Aerospace Science and Technology 31 (2013) 59–72

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Large eddy simulations of 2-D and 3-D spatially developing mixing layers

Chandra S. Martha[a,*], Gregory A. Blaisdell[a], Anastasios S. Lyrintzis[b]

[a] School of Aeronautics and Astronautics, Purdue University, West Lafayette, IN 47907, United States
[b] Embry-Riddle Aeronautical University, Datona Beach, FL 32114, United States

ARTICLE INFO

Article history:
Received 12 October 2012
Received in revised form 11 September 2013
Accepted 20 September 2013
Available online 1 October 2013

Keywords:
Large eddy simulation
Mixing layer
Inflow forcing
Vortex method
Translative instability
Vortex pairing
Streamwise vortices
Rib vortices
Buffer zone
ANSYS-Fluent™

ABSTRACT

The present work focuses on evaluating and improving the modelling capabilities of the second-order accurate ANSYS-Fluent™ LES implementation for a spatially developing mixing layer. The LES methodology is evaluated by modelling a widely studied problem, a spatially-developing, planar mixing layer and improved by modifying the inflow forcing algorithm available in Fluent to better suit the mixing layer. The latter is one of the main contributions of this work. The suggested inflow modification is applicable to any CFD code, not only Fluent. A low Reynolds number mixing layer is simulated in two- and three-dimensions without the splitter plate walls. The effects of inflow forcing and a buffer zone at the domain exit are investigated using 2-D simulations. The sensitivity of the 2-D results to time-step and grid sizes is also investigated. We present a modified random vortex method (VM) algorithm for inflow forcing in 3-D. The modified VM is based on modelling inflow velocity perturbations through hair-pin vortices of the splitter plate boundary layers to better capture the translative instability and the streamwise vortices of the mixing layer.

1. Introduction

Recent advances in the processing power of computers have dramatically changed the landscape of computational fluid dynamics (CFD). High-fidelity CFD tools, *e.g.*, large eddy simulation (LES) and direct numerical simulation (DNS), have become viable means of understanding complex fluid dynamics phenomena. However, satisfactory simulation methodologies need to be established before these tools can be used effectively. The current work deals with evaluating and improving the LES methodology available in ANSYS-Fluent™. The LES methodology is evaluated by modelling a widely studied problem, a spatially-developing, planar mixing layer and improved by modifying the inflow forcing algorithm available in Fluent to better suit the mixing layer.

A planar mixing layer is formed at a planar interface of two fluid streams moving one over the other with different velocities. Mixing layers belong to an important class of free shear flows and commonly occur in many engineering applications involving chemically reacting flows, scalar mixing and jets, etc. A clear understanding of the characteristic features of the mixing layers is prerequisite to controlling the process of mixing in such applications.

Rayleigh [26] showed that a mixing layer is unstable for certain wavy disturbances. The instability modes of the mixing layer can be classified broadly into two categories: (a) two-dimensional modes, formed as a consequence of spanwise invariant disturbances, that cause coherent spanwise vortex rollers (Kelvin–Helmholtz instability mode) and their subsequent pairing downstream (subharmonic instability mode), and (b) three-dimensional modes, formed when the perturbations have spanwise variation, that result in translative instability [25], helical pairing [9], and formation of streamwise vortices. Pierrehumbert and Widnall [25] reported that the "translative instability" of a mixing layer bends the cores of the spanwise vortex rollers and that the instability mode is most unstable for perturbations with spanwise wavelengths approximately 2/3 of the spacing between the 2-D spanwise vortex rollers. This instability is attributed to the development of counter-rotating streamwise (rib) vortices, which eventually leads to three-dimensionality in a mixing layer. A sketch of the topology of streamwise vortices is shown in Fig. 15 of Bernal and Roshko [7]. Rogers and Moser [27] studied the formation of these rib vortices through numerical simulations and concluded that the rib vortices develop in the braid region between the spanwise vortex rollers and extend from the bottom of one roller to the

---

* Corresponding author.
E-mail address: chandra.martha@intel.com (C.S. Martha).

1270-9638/$ – see front matter © 2013 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.ast.2013.09.007

Nomenclature

BCD	Bounded Central Differencing
CFL	Courant–Friedrichs–Lewy
DNS	Direct Numerical Simulation
FTC	Flow-Through Cycle
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier–Stokes
VM	Vortex Method

top of its neighbor. Additional information on the rib/hair-pin vortices can be found in Adrian et al. [1,2]. The interaction of the two- and three-dimensional instabilities initiates the transition of the mixing layer to turbulence leading to self-similarity further downstream. In the self-similar region, the mixing layer exhibits a linear growth rate and collapse of the profiles of appropriately scaled mean velocity and Reynolds stresses.

The perturbations at the inflow dictate the evolution of the mixing layer. The experimental work of Ho and Huang [13] shows that the growth rate of a mixing layer can be greatly manipulated by forcing the mixing layer near a subharmonic of the most-amplified frequency (or fundamental frequency) at a very low forcing level. The development of a mixing layer is highly sensitive to the boundary conditions. The splitter plate geometry [11], free-stream turbulence intensity [24] and velocity ratio [22,23] affect the evolution of the mixing layer. Mixing layers have been investigated experimentally in [6,32,39] and [15] among others. They have been studied computationally in [4,21,27–29,33,35] among others.

The mixing layer is computationally investigated without including the splitter plate by two different approaches: (a) temporally evolving mixing layer, and (b) spatially evolving mixing layer. In the former approach, the evolution of the flow structures is investigated as they convect downstream with respect to an observer moving with these structures. Therefore, the changes in flow structures are seen in time in this approach. Periodic boundary conditions (BCs) are used in the streamwise direction as the flow is assumed homogeneous in that direction. Because of the modelling simplicity, numerous computational investigations have dealt with this approach some of which are [5,10,21,27,28,36,37] among others. The latter approach deals with the study of the flow structures as they convect downstream with respect to a fixed laboratory frame of reference. In this approach, the changes in flow structures are seen in space and hence, streamwise periodic BCs are not suitable. Inflow forcing is employed to introduce artificial perturbations about the mean flow at the inflow boundary to ensure realistic evolution of the mixing layer. Refs. [4,29,33–35,40] among others deal with the simulation of spatially developing mixing layers. Inflow forcing is an active area of research as there is no unified method for all problems and it is chosen to meet the specific needs of the problem being simulated (see [12]).

The perturbations are typically based on the superposition of harmonic modes at specified frequencies for 2-D simulations or on a broad-band spectrum representing isotropic turbulence for both 2-D and 3-D simulations [33]. A precursor simulation with the rescale-recycling technique [16] can also be used for inflow forcing. There are other relatively new inflow forcing methods that are yet to be used for 3-D spatially developing mixing layers. The random flow generation (RFG) technique of Smirnov et al. [31] based on divergence-free velocity field helps generate anisotropic turbulent velocity fluctuations from user-defined Reynolds stresses at the inflow boundary and it was tested satisfactorily for a flat-plate wake flow among other cases. The random vortex method (VM) generates velocity perturbations via a fluctuating two-dimensional vorticity field within the inflow plane and has been shown to work well for a fully developed pipe flow case [20].

Our objective in this study is not to examine the structure of the mixing layer in detail but rather to focus on developing a successful simulation methodology to perform an LES using ANSYS-Fluent™. Our intent is to predict the characteristic features of the mixing layer through spatially developing mixing layer simulations and to assess the applicability of using a second-order spatial discretization scheme in the context of LES on coarse meshes. The walls of the splitter plate are not modeled to keep the computational cost low. The 2-D mixing layer simulations, although not physical for turbulence problems, are performed in order to establish a simulation methodology. A rigorous approach is followed through the 2-D mixing layer simulations to study the sensitivity of the LES results to the temporal and spatial resolutions. The effectiveness of a buffer zone near the end of the physical domain is also investigated. For 3-D mixing layer simulations, the deficiencies of the ANSYS-Fluent implementation of the inflow forcing algorithm, which is based on the random vortex method (VM) as described by Mathey et al. [20], are discussed. One of the main contributions of this work is the modified vortex method to better suit 3-D mixing layer simulations.

The article is organized as follows. The computational procedure dealing with the mixing layer simulations is described in Section 2 along with details on the original vortex method (VM) and its modifications for the 3-D simulations. The results of the 2-D and 3-D mixing layer simulations are discussed in Section 3. Finally, conclusions are given in Section 4. A preliminary version of this work is presented in [18].

2. Computational procedure

The commercial CFD tool ANSYS-Fluent™ (v. 6.3.26) is used in the present study. The tool is based on cell-centered finite volume discretization. The detailed description of its LES solver and the numerical schemes can be found in ANSYS-Fluent [3]. A description of the chosen numerical methods and simulation procedure in this study is given in Section 2.5.

2.1. Parameters of the planar mixing layer

The parameters are chosen to match the 2-D DNS mixing layer case considered in Uzun [35]. An isothermal mixing layer at a temperature $T_\infty = 298$ K is considered. The mean streamwise velocity at the inlet is specified as

$$\bar{u}(y) = \frac{U_1 + U_2}{2} + \frac{U_1 - U_2}{2} \tanh\left(\frac{2y}{\delta_\omega(0)}\right); \tag{1}$$

and the mean cross-stream velocity as $\bar{v}(y) = 0$. The terms U_1 and U_2 represent the velocities of high-speed and low-speed streams, respectively. And, the initial vorticity thickness is given by

$$\delta_\omega(0) = (U_1 - U_2)/|\partial\bar{u}/\partial y|_{y=0}. \tag{2}$$

This velocity profile is a good approximation to the mean flow with a splitter plate far enough downstream that the wake effects have vanished. The convective velocity of the large-scale eddies of the mixing layer $U_c = (U_1 + U_2)/2 = 0.375c_\infty$; and the relative convective Mach number of the mixing layer $M_c = (U_1 - U_2)/2c_\infty =$