European Journal of Mechanics B/Fluids 46 (2014) 172–180

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

The secondary vortex rings of a supersonic underexpanded circular jet with low pressure ratio

Huanhao Zhang, Zhihua Chen *, Baoming Li, Xiaohai Jiang

Key Laboratory of Transient Physics, Nanjing University of Science & Technology, Nanjing, 210094, China

ARTICLE INFO

Article history:
Received 16 December 2013
Received in revised form
1 March 2014
Accepted 31 March 2014
Available online 12 April 2014

Keywords:
Compressible vortex ring
Underexpanded jet flow
Supersonic jet
Large eddy simulation

ABSTRACT

Study of the secondary vortex ring (SVR) is essential to understand the complicated flow structures of supersonic impulsive jets. In the present study, the main characteristics of compressible secondary vortex ring and the primary vortex ring (PVR) in the starting three-dimensional (3D) flow field of a supersonic underexpanded circular jet are investigated numerically for $Ma = 1.2$, 1.4 and 1.8, at a low pressure ratio (jet flow pressure/ambient pressure) of 1.4. The governing equations of large eddy simulation (LES) for compressible flow have been employed and are solved numerically with the combination of high-order hybrid schemes. Our results illustrate the reason for generation of the SVRs by supersonic underexpanded jets, and it is the rolling up of the shear layer which is resulted from the combination of two slip lines when their two triple points on the embedded shock wave interact with each other. After formation, the SVR interacts with the PVR, and rolls over the periphery of PVR and moves upstream. For a higher Mach number of 1.8, multiple SVRs form during the evolution.

1. Introduction

Due to the important applications in the field of propulsion and projectile launch etc., compressible vortex rings generated by supersonic jets have been extensively studied [1–5]. For a supersonic jet, the precursor shock wave front diffracts and expands while exiting the nozzle [6]. Then, high pressure gases within the tube are discharged to form a shear layer and roll up into a vortex ring under the baroclinic effects. This large vortex moves downstream and expands immediately through entrainment of ambient gas into the core, which causes a fast increase of its diameter [7]. For a supersonic flow field, there are shock waves such as shock cells, vortex-induced shock pair and embedded shock appearing in the flow field and evolving with the whole flow field [8], and the flow structures are also distinct for different nozzle geometries [9,10].

For vortex rings emanating from shock tubes, there are mainly three distinct jet flow fields [11]. For low Mach number, there is no shock wave existing, and the vortex ring is conventional and characterized by very thin core. At a higher Mach number (*Ma*) of about 1.4, the vortex-induced shock pair appears on opposite sides of the shear layer of the main vortex core [12], and an embedded rearward facing shock arises as part of the matching process between the expanded gases with the low-pressure upstream and high-pressure downstream. This embedded shock propagates along with the vortex ring and its ends connect with the shock pair [8,13]. With the increase of Mach number to about 1.5–1.65 [7,11,14], a secondary counter-rotating vortex ring (CRVR) appears ahead of the primary vortex ring. It is believed that the cause of the SVR is the rolling up of the shear layer ahead of the embedded shock due to Kelvin–Helmholtz instability [2,7,8].

There are few numerical investigations performed to illustrate the generation and evolution of SVR. Ishii et al. [2] performed both the experimental and numerical simulations on a circular jet, in which a finite-difference TVD scheme was used to solve the Euler equations for an axially symmetric flow. Even though their research focused on the jet evolution, they pointed out that the Kelvin–Helmholtz (K–H) instability along the slip line originating from the triple point of the Mach disk is responsible for the generation of SVRs. Based on the two-dimensional (2D) axisymmetric compressible Navier–Stokes equations and using AUSM+ scheme to approach convective terms, Murugan et al. [13] also studied the formation and evolution of counter rotating vortex rings, they observed that the shear layer formed near the Mach disk plays a dominant role in the generation of SVR.

Although there are many investigations preformed on the vortex rings of circular jets, there are a few studies on the generation mechanism of SVR formed ahead of the PVR, and the numerical studies were conducted under the assumption of two-dimensional

* Corresponding author.
E-mail address: chenzh@mail.njust.edu.cn (Z. Chen).

http://dx.doi.org/10.1016/j.euromechflu.2014.03.016
0997-7546/

ideal/laminar flow, therefore, the details of SVR generation and its evolution along with PVR during the turbulent transition are not very clear. In this paper, with the employed LES and high-order hybrid numerical methods, the starting flow fields of a 3D supersonic circular jet with different Mach numbers, $Ma = 1.2$, 1.4 and 1.8, have been simulated at a low pressure ratio of 1.4. The characteristics of the PVR geometry and its evolution have been discussed, and the generation mechanism of both the single and multiple SVRs has also been illustrated in detail.

2. Numerical method and physical model

2.1. Numerical method

The compressible LES equations can be obtained by Favre filtering the compressible Navier–Stokes equations in the Cartesian coordinate system.

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j) = 0, \tag{2.1}$$

$$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_i\tilde{u}_j) = -\frac{\partial \bar{p}\delta_{ij}}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} - \frac{\partial \tau_{ij}^{sgs}}{\partial x_j}, \tag{2.2}$$

$$\frac{\partial \bar{\rho}\tilde{E}}{\partial t} + \frac{\partial (\bar{\rho}\tilde{E} + \bar{p})\tilde{u}_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\bar{\kappa}\frac{\partial \tilde{T}}{\partial x_j}\right) + \frac{\partial \sigma_{ij}\tilde{u}_i}{\partial x_j} - \frac{\partial q_j^{T-sgs}}{\partial x_j}, \tag{2.3}$$

$$\frac{\partial \left(\bar{\rho}\tilde{z}\right)}{\partial t} + \frac{\partial (\bar{\rho}\tilde{u}_j\tilde{z})}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\bar{\rho}\bar{D}\frac{\partial \tilde{z}}{\partial x_j} - \rho q_j^{z-sgs}\right], \tag{2.4}$$

where the filtered Newtonian stress tensor σ_{ij}, pressure $\bar{p}$ and total energy $\bar{\rho}\tilde{E}$ are expressed by

$$\sigma_{ij} = \bar{\mu}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \tilde{u}_k}{\partial x_k}\right),$$

$$\bar{p} = \bar{\rho}R^0\tilde{T}\sum_{m=1}^{2}\frac{\tilde{Y}_m}{W_m} + \bar{\rho}R^0\sum_{m=1}^{2}\frac{T_m^{sgs}}{W_m},$$

$$\bar{\rho}\tilde{E} = \frac{\bar{p}}{(\tilde{\gamma}-1)} + \frac{1}{2}\bar{\rho}\left(\tilde{u}_k\tilde{u}_k\right) + \frac{1}{2}\tau_{kk}.$$

The subgrid terms including the stress tensor τ_{ij}^{sgs}, turbulent temperature flux q_j^{T-sgs}, scalar transport flux q_j^{z-sgs} and the temperature–species correlation term T_m^{sgs} are given by

$$\tau_{ij}^{sgs} = \bar{\rho}\left(\widetilde{u_iu_j} - \tilde{u}_i\tilde{u}_j\right), \tag{2.5}$$

$$q_j^{T-sgs} = \bar{\rho}\left(\widetilde{c_pTu_j} - \tilde{c}_p\tilde{T}\tilde{u}_j\right), \tag{2.6}$$

$$q_j^{z-sgs} = \bar{\rho}\left(\widetilde{zu_j} - \tilde{z}\tilde{u}_j\right), \tag{2.7}$$

$$T_m^{sgs} = \widetilde{TY_m} - \tilde{T}\tilde{Y}_m. \tag{2.8}$$

The mixture fraction z is defined as: $z = (Y - Y_1)/(Y_2 - Y_1)$, where Y_1 refers to the mass fraction of oxidizer (ambient gas), and Y_2 denotes the fuel (jet gas) mass fraction. With this definition, z takes the value 0 in the gas of jet and 1 in the ambient air. $\bar{\kappa}$, $\bar{\mu}$ and $\bar{D}$ are the filtered heat conduction, dynamics viscosity and molecular diffusivity, respectively. They are obtained from binary mixing rules and the pure component mixing properties. W_m denotes the molecular weight of component m, and R^0 is the gas constant, $R^0 = 8.3143$ J/(mol · K).

The subgrid terms (2.5)–(2.8) need to be modeled for the closure of the multi-component LES equations (2.1)–(2.4). We choose the recently developed stretched-vortex subgrid scales (SGS) model for multicomponent, compressible flows to approach the unresolved subgrid terms [15]. The stretched-vortex SGS model which uses stretched vortices to represent the subgrid scales was proposed originally for incompressible flow [16], and has been extended for different purposes [17–19]. The stretched vortex is a physical model for turbulent fine scales which is assumed to consist of tube-like structures with concentrated vorticity [20]. The stretched-vortex SGS model is designed for simulating turbulent fine scales and has the capability of predicting subgrid scale quantities systematically. In this model, the subgrid turbulent kinetic energy takes the Lundgren form [21].

The supersonic circular jet flow includes shock–vortex interaction and turbulent shear flow, and complex phenomena, therefore, the spatial and temporal resolution requirement varies largely. The fluid-solver framework AMROC (Adaptive Mesh Refinement in Object-oriented C++) [22] has been proven to be advantageous for the LES of supersonic flows [15,23], it includes the block-structured adaptive mesh refinement (SAMR) method [24] and a hybrid numerical method of tuned centered difference-weighted essentially non-oscillatory (TCD–WENO) [21] for simulating the shock induced compressible flow.

The SAMR method in AMROC is one kind of adaptive mesh refinement algorithm designed especially for hyperbolic partial differential equations [25]. The computational domain consists of blocks with rectangular grids, and SAMR uses a hierarchical block data structure, therefore each block can be solved as a single grid which makes the computation more effective than other methods. Further information can be found in Refs. [22,24,25].

AMROC provides an object-oriented framework implementation of the SAMR method mentioned above, in which the SAMR method has been decoupled from a particular scheme, therefore, different schemes can be chosen for the solving. It characterizes with the efficient parallelization strategy on distributed memory machines and can run on all high-performance computers with MPI-library installed.

The hybrid TCD–WENO method was initially proposed in Ref. [21] to satisfy the different resolution requirements within the regions with different flow features, such as shock waves, and turbulence. WENO schemes can effectively eliminate the spurious wave during the calculation, and it has higher order accuracy and excellent discontinuities capturing ability. However, its ability to distinguish shortwave for dissipation error is lower than the high-order difference scheme such as centered-difference methods. TCD uses the Ghosal truncation error as an object function, and optimizes the coefficients of center difference template to eliminate the dispersion error. The hybrid TCD–WENO scheme takes advantage of the virtues of both schemes. The high order WENO scheme is used around discontinuities (such as shock waves and contact surfaces), while TCD scheme (but in skew-symmetric form) is employed to handle the smooth or turbulent regions of the flow. For the hybrid TCD–WENO method, both the WENO and centered-difference methods are specially tuned to minimize dispersive errors at those locations where scheme switching takes place. The WENO stencil coefficients have been adjusted such that the optimal stencil matches the TCD stencil. This modification largely eliminates any dispersion errors that result when transitioning between schemes.

In this paper, the derivatives of inviscid fluxes are presently computed using a hybrid 7-point tuned centered-difference (TCD) scheme and WENO upwind scheme, this leads to 5th order precision. Time advancement is achieved with the fourth-order Runge–Kutta method.

2.2. Physical model

In this paper, the three-dimensional computational domain is taken as a cuboid. The circular nozzle diameter is chosen to be $D = 0.01$ m, and locates at the center of the cuboid's left plane.