

An Assessment of CFD Applied to Steady Flow in a Planar Diffuser Upstream of an Automotive Catalyst Monolith

Sophie Porter
Coventry Univ.

Ahmad Kamal Mat Yamin
Universiti Teknikal Malaysia Melaka

Svetlana Aleksandrova, Stephen Benjamin, Carol A. Roberts, and Jonathan Saul
Coventry Univ.

ABSTRACT

Flow maldistribution across automotive exhaust catalysts significantly affects their conversion efficiency. Flow behaviour can be predicted using computational fluid dynamics (CFD). This study investigates the application of CFD to modelling flow in a 2D system consisting of a catalyst monolith downstream of a wide-angled planar diffuser presented with steady flow. Two distinct approaches, porous medium and individual channels, are used to model monoliths of length 27 mm and 100 mm. Flow predictions are compared to particle image velocimetry (PIV) measurements made in the diffuser and hot wire anemometry (HWA) data taken downstream of the monolith. Both simulations compare favourably with PIV measurements, although the models underestimate the degree of mixing in the shear layer at the periphery of the emerging jet. Tangential velocities are predicted well in the central jet region but are overestimated elsewhere, especially at the closest measured distance, 2.5 mm from the monolith. The individual channels model is found to provide a more consistently accurate velocity profile downstream of the monolith. Maximum velocities, on the centre line and at the secondary peak near to the wall, are reasonably well matched for the cases where the flow is more maldistributed. Under these conditions, a porous medium model remains attractive because of low computational demand.

CITATION: Porter, S., Mat Yamin, A., Aleksandrova, S., Benjamin, S. et al., "An Assessment of CFD Applied to Steady Flow in a Planar Diffuser Upstream of an Automotive Catalyst Monolith," *SAE Int. J. Engines* 7(4):2014, doi:10.4271/2014-01-2588.

INTRODUCTION

Catalytic converters are widely used in the automotive industry in order to reduce harmful exhaust emissions and conform to emission regulations. For optimum conversion efficiency, uniform flow is required across the monolith. Computational fluid dynamics (CFD) is increasingly being used to assess the flow distribution of different design concepts; it is therefore necessary that predictions of flow behaviour offered by CFD are reliable and of sufficient accuracy.

The catalyst is typically a ceramic monolith structure comprised of many parallel channels of square cross-section and small hydraulic diameter (~1 mm). The precious metals of the catalyst are embedded in a thin washcoat applied to the channel walls, thus providing a large surface area for reaction of exhaust pollutants. The size and shape of monolith depends on vehicle size and packaging constraints, however a typical passenger vehicle monolith would have a circular or oval cross-section and a cell density between 31 and 140 cells/cm².

Due to space constraints, wide-angled diffusers are used to connect the exhaust pipe to the front face of the catalyst. This results in flow separation at the diffuser inlet and a non-uniform flow distribution within the monolith.

Modelling the flow is challenging. The model should not only adequately simulate the flow in the diffuser but also account for the pressure loss associated with the monolith itself. One approach would be to solve the flow field both within the diffuser and in each of the individual channels. This is computationally demanding as it requires discretising the flow domain over several thousand channels. An alternative approach, widely used, is to treat the monolith as a porous medium or equivalent continuum. This reduces the computational effort but requires an accurate prescription of the monolith resistance. A common assumption is that this can be adequately described by losses associated with unidirectional flow in the channels. These can be obtained from theoretical considerations or can be measured directly on a flow rig. Benjamin et al. [1] found that using this approach for

axi-symmetric systems resulted in CFD under-estimating the degree of flow maldistribution. This was attributed to an inaccurate prescription of losses at locations in the monolith where the flow approached the channels obliquely.

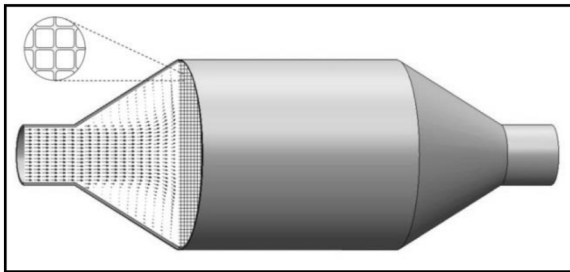


Figure 1. Schematic of flow in a catalyst assembly.

This can be illustrated by Figure 1, which shows an example of flow through an axi-symmetric wide-angled diffuser placed upstream of a monolith. The flow separates at the inlet, producing a confined jet that traverses the diffuser. Near the monolith the jet decelerates and spreads radially. Part of the flow enters the channels, the rest feeds the recirculating gas located between the jet and the diffuser wall. The net effect is that the flow entering the monolith is maldistributed, with the highest velocity occurring in the channels near the centre of the monolith. On the centre-line flow enters the channels axially, whereas at larger radial distances it approaches them obliquely. This can cause flow separation at the channel entrance, especially at high incidence, and hence an increased resistance at these locations. The radial flow stagnates as it approaches the diffuser wall near the front face of the monolith, causing a rise in pressure and encouraging flow through the outermost channels. This results in secondary velocity peaks for channels close to the wall.

Losses due to oblique entry have been measured by Quadri et al. [2] and Persoons et al. [3]. Küchemann and Weber [4] derived an expression for this pressure loss, given in Eq. (1), based on considerations of oblique entry into flat plate heat exchangers.

$$\Delta P_{obl} = \frac{\rho v^2}{2} \quad (1)$$

Here ρ denotes fluid density and v the transverse velocity component of flow upon entry into the heat exchanger. Incorporating Eq. (1) into CFD simulations, Benjamin et al. [1] achieved much-improved maximum velocity predictions; however, the secondary peaks were over-predicted. This was attributed to very high levels of resistance at high incidence, promoting increased flow towards the outer channels. A critical angle approach was therefore proposed by Quadri et al. [2], where the pressure losses were capped at higher angles of attack. For a given critical angle α_c , the pressure loss was assumed to satisfy

$$\Delta P_{obl} = \begin{cases} \frac{\rho v^2}{2}, & \alpha < \alpha_c \\ \frac{\rho v_c^2}{2}, & \alpha \geq \alpha_c \end{cases} \quad (2)$$

where α denotes the angle of attack and v_c the transverse velocity component of flow entering at α_c . This improved predictions of the velocity distribution, although the distribution away from the centre-line was shown to be quite sensitive to the chosen value of the critical angle.

These studies compared predictions with velocity measurements at the monolith exit and hence implicitly assumed that predictions within the diffuser were correct. Development of optical methods such as particle image velocimetry has now enabled measurement of the flow field upstream of the monolith. Breuer et al. [5] used PIV and hot wire anemometry downstream of a catalytic converter to validate CFD, while Turner et al. [6] used PIV for the validation of CFD applied to flow in the inlet diffuser upstream of a diesel particulate filter. Quadri et al. [7] measured velocity profiles upstream and downstream of monoliths for a planar geometry with steady flow. Mat Yamin et al. [8] obtained similar data under both steady and pulsating flow. This paper uses the steady flow data from [8] and extra measurements performed with a fitted sleeve to compare porous medium model predictions with measurements both within the diffuser and in the monolith (as measured downstream). The planar geometry provides full optical access in the diffuser and permits detailed comparisons to be made. Measurements downstream of the monolith also provide the opportunity to assess the validity of the pressure loss expressions discussed previously.

The porous medium approach is computationally efficient but requires prescription of the pressure loss as discussed above. A model which includes the geometry of all the monolith channels provides a detailed representation of the geometry and avoids the need to prescribe these losses a-priori. However it is achieved at the expense of a high computational demand. The 2D planar geometry described in [8], however, does provide the opportunity of assessing this type of model by only requiring the modelling of channels across a section of the catalyst. Such a model is also described in this paper and is compared with the porous medium approach.

METHODOLOGY

Experimental Data Collection

Figure 2 shows a schematic diagram of the isothermal flow rig used for steady state flow measurements. The experimental set-up is presented by Mat Yamin [8, 9] and is reiterated here.

The high pressure line supplies air through a viscous flow meter into the flow rig (1) where a plenum (2) with flow straightener (3) and axisymmetric nozzle (4) leads to a pulse generator (5). Measurements of non-pulsating flow, used in the current study, were obtained by fixing the rotor of the pulse generator in a fully open position. Air then flows through another flow straightener (6) and past a resonator box (7), installed to shape pulses during pulsating flow studies. A particle generator (9) supplies seeding to a second plenum (8) with flow straightener (10) to minimise any swirl components. A two-dimensional nozzle (11) provides uniform flow to the planar diffuser (12), thus generating well-defined inlet conditions for CFD. The diffuser has inlet dimensions 24×96 mm, outlet dimensions 78×96 mm, length 48 mm and total included angle of approximately 60° . The diffuser is made of crown glass for optimal optical conditions for PIV. The diffuser outlet attaches to an unwashcoated cordierite monolith (13) of length 27 mm or 100 mm. The monoliths had channel hydraulic diameter 1.12 mm, a nominal cell density of 62 cells/cm² and porosity 0.77. An outlet sleeve (14) of length 50 mm minimised any influence from air outside the rig. The sleeve had dimensions 125×137 mm and thus provided an expansion at the rear of the monolith. As part of the present investigation a fitted sleeve of the same dimensions as the diffuser outlet was used and HWA profiles were measured at the rear of the monolith. These measurements permitted an assessment of the performance of the models in predicting flow within the monolith and mixing downstream of the monolith.

Flow within the diffuser was measured using a TSI PIV system. A six-jet atomiser at 25 psi produced olive oil droplets of approximately $0.6 \mu\text{m}$ diameter. A cylindrical lens of -25 mm focal length was combined with a spherical lens of 500 mm to transform the circular beam from a 120 mJ solid-state Nd:YAG laser into an approximately 1 mm thick light sheet at a stand-off distance of 0.5 m to illuminate the seeded flow. A 4-megapixel CCD camera with resolution of 2048×2048 pixel (1 pixel = $7.4 \mu\text{m}$) was used to capture the flow field. The camera, coupled with a 105 mm lens, was placed 0.8 m from the measurement plane to cover an 80×60 mm field of view resulting in a magnification factor of 0.155. An f number of 11 enabled a particle image diameter above 2 pixels, therefore avoiding pixel locking. INSIGHT-3G software was used for image processing. The recursive Nyquist method with a 64×64 initial grid and a final grid of 32×32 pixels yielded 95% valid vectors in each field and vector resolution of 0.76 mm.

Axial velocity profiles at the nozzle exit and downstream of the monolith were obtained using a TSI IFA 300 hot-wire anemometry system. $5 \mu\text{m}$ platinum-plated tungsten wires (Dantec 55 P11) were calibrated on an automatic TSI 1129 calibration rig. A 1 MHz 4 channel 12 bit A/D converter converted the IFA output voltage (± 5 V) to a digital signal to be processed by ThermalPro software.

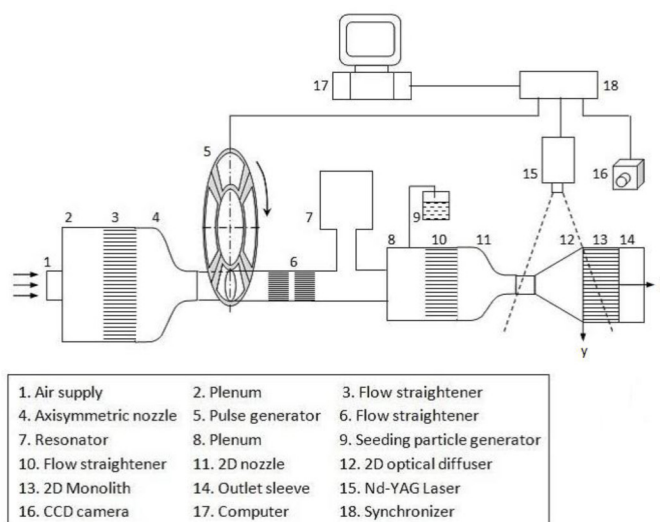


Figure 2. Schematic diagram of 2D isothermal flow rig

CFD Design

Flow simulations were performed using the commercial CFD code Star-CCM+ on a flow domain consisting of a diffuser, substrate and outlet sleeve (Figure 3). Velocity profiles across horizontal and vertical planes at the nozzle exit were found to be acceptably uniform [8], thus flow at the diffuser inlet may be assumed one-dimensional for simulation purposes. Velocity profiles at the monolith exit for various z positions [9] show little variation and symmetry; it is therefore acceptable to model the system as two-dimensional and symmetrical with respect to the plane $y=0$. The fluid is modelled as a constant density air flow. Reynolds averaged Navier Stokes (RANS) equations are combined with the v^2f turbulence model, proven to improve accuracy for separating flows [10].

The computational method entailed in this study comprises of two distinct approaches to the same problem. The first is the more common method of modelling the monolith as a porous medium with a prescribed resistance. The second technique allows the monolith to retain its solid properties at the channel walls and therefore employs a geometry design inclusive of small, parallel channels through which fluid flows. The model requires a three-dimensional mesh with prism layers at the channels walls. Table 1 compares the computational resources required for each approach. The meshes cited were subject to a mesh independence study, which showed negligible change to solutions for a more refined mesh.

Table 1. Computational resources for CFD models

Model	Porous Medium		Individual Channels	
Monolith length	27 mm	100 mm	27 mm	100 mm
# cells	28,423	47,354	3,962,318	8,824,908