3.44}{\sqrt{X^+}} + \frac{fRe_c + \frac{K(\infty)}{4(X^+)} - \frac{3.44}{\sqrt{X^+}}}{1 + C(X^+)^{-2}} \tag{4}$$

Many studies have been performed over the years to investigate the pressure drop of laminar flow in the automotive catalyst converter by using Shah's formulation. As stated in previous research of investigate the pressure loss in automotive catalyst monolith for an instance, characterised by the investigation of an oblique entry pressure loss [6]. A single channel approach gives a better result for the pressure drop prediction using simulation [7]. It is found that the difficulties to predict the pressure drop of catalyst monoliths was the main problem that has to face. However, the monolith channel was designed as a single channel to represent the monolith catalyst and was predicted using CFD simulation [8]. It is known that CFD is a powerful tool for anticipating the flow field through calculations at the inlet and outlet pipe of the catalytic converter.

CFD simulation was proven to be the simplest and the most accurate method used to predict the development of boundary layer of the laminar flow along the monolith channel [9-10]. Using CFD simulation also can predict the air temperature and air flow will reduce the energy and cost to achieve the thermal comfort [11]. Besides, the research to date tends to focus on investigating the pressure drop of the monolith catalyst rather than investigating the development of the boundary layer along the monolith channel. Other than that, the CFD simulation also can predict the effectiveness of simulating the painting line by using CFD method to identify the air flow and turbulent pattern [12]. In addition, a research gap was discovered in the prediction of the development of boundary layer through the single channel using CFD simulation.

The purpose of this paper is using the turbulence model to predict the pressure drop along the monolith channel by comparing with the measurement data and also the formulation data e.g. H-P and Shah Formulation. Meanwhile, the prediction of the boundary layer along the monolith single channel also would be observe. The development of boundary layer along the monolith single channel would depends on the Re number and the hydrodynamic entry length.

2. Methodology

In this study, the determination of the pressure drop in single channel was conducted through both CFD prediction and experimental study. The single channel is predicted using Fluent solver to predict the pressure drop along the monolith channel by considering the boundary condition, mesh, and grid. For the experimental studies, the test was conducted under isothermal condition at the ambient temperature and at low Re number which is at laminar region (Re < 2000).

2.1 CFD Setup

Flow simulations of the single channel were performed using the commercial ANSYS WORKBENCH 16.0 system and the Fluent solver is chosen. The single channel of monolith catalyst was designed based on the specific characteristics that are stated in Table. 1. The monolith channel with the porosity (0.814) and the fluid properties of air with density (1.18 kg/m^3) was used. The Realizable k-ε (RKE) of the single channel was used in the Fluent solver setup. The boundary condition of the coarse, medium and fine monolith single channel with the number of element was stated in Table 1. Below. By choosing the best mesh independency test and the number of element shows that medium state was more mesh independent. The grid independent study also used for the square-cell shape by using the sub-grid scale modelling [13].

Table 1
Mesh Dependency Test

	Coarse	Medium	Fine
No of element	264600	477400	880600
Pressure drop (ΔP)	851	870	881

Based on geometrical and parameter of the single channel were applied to create the dynamic view of the solid model of single channel of the catalyst monolith. To produce this geometrical model of single channel is from designing the model. For designing the geometrical model, Fluent solver is chosen to sketch and choose the right planes of designing. After that, the surface is separated and named into a different functional group such as inlet and outlet for the single channel model.

As stated in Figure 2, the meshing of single channel (a) front face and (b) symmetry plane. The meshing for a single channel is chosen from the toolbar of the workbench to setup the meshing. An edge sizing along the single channel is chosen to be 1400 by using the number of division at the outlet which is 11 number of division. By setting the inflation setup, 5 number of layer inflation is chosen with the growth rate of 1.2 and the maximum thickness of inflation is 9.22e-005m.

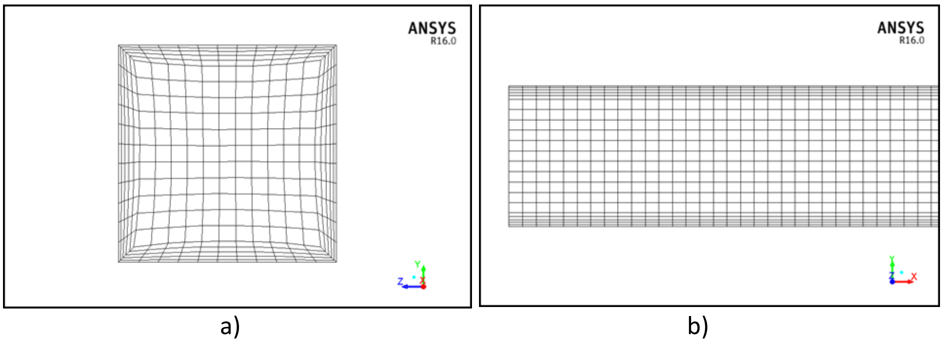
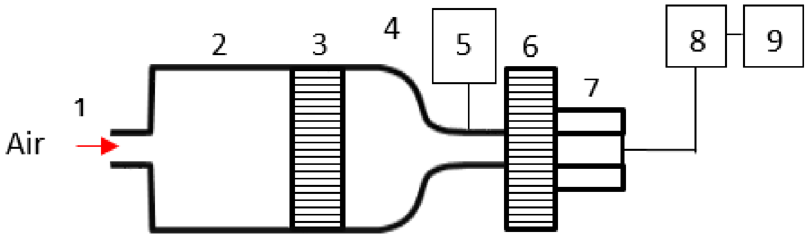


Fig. 2. Meshing of single channel at (a) front face and (b) symmetry plane

From the CFD simulation of the single channel monolith the result of the monolith resistance can be predicted.

2.2 Laboratory Setup

After done the CFD prediction test, the experimental study needs to be run to validate the data from the CFD simulation. Figure 3 shows the schematic drawing of the isothermal flow rig. Air was supplied from the air compressor entering the inlet sleeve (1) via the plenum (2) and the mixing air will aid in the reduction of the energy of the air particles. Air particles will flow through the air straightener (3) to reduce the particle energy and straighten the air particle before entering the axisymmetric nozzle (4). After air entering the nozzle to get the uniform flow, the pressure transmitter (5) was placed before the monolith as the pressure sensor of Model 5 (± 10kpa) and Furness control Limited to measure the air flow rate and the other one was exposed to the atmosphere. Air entering the monoliths (6) of the lengths 129mm was positioned downstream of the diffuser. Finally, the air entering the outlet sleeve (7) of the length 50 mm was used to minimise disturbance by surrounding air. While, the pitot tube (8) and pressure transmitter (9) was placed to detect the air flow rate leaving the monolith at 30 mm from the monolith. The pitot tube as pressure sensor model of Model 5 (± 10kpa) was manufacture from Furness Control Limited with the scale of 0 – 91.9m/s (0 – 5kpa sqr rooted).



1. Air supply
2. Plenum
3. Flow straightener
4. Axisymmetric nozzle
5. Pressure transmitter
6. 2D Monolith
7. Outlet sleeve
8. Pitot tube
9. Pressure transmitter

Fig. 3. Schematic diagram of 2D isothermal flow rig

3. Results

Figure 4 shows the pressure drop along the catalyst monolith channel with the cell density is 600 cpsi obtain from CFD prediction and theoretical formulation. The cell density of the single channel monolith was categories from the three different cell densities which is 600/3.5, 600/4.0 and 600/4.5 cpsi. The pressure drop data was plotted against the channel Reynolds number, Re_c at the CFD and Shah. Both graphs show when the Re_c increase, it will gradually increase the pressure drop of the monolith channel. Shah formulation was derived that the flow is developing laminar flow. The result of CFD predictions shows it was agreed with Shah's formulation data is due to the developing flow [14].

From Figure 5, the data was normalised of non-dimensionless monolith length against the dynamic pressure as shown in Figure 6. The measurement data are closed with the CFD prediction data and also the Shah Formulation data. The normalised data shows the air flow inside a single channel is developing laminar flow. The coefficients were included in the CFD predictions to present the axial pressure loss. The comparison between measurement, CFD predictions, Shah's equations, and H-P formulation shows that the higher channel velocity of the monolith, the higher the value of pressure drop. The pressure loss coefficient of the 600 cpsi monolith of 4.0 mil was much smaller than those 3.5 mil and 4.5 mil because the developing boundary layer as a small fraction of the total hydraulic diameter of the monolith can cause a smaller pressure drop per unit length. The similar trend has been observed [15] and proved that the pressure drop is proportional to the fluid velocity.

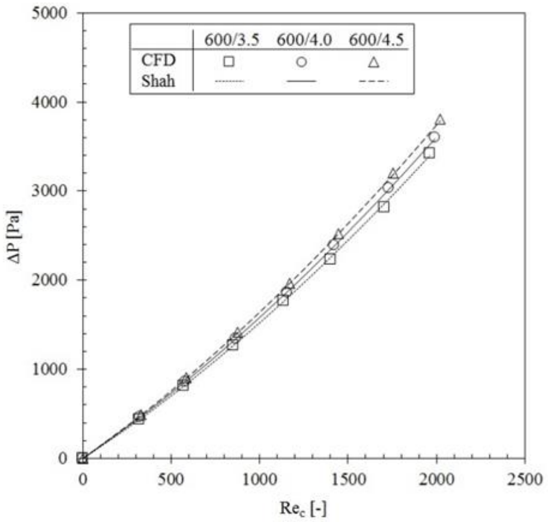


Fig. 4. Pressure drop, ΔP versus Reynolds's number inside the channel, Re_c of CFD and Shah

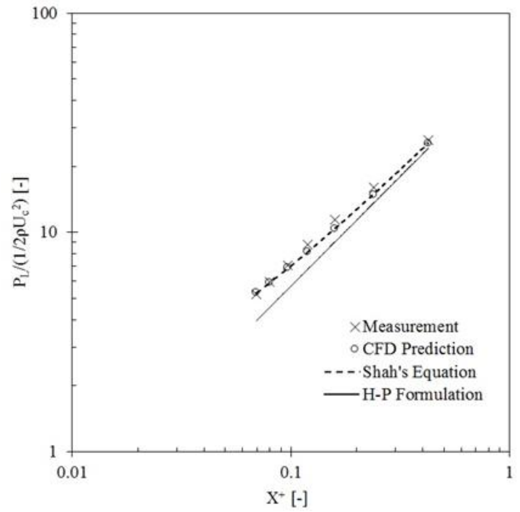
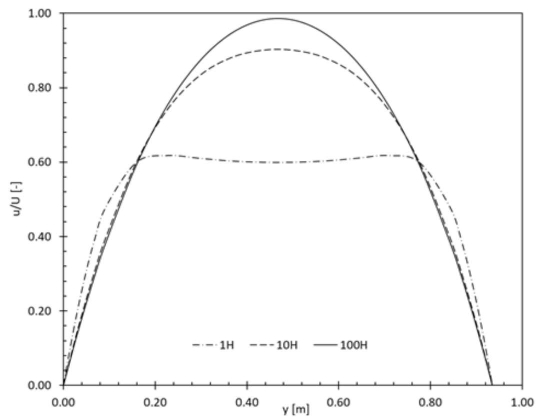
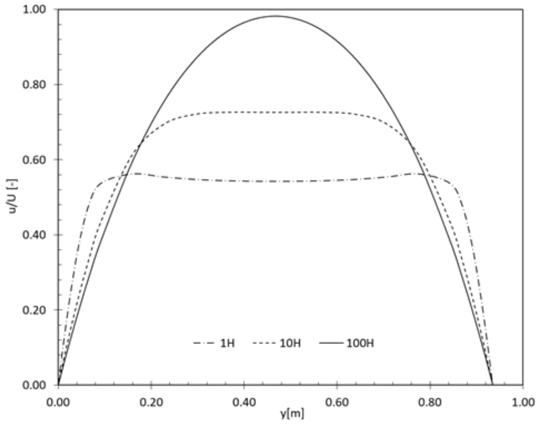


Fig. 5. $P_L/ (1/2\rho U_c^2)$ versus X^+

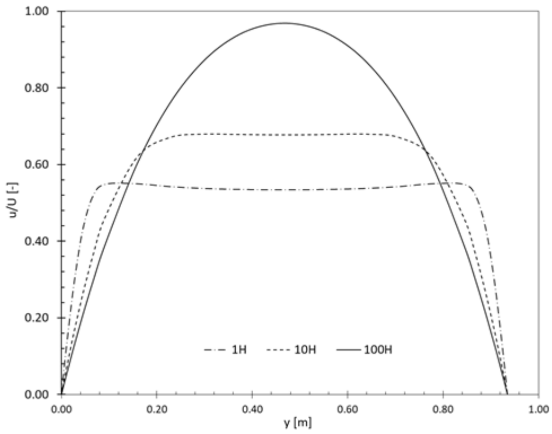
Figure 6 shows the plug profile of the single channel at the Re number of 325, 1154 and 1991. The plug profile shows the developing boundary layer along the monolith single channel. The boundary layer development is observed at three different entry length of the monolith single channel e.g. 1H, 10H, and 100H. At the lower Re number of 325, the boundary layer development occurred at the lowered entry length of the monolith single channel.



(a) Re = 325



(b) Re = 1,154



(c) Re = 1,991

Fig. 6. Plug profile of single channel at (a) Re=325, (b) Re=1,154 and (c) Re=1,991

The simulated flow behaviour and plug profile of single channel were concluded in Figure 6. Figure 7 shows the wall shear stress against the monolith length of the monolith single channel. The variation of wall shear stress in the flow direction for flow inside the single channel at the entrance region to fully developed region. For the shorter the channel height, the shorter turbulent flow inside the monolith single channel as expected and also depends on the lower Re number.

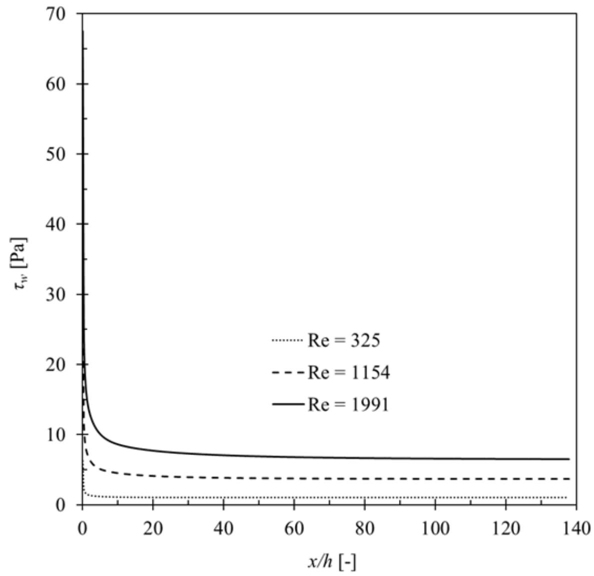


Fig. 7. Wall shear stress against monolith height

4. Conclusions

CFD studies have been performed in the simplest model to represent the catalyst monolith channel of the automotive catalytic converter. The results positively show that the pressure drop of CFD modelling corresponds with the measurement and empirical formulations. However, the parabolic profile of the plug profile and the shear stress of the monolith single channel shows that the higher Re number, the higher the hydrodynamic entry length.

Acknowledgment
This work is supported by University Teknikal Malaysia Melaka (UTeM). This financial support is funded by a grant from the Ministry of Higher Education of Malaysia (FRGS/2/2014/TK01/FKM/02/F00231)

References
[1] Pardiwala, Julie M., Femina Patel, and Sanjay Patel. "Review paper on catalytic converter for automotive exhaust emission." *gas* 18 (2011): 19.
[2] L. Venkatesh, R. Logeshkumar, G. Jayaprakash, and N. Dinesh. "Control and Reduction of Emission Using Catalytic Converter. " *International Research Journal of Engineering and Technology* 4, no. 11 (2017): 788-794.
[3] Anupam Mukherjee, Kunal Roy, Jayabrata Bagchi, and Konika Moldan. "Catalytic Converter in Automotive Exhaust Emission." *Journal for Research* 2, no. 10 (2016): 29-33.
[4] Balogh, Ramon-Mihai, Ioana Ionel, Dan Stepan, Hans-Peter Rabl, and Andreas Pfaffinger. "NOx reduction using Selective Catalytic Reduction (SCR) system–α variation test." *Termotehnica* 2 (2011): 38-42.
[5] Hayes, R. E., Anton Fadic, Joeseph Mmbaga, and A. Najafi. "CFD modelling of the automotive catalytic converter." *Catalysis today* 188, no. 1 (2012): 94-105.
[6] Benjamin, S., C. Roberts, and J. Wollin. "A study of pulsating flow in automotive catalyst systems." *Experiments in fluids* 33, no. 5 (2002): 629-639.
[7] Porter, Sophie, Ahmad Kamal Mat Yamin, Svetlana Aleksandrova, Stephen Benjamin, Carol A. Roberts, and Jonathan Saul. "An assessment of CFD applied to steady flow in a planar diffuser upstream of an automotive catalyst monolith." *SAE International Journal of Engines* 7, no. 4 (2014): 1697-1704.
[8] Quadri, S. S., Stephen F. Benjamin, and Carol A. Roberts. "An experimental investigation of oblique entry pressure losses in automotive catalytic converters." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 223, no. 11 (2009): 2561-2569.
[9] Tanno, Kenji, Hisao Makino, Ryoichi Kurose, Satoru Komori, Hiroshi Shimada, and Seung-Min Hwang. "Effect of turbulent–laminar flow transition on degradation of de-NOx catalyst." *Fuel* 89, no. 4 (2010): 855-858.

[10] Guojiang, Wu, and Tan Song. "CFD simulation of the effect of upstream flow distribution on the light-off performance of a catalytic converter." *Energy Conversion and Management* 46, no. 13-14 (2005): 2010-2031.

[11] Muhieldeen, M W. "Saving Energy Costs by Combining Air-Conditioning and Air- Circulation Using CFD to Achieve Thermal Comfort in the Building. " *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, no. 1 (2019): 84–99.

[12] Yosri, Muhammad Hafizan, Pauziah Muhamad, and Norfazrina Hayati Mohd Yatim. "Evaluation of Air Flow Pattern for Conceptual Design of Automotive Painting Line Using Computational Fluid Dynamic (CFD) for Better Dust Particle Reduction." *CFD Letters* 11, no. 2 (2019): 42–49.

[13] Amirnordin, Shahrin Hisham, Suzairin Md Seri, Wan Saiful-Islam Wan Salim, Hamimah Abd Rahman, and Khalid Hasnan. "Pressure drop prediction of square-cell honeycomb monolith structure." In *2011 International Conference on Environment Science and Engineering IPCBEE*, vol. 8. 2011.

[14] Shah, R. K. "A correlation for laminar hydrodynamic entry length solutions for circular and noncircular ducts." *Journal of Fluids Engineering* 100, no. 2 (1978): 177-179.

[15] Yamin, Ahmad Kamal Mat. "Pulsating flow studies in a planar wide-angled diffuser upstream of automotive catalyst monoliths." PhD diss., Coventry University, 2012.