Computers & Fluids 101 (2014) 183–193

Contents lists available at ScienceDirect

Computers & Fluids

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

A parallel multidomain strategy to compute turbulent flows in fan-stirred closed vessels

A. Bonhomme [a,*], F. Duchaine [c], G. Wang [d,1], L. Selle [a,b], T. Poinsot [a,b]

[a] Université de Toulouse, INPT, UPS, IMFT (Institut de Mécanique des Fluides de Toulouse), Allée Camille Soula, F-31400 Toulouse, France
[b] CNRS, IMFT, F-31400 Toulouse, France
[c] CERFACS, GlobC Team, 42 Avenue Gaspard Coriolis, 31057 Toulouse Cedex 01, France
[d] CERFACS, CFD Team, 42 Avenue Gaspard Coriolis, 31057 Toulouse Cedex 01, France

ARTICLE INFO

Article history:
Received 30 December 2013
Received in revised form 15 May 2014
Accepted 6 June 2014
Available online 20 June 2014

Keywords:
Code coupling
Fan-stirred vessel
Homogeneous isotropic turbulence
Turbulent flame speed

ABSTRACT

This paper presents a parallel multidomain strategy to compute the turbulent flow in a closed vessel stirred by six fans. The method is based on running multiple instances of the same solver, working on different subdomains and communicating through small overlapping zones where interpolations allow to handle moving meshes. First the accuracy of this Multi Instances Solver Coupled on Overlapping Grids (MISCOG) approach is evaluated for the convection of a single vortex. Load balancing issues on parallel machines are discussed and a performance model is proposed to allocate cores to each code instance. Then, the method is applied to the LES of a closed vessel stirred by six fans. Mean and fluctuating fields obtained by the LES are compared to experimental data. Finally, the structure of the turbulence generated at the center of the vessel is studied and the mechanisms allowing turbulence to travel from the fans to the center of the vessel are analyzed.

1. Introduction

Turbulence has been studied for decades in its most canonical form: homogeneous isotropic turbulence (HIT) [1–7]. This limit case is the cornerstone of multiple theoretical approaches as well as the building brick of Large Eddy Simulation (LES) models where the Kolmogorov cascade assumption allows to model the effects of small scales from information available for the resolved ones [8,9]. HIT is also the only generic case where the interaction of other phenomena with turbulence can be defined using a limited number of parameters: evolution of large droplets in HIT [10–12], interaction of evaporating droplets with HIT [13], flame/turbulence interaction [14–16].

While defining HIT theoretically or numerically is a reasonably simple and clear task, creating HIT experimentally is more challenging. This paper focuses on one classical technique used to generate HIT: fan-stirred closed vessels. Sometimes these apparatus are called ‘bombs’, a denomination that will be used in this paper. Stirring vessels with fans to study turbulent flame propagation has been used for more than a century (see Laffitte’s book [17]). A classical paper where this turbulence was qualified as HIT is due to Semenov [18] who showed that properly designed bombs with multiple fans were able to generate reasonable HIT in a zone located near the center of the chamber where the mean flow is almost zero and turbulence is homogeneous and isotropic. A significant amount of work has been based on correlations obtained in such bombs. The most famous example is probably the quest for ‘turbulent flame speed’ correlations in which the speed s_T of premixed turbulent flames is expressed as a function of the initial turbulent velocity u'. Such correlations continue to be frequently published [19–23] and interestingly, few of them agree. One reason for this may be that the notion of a generic turbulent flame speed depending only on a limited number of flow and flame parameters may not be relevant [14]. Another one could be that the initial turbulence in such bombs is not really close to HIT and that more parameters should be taken into account. Therefore, since most models are based on measurements performed in bombs, an interesting question is to study whether the flow created in a fan-stirred bomb really mimics HIT and over which spatial extent. This question has been investigated experimentally [18,24,25] but using CFD would be a useful addition.

Even though the largest CFD simulations to date have been published for HIT with meshes up to 64 billion points [26], all of them were performed in simple cubic meshes, initialized with a flow which has all the properties of theoretical HIT. None of these

* Corresponding author. Tel.: +33 0534322893.
E-mail addresses: adrien.bonhomme@imft.fr (A. Bonhomme), laurent.selle@imft.fr (L. Selle), thierry.poinsot@imft.fr (T. Poinsot).
[1] Now at Sherbrooke University.

http://dx.doi.org/10.1016/j.compfluid.2014.06.010
0045-7930/

simulations address the question of how HIT is created (if it is) in a real fan-stirred bomb. This question is much more complicated and existing experimental diagnostics are not always sufficient to guaranty that the flow in this situation matches all properties of theoretical HIT: in a bomb, fans obviously induce a strong mean, pulsated flow. In the center of the vessel, the mean flow is expected to be zero and turbulence assumed to diffuse to a central zone where HIT is expected. This involves a series of questions which are rarely addressed:

- By which mechanisms does turbulence transfer from the fan region to the central zone?
- Since the number of fans is usually limited, are there preferential straining axes in the bomb which could affect isotropy near its center?
- The fans flow being by nature unsteady, is turbulence at the center of the apparatus sensitive to the pulsating nature of the flow created by the blades rotation?
- How large is the zone where HIT is obtained?

The objective of this paper is to show how the turbulent flow in a fan-stirred vessel can be studied using high-resolution LES to complement experimental diagnostics. To reach this objective, the simulation code must satisfy three criteria:

- Considering the complexity of the objects to mesh, the need to correctly capture the blade geometry and the necessity to handle moving objects, unstructured meshes are required so that classical DNS codes used for HIT (spectral methods [27,28], high-order compact schemes [29–31]) cannot be used.
- The configuration includes a large number of moving objects (the fans) close to each other. Classical techniques such as ALE (Arbitrary Lagrangian Eulerian) [32–34] are difficult to implement for a flow with multiple fans because of meshing issues. Immersed Boundary methods [35–37] are easier to develop for moving objects but are usually associated to a low order of accuracy which is not acceptable in a LES framework. Here, a new multidomain high-order LES technique with mesh overlapping developed by Wang et al. [38,39] is used on a real configuration.
- To resolve turbulent structures accurately, a high-fidelity explicit (in time) LES solver is needed and the corresponding CPU cost is expected to be large so that the implementation of the multidomain method must be fully parallel.

This paper is organized as follows: first, the numerical methodology is described in Section 2. It is based on the simultaneous execution of multiple instances of the same solver, called MISCOG for Multi Instances Solver Coupled on Overlapping Grids. These instances are coupled on parallel computers using the OpenPalm coupler [40,41]. This coupler is well suited for this task, however, one limitation is that only two instances can exchange at the same time so that the balancing strategy becomes much more complex than it was for a single instance, which is also discussed in Section 2.

A validation test case of the MISCOG approach is presented in Section 3. It consists in propagating a single vortex across two overlapping computational domains. It is thought as an elementary validation of the ability to convect turbulent structures. The method is then applied to a fan-stirred bomb experiment developed in Orléans [42], where 7 instances are required to compute the bomb and the six fans. Section 4 describes this configuration, the numerical set-up and the parallel efficiency of the global simulation.

Flow results are discussed in Section 5: quantities that can be obtained both from LES and PIV are first compared (mean flow fields and RMS values for all three velocity components). LES results are used to analyze quantities which cannot be obtained experimentally such as the velocity tensor – to identify the structure of the turbulence – or the budget of turbulent kinetic energy in order to understand how turbulence reaches the center of the vessel.

2. Numerical methodology

The filtered LES unsteady compressible Navier–Stokes equations that describe the spatially filtered mass, momentum and energy conservation are solved by the unstructured compressible LES solver, AVBP [43]. These equations can be written in the conservative form:

$$\frac{\partial \mathbf{W}}{\partial t} + \vec{\nabla} \cdot \vec{\mathbf{F}} = 0 \tag{1}$$

where $\mathbf{W}$ is the vector containing the conservative variables $(\rho, \rho U, \rho E)^T$ and $\vec{\mathbf{F}} = (\mathbf{F}, \mathbf{G}, \mathbf{H})^T$ is the flux tensor. The flux is divided into two components: the convective flux depending only on $\mathbf{W}$ and the viscous flux depending on both $\mathbf{W}$ and its gradient $\nabla \mathbf{W}$. The contributions of Sub-Grid Scale (SGS) turbulence models are included in the viscous flux through the addition of the so called turbulent viscosity ν_t. Two schemes are used in this study: Lax–Wendroff [44] (LW, with 2nd-order accuracy in time and space) and the two-step Taylor–Galerkin finite element scheme TTGC [45] (3rd-order in time and space). The LW scheme, which is faster than TTGC is used for transient phases while all statistics are gathered (when steady state is reached) using the TTGC scheme.

To compute the whole configuration and the flow created by the fans the code must be able to deal with moving parts (in this case, six rotating fans). Immersed Boundaries Methods [35,36] were tested but were not able to represent correctly the blade geometry of the fan because the entire zone spanned by the fans must be meshed with a very fine grid size leading to a prohibitive cost in term of CPU time. ALE methods with mesh deformation [46,47,34] were also considered but introduced excessive deformation of cells and frequent interpolation phases [48].

To solve this problem, the MISCOG approach, developed initially for turbomachinery [38,39], was extended to bomb configurations. In MISCOG, two or more instances of the same LES solver (namely AVBP), each with their own computational domain, are coupled through the parallel coupler OpenPALM [40,41]. For the bomb case, the whole flow domain is initially divided into 7 parts: the bomb itself has a static mesh (AVBP01) while each fan is computing in a moving framework (AVBP0i, $i \in [2; 7]$). For moving parts, the code uses the ALE block rotation approach [46,47,34]: the grid is rotated without deformation. The remaining unit AVBP01 simulates the flow in the static part of the bomb in the same coordinate system. The solution retained to handle interfaces between the units involving rotating and non-rotating parts consists in reconstructing the residuals using an overset grid method and exchanging by interpolation the multidomain conservative variables wherever needed. To do so an efficient distributed search algorithm is implemented in the OpenPALM coupler to locate the points in parallel partitioned mesh blocks and a linear method is used to interpolate residuals (the interpolation is of 2nd order). This coupling phase is implemented outside the CFD instances in conjunction with second order interpolation.

The computational domain corresponding to the experiment of Orléans is displayed in Fig. 1: six cylindrical rotating domains ($i = 2$–7) are used for each fan zone while one domain ($i = 1$) is used for the rest of the bomb. In general, the number of cells used for each domain can be different. Here the grids for the six fans (AVBP02–AVBP07) have the same number of cells but the bomb