# STABILIZATION OF A TURBULENT PREMIXED FLAME

**Laurent Vervisch-Guichard**
CRIHAN
32 rue Raymon Aron
76130 Mont Saint Aignan, France

**Julien Réveillon**[*] **and Raphaël Hauguel**
LMFN / CORIA UMR CNRS 6614
University of Rouen INSA, Avenue de l'université
76800 Saint Etienne du Rouvray, France

## ABSTRACT

This paper proposes a numerical procedure to simulate a spatially decaying turbulence with constant time averaged properties.

A forced isotropic turbulence with statistically constant properties is generated thanks to a direct numerical simulation (DNS) in spectral space. Then, parts of this spectral DNS are used as boundary conditions for a finite difference DNS solver. The subsonic turbulent flow is injected at a very low convective mach number although fully compressible Navier-Stokes equations have been used. To evaluate the procedure, a planar turbulent premixed flame has been stabilized in a grid turbulence.

## INTRODUCTION

Premixed turbulent combustion is a common procedure in industry operating systems or in laboratory experiments. The understanding and control of interactions between the turbulence and the flame are important issues when optimizing combustion processes to improve the economical and ecological output of the device. Accurate direct numerical simulations of such devices have to reproduce the whole characteristics of the flow. However, numerical difficulties arise when simulating a turbulent injection with Navier-Stokes equations. Therefore, basic studies of turbulent premixed combustion are mostly based on decaying homogeneous turbulence simulations (Poinsot *et al* 1996) or are using a low mach number formulation (Louch 1998) which does not allow for the pressure effects.

This paper presents a numerical procedure to simulate a spatially decaying turbulence with time averaged properties that remain constant. The subsonic turbulent flow may be injected at a very low convective mach number although it is described by a fully compressible solver. Following this procedure, a planar premixed turbulent flame has been stabilized with a direct numerical simulation. Therefore, it is possible to study on a large period of time the physic of turbulent combustion (flame velocity, stretching, quenching, ...) with statistically constant turbulent parameters (kinetic energy, dissipation, integral scale, ...).

The present paper proposes a summarized description of the injection procedure (Vervisch-Guichard 1999). Then, its ability to stabilize a flame in a grid turbulence is showed.

## NUMERICAL PROCEDURE

The simulation of a spatially decaying turbulence requires the generation of fluctuations which generate and maintain this turbulence. The adopted approach to the problem consists to prescribe an established turbulent flow on the inlet of the simulation domain. Three major stumbling blocks must be solved.

1. Turbulent fluctuations must be generated and maintained following some prescribed parameters (spectrum, integral length scale, ...).
2. Appropriate boundary conditions have to be developed for the subsonic inlet.
3. Coupling between the fluctuations and the inlet conditions must be carried out.

It is obvious that the spatial and temporal variations of the fluctuations which are imposed at the inlet are at the origin of the main properties of the decaying turbulence. However, they have, in the same way, an essential role in the production of acoustic phenomena. Indeed, turbulence properties and acoustic phenomena

---

[*] Corresponding author - reveillon@coria.fr

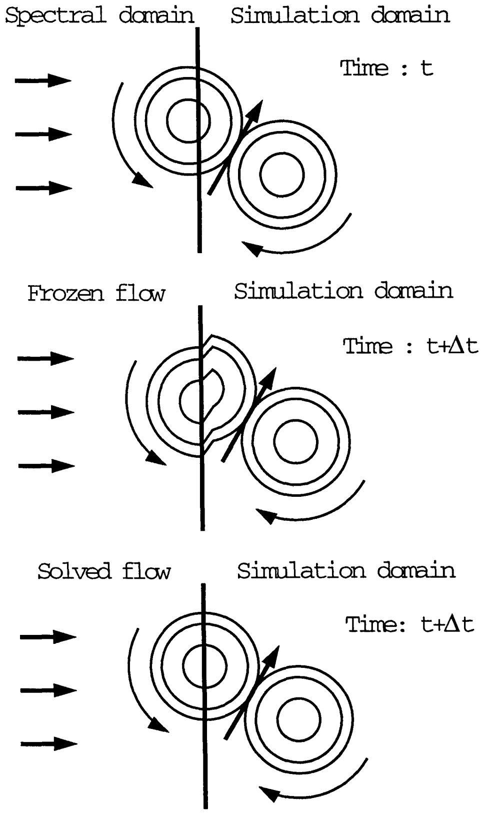

FIG. 1 – **Sketch of the main property of the method: vortex continuity. top: flow at a given time *t*, center: consequence of a frozen flow injection, bottom: the proposed method avoid the vortex shearing**

that are likely to be developed depend on the velocity gradients near the entry. These gradients result from the interaction between the flow in the domain and the fluctuations which are injected during the simulation. Thus, spatial and temporal variations of these fluctuations cannot follow unspecified evolutions and must agree as well as possible with the movement of the turbulent structures which are simulated. Therefore, the key point of the method is to ensure the continuity equation at the inlet boundary.

The main property of the numerical procedure suggested in this paper is an optimal compatibility of the fluctuations imposed at the inlet with the simulated turbulent flow (fig. 1). The solution which was adopted consists in simulating a 'ghost' turbulence whose characteristics (spectrum, velocity root mean square, ...) correspond to those required at the inlet of the computational domain. The 'ghost' field is generated by a spectral code which is used to simulate a forced isotropic homogeneous turbulence. One of the plans of this spectral turbulence, is then interpolated to provide the fluctuations (fig. 2) which are used as boundary conditions to simulate a spatially decaying compressible turbulence.

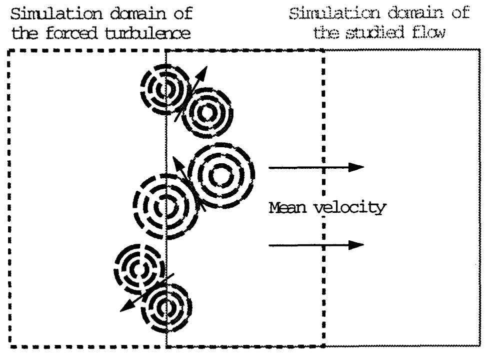

FIG. 2 – **Sketch of the coupling between spectral solver ('ghost' domain) and finite difference solver (computational domain).**

## Turbulence forcing

There are various ways in which the forcing of an isotropic homogeneous turbulence can be achieved in a spectral DNS. First, it is possible to froze the magnitude of the largest structures of the velocity field. But stabilization of the parameters needs many eddy turn over times and the results are statistically dependent of the anisotropic large structures. Another solution is to use stochastic schemes (Eswaran and Pope, 1988) Energy is added randomly in the low wave number range. These schemes appear to be efficient and statistically independent. However, fluctuations of some turbulence parameters around their prescribed values are important. Moreover, it adds randomness to the flow and it may influence the result of the studies based on the already stochastic nature of the turbulence. A third way is to use a deterministic forcing scheme which forces the lower-wave number modes with a controlled amount of energy. The most effective work in this domain has been carried out by Overholt and Pope (1998). Their deterministic scheme forces the simulated spectrum toward a model spectrum by using a time-dependent and wave number-dependent coefficient. It is a robust and efficient method and it doesn't introduce artificial stochasticity into the computation which reaches quickly stationarity. Therefore, a method close to the Overholt and Pope (1998) forcing scheme has been used to control the turbulence parameters in the spectral space.

## Multidimensional characteristic boundary conditions

The evolution of the flow at a given time is solved by the equations of Navier-Stokes. They express the local behavior of the flow according to the surrounding disturbances. However, at the boundaries of the computational domain, part of information is missing and must be replaced by boundary conditions. They must represent the disturbances that the external flow imposes at the boundaries of the domain. According to the propagation of the disturbance, it is possible to determine the number of boundary conditions and the way of imposing them on the simulation. Many works have been dedicated to the treatment of the boundary conditions. The procedure to inject turbulent flows is based on the Navier-Stokes characteristic boundary conditions (NSCBC) described by Poinsot and Lele (1992).

The Eulerian terms of the transport equations are represented in the forms of characteristic waves. The behavior of the waves being propagated towards the outside of the field is defined by the information contained in the field. No condition is to be specified for these waves and the boundary conditions should not interfere with their propagation. On the other hand, the waves being propagated inside the field require to be prescribed. Moreover, they have to be compatible with the boundary conditions imposed on the computational domain. Poinsot and Lele suggested to determine the time variation of the waves amplitude (TVWA) thanks to the monodimensional equations of Euler. This approximation is usually adequate as long as the transverse terms are negligible. Thus, NSCBC must not be used for the injection of a turbulent flow which is multidimensional. Therefore the NSCBC method has been extended from a monodimensional to a multidimensional description and may be applied following four stages.

1. Correction of the diffusion and dissipation terms
2. Computation of the TVWA of the characteristic wave propagating outside the computational domain (fig. 3).
3. Evaluation of the TVWA of the waves being propagated towards the interior of the computational domain (fig. 3).
4. Computation of the amplitude variation rate of the conservative variables thanks to a complete set of equations rebuild with the whole of the preceding corrections.

## Spectral and physical spaces coupling

First, it is necessary to make clear that the injection method described in this paper does not depend on the numerical methods (time and space integration) used to solve the flow evolution in both spectral and physical spaces. Therefore, no detail is given on the codes specificity's. The coupling is done only through the evaluation of the boundary conditions and the treatment of the entering acoustic waves.

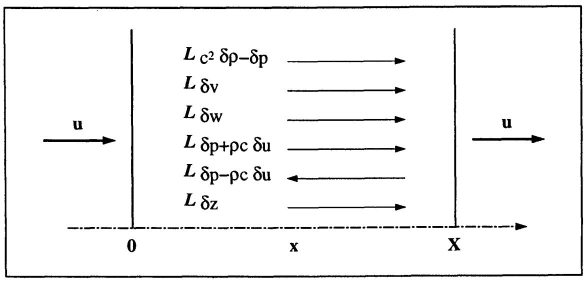

FIG. 3 – **Directions of propagation ($u < c$) of the time variation of the waves amplitude ($\mathcal{L}_{(c^2\delta\rho-\delta p)}, \mathcal{L}_{\delta v}, \mathcal{L}_{\delta w}, \mathcal{L}_{(\delta p+\rho c\delta u)}, \mathcal{L}_{(\delta p-\rho c\delta u)}, \mathcal{L}_{\delta z_\beta}$) associated to the variables ($u, v, w, p, \rho, z$) (resp: velocity components, pressure, density, inert scalar). $c$ is the sound velocity. All the waves except one are propagating inside the domain and have to be estimated by the multidimensional boundary conditions**

The turbulence at the inlet of the studied flow is maintained by a distinct simulation of a turbulent flow. Thus, two computations are processed simultaneously thanks to a spectral code (SP) and a finite difference code (FD). The SP code is totally independent and generates anisotropic homogeneous turbulence, properties of which are controlled and statistically stationary. The main solver FD simulates the evolution of the main flow. It depends on the upstream conditions generated by the SP code. The mean injection velocity which can vary during the computation is the same in both SP and FD solvers. Time advancement of the two codes is done with the same time steps and sub time steps according to the numerical method used. For each iteration, the time advancement of the SP code is first carried out because of its independence. This preliminary stage establishes all necessary information to define the boundary conditions of the FD code. Various parameters computed by the spectral code (velocity, gradients of velocity, etc.) are interpolated towards the positions which correspond to the inlet plan of the FD simulation. This information is memorized before being used for the calculation of the time advancement of the FD code.

## Inflow boundary conditions.

In this section we shall describe the spatial and temporal conditions imposed to estimate the characteristic waves during the injection of the turbulent flow.

Multidimensional NSCBC boundary conditions are used. On the whole of the inlet plan (considered on the left of the computational domain, main flow propagating from left to right), celerities $(u+c)$ and $(u-c)$ are respectively positive and negative. Thus, TVWA $\mathcal{L}_{(\delta p+\rho c\delta u)}$, defined figure 3 must be estimated using a boundary condition whereas $\mathcal{L}_{(\delta p-\rho c\delta u)}$ is calculated in the FD domain. The main boundary condition relates to the evolution of the normal component of the velocity : $u$ (or streamwise component). $u$ is imposed according to the value $u_s$ given by the spectral resolution.

$$\frac{\partial u}{\partial t}=\frac{u_s(t+\delta t)-u(t)}{\delta t}$$

With this boundary condition, the estimation of the amplitude variation rate $\mathcal{L}_{(\delta p+\rho c\delta u)}$ results from the characteristic wave propagation analysis:

$$\begin{aligned}\mathcal{L}_{(\delta p+\rho c\delta u)} &= \mathcal{L}_{(\delta p-\rho c\delta u)}+2\,c\,(s_{\rho u}-u\,s_\rho)\\ &- 2\,\rho c\,\frac{u_s(t+\delta t)-u(t)}{\delta t}\end{aligned}$$

where $s_{\rho u}$ and $s_\rho$ are known. $s_{\rho u}$ and $s_\rho$ are the spanwise and the non-Eulerian terms of the Navier-Stokes equations written at the boundary. However, this estimation, which depends on the TVWA $\mathcal{L}_{(\delta p-\rho c\delta u)}$ determined in the domain of simulation, reflects the numerical waves. This problem can be solved by adopting an estimation which is totally independent of any characteristic waves resulting from the field of simulation. To this end, the two following conditions are proposed:

$$\left\{\begin{aligned}\frac{\partial u}{\partial t} &= \frac{u_s(t+\delta t)-u(t)}{\delta t}\\ \frac{\partial u}{\partial x} &= -\frac{\partial v}{\partial y}-\frac{\partial w}{\partial z}\end{aligned}\right.$$

Then, thanks to the Navier-Stokes equations decomposed into characteristic waves, $\mathcal{L}_{(\delta p+\rho c\delta u)}$ may be written:

$$\begin{aligned}\mathcal{L}_{(\delta p+\rho c\delta u)} &= (u+c)\;[s_{\rho u}-u\,s_\rho\\ &- \rho\,\frac{u_s(t+\delta t)-u(t)}{\delta t}\\ &+ \rho\,(u-c)\,\left(\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)]\end{aligned}$$

The determination of the other amplitude variation rates $(\mathcal{L}_{(c^2\delta\rho-\delta p)}, \mathcal{L}_{\delta v}, \mathcal{L}_{\delta w}, \mathcal{L}_{\delta z_\beta})$ (fig. 3) depends on the sign of the normal component $(u)$ which is mainly positive but which can locally take negative values for highly turbulent flows. In the areas of the inlet plan where the momentum is negative, these amplitude variations of the characteristic waves are calculated in the field of simulation otherwise they are deduced from the spectral field.

## STABILIZED TURBULENT FLAME

One of the major application of the numerical simulation of turbulent flows is the study of turbulent combustion. Indeed, DNS allows for an accurate understanding of the complex physical mechanisms present in the flames and turbulence interactions. Therefore, models can be developed and preliminary tested before being validated with more complex flows. Until now, for a fully compressible formulation of the transport equations, DNS of turbulent premixed flames were carried out in configurations were the energy of turbulence decreases temporally. These nonstationary configurations produce analysis based on small statistical samples, results of which are influenced by the initial conditions. The injection method described in this paper allows us to avoid these stumbling blocks while preserving a fully compressible description of the flow.

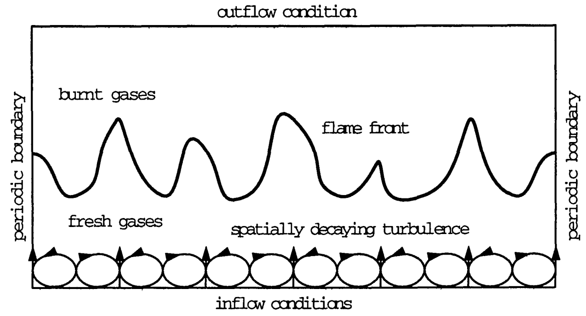

FIG. 4 – **Sketch of the computational configuration**

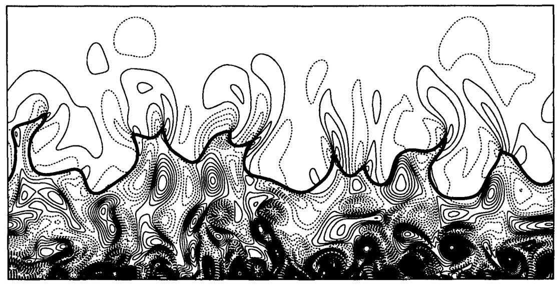

FIG. 5 – **Result of the simulation, isovorticity and temperature**

A two-dimensional configuration has been used (fig. 4) to evaluate the ability of the method to simulate the propagation of a stabilized turbulent premixed flame in a spatially decaying turbulence (fig. 5). There are two major difficulties embedded in this type of simulation. On the one hand, the dimension of the computational domain must be significant to contain the whole flame front which is strongly wrinkled by turbulence. On the other hand, because of the constant evolution of the instantaneous flame surface, the mean propagation velocity of the premixed flame is variable during the simulation. This is particularly confirmed before the flame reaches a statistically stationary state (fig. 6). Indeed, the initial condition is built starting from a stationary plane flame embedded in a turbulent flow whose fluctuations around the laminar flame were damped while respecting a divergence free assumption. During the computation, the turbulence fluctuations disturb the flame front whose surface increases along with the turbulent flame velocity. To avoid propagation of the flame towards the inlet of the computational domain, the average position (along the spanwise direction) of the flame is stabilized. An effective way to modify the velocity field without disturbing the physical phenomena is to perform a Galilean transformation. Therefore, a velocity field ($u$) which is not adapted to the propagation velocity of the flame becomes ($u + \Delta u$). The kinetic energy of the flow is corrected as well. Many criteria are possible to determine the correction velocity. In this work, it was decided that the total amount of fuel in the domain must keep a constant value.

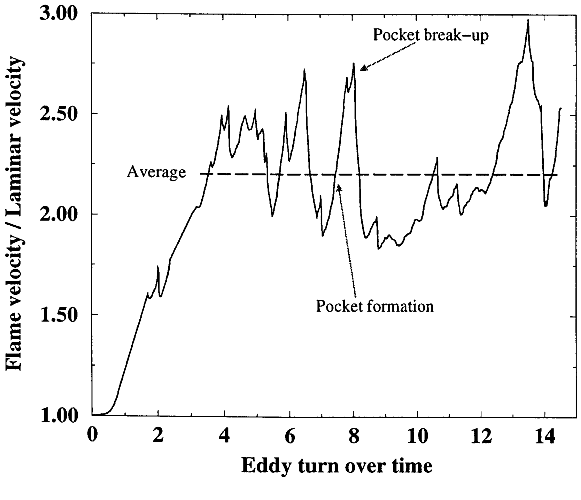

FIG. 6 – **Sketch of the computational configuration**

The whole details concerning the chemical parameters of the computation are not specified. Combustion has been solved following a single step reaction solver using a classical Arrhenius law. Flamelet regime prevails with a unity Lewis number. Damkhöler number (ratio between an integral scale of the turbulence and the chemical characteristic time) is equal to 3.7. Karlovitz number (ratio between the chemical characteristic time and the Kolmogorov time) is equal to 0.91.

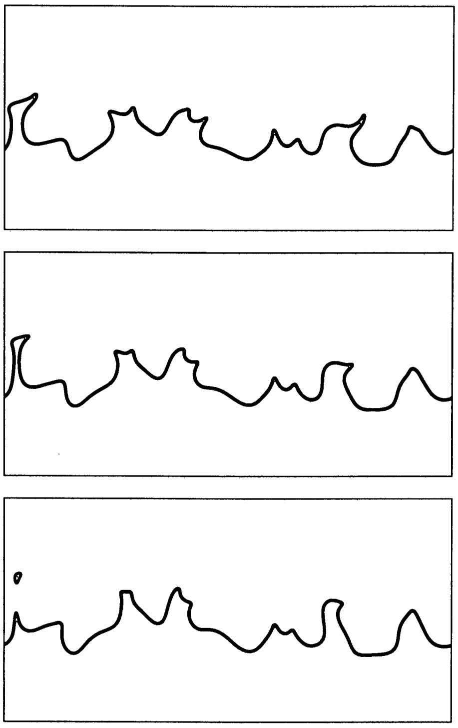

FIG. 7 – **Heat release isocontours**

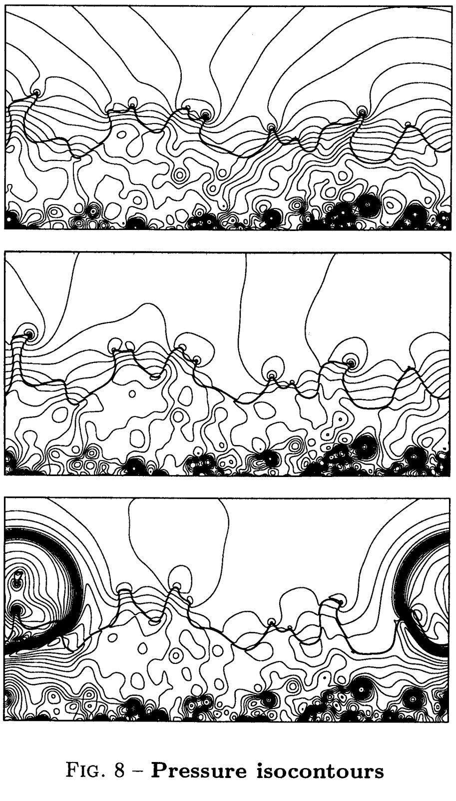

FIG. 8 – **Pressure isocontours**

Initially, the action of the turbulence wrinkles the flame and increases its surface (fig. 7). During this stage, the mean propagation velocity of the flame increases. Then it reaches a statistically stationary state (fig. 6) which allows for the temporal analysis of any physical parameters. It is also possible to examine non-stationary behavior. For example, during this computation, the formation of fresh gases pockets in the burnt gases area may be observed as it is shown figure 8. The pocket formation is induced by front to front interaction of two parts of the flame. When they meet, burnt gases are confined and a pressure wave is generated and propagates (fig. 8) along with a divergence of the velocity field (fig. 9).

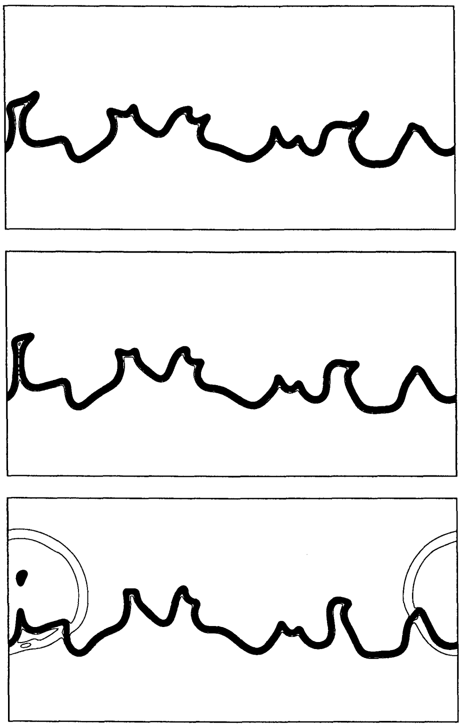

FIG. 9 – **Velocity divergence**

## CONCLUSION

The numerical procedure presented in this paper (fig. 10) allows for direct numerical simulation of spatially decaying turbulence. Statistical properties of the flow are stabilized and controlled. The fully compressible Navier-Stokes equations are solved with a classical finite difference DNS solver but specific boundary conditions have been developed for the turbulence injection. The proposed method ensure the continuity of the turbulent structures being injected. Moreover the flow has the possibility of going back the inlet boundary following the rotation intensity. A turbulent premixed flame (flamelet regime) has been stabilized with this procedure.

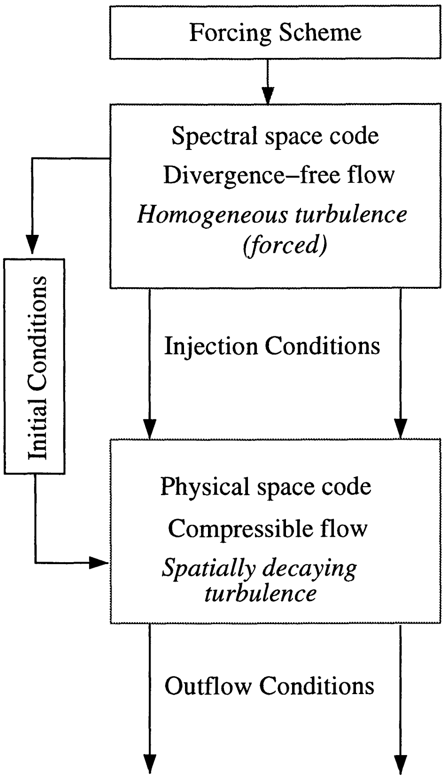

FIG. 10 – **Sketch of the numerical procedure**

## REFERENCES

Eswaran and Pope S.B. (1988) Direct numerical simulations of the turbulent mixing of the passive scalar. *Phys. Fluids*, 31(3):506–520.

Louch, D. S. (1998). *Vorticity and turbulent transport in premixed turbulent combustion.* PhD thesis, University of Cambridge, U.K.

Overholt and Pope, S. B. (1998). A deterministic forcing scheme for direct numerical simulation of turbulence. *Computer and Fluid*, 27:11–28.

Poinsot, T., Candel, S., and Trouvé, A. (1996). Direct numerical simulation of premixed turbulent combustion. *Prog. Energy Combust. Sci.*, 12:531–576.

Poinsot, T. and Lele, S. K. (1992). Boundary conditions for direct simulations of compressible viscous flows. *J. Comput. Phys.*, 1(101): 104–129.

Vervisch-Guichard, L. (1999). *Développement d'outils numériques dédiés à l'étude de la combustion turbulente.* PhD thesis, University of Rouen, France