LES of spatially developing 3D compressible mixing layer

Laetitia DORIS, Christian TENAUD, Loc TA PHUOC

Laboratoire d’informatique et de mécanique pour les sciences de l’ingénieur, UPR CNRS 3251, BP 133, 91403 Orsay cedex, France
E-mail: Laetitia.Doris@limsi.fr

(Reçu le 15 janvier 2000, accepté après révision le 22 avril 2000)

Abstract. The spatial development of a turbulent compressible mixing layer is investigated by means of large eddy simulation (LES). The subgrid viscosity is represented by the so-called mixed-scale model, adapted to compressible flows. Two different shock capturing schemes and three sets of inlet white-noise perturbations are investigated. The comparison between numerical and experimental results gives an overall good agreement.

SGE du développement spatial de couches de mélange compressibles turbulentes 3D

Résumé. Cette étude concerne le développement spatial d’une couche de mélange compressible turbulente par la méthode de la simulation des grandes échelles (SGE). La viscosité de sous-maille est représentée par le modèle d’échelles mixtes adapté aux écoulements compressibles. Deux schémas numériques à capture de choc, ainsi que trois combinaisons de bruits blancs, perturbant les vitesses d’entrée, sont testés. La comparaison entre les résultats numériques et expérimentaux donne un bon accord général.

Version française abrégée

1. Introduction

Cette étude s’intéresse à la prédiction d’écoulements libres compressibles turbulents par la simulation des grandes échelles. L’écoulement choisi est la couche de mélange se développant spatialement en aval d’une plaque plane. L’influence de l’amplitude des perturbations du profil de vitesse à l’entrée, ainsi que celle du schéma numérique, sont étudiées. Le nombre de Mach convectif est 0,64, pour lequel les effets de compressibilité sont relativement faibles, et le nombre de Reynolds est de 60000.

2. Équations et modélisation

Les équations de Navier–Stokes pour les écoulements compressibles sont filtrées spatialement. Plusieurs termes de sous-maille apparaissent, dus à la présence de non-linéarités dans les équations. Seuls deux
d’entre eux sont modélisés dans cette étude, les autres étant supposés négligeables. Le premier est le tenseur des corrélations de vitesse de sous-maille, modélisé par une formulation de type Boussinesq; la viscosité de sous-maille est basée sur le modèle d’écailles mixtes (équation (1)). Le second est le flux de chaleur de sous-maille, modélisé par une formulation de type premier gradient (équation (2)); le nombre de Prandtl de sous-maille est fixé à 0,6.

3. Méthode numérique et configuration étudiée

Le domaine de calcul est constitué de $512 \times 59 \times 59$ points. Deux schémas numériques différents, tous deux à capture de choc et à variation totale décroissante (TVD) sont comparés : le premier est le schéma de Harten–Yee avec le limiteur de Van Leer [5]; le second est un schéma WENO d’ordre plus élevé [6]. Les conditions aux limites externes s’appuient sur la méthode des caractéristiques. Le cas considéré est similaire à une configuration expérimentale déjà étudiée dans la littérature [9,8]. Trois simulations différentes sont effectuées avec le schéma Harten–Yee, pour lesquelles seules les amplitudes des bruits blancs superposés aux profils de vitesse d’entrée varient. Une autre simulation est effectuée avec le schéma WENO de façon à observer l’influence de la dissipation numérique sur les résultats.

4. Résultats des simulations tridimensionnelles

On étudie le champ de vitesse moyennée en temps et suivant la direction transverse. On néglige dans ce champ moyen toutes les contributions de sous-maille. L’évolution longitudinale de l’épaisseur de vorticité est comparée aux données expérimentales (voir figure 2). Le cas pour lequel les perturbations sur $u$ sont les plus élevées permet d’obtenir la similitude plus proche du bord de fuite de la plaque. De plus le schéma WENO donne un taux d’épanouissement plus proche de l’expérience que le schéma Harten–Yee. La similitude est bien respectée par les profils de vitesse longitudinale dans la couche de mélange pour les trois cas. Pour ce qui est des intensités turbulentes $\sigma_u$ et $\sigma_v$, le cas avec une perturbation de 10% sur $u$ et avec le schéma WENO donne un comportement de similarité et un accord avec l’expérience satisfaisant. Pour ce cas, les coefficients de dissymétrie $S_u$ et $S_v$ correspondent bien aux résultats expérimentaux.

1. Introduction

This study deals with the simulation of a high-speed compressible 3D mixing layer, by means of large eddy simulation. The goal is to validate this method for complex, inhomogeneous and external flows in a compressible case. Many temporal DNS or LES of compressible mixing layers can be found in the literature (see, for example, [1,2]), but very few studies deal with the 3D spatial case. In order to simulate the flow structure observed experimentally, our computational domain is 3D, starts at the trailing edge of the flat plate and includes both the transitional wake and mixing-layer regions. The difficulties of this type of simulation concern mainly the settings of the inlet and outlet boundary conditions, since they should reproduce as precisely as possible the experimental boundary conditions. In this paper, we report the influence of the magnitude of the inlet perturbations on the setup of the mixing layer, as well as the influence of the numerical scheme. The convective Mach number investigated is $M_c = 0.64$, for which the compressibility effects are still relatively weak. This value is however high enough for the flow to be highly three-dimensional. The Reynolds number, based on the inlet high-speed boundary-layer thickness, is $Re = 60 000$. 

568
LES of 3D compressible mixing layer

2. Governing equations and subgrid modeling

2.1. Filtered equations

The governing equations are the compressible Navier–Stokes equations written in a conservative form, representing the conservation of mass, momentum and total energy by unit of mass. These equations are filtered with an implicit spatial filter combined with the Favre decomposition: \( \rho = \overline{\rho} + \rho' \), \( \overline{u_i} = \overline{\rho u_i} / \overline{\rho} \). \( \overline{u_i} \) is the resolved part of the density-weighted velocity, and \( u_i' \) its subgrid or unresolved part: \( u_i = \overline{u_i} + u_i' \).

Due to nonlinearities, two subgrid terms appear in the momentum equation: the subgrid part of the viscous diffusion, and the subgrid stress tensor \( \tau_{ij} = \overline{\rho(u_i u_j - \overline{u_i} \overline{u_j})} \). In the energy equation, we have chosen to use the real filtered pressure, which is a function of \( \tau_{kk} \). Hence four subgrid terms appear (see [3]): the subgrid viscous diffusion, the subgrid heat diffusion, the subgrid third-order moment of the velocity, and the subgrid heat flux \( \Theta_j = C_p \overline{u_j \overline{T} - \overline{u_j} \overline{T}} \). In this study, only \( \tau_{ij} \) and \( \Theta_j \) are modeled. The other terms are supposed to be at least one order of magnitude lower, according to the conclusions of Vreman et al. [1] concerning a 2D temporal mixing layer.

2.2. Subgrid modeling

The modeling of the subgrid stress tensor is based on the concept of subgrid viscosity \( \mu_{sg} \). This subgrid viscosity is modeled by the mixed-scale model, which is defined as the algebraic average between the vorticity model and the turbulent kinetic energy model [4]:

\[
\mu_{sg} = \left( \frac{\overline{\rho C_\omega \Delta^2 |\nabla u|}}{\overline{\rho C_k \Delta^2 \sqrt{k}}} \right)^{1/2} \tag{1}
\]

In the vorticity model (\( \mu_\omega \)), the definition of the time scale is based on the magnitude of the resolved part of the vorticity (\( \overline{\omega} \)), with \( C_\omega = 0.01 \). \( \Delta \) represents the cutoff length of the spatial filter. The turbulent kinetic energy model (\( \mu_k \)) is based on the subgrid kinetic energy \( k = u_i' u_i'/2 \), with \( C_k = 0.126 \). \( k \) is obtained by the scale similarity assumption [4], and by means of a double-filtering technique: \( k = (\overline{\nabla u_k^2} \overline{u_k^2})^2/2 \), where (\( \overline{\cdot} \)) represents an explicit filter with a cutoff length of \( 2\Delta \).

The mixed-scale model takes into account both the large and small velocity scales: the first one through the resolved vorticity and the second one through the subgrid kinetic energy. It is a self-adapted model because the eddy viscosity vanishes automatically at the wall and in the regions of the flow where all the structures are well resolved.

A comparison of three different models of subgrid viscosity (the TKE model, the mixed-scale model and the Smagorinsky model) has been made for a 2D configuration in [3]. The mixed-scale model has been selected for this 3D study, since it has given satisfactory results for the 2D computations.

Concerning the modeling of the subgrid heat flux, a first gradient formulation is used, where the subgrid Prandtl number \( Pr_{sg} \) is assumed to be constant, and set to 0.6:

\[
\Theta_j = \frac{\mu_{sg} C_p}{Pr_{sg}} \frac{\partial \overline{T}}{\partial x_j} \tag{2}
\]

3. Numerical approach and configuration studied

The computational domain starts at the trailing edge of the flat plate. It contains 512 points in the longitudinal \( (x) \) direction, and 59 points in both vertical \( (y) \) and spanwise \( (z) \) directions. The mesh is tightened in the \( y \) direction around the centerline of the mixing layer. In the two other directions the mesh is homogeneous. The dimensions of the domain are \( L_x = 80 \delta_1 \), \( L_y = 50 \delta_1 \), and \( L_z = 30 \delta_1 \), where \( \delta_1 \) is the high-speed boundary-layer thickness at the inlet (\( \delta_1 = 10 \text{ mm} \)).
The governing equations have been solved through a finite volume method, using two different numerical schemes. The first one is a spatially second-order upwind shock-capturing TVD scheme originally developed by Harten and Yee [5]. The limitor used in our computations is the Van–Leer limitor function. The second numerical scheme is a WENO scheme, with \( r = 3 \) [6], which is supposed to be accurate up to the fifth order in the smooth regions. Concerning the diffusive fluxes, a central differencing scheme is applied, giving a second order accuracy in space. The time integration is based on a third-order Runge–Kutta method [6].

At the inlet boundary, which corresponds to the trailing edge of the flat plate, the non-reflecting boundary condition method of Poinsot and Lele [7] is applied. The longitudinal mean velocity profile is initialized using a Whitfield boundary layer profile. The vertical and spanwise components of the mean velocity are set to zero. Three unsteady white noises are superimposed on the three components of the mean velocity. Three cases have been investigated. The magnitudes of these white noises are displayed in the table, in percents of the high speed \( U_1 \).

At the upper and lower part of the domain, and at the outlet, non-reflecting conditions are also applied, in order to avoid the confinement of the mixing layer, and so we expect that the structures within the mixing layer can freely grow as they are convected downstream. In the spanwise direction, which is supposed to be homogeneous, a periodic condition is used.

The physical configuration studied is similar to the one investigated experimentally by Samimy et al. [8] and by De Bisschop [9]. The subscripts 1 and 2 represent the high-speed and low-speed parts of the domain, respectively. The Reynolds number is known to have very little influence on the growth rate of the mixing layer. For instance, the two experiments we refer to in this study do not have the same Reynolds numbers but have very close growth rates (see the next section). Hence, in order to reduce computational costs, the Reynolds number of the simulation is 15 times smaller than the one in the experiments of De Bisschop. Its value, based on \( U_1 \) and \( \delta_1 \), is \( Re = 6.0 \cdot 10^4 \). The convective Mach number is \( M_c = 0.64 \). The velocity ratio \( r = U_2/U_1 \) is 0.27, the density ratio \( s = \rho_2/\rho_1 \) is 0.57.

4. Results of 3D simulations

In order to compare the numerical results to experimental ones, the 3D velocity field is averaged in time and along the \( z \)-direction. The nondimensional integration time is \( \Delta t = 60 \) to 120 depending on the case. The reference speed is \( (U_1 + U_2)/2 \) and the reference length is the inlet boundary layer thickness \( \delta_1 \). The mean values of the subgrid part of all the quantities are supposed to be zero.

In the left part of figure 1, the longitudinal evolution of the vorticity thickness \( \delta_\omega = \Delta U/(\partial u/\partial y)_{\text{max}} \), with \( \Delta U = U_1 - U_2 \), is plotted for the three cases of inlet perturbations with the Harten–Yee scheme. In each simulation curve, two parts can be distinguished: the first one represents the wake of the flat plate. Then, a transition occurs and the evolution becomes linear. This linear part represents the mixing layer itself, which has reached a self-similar behavior. If we plot a matching line of this part of the curve, its intersection with the \( x \)-axis gives the virtual origin of the mixing layer. The slope of this part of the curve is
called the expansion rate of the mixing layer. We can notice that the three expansion rates are equal to each other, which means that the spatial development of the mixing layer is not influenced by the inlet noise. However, the three virtual origins are not the same: in case 2, the virtual origin is closer to the flat plate than in cases 1 or 3. This could mean that only the perturbation along the $x$ direction has a significant influence on the position of the transition between wake and mixing layer.

In the right part of figure 1, the two numerical schemes are compared to each other and to the experimental results of De Bisschop [9] and of Samimy et al. [8]. The numerical curve has been shifted by a certain value of $x_0$ in order to have the same virtual origin as the experimental data. The agreement between simulation and experiment is much better for the WENO simulation than for the Harten–Yee simulation. This shows that the dissipation of the numerical scheme has a nonnegligible influence on the spatial development of the mixing layer.

Note that in the experiments the wake of the flat plate is not observed, contrary to our simulations. This could be explained either by the fact that the Reynolds number of the simulations is not high enough, or by the fact that the inlet boundary layer profile does not take into account the growth of the experimental boundary layer along the flat plate.

Once the flow has reached a self-similar state, all the dimensionless profiles should collapse. In figure 2 (right) the nondimensional longitudinal velocity $U^* = (U(y) - U_2)/\Delta U$ profiles are plotted for case 2, at six locations situated in the self-similarity region, as a function of the nondimensional vertical coordinate.
L. Doris

Figure 3. Skewness of longitudinal (left) and vertical (right) velocities versus dimensionless vertical coordinate for case 2, using the WENO scheme.

Figure 3. Coefficient de dissymétrie des vitesses longitudinale (à gauche) et verticale (à droite), en fonction de la coordonnée verticale adimensionnée, pour le cas 2, avec le schéma WENO.

\[ y^* = (y - y_0)/\delta_\infty \], where \( y_0 \) is taken such as \( U(y_0) = (U_1 + U_2)/2 \). The six profiles gather very well together. Moreover they fit the experimental profiles quite satisfactorily. It is worth noting that the same result is also obtained in cases 1 and 3.

The nondimensional r.m.s. of the longitudinal (\( \sigma_u/\Delta U \)) and vertical (\( \sigma_v/\Delta U \)) velocities are plotted in figure 2, for case 2 using the WENO scheme. The similarity behavior is well recovered for both intensities, and moreover they are well predicted at the edges of the mixing layer, except for \( \sigma_u \) on the low-speed side \( (y^* \approx -1) \). This may be due to the high inlet perturbations of the longitudinal velocity. At the center of the mixing layer, \( \sigma_u \) is overpredicted by about 25%. This overprediction has already been observed in other configurations [10], with different subgrid models. No convincing explanation has yet been found for this behaviour.

Concerning the third order moments, the skewness of the longitudinal and vertical velocities are plotted in figure 3, only for case 2 using the WENO scheme. We can observe that the agreement with experiment is very good within the mixing layer region, which proves that the turbulence organisation is well recovered. However in the vicinity of the mixing layer, the noise due to the inlet perturbations is still important and leads to hazardous prediction of \( S_u \) and \( S_v \) coefficients.

5. Conclusion

3D simulations of the spatial development of a compressible mixing layer at a convective Mach number of 0.64 have been run, with three combinations of inlet white noise perturbations and with two different numerical schemes. The longitudinal evolution of the vorticity thickness is influenced both by the amplitudes of these white noises and by the numerical scheme. The case which gives the best results is the one with the highest perturbation on the longitudinal velocity, and with the WENO scheme. In this case a similarity behavior is obtained for \( \sigma_u \) and for \( \sigma_v \) but the former is overpredicted at the center of the mixing layer. The skewness coefficients are however well recovered in this part of the flow. These encouraging results seem to prove the ability of large eddy simulation to represent correctly this weakly compressible flow. In order to obtain a better understanding of compressibility effects found in the experiments, the study of a highly compressible mixing layer (at \( M_c = 1 \)) should be undertaken in the future.

Acknowledgements. The calculation time has been provided by IDRIS, CNRS. This work is financially supported by the DGA, ONERA. The authors greatly acknowledge these institutions.

572
References