Combustion and Flame 188 (2018) 469–482

Contents lists available at ScienceDirect

Combustion and Flame

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

Modal decomposition of the unsteady flow field in compression-ignited combustion chambers

Antonio J. Torregrosa, Alberto Broatch, Jorge García-Tíscar*, Josep Gomez-Soriano

CMT–Motores Térmicos, Universitat Politècnica de València, Camino de Vera, Valencia 46022, Spain

ARTICLE INFO

Article history:
Received 6 April 2017
Revised 18 June 2017
Accepted 5 October 2017
Available online 5 November 2017

Keywords:
DMD
POD
NVH
CFD
ICE
Noise

ABSTRACT

In this paper, the unsteady behaviour of a compression-ignited (CI) engine combustion chamber is studied by analysing the results of a Computational Fluid Dynamics (CFD) model through the application of different flow decomposition techniques, aiming to resolve the underlying modal structure of the process. Experimental validation for the combustion simulation is provided, and a methodology for extracting coherent pressure information is proposed in order to provide a suitable input for different analysis methods. These range from straightforward Fourier transform techniques to more sophisticated modal decomposition approaches. In particular Proper Orthogonal Decomposition (POD) is shown to provide valuable insight into the time-spatial structure of the combustion flow field, allowing the establishment of correlations between pressure modes and physical parameters of the combustion, such as the injection timing or the chamber geometry. Dynamic Mode Decomposition (DMD) on the other hand is proven to successfully highlight the link between the frequency of the unsteady energy components and their spatial distribution within the chamber. Advantage is then taken of the modal characterization of the unsteady behaviour in the chamber to showcase how physical parameters such as the spray angle can be modified to optimize the acoustic signature of the combustion process, helping CI internal combustion engines reduce their acoustic environmental impact.

1. Introduction

Having served for more than a century as the primary means of powering transportation in industrialised societies, the reciprocating internal combustion engine is nowadays facing ever-increasing competition in the automotive market from another old [1] competitor, the electric motor.

While the main reason leading to regulatory pressure on the combustion engine is that of harmful combustion emissions, different solutions can be found to counter this issue, such as the use of catalytic converters, particle traps, advanced fuels and intelligent combustion strategies.

As oil, gas and coal-fired power plants are still notably featured in the generation mix of several countries, it is conceivable that the effective environmental footprint of electric vehicles could be matched by their thermal counterparts, at least until cleaner energy sources become widespread.

There is another aspect however in which electric cars enjoy a clear advantage over combustion-powered ones, that of NVH (Noise, Vibration and Harshness). Combustion implies, by definition, a sudden energy release which is in turn inherently noisy: energy is not only transferred to the piston; it also causes an unsteady flow field that resonates in the chamber and propagates through the engine block into the cabin and the environment [2].

Although the combustion chamber is not the sole source of noise in passenger cars, since other sources such as turbochargers, pumps, wheels or aerodynamics are also relevant, it is certainly the most relevant one at low speeds and, thus, in urban environment where some authorities are already considering a permanent ban on thermal vehicles.

Therefore, it is essential to not only optimize combustion chambers of passenger cars to reduce the formation of pollutants, but to also consider carefully how the reduction of acoustic emissions may be achieved in the design process [3].

One of the most useful tools in the pursuit of optimized designs is Computer Fluid Dynamics (CFD), as a huge number of parameters can be tested without the need of costly and time-consuming experimental testing [4]. Once that a CFD solution has been validated, multiple changes can be simultaneously applied to the setup and solved in parallel, following strategies such as design of experiments [5], genetic algorithms [6,7] and machine learning that autonomously refine a solution until an optimum is found.

* Corresponding author.
E-mail address: jorgarti@mot.upv.es (J. García-Tíscar).

https://doi.org/10.1016/j.combustflame.2017.10.007

However, to apply these algorithms and accomplish better designs of combustion chambers that not only address chemical pollutants but also the issue of combustion noise [8,9], it is necessary to perform a detailed analysis of the CFD results, that allows the characterization of the unsteady behaviour of flow field in the chamber [10,11].

In this paper, we apply different postprocessing techniques to a CFD model of a CI combustion chamber whose overall results in terms of pressure are also experimentally validated. We show how these techniques can reveal a better understanding of the combustion unsteady behaviour, reducing the need for guesswork and manual CFD postprocessing, alleviating the workload of design engineers and offering actionable information that can translate into quieter and more competitive combustion engines.

2. Background

In order to provide context for the problem being addressed in this work, a brief introduction of the state of the art in combustion noise issues produced by CI engines is provided in this section, along with the formulation of different analysis techniques of interest to better understand this problem.

2.1. Resonant combustion noise of CI engines

Already since the work of Draper in 1938 [12], knock effects of SI combustion engines were investigated by comparison with the ideal vibration modes of a cylinder. One of the first investigations into noise production by CI engines was presented by Priede [13] who noted that *"the rapid pressure rise caused by the sudden combustion of an appreciable fraction of the fuel-air mixture initiated also pronounced gas oscillations in the combustion chamber, and that these oscillations produce a broad peak on the cylinder-pressure spectrum and thus enhance the emitted noise in that frequency range"*.

He also indicated that *"the frequency of gas oscillations is determined by the geometric shape of the combustion chamber"*, a concept that would be linked to resonance modes in a follow-up paper [14], in which he mentions the simple side-to-side mode already detected by Draper when sudden knocking events occurred in SI engines. However, neither had the means to resolve the actual, 3D gas oscillation modes caused by the combustion and the chamber geometry.

Targeting open chambers and assuming several simplifications, Hickling et al. [15] tried to improve upon purely theoretical formulas using a Finite Element Model (FEM). Still, his 16-element, quiescent gas FEM model lacked the capability to resolve actual flow or combustion, or even small-scale gas oscillations and thus, only a slight variation from the theoretical modal coefficients was obtained.

The potential of CFD for analysing combustion chamber resonance modes was first demonstrated by Torregrosa et al. [8,10]. However, although wave motion across the chamber was successfully resolved and compared with theoretical cylinder modes, no realistic initial flow field or combustion model were used, introducing instead small initial regions of high temperature and pressure simulating the ignition points.

Both shortcomings are overcome in the present investigation where three whole engine cycles are simulated, including the scavenging process, piston movement and valve movement profile. A realistic flow field including the swirling air motion is thus obtained in which the injection and combustion processes are modelled, accurately reproducing the pressure oscillations that are present in the real engine and are responsible for resonant combustion noise.

This numerical pressure data of the whole system, once validated, can then be analysed through different techniques to reveal the real combustion modes directly, without the need of relaying on comparison or correlation with idealized analytic vibration modes of simple cylinders or cavities.

2.2. Numerical identification of resonant modes

A way to explore the spatial distribution of the acoustic flow field for different frequency-related phenomena of interest is to perform the Fourier transform at each cell record in the considered domain. Then, the amplitude of the transformed signal at the frequency of interest is used to colour each cell, be it directly or through a smoothing interpolation.

This technique can be exploited in an attempt to identify the aforementioned actual modes of the combustion unsteady field. The spatial representation of selected Fourier traces that are manually selected based on inspection of the spectra can give a hint at the influence of the chamber geometry on the pressure oscillation patterns.

However, while this Fourier analysis method allows the obtaining of interesting results in a simple, straightforward way which moreover can be found already implemented in some commercial CFD codes, it is not without disadvantages.

Among them is the need to manually select the frequencies of interest, thereby having the risk of missing relevant frequencies, which are not known beforehand, and representing an increase of the workload of the design engineers. Time evolution of these spatial features is also lost unless Short-time fast Fourier transform (STFFT) is attempted, at the cost of losing frequency resolution.

Also, in more complex geometries or flow structures such as those of turbulent jet ignition chambers, it could be difficult to exactly pinpoint relevant features, thereby requiring specific and intricate visualisation methods to reconstruct these modal structures.

2.3. Proper Orthogonal Decomposition

In order to address these limitations, more sophisticated analysis can be carried out to obtain the modal decomposition of the unsteady flow field. One of the most used among these is probably the Proper Orthogonal Decomposition (POD), also called Principal Component analysis (PCA) or Karhunen–Loève expansion [16], which originated in the field of probability theory and was first applied to the analysis of turbulent flow in 1967 [17]. As (rather poetically) put by Aubry in the aptly named paper *"On the Hidden Beauty of the Proper Orthogonal Decomposition"* [18], the objective of this technique is that:

> *"The flow is decomposed into both spatial and temporal orthogonal modes which are coupled: each space component is associated with a time component partner. The latter is the time evolution of the former and the former is the spatial configuration of the latter."*

In this way, this decomposition allows the identification of which spatial structures comprise the most energy of the flow field, which is understood as the superposition of all modes. The ordering of the contribution of each mode however, allows in principle a simplified yet meaningful reconstruction of the flow field.

Generally, information on the flow field evolution coming from either numerical simulations or experimental measurements[1] will be naturally presented in a sequence of N vectors $\mathbf{v}_i$ representing temporal snapshots that can be gathered in a matrix $\mathbf{V}$:

$$\mathbf{V}_1^N = \{\mathbf{v}_1, \mathbf{v}_2, \dots, \mathbf{v}_N\} \tag{1}$$

Each snapshot contains a quantity M of scalar flow magnitudes which in the case of experimental results usually represent the

[1] For instance, from experimental Particle Image Velocimetry (PIV) measurements that capture a sequence of velocity field snapshots.