Physics Letters A 379 (2015) 1256–1262

Contents lists available at ScienceDirect

Physics Letters A

www.elsevier.com/locate/pla

The initial flow characteristics of supersonic jets with different geometries ☆

Huanhao Zhang, Zhihua Chen [*,1], Xiaohai Jiang, Zeqing Guo

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

A R T I C L E   I N F O

Article history:
Received 28 April 2014
Received in revised form 5 February 2015
Accepted 15 February 2015
Available online 18 February 2015
Communicated by F. Porcelli

Keywords:
Supersonic jet
Non-circular jet
Compressible vortex loop
Axis switching
Large eddy simulation

A B S T R A C T

Based on large eddy simulation (LES) combined with the high-order hybrid schemes, the flow characteristics of both supersonic circular and square jets were investigated numerically for $Ma = 1.4$. Our results illustrate the three-dimensional (3D) structures and their evolutions of both circular and square jets, as well as the shock structures of both jets. Moreover, the phenomena of 45° axis switching and stream-wise vortices enhancing the entrainment and mixing process of the square jet were also identified and discussed in detail.

0. Introduction

Supersonic jets have been applied in many practical applications, including jet and rocket propulsion, projectile launch, mixing augmentation, enhanced combustor performance and thrust vector control [1–5]. As an efficient technique of passive flow control, non-circular jets, such as elliptical, rectangular, square and triangular jets, have been extensively investigated to control the mixing and entrainment rate because they enhance the entrainment properties at relatively low cost [6–16]. The most crucial interest motivated by the engineering applications is to recognize and improve the understanding of the flow properties of non-circular jets; many investigations are underway to elucidate the physical properties of such flows.

Previous studies have determined that the major flow structure of a jet is strongly related to the curvature of the nozzle geometry [10]. As the change of curvature is introduced to the nozzle, it leads to a nonuniform translational velocity downstream [17]. The instability produced by the shape corners, which accelerates the vortex loop deformations, eventually results in an enhancement in mixing. As a non-circular jet spreads, the mean-flow cross-section can evolve through a shape similar to that at the nozzle but with its minor and major axes interchanged. This phenomenon is denoted as the axis switching and has been observed in various non-circular jets [2,5–16]. Moreover, Zare-Behtash et al. [2,5] depicted experimentally the phenomenon of axis switching of the elliptic and square vortex loop. Recently, experimental and numerical research studies [8–12] confirmed that axis switching is the primary mechanism responsible for the enhanced entrainment properties of non-circular jets relative to those of comparable circular jet.

The axis switching of the square jet was observed primarily for the subsonic jets. For the supersonic and unsteady jets, the flow and shock wave structures are not highly clear. In this paper, we aim to present a complete description of the 3D characteristics of the initial flow fields of a supersonic square jet and its comparison with a supersonic circular jet. The large eddy simulation (LES) and high resolution hybrid schemes are used for the numerical simulation. The geometrical characteristics of the primary vortex loops and the evolution of both the circular and square jets is illustrated. Moreover, the process of the axis switching phenomenon and azimuthal instability of the square vortex loop are discussed in detail.

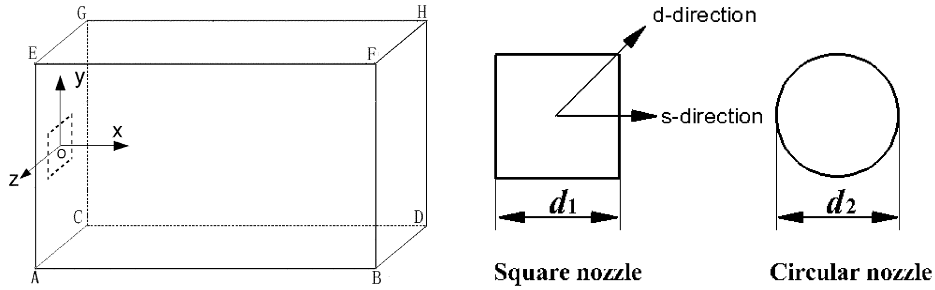
1. Numerical method and physical model

LES is considered to be the most suitable approach to obtain a turbulent flow with high Reynolds number, in which only the large scales of the flow are simulated directly and the small scales are modeled. The compressible LES equations can be obtained by Favre filtering the compressible Navier–Stokes equations in the Cartesian coordinate system [18]. The mixture fraction f is defined as:

☆ Project supported by the National Natural Science Foundation of China under Grant (No. 11272156).

* Corresponding author.
E-mail address: chenzh@mail.njust.edu.cn (Z. Chen).
[1] Professor, Ph.D.

http://dx.doi.org/10.1016/j.physleta.2015.02.027
0375-9601/



(a) Computational domain (b) Geometry of the nozzle

Fig. 1. Schematic of the computational model.

$f = (Y - Y_1)/(Y_2 - Y_1)$, 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, f takes the value 1 in the jet gas and 0 in the ambient air.

The sub-grid terms need to be modeled for the closure of the multi-component LES equations. We choose the recently developed stretched-vortex sub-grid scales (SGS) model for multi-component, compressible flows to approach the unresolved sub-grid terms [19]. The stretched-vortex SGS model is designed for simulating turbulent fine scales and has the capability of predicting sub-grid scale quantities systematically. In this model, the sub-grid turbulent kinetic energy takes the Lundgren form [20].

A hybrid method combining the tuned centered-difference (TCD) stencil with a weighted essentially non-oscillatory (WENO) method is used for solving the LES equations of the compressible and shock-driven flows [21,22]. The hybrid TCD–WENO method [22] has been applied successfully for simulating the shock induced compressible flow [18,23,24]. The TCD–WENO method is proposed to satisfy the different resolution requirements within the regions with different flow features, such as shock waves and turbulent. The WENO scheme has higher-order accuracy and excellent discontinuities capturing ability. However, its ability to distinguish shortwave for dissipation error is lower than the 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 the TCD scheme is used 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 occurs. The WENO stencil coefficients were adjusted to make them match the TCD stencil. This modification largely eliminates any dispersion errors that result when transitioning between schemes. The derivatives of inviscid fluxes are presently computed using a hybrid numerical method with a seven-point TCD scheme and a seven-point WENO upwinded scheme, which leads to 5th order precision. Time advancement is achieved with the fourth-order Runge–Kutta method.

The three-dimensional computational domain and the corresponding coordinate system are shown in Fig. 1(a). The computational domain is chosen to be a cuboid, and the original point of coordinate system is located at the center of the jets. The geometries of square and circular nozzles are displayed in Fig. 1(b). The length (AB), height (AE) and width (AC) of the cuboid are chosen to be $Lx = 75$ mm, $Ly = 56$ mm and $Lz = 56$ mm.

The nozzle is located at the center of the left plane of the cuboid (plane AEGC). The side length (d_1) of the square nozzle and the circular nozzle diameter (d_2) are chosen to be the same, $d_1 = d_2 = 10$ mm. For a square jet, the side wall centerline denotes the direction normal to the flat side and is called s-direction (minor axis), while the diagonal direction is named as the d-direction (major axis).

Initially, the gases within the entire computational domain are chosen to be the same; their specific heat ratio, dynamic viscosity coefficient and temperature are $\gamma = 1.4$, $\nu = 1.73 \times 10^{-5}$ kg/m^2 and $T = 300$ K, respectively. The ambient gas is considered to be standard, its velocity and pressure are $u_0 = 0$ (rest) and $p_0 = 1.0$ atm, respectively. The supersonic gas is injected into the computational domain from the nozzle exit, and the jet Mach number and pressure are taken to be $Ma = 1.4$ and $p_j = 1.4p_0$, respectively. Therefore, the corresponding Reynolds number of both the circular and square jets is the same, $\mathrm{Re} = 2.8 \times 10^5$.

To minimize the amplitude of reflective acoustic waves at the boundaries of the computational domain, non-reflective boundary conditions [25] are applied to the outflow boundaries. The uniform Cartesian grid is used. After the converging tests, the corresponding uniform Cartesian grid number is chosen to be $525 \times 392 \times 392$.

2. Numerical results and discussion

To describe the three-dimensional evolution of the supersonic circular and square jets during the initial stages of propagation, sequential 3D isosurface of mixture fraction ($f = 0.83$) for supersonic circular and square jet are presented in Figs. 2 and 3, respectively.

From both figures, with the initial exiting of supersonic gases, the primary vortex loop (PVL) generates under the baroclinic effects. This large vortex moves downstream and expands immediately through the entrainment of ambient gas into the core, which causes a rapid increase of its diameter. For the circular nozzle, due to the constant azimuthal curvature, its flow expands uniformly in all directions, and the primary vortex core exhibits as a ring, as shown in Fig. 2(a). During its evolution, the core remains circular (Figs. 2(b)–2(c)). Therefore, the vortex diameter in any cross-sectional plane is the same.

For the square nozzle (Fig. 3), the primary vortex core appears to be square and coplanar when it is initially generated (Fig. 3(a)), and then it twists very quickly (Figs. 3(b)–3(c)). Following the motion of the square vortex loop, the square vortex core at different positions in the flow field moves in different directions with different speeds, which accelerates three-dimensionality of the square vortex structure.

Fig. 4 shows the head-on pressure contours of the primary vortex core (VC) at different times. As indicated in Fig. 4, the lowest pressure region (blue) refers to VC, and it divides the flow field into two parts, which are visualized as two concentric loops with high pressure. The exterior boundary of the outer loop refers to the precursor shock wave (PS), while the inner one is the jet shear layer (SL). For the circular jet, the PVL is circular, and it expands and attenuates rapidly. Therefore, the cross flow field of the circular jet appears symmetric and uniform before $t = 90$ μs (circular