International Journal of Heat and Mass Transfer 127 (2018) 546–560

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Experimental and numerical investigations of flow through catalytic converters

Hesham A. Ibrahim [a], Wael H. Ahmed [a,*], Sherif Abdou [b], Voislav Blagojevic [b]

[a] University of Guelph, School of Engineering, 50 Stone Road, Guelph, ON NIG 2W1, Canada
[b] Vida Fresh Air Corp, 101 – 5500 North Service Rd., Burlington, ON L7L 6W6, Canada

ARTICLE INFO

Article history:
Received 14 February 2018
Received in revised form 8 July 2018
Accepted 9 July 2018

Keywords:
Fluid flow
Catalytic converter
Thermal
CFD
Experimental

ABSTRACT

The need for improved fuel economy, while meeting more stringent global vehicle emission standards, continues to grow with the increasing demand for environmental protection and rising fuel prices. A new generation of catalytic converters, designed and patented by Vida Fresh Air Corp., offers emissions reduction while improving fuel economy. In this design, a thin layer of insulating material is placed inside the ceramic honeycomb channels, creating a multi-chamber catalytic converter. The improvement in performance of the catalytic converter is attributed to the change in both the flow distribution and the controlled heat diffusion from the inner to the outer chambers. On engine performance tests have shown significant improvements in both fuel economy and emissions, however, the theory of operation of this design needs to be validated for potential design improvements to achieve an optimum performance. In this study both experimental and numerical investigations are carried out in order to understand the flow through the catalytic converter, using different monolith cell densities. A dynamically scaled-down model for a typical flow through catalytic converter was utilized for this study. Detailed experiments were conducted using hot air as the working fluid in order to evaluate the thermal and fluid flow characteristics of the new catalytic converter technology without the effect of chemical reactions. The measurements were performed at a Reynolds number of 43,000 with a free stream temperature of 177 °C. These conditions were selected in order to achieve thermal and hydraulic similarity to actual flow conditions for a typical catalytic converter. Numerical modelling of the flow through the setup under investigation was found to adequately replicate the experimental measurements for temperature, velocity and turbulence intensity within ±3%, ±5% and ±8% respectively. The use of a new design of the catalytic converter found to improve the thermal performance by 18% and the hydraulic performance by 5% without a significant increase of the pressure drop across the catalytic converter.

1. Introduction

Since their inception, automobiles have mostly been powered with an internal combustion engine. The combustion processes associated with those engines are responsible for releasing harmful emissions including carbon monoxide (CO), unburnt hydrocarbons (HC) and nitrogen oxides (NOx) which have severe negative effects on humans and the environment. This has led to the development of emission control systems to treat exhaust gases and convert them to less harmful products called catalytic converters [19,28,4].

In the thirties, automobile manufacturers realized the need to treat the products of the combustion process that occurred within the internal combustion engines [39]. The initial solution for this challenge was the utilization of pellet catalytic converters. The pellets were spherical particles with diameters ranging from 2.5 mm to 5 mm and made of gamma-alumina. The pellets were enclosed in a steel shell to form the catalytic converter and loaded with noble metals and stabilizers for exhaust emission treatment. This catalytic converter technology had many disadvantages. Due to the design of these pellet catalytic converters, a large pressure drop occurred across the converter which directly affected the performance of the engine. Moreover, the risk of losing the catalyst was higher due to particle wear [40]. These disadvantages encouraged scientists and engineers to develop monolithic catalytic converter found in vehicles today.

Monolith substrates are the main component of exhaust line after treatment systems found in automobiles today. They provide superior performance in comparison to the pellets support type. Monolith substrates are typically characterized by their cell

* Corresponding author.
E-mail address: ahmedw@uoguelph.ca (W.H. Ahmed).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.07.052
0017-9310/

density and channel wall thickness. Due to the large total surface area of the channels and the small thermal mass of the substrate, heat transfer is greatly enhanced, which improves the conversion efficiency indicating an improved thermal performance [33]. The thermal performance of a catalytic converter is typically measured in terms of the time needed for the catalytic converter to reach the light-off temperature. The "light-off" temperature is quantified as the temperature at which the conversion efficiency of pollutants reaches 50% [13]. On the other hand, due to the geometrical design of the catalytic converter can, flow reaching the front face of the substrate is typically not uniform. Therefore, improving the thermal and hydraulic performance of the monolith substrate enhances light off characteristics of the catalytic converter and improves efficiency of the converter [12,30].

Due to the difficulty of instrumenting catalytic converter monoliths and the fact that measurement instrumentation affects the flow through the catalytic converter, CFD simulations have lent themselves as a valuable tool used to evaluate the performance of catalytic converters for the last three decades. Many techniques to mathematically model the flow field inside the substrate starting from 1D unidirectional models to full and comprehensive 3D models are suggested by Chen et al. [5], Kumar and Mazumder [18], Ozhan et al. [23] and Hayes et al. [10]. In these studies, CFD (Computational Fluid Dynamics) analysis was used to predict the behaviour of the flow, thermal characteristics and conversion efficiency of monolith substrates. Moreover, Ferguson and Finlayson [8], Heck et al. [11] and Young and Finlayson [32] developed the earliest mathematical models to study the physical and chemical processes within the catalytic converter. Their model incorporated heat and mass transfer effects inside the monolith and the laminar flow inside the monolith channels. Their models were the basis for most of the developed models. With huge advancements in computational power over the past three decades, mathematical modelling has become economical and faster to use and researchers now rely extensively on CFD tools to simulate and evaluate the performance of catalytic converters.

Early work done on characterizing the flow behaviour and thermal characteristics of catalytic converters involved steady state and transient analysis using different boundary conditions. Many researchers investigated and simulated steady state flow under reacting flow conditions [9,30,25,29] and other researchers investigated the non-reacting steady state flow inside a catalytic converter [12,21,20]. Transient flow simulations were also used by some researchers to investigate the performance of the catalytic converter during cold-start period including [37,3,10,45]. Moreover, other researchers studied only the hydraulic behaviour of the flow within the monolithic substrate under steady state cold flow conditions [12,2,10].

Koltsakis [17] and Shelef and McCabe [26] reviewed catalytic converter systems for automobile emission control. Their studies covered the main principles and performance of catalytic converters. They discussed the durability of catalytic converters as well as the effect of catalytic converter performance on the thermal management of the engine. In addition, a comprehensive review of the concurrent mathematical models developed to monitor and optimize catalytic converter performance as well as studying the catalytic reactions and the physical characteristics was discussed. Other the other hand, Chen et al. [5] presented an overview of the state of the art of the mathematical models used to model the monolith and the catalytic reactions. They discussed various aspect of the modelling process including steady state and transient modelling, gas uniformity and flow distribution, chemical kinetics and conversion characteristics and heat and mass transfer within the monolith.

Improving the hydraulic performance means improving the flow distribution of the flow within the substrate and minimizing the generated back pressure. A more uniform flow distribution enhances the conversion efficiency and durability of the catalytic converter [1,14,13,16,22]. When flow is not uniformly distributed, the catalyst within the ceramic substrate becomes partially utilized and that leads to a reduction in the conversion efficiency and the life time of the monolith. Moreover, lower back pressure improves engine performance resulting in more fuel-efficient vehicles. This leads to less GHG (Green House Gas) emissions [7,36,10]. On the other hand, studies including Cho et al. [6], Chen and Schirmer [42] and Shuai and Wang [27] focused on the effect of exhaust manifold design and monolith properties on the flow distribution and hydraulic performance of catalytic converters.

Lai et al. [20] studied the effect of exhaust manifold geometry on the flow distribution as bent pipes tend to distort the flow and increase flow maldistribution. They utilized 3D simulations incorporating the brick resistance into the simulation to obtain accurate predictions. They concluded that the flow becomes more uniformly distributed when the inlet pipe is shorter in length and the bending angle is smaller. Moreover, they investigated the effect of brick properties on flow distribution concluding that the higher the brick resistance the more uniform the flow distribution observed. Similar conclusions were reached by Amirnordin et al. [36], Karvounis and Assanis [16], Ramanathan et al. [45], Weltens et al. (1993), Chen et al. (2004) and Lun et al. (2010). They also added that, with multi-brick catalytic converters, the second brick tends to show more uniform flow distribution than the first one due to the gap between the two bricks which allows flow to redistribute more uniformly. They specifically suggested using smaller channel hydraulic diameters and shorter ceramic substrates to overcome the pressure drop developed by higher brick resistance.

On the other hand, Jeong and Kim [13], Lai et al. [20] and Weltens et al. (1993) utilized CFD models to investigate the effect of the inlet Reynolds number on flow distribution and they reported that flow uniformity decreases when increasing the inlet flow velocity and Reynolds number. However, when testing the effect of pulsating flow on flow distribution they found that flow pulsations tend to enhance the flow uniformity. Liu et al. [43] and Liu et al. [44] performed an experimental and a numerical study on a reverse flow catalytic converter for a natural gas/diesel dual engine. The simulation involved a 1-dimensional single channel model to model the substrate. They concluded that the conversion efficiency of CO and HC was improved for the reverse flow catalytic converter for low inlet temperature and light engine load only when the catalytic converter's initial temperature is high enough.

The effect of pressure drop on the hydraulic performance of catalytic converters has been investigated by many researchers ([38,2,7,35,10]). They investigated the effect of inlet flow conditions, substrate properties and catalytic converter geometry on the pressure drop utilizing various modelling strategies. On the other hand, the thermal behaviour of catalytic converters has been investigated by many researchers Chakravarthy et al. [3], Shamim et al. [46], Hayes et al. [10], Jeong and Kim [13] and Chung et al. [41]. Conversion efficiency and thermal response of the monolith under different load conditions are crucial to understand the thermal characteristics of the catalytic converter. Understanding the interactions between the chemical and physical processes within the monolith channels is a key factor in modelling monolith reactors [5].

The need for better automotive technologies to improve fuel economy, while meeting more stringent global vehicle emission standards, continues to grow with the increasing demand for environmental protection and rising fuel prices. Therefore, improving the thermo-fluid performance of catalytic converters is necessary to meet emission regulations. Improving the performance of catalytic converters requires intensive experimental investigations to study flow behaviour inside the catalytic converter. This