

# Comparison of Experimental PIV Data and CFD Simulations for Flow in a Diesel Particulate Filter Inlet Diffuser

2011-01-1241

Published  
04/12/2011

Christopher Turner, David Thornhill and Geoffrey McCullough  
Queen's Univ., Belfast

Samir Patel  
Ansys Europe

Copyright © 2011 SAE International

doi:[10.4271/2011-01-1241](https://doi.org/10.4271/2011-01-1241)

## ABSTRACT

Flow maldistribution of the exhaust gas entering a Diesel Particulate Filter (DPF) can cause uneven soot distribution during loading and excessive temperature gradients during the regeneration phase. Minimising the magnitude of this maldistribution is therefore an important consideration in the design of the inlet pipe and diffuser, particularly in situations where packaging constraints dictate bends in the inlet pipe close to the filter, or a sharp diffuser angle. This paper describes the use of Particle Image Velocimetry (PIV) to validate a Computational Fluid Dynamic (CFD) model of the flow within the inlet diffuser of a DPF so that CFD can be used with confidence as a tool to minimise this flow maldistribution.

PIV is used to study the flow of gas into a DPF over a range of steady state flow conditions. The distribution of flow approaching the front face of the substrate was of particular interest to this study. Optically clear diffusing cones were designed and placed between pipe and substrate to allow PIV analysis to take place. Stereoscopic PIV was used to eliminate any error produced by the optical aberrations caused by looking through the curved wall of the inlet cone.

In parallel to the experiments, numerical analysis was carried out using a CFD program with an incorporated DPF model. Boundary conditions for the CFD simulations were taken from the experimental data, allowing an experimental validation of the numerical results. The CFD model incorporated a DPF model, the cement layers seen in

segmented filters and the intumescent matting that is commonly used to pack the filter into a metal casing. The mesh contained approximately 580,000 cells and used the realizable  $\kappa$ - $\epsilon$  turbulence model. The CFD simulation predicted both pressure drop across the DPF and the velocity field within the cone and at the DPF face with reasonable accuracy, providing confidence in the use the CFD in future work to design new, more efficient cones.

## INTRODUCTION

With modern vehicle emission legislation becoming ever more stringent, it has become common practice to fit diesel vehicles with DPFs to reduce the amount of potentially carcinogenic particulate matter released into the atmosphere. Space constraints in vehicles often mean that diffusing cones with very sharp included angles are used to convert the cross sectional area of the exhaust pipe to that of the DPF. [Figure 1](#) shows an example of the type of area change seen within an exhaust system when space constraints are a major factor of the exhaust system design. Designs such as this lead to a jet of exhaust gas acting on a small area of the filter, reducing the effective flow area and hence reducing the efficiency of the substrate.

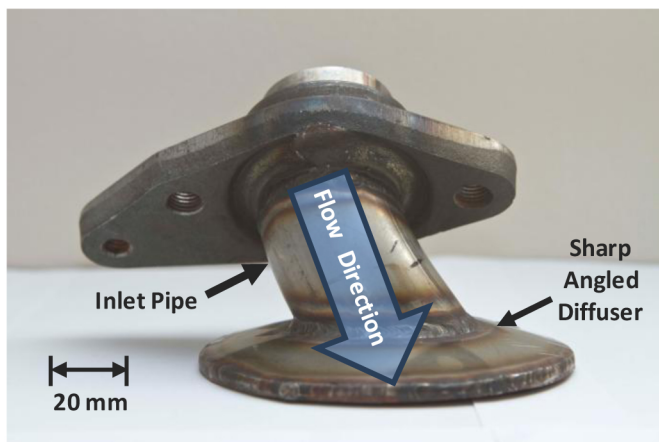


**Figure 1. Sudden Expansion into an Exhaust Substrate**

The impinging jet produces an uneven flow velocity entering the face of the DPF substrate, referred to here-in as maldistribution of flow. This phenomenon is likely to cause uneven soot distribution, which in turn will create non-uniform heat release during the regeneration process. The non-uniform exothermic reaction leads to thermal gradients within the substrate, potentially causing mechanical failure due to thermal stress.

Many studies have been conducted in the past to understand the flow entering and leaving DPF substrates and to attempt an improvement in the performance of DPFs. CFD has become a very useful tool for this purpose, with simulations of flow through DPFs becoming more complex in recent years. Porous media models have been used in the past to simulate the flow through DPFs, for example, Zhang et al [1] modelled the DPF as a porous media with laminar flow enforced and increased resistance to flow in lateral directions to ensure no lateral flow within the substrate. More recently there have been many DPF pressure drop models published based on the work by Konstandopolous [2, 3]. In 2009 Koltsakis et al [4] and Bensaid et al. [5] published complex DPF models that allow the study of the soot loading and regeneration cycle. In recent years Charbonnell and Opris [6] developed a pressure drop model that is fundamentally different to that prescribed by Konstandopolous. The model was based on previous work by Opris [7] and uses Navier Stokes equations to derive the back-pressure equation. The approaches differ in that the Konstandopolous method contains semi-empirical formulae, whereas the approach by Charbonnell and Opris is free from such formulae, leaving it slightly more flexible. While these approaches are fundamentally different, both models have been fully validated, Konstandopolous using extensive pressure drop experiments and Charbonnell and Opris using CFD with empirical inputs, neither model has been proven to perform better than the other.

The numerous CFD simulations of DPF flow properties have yielded some very interesting results. The study by Koltsakis et al showed the self balancing effects that soot agglomeration has on the flow distribution through the DPF, an effect that has been demonstrated experimentally by Stratakis and Stamatelos [8] in 2004. The CFD studies have also confirmed that the inlet geometry leading up to the front face of the DPF plays a vitally important role in the flow distribution entering the substrate. Koltsakis et al. [4] carried out numerical simulations on different types of inlet geometries including shallow and steep angle straight inlet cones, asymmetric cones, and 90 degree bends immediately before the inlet cones, demonstrating the different effects on flow distribution resulting from these geometries. Zhang et al. [1] studied the effect that altering the approach angle of inlet cones had on the flow uniformity entering the DPF substrate. These parametric studies showed that it is possible to optimize the inlet geometry to provide a significant improvement in flow uniformity at the DPF face. While there are both experimental studies and numerical studies on the flow through DPFs and their inlet cones, rarely are the numerical models developed alongside an experimental program of work. Studies by Koltsakis, Zhang, and Bensaid while validated using other methods such as using single channel models, contain no experimental data to compare with the numerical solution.

Experimental studies on the flow distribution through exhaust substrates such as DPFs and catalytic converters have been carried out in the past using techniques such as Hot Wire Anemometry (HWA) and Laser Doppler Velocimetry [8, 9, 10]. These techniques have proven to be capable of measuring flow properties in numerous studies since their development; however HWA and LDV are, in principle, point measurement techniques with a high temporal resolution. Time averaging techniques are therefore used to gain velocity profiles of an entire flow field. The use of time-averaging makes it difficult to use these techniques in a flow that is unsteady and, as such, experiments are usually conducted at steady flow conditions, or the results from unsteady flow are ensemble averaged. As HWA is intrusive it is difficult to measure the flow entering the DPF without altering its structure. Stratakis and Stamatelos avoided this problem by measuring at the exit of the DPF without the cone present - a technique that the study by Zhang [1] suggested could over predict the flow uniformity by up to 7%. PIV is a different form of velocity measurement and offers the user a chance to record the velocity distribution throughout a large area at an instantaneous point in time without intruding on the flow itself.

PIV is a flow visualisation technique that involves capturing two images of an investigation region in very quick succession. The images are separated by a carefully chosen, known time interval. The technique utilises a very thin light sheet (~1mm thick) to illuminate micro-particles that have

been dispersed throughout the flow in question. Once the cameras are calibrated for displacement, the known time interval allows the user to calculate the kinematics of the seed particles in the fluid flow [11]. The nature of PIV allows the user to gather a large amount of velocity data at a single instant in time: if this velocity data can be obtained for the same investigation region but at slightly different moments in time, great insight can be gained into the properties of highly unsteady flows. PIV allows the user to produce vector and contour plots of flow velocity that can be directly compared to those seen in many CFD simulations.

PIV has been used to study vehicle exhaust substrates in the past. Bruer et al [12] used PIV to evaluate how well HWA could determine the mass-flow distribution across a catalytic converter face, while Persoons et al [13] demonstrated that PIV is a useful tool when studying 3-D flow immediately upstream of a catalyst. PIV results matched well with those obtained by LDV, except where PIV out-performed LDV; PIV allowed the user to obtain measurements up to 0.2mm from the substrate face, while LDV only allowed measurements up to 1mm from the substrate face.

This paper describes the validation of a numerical model using experimental data. A range of steady flow scenarios were modelled and compared with experimental data predominantly obtained using PIV. An analysis of the pressure drop across a DPF assembly and of the velocity field within the diffusing section immediately in front of the DPF is discussed and the potential for future work from this study involving unsteady flow, and cone optimisation is outlined.

## EXPERIMENTAL APPARATUS

The apparatus used for this study is split into two sections, the PIV flow apparatus and the steady flow apparatus. The PIV apparatus consists of optically clear pipes and cones, while the steady flow apparatus includes a method of measuring mass flow using an orifice plate. Each of these sets of apparatus are described in further detail in their respective sections.

### PIV FLOW APPARATUS

An optically clear cone with an internal angle of 90°, seen assembled in Figure 2, was designed and manufactured from thermoformed sheet acrylic. Chloroform was used as an adhesive in the manufacture of the cone as it produces a very clean joint which is beneficial when trying to obtain PIV measurements close to the walls as it minimises laser reflections. Inlet and outlet pipes were constructed using clear acrylic tube and flanges.

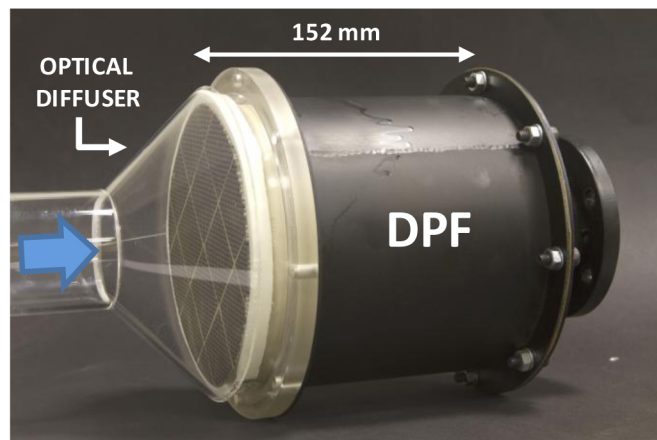


**Figure 2. DPF with Optical Cone in Position**

For all experiments, the DPF used was a SiC substrate with a diameter of 143mm and a length of 153mm. The nominal cell density is 300 cpsi with a wall thickness of 0.015 inch (0.381 mm). The substrate itself is made from 16 individual blocks joined with a cement layer approximately 2mm thick. The DPF was encased in a steel holder and held in place by a layer of intumescent matting 4 mm thick. While calculations showed that the mass of seed material introduced to the flow during experiments was negligible, the DPF was regenerated for two hours at a temperature of 500°C within an infrared oven at the end of each day's experiments to ensure any seed material trapped within the DPF was removed.<sup>1</sup>

Pressure readings were taken at four points in the experimental apparatus and the positions of the transducers can be seen in Figure 3.

The positioning of the pressure transducers was highly dependent on future work involving unsteady flow. This future work involves investigating unsteady gas dynamic pulses through the DPF using specially designed apparatus that can produce a repeatable pressure wave representative of that found in the exhaust system of an IC engine [14]. To prevent the unsteady pulses overlapping and producing unclear data, the inlet and outlet pipes were made 2m long and the pressure transducers placed appropriately to eliminate wave superposition. Therefore, the pressure drop across the DPF discussed in this work includes the inlet and exit cones along with 300 mm of pipe on either side of the DPF assembly.

<sup>1</sup>While regeneration of a DPF that contains soot requires a temperature of around 650°C, a thermo-gravimetric analysis of the olive oil used as seed material was conducted which showed that when heated to 450°C the remaining mass of olive oil residue was negligible.

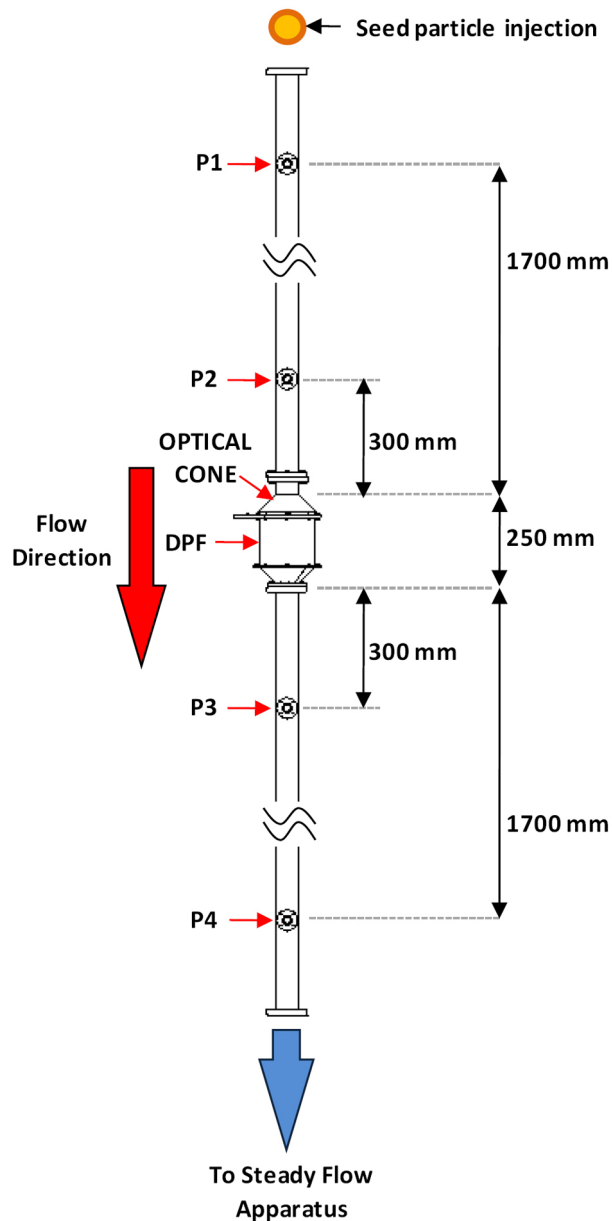


**Figure 3. Positions of Pressure Transducers**

The laser used in the experiments was a Class 4 Dual Head Nd: YAG operating in Q-Switch pulse mode. Each head of the laser produces a circular beam of wavelength 1064nm which is then converted to a wavelength of 532nm, corresponding to the green part of the visible light spectrum. The laser pulses were controlled by a pulse generator for calibration and steady flow testing. The circular beam was transformed into a thin light sheet of approximately 1mm thickness by a series of convex and concave cylindrical lenses. The lasers were triggered and images captured with

image separations of between 17 and 30  $\mu$ s, depending on the mass flow rate being investigated.<sup>2</sup>

The two cameras used for stereoscopic PIV were digital PCO SensiCams. The images were recorded at a resolution of 1280×1024 greyscale pixels. A specially designed calibration target was used to calibrate the system in three planes for stereoscopic PIV. Scheimpflug optical mounts, as seen in [Figure 4](#), were used to ensure the images were correctly focussed across the entire field of view.

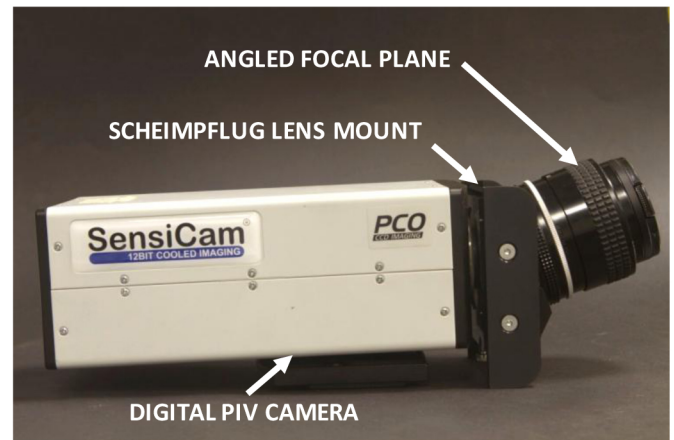


**Figure 4. Digital Camera with Scheimpflug Mount**

## STEADY FLOW APPARATUS

A 75m<sup>3</sup> vacuum tank was evacuated to -0.8 bar gauge pressure using three vacuum pumps. The evacuated tank was used to drive a steady flow representative of the mass flow rate seen in the exhaust system of a vehicle at typical part load driving conditions. The steady flow apparatus that was adapted for this study was designed to BS EN ISO 5167 and can provide steady-state testing at air flow rates of up to 0.2kg/s. The mass flow rate was measured downstream of an internally baffled stagnation tank using an orifice plate and the flow rate was controlled by a 12V motorised valve. The PIV apparatus was connected to the steady-flow apparatus via flexible pipe that fed into the stagnation tank. This apparatus contains five absolute pressure transducers that were calibrated to a Druck® pressure transducer calibrator. A schematic of the steady flow apparatus is given in [Figure 5](#).

<sup>2</sup>The period of separation is chosen to ensure that the majority of particles travelled approximately 8 pixels in either or both X and Y directions between each image. This ensures that majority of particles will remain within the 32×32 pixel interrogation regions between each image while maximizing the spatial resolution.

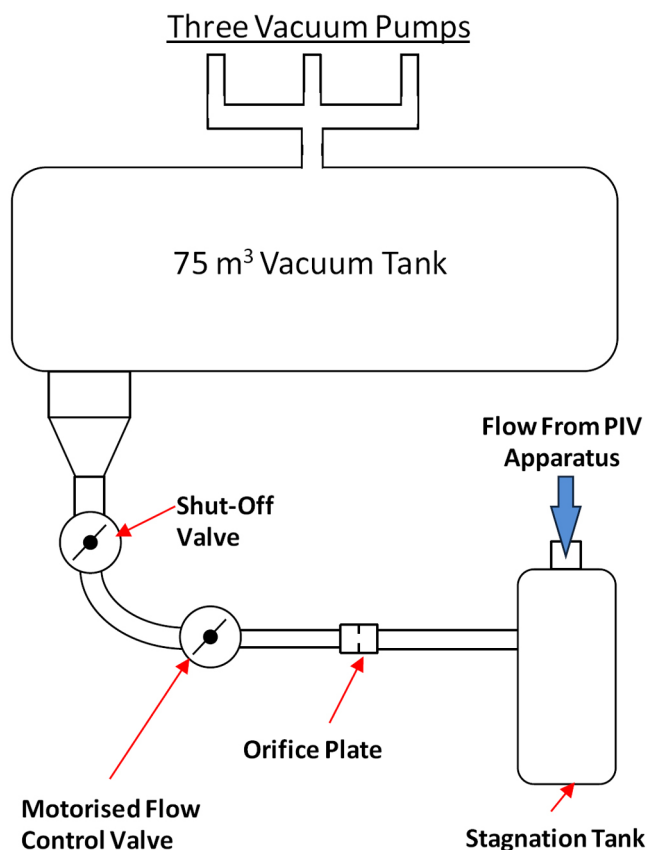


Figure 5. Steady Flow Apparatus

## EXPERIMENTAL PROCEDURE

The steady flow apparatus was used to produce a range of mass flows through the DPF. The flow rate started at 0.022kg/s and was increased in increments of 0.005kg/s to a value of 0.047kg/s to represent typical part load driving conditions of an 1800cc diesel engine. Differential pressure readings were obtained at each of the mass flow rates using an inclined manometer. Due to very slight fluctuations in the mass flow readings, each differential pressure was recorded 10 times along with the mass flow for each reading. An average of both differential pressure and mass flow was then recorded.

Due to optical aberrations produced by the curvature of the clear cone, stereoscopic PIV was used to obtain the PIV images. The two cameras were placed so that the image plane was at 80° to the object plane, allowing the user to view one half of the diffusing cone. Symmetry was assumed in the diffusing cone to negate the need to record two sets of images and stitch the information together. Figure 6 is a representation of the camera set-up. It shows the camera positions relative to the DPF using an axial (a), longitudinal (b) view.

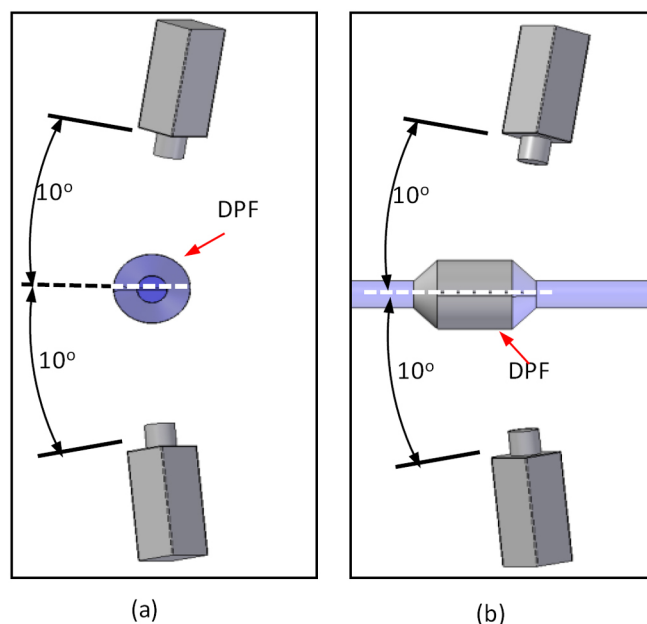


Figure 6. Stereoscopic Camera Positions: (a) Axial View; (b) Longitudinal View

## MODEL PARAMETERS AND BOUNDARY CONDITIONS

The modelled geometry consisted of a DPF region, inlet and outlet diffusing sections and inlet and outlet pipes. The inlet pipe was modelled as 2m in length and the outlet pipe as 1.7m in length to coincide with the position of pressure transducer 4 in the experimental apparatus. Discussed previously, the full length of pipe was modelled as the next step in this research is to model a highly unsteady gas dynamic flow through the DPF.

Mesh independence was conducted using a steady flow simulation of the highest mass flow rate to ensure cells in high pressure and velocity gradient regions were resolved to a satisfactory standard. The final model contained a total of 580,000 cells. Tetrahedral cells were used for the inlet and outlet cones as future work on this model involves a change of geometry in these regions. While tetrahedral meshing increases cell density and therefore solution time, it was found that it allowed for a more consistent cell size between the different geometric models of inlet and outlet geometry than using hexahedral cells exclusively. Figure 7 shows the most important aspects of the simulation geometry and displays the face mesh of the DPF.