

Development of Flow Uniformity Indices for Performance Evaluation of Aftertreatment Systems

2011-01-1239

Published
04/12/2011

Achuth Munnannur, Christopher M. Cremeens and Z. Gerald Liu
Cummins Emission Solutions

Copyright © 2011 SAE International

doi:10.4271/2011-01-1239

ABSTRACT

With the on- and off-road diesel engine emission regulations getting more stringent across the world, diesel aftertreatment systems are expected to deliver outstanding performance and reliability. These objectives should be met by fulfilling tight packaging constraints and incurring only modest material and testing costs. A major strategy for meeting these often conflicting requirements is the effective use of simulation tools such as computational fluid dynamics (CFD) in system design and performance evaluation. Prerequisites for using this CFD analysis-led-design approach, however, are knowledge of the confidence level of the predictions and knowledge of the appropriate transfer functions that establish the relationships between the measured performance parameters and model predictions. The primary aim of the present work is to develop statistically and physically relevant measures that assess the uniformity of flow in aftertreatment systems. Most commonly, a flow distribution index based on $L1$ norm has been used for quantifying flow uniformity. The index proposed in the present work is based on the $L2$ norm thereby making it more sensitive to flow maldistribution. Key statistical advantages of the proposed index are first discussed. Then, CFD predictions of flow velocities and indices are compared with experimental measurements made under realistic flow and measurement location conditions for various geometries and flow rates. Main sources of uncertainties in measurements and predictions are identified and discussed. Discussions on the use of $L2$ norm based indices in assessing exhaust gas species and temperature uniformity are also provided.

INTRODUCTION

Monolithic catalyst substrates are among the most common devices used in diesel emission control with aftertreatment

systems. Heterogeneous chemical reactions that occur within the catalyst directly affect the system performance, and understandably, catalyst dimensions, washcoat thickness and formulation, operating temperature are of importance. However, flow and thermal conditions upstream of the catalyst are also crucial in deciding the performance of a catalyst. The uniformity of flow at the inlet of a catalyst can have a significant impact on emission conversion and catalyst aging [1]. Obtaining the desired flow distribution at the catalyst inlet is an optimization problem, since the flow distribution is decided not only from specifics of system design including the diameter, length and orientation of inlet pipes, expansions/contractions, but also from operating parameters such as exhaust flow rate and temperature. Computer-aided simulations have quickly established themselves as fast, cost-effective alternatives to physical testing in the design and optimization of aftertreatment systems. Of special importance are the multi-dimensional computational fluid dynamic (CFD) simulations that allow the relevant details of fluid/ thermal/ reactive processes of interest to be investigated with a pre-determined accuracy. Several examples of the use of CFD in after treatment system design optimization can be found in literature. Weltens *et al.* [2], formulated a flow distribution index to characterize the degree of uniformity of flow velocity and systematically investigated the effect of several design parameters on this index through CFD simulations. Karvouis and Assanis [3] demonstrated a methodology to analyze the effect of inlet flow distribution on catalytic conversion efficiency by using CFD predictions in conjunction with an analytical mass transfer limited conversion model. A similar approach was used by Benjamin *et al.* [4] to predict the geometry of a catalytic converter that can provide a uniform conversion efficiency across the monolith. However, in addition to improving the capabilities of analysis tools themselves, it is important to develop statistically and physically relevant

predictive measures to improve capabilities of, and confidence in, the CFD Analysis-Led-Design process. This work presents details of the development and validation of an index that is useful in characterizing the uniformity of flow in aftertreatment systems. The index was proposed based on physical and statistical considerations. For validation, experimental velocity data obtained under realistic flow and measurement location conditions for various geometries and flow rates were compared with CFD predictions.

MEASURES OF FLOW UNIFORMITY

A good measure of flow uniformity should be,

1. of physical and/or statistical significance
2. a directly representative measure of the important features of the flow field
3. sensitive to flow maldistribution to a sufficient degree
4. measurable in an appropriate experimental set-up
5. having finite upper and lower bounds

The most commonly used measure of flow uniformity in aftertreatment applications is the flow distribution index/flow uniformity index defined as [1,2,5],

$$\gamma_{L1} = 1 - \frac{1}{2} \left(\sum_{i=1}^n \frac{A_i}{A_{tot}} \frac{|v_i - v_{avg}|}{v_{avg}} \right) \quad (1)$$

where n is the number of cells on the face of interest (typically, the inlet face of the catalyst), i is the cell index, A is the face area of a cell, v is the axial component of velocity, 'tot' denotes total and 'avg' denotes area weighted average. This measure uses the Average Absolute Deviation (AAD), and is thus an $L1$ norm based metric. Division by the scaling factor 2 ensures that results are physically consistent. For instance, for a half-closed substrate with equal area cells and uniform velocity gas flow, the value will be 0.5 with the scaling factor, while it will be 0.0 without the scaling factor [2]. This formulation has the advantage that it is bound at the upper end with the maximum value of 1.0 (for a perfectly uniform flow). However, as a statistical measure of dispersion, AAD is less sensitive to extreme values in the distribution than the standard deviation [6]. Therefore, it is a common practice to use a secondary measure known as the velocity ratio (VR) to assess the deviation of maximum velocity from the average. It is defined as,

$$VR = \left(\frac{v_{max}}{v_{avg}} \right) \quad (2)$$

where the subscript 'max' denotes the maximum velocity at the face of interest. Zhang *et al.* [7] reports the use of an

average deviation index. Following the notations used in this work, it can be expressed as,

$$D = \left(\sum_{i=1}^n \frac{A_i}{A_{tot}} \frac{|v_i - v_{avg}|}{v_{avg}} \right) \quad (3)$$

It can be seen that this measure can be derived from Eq. (1). Its definition is similar to that of a coefficient of variation, but using AAD instead of the standard deviation. Therefore, lower values indicate better uniformity. The disadvantages are that it does not have an upper bound and it uses AAD making it less sensitive to extreme values of the distribution.

In the present work, a new $L2$ norm based index that uses the standard deviation is proposed as follows:

$$\gamma_{L2} = 1 - \frac{1}{2v_{avg}} \left(\sqrt{\sum_{i=1}^n \frac{A_i}{A_{tot}} (v_i - v_{avg})^2} \right) \quad (4)$$

The use of standard deviation instead of AAD is proposed due to the fact that the former is considered to be a statistically superior measure of dispersion. For instance, Fischer [8] demonstrated that standard deviation of a sample produces the smallest probable error when used as an estimate of the corresponding population parameter ("true value"). It was also shown that the additional error incurred with the use of AAD instead of standard deviation will be 14% for a Gaussian distribution [8]. Furthermore, Ellison *et al.* [6] warn that robust estimators such as AAD are not recommended when the data may follow non-Gaussian distributions including skewed and multi-modal shapes. Since shape of velocity distributions in aftertreatment systems can deviate appreciably from Gaussian, the use of an $L1$ norm based index for any general case may not be statistically justified.

Since the AAD of any distribution is always less than or equal to the standard deviation [9], it follows that $\gamma_{L2} \leq \gamma_{L1}$. To illustrate this, Fig. 1 shows two skewed distributions generated from 50,000 data points. For the Gamma distribution, values of $L1$ and $L2$ norm based indices were 0.73 and 0.65 respectively. For the more heavily-skewed Exponential distribution, values of $L1$ and $L2$ norm based indices were 0.63 and 0.50 respectively. Figure 2 compares the values of the two indices for seven different Gaussian distributions that had the same mean but different variances. Some of the generated data points for the distributions with high variances had negative values, thereby representing a reversed flow condition. These can lead to negative values of the flow distribution indices as seen in Fig. 2. Yet again, it is observed that $\gamma_{L2} \leq \gamma_{L1}$.

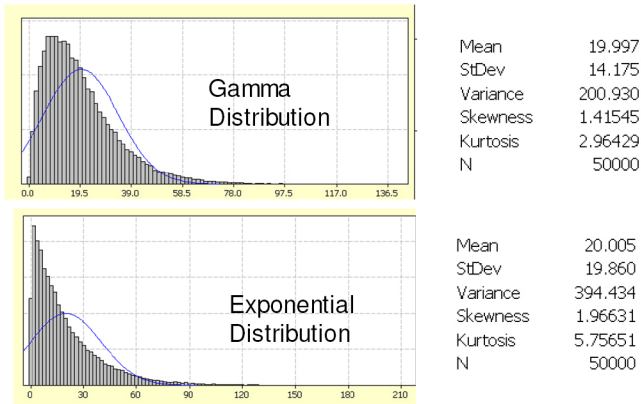


Fig. 1. Two statistical distributions for assessing uniformity measures

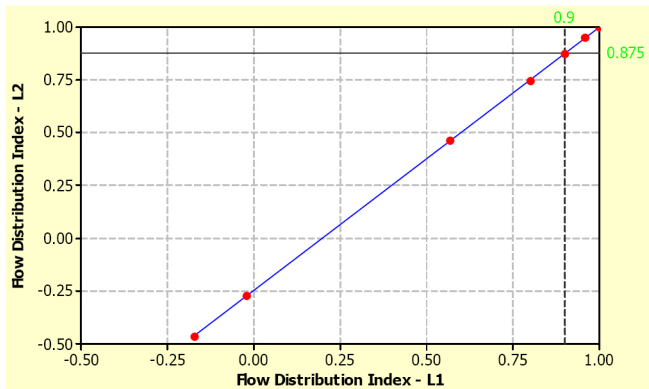


Fig. 2. Comparison between uniformity measures for various Gaussian distributions

Another interesting statistical feature of an $L2$ norm based flow uniformity measure is that a confidence level based estimate of the velocity ratio (Eq. (2)) can be obtained if the average velocity and γ_{L2} are known. For instance, for a Gaussian distribution, for 99% confidence interval,

$$v_{max} = v_{avg}(1 + 3\sigma) \quad (5)$$

where σ is the standard deviation. From Eqs. (2) and (4),

$$VR \sim [1 + 6(1 - \gamma_{L2})] \quad (6)$$

In general, for an arbitrary distribution, we can expect a linear relationship between the velocity ratio and γ_{L2} as,

$$VR = C_1 - C_2\gamma_{L2} \quad (7)$$

where C_1 and C_2 are constants determined from the expected confidence interval.

Hence, it can be seen that many of the desirable characteristics of γ_{L1} , including the bounded upper limit of 1.0 and the physically consistent scaling are retained in γ_{L2} . However, this measure has better statistical significance and is more sensitive to extreme values of the velocity distribution (i.e., low and high velocities).

MEASUREMENTS AND SIMULATIONS - SET-UP

A cold flow blower that can produce flow rates in the range of 0.033 SCMS to 0.944 SCMS (i.e., 70 SCFM - 2000 SCFM) was used to provide the required flow rates in the experiments. The blower was connected to the inlet section of an aftertreatment system (shown in Fig. 3) that can be fitted to a Cummins medium-duty engine. It was fitted with a flange to which various flow distribution devices can be attached. Velocity measurements were made at an axial distance of about 0.2 times the pipe diameter downstream of the flange, since it represents the location of the inlet face of a Diesel Oxidation Catalyst. The three flow distribution devices investigated in this study are shown in Fig. 4. These were a perforated plate with 38% open area, a D-section with 60% open area and a plus-section with 50% open area; these will henceforth be referred to as the Perf geometry, Dee geometry and Plus geometry respectively. For each geometry, velocity measurements were conducted at 0.12 SCMS, 0.24 SCMS and 0.36 SCMS (i.e., 250 SCFM, 500 SCFM and 750 SCFM), which will henceforth be referred to as the Low flow, Medium flow and High flow cases respectively. The lower limit of flow rate was chosen based on the ability of the blower to maintain the commanded flow rate steady and the least count of the pressure transducer. The upper limit was chosen based on the consideration that the velocity probe vibrations are not excessive. These flow rates, however, did cover the range of flow rates that the system is expected to experience.



Fig. 3. Test geometry. Location of flange and measurements are also shown.

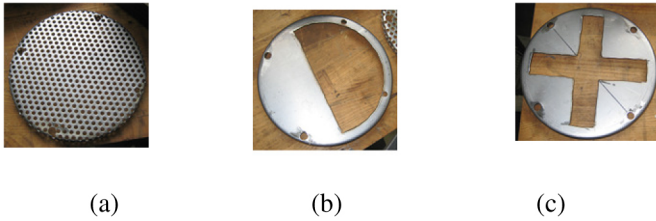


Fig. 4. Flow distribution devices (a) Perf (b) Dee (c) Plus

Velocity measurements were made using a 3 mm outer diameter Pitot tube connected to a 0-10 Torr DATAMETRICS Barocel 590 differential pressure transducer and a data acquisition system to log the data. Temperature data at the measurement point was collected and logged using a K-type thermocouple fastened to the back side of the Pitot tube, so as not to disturb the flow. ISO guidelines for flow rate measurements recommend that velocity measurements are made well downstream of a flow distribution devices (usually 5 times the duct diameter) after ensuring that flow is substantially parallel to the conduit axis [10]. In the present study, significant non-uniformity in flow was anticipated due to the presence of the flow distribution devices and the choice of realistic measurement location. It was also realized that the Pitot tube may not always be aligned with the flow direction. ISO guidelines available for measurements in ducts with symmetric flow and parallel streamlines [10] and for moderately swirling/asymmetric flow conditions [11] were consulted. To obtain high resolution measurements, the Pitot tube was mounted on and positioned by a Cartesian actuator system similar to that described by Benker *et al.* [12]. Due to a malfunction of the system controller output, position was measured via linear position sensors, and the precision of the probe positioning to a commanded location was within ± 1 mm. Velocity data was acquired at 50 points, such that measurement at each point represents the average velocity of flow through one of the 50 equal area sections. The point locations were determined by Log-Chebyshev method [10] and are shown in Fig. 5. The actuator was programmed such that at each location, the probe remained stationary for 5 seconds, collecting data at 10 Hz; thus the reported value at each location represents an average of 50 readings.

A Pitot tube uses the pressure differential between static (P_{stat}) and total pressures (P_{tot}) to estimate the flow velocity as,

$$v = \alpha \sqrt{2 \left(\frac{P_{tot} - P_{stat}}{\rho} \right)} \quad (8)$$

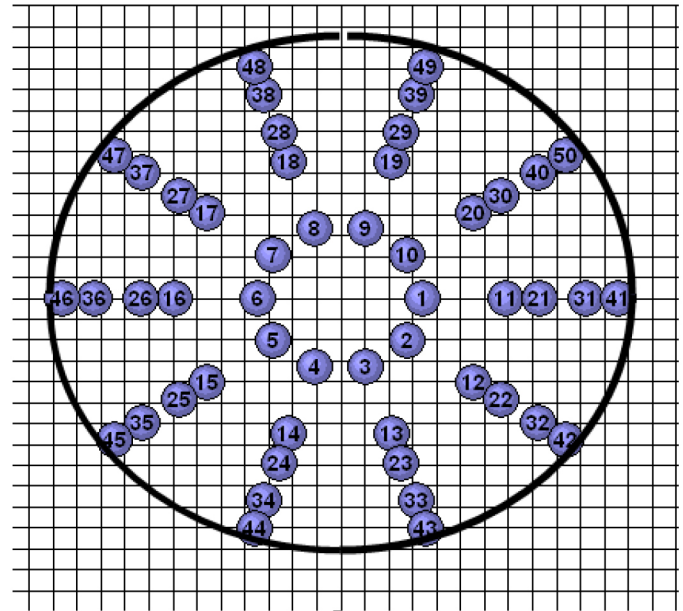


Fig. 5. Locations of velocity measurements

where α is the calibration factor of the Pitot tube and ρ is the density of fluid. For the present pitot tube, $\alpha = 1.0$. Density was computed by measuring the upstream fluid pressure (P_{abs}) using an absolute pressure transducer and the fluid temperature (T_{abs}) using a K-type thermocouple as,

$$\rho = \frac{P_{abs}}{RT_{abs}} \quad (9)$$

where R is the characteristic gas constant.

For simulations, the finite volume CFD code FLUENT 6.3.26 was used. FLUENT can solve unsteady, turbulent, compressible, reacting multi-phase flow problems using unstructured grids and the details of this code can be found elsewhere [13]. Geometries were created in Pro-Engineer and were meshed using the pre-processor GAMBIT 2.4.6. Hexahedral and tetrahedral cells were used and the total cell count was between 1 million and 3 million for the three geometries investigated. Steady gas flow analyses were conducted using the pressure-based solver. Ideal-gas equation was used to account for spatial variations in density and Sutherland's three coefficient method was used for computing viscosity. Realizable κ - ϵ model was used for turbulence computations. Due to prohibitive computational time and potential for numerical convergence issues, perforated flow devices are traditionally modeled as porous jumps in CFD simulations [c.f., Chen and Schirmer [14]]. A porous jump is a thin membrane that has known velocity - pressure drop characteristics and over which the change in pressure can occur due to viscous and inertial losses [13]. In the present study too, the Perf flow distribution device was modeled a