European Journal of Mechanics / B Fluids 67 (2018) 158–167

Contents lists available at ScienceDirect

European Journal of Mechanics / B Fluids

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

# Novel characteristics of wavy cylinder in supersonic turbulent flow

Chang-Yue Xu *, Zhu-Qing Ni

*College of Aerospace Engineering, Nanjing University of Aeronautics & Astronautics, Nanjing 210016, China*

ARTICLE INFO

*Article history:*
Received 21 February 2017
Accepted 10 September 2017
Available online 18 September 2017

*Keywords:*
Supersonic flow
Quasi-steady flow
Scale-adaptive simulation
Wavy cylinder
Circular cylinder

ABSTRACT

Supersonic flow past a wavy cylinder has been investigated numerically using scale-adaptive simulation technique. The free-stream Reynolds number and Mach number for this supersonic flow are chosen as $2 \times 10^5$ and 1.7, respectively. For the sake of comparison and discussion, a corresponding circular cylinder for the same computational parameters also has been calculated. Unlike the subsonic operating condition, drag reduction and fluctuating lift suppression of wavy cylinder in supersonic flow are less dramatic. Results indicate that wavy surface as a flow control technique is invalid for supersonic flow over a circular cylinder. This is because the existence of bow shock-wave and no large-scale vortex shedding in the flow field of circular cylinder. Further, two different flow states have been found in the near wake of wavy cylinder. One is a quasi-steady flow state behind the nodal position with converged wake, and the other is an unsteady flow state behind the saddle position with obvious vortex street. Based on the analyses of flow patterns, pressure power spectral densities, and proper orthogonal decomposition of pressure fields, some novel characteristics of wavy cylinder in supersonic flow are systematically investigated.



## 1. Introduction

Turbulent flow past a bluff body is of a basic physical problem, which has been studied for many years. From the fundamental study view, the researchers are prone to choose some canonical flow configurations, such as circular cylinder, square cylinder, sphere, and so on. As a typical bluff body, circular cylinder is used usually by many computational or experimental researches because of its simple geometry. Changing the geometry of circular cylinder, some special force and flow characteristics can be obtained, such as drag reductions, fluctuating force and vortex shedding suppressions, etc. Wavy surface as a simple modification of the cylinder, such as the wavy cylinder, has attracted many researchers due to its flow control characteristics.

Extensive works have been conducted on the incompressible flow around a wavy cylinder [1–10], including experiments and numerical simulations. Selecting the appropriate geometric parameters, i.e., wavelength and amplitude of wavy surface, drag reductions and fluctuating lift suppressions can be obtained. Drag reductions of wavy cylinder are closely associated with modifications of flow characteristics. For example, vortex shedding is suppressed or delayed downstream, dead-air zones behind the body are elongated. People have a better understanding of force and flow characteristics of wavy cylinder in incompressible flow. Unfortunately, relevant researches of wavy cylinder in the compressible flow are relatively sparse. In our previous work [11], the transonic flow past a wavy cylinder has been investigated at a free-stream Mach number 0.75. Besides the similar force behaviors and flow characteristics to the incompressible flow, several special fundamental mechanisms corresponding to the compressible effect have been studied. Comparing to the incompressible flow, some different flow phenomena have been observed reliably, such as shock and shocklet elimination. For wavy cylinder in the supersonic regime, the reader may have a question: does there have some different characteristics from the subsonic regime? To our best knowledge, no experimental or numerical works have been found to determine the flow features and force characteristics of a wavy cylinder in supersonic flow. The aim of the present study is to make such a determination.

We know that circular cylinder is the prototype of wavy cylinder. To investigate the flow characteristics of a wavy cylinder in the supersonic flow, the compressible flow features of a corresponding circular cylinder need to be identified. For the transonic flow past a circular cylinder [12], two different flow states can be distinguished according to a critical Mach number $M_{cr}$ ($\approx 0.9$). When $M_\infty < M_{cr}$, the flow field acts as the unsteady state characterized with vortex shedding. While, a quasi-steady flow state with small disturbances appears in the wake for $M_\infty > M_{cr}$. The wake converges sharply to a neck with two symmetrical trailing shock-waves standing there, which is similar as the supersonic regime [13]. Recent studies [14] suggest that the small oscillations in supersonic flow around circular cylinder are likely driven

* Corresponding author.
*E-mail address:* cyxu@nuaa.edu.cn (C.-Y. Xu).

http://dx.doi.org/10.1016/j.euromechflu.2017.09.006

by pressure wave along the shear layer, unlike the oscillations originated from vortex shedding in the subsonic flow. Unsteady flow state has been observed in the transonic flow past the wavy cylinder at $M_\infty < M_{cr}$ [11]. When $M_\infty > M_{cr}$, especially for supersonic flow over a wavy cylinder, does there have some special flow features? Perhaps, it has the similar flow characteristics as circular cylinder? Up to now, all of researches cannot give a definitive answer. To address this question, a more in-depth analysis of supersonic flow over a wavy cylinder has been performed in this paper.

To investigate flow characteristics of wavy cylinder in the supersonic turbulent flow, an appropriate numerical approach applied to the compressible turbulence needs to be checked. Reynolds averaged Navier–Stokes equation (RANS) method is used commonly for the engineering problem. RANS has the advantage of small computer resource in the computation. While the RANS turbulence model produces too much eddy viscosity, which often over-damps the unsteady flow features in the flow field. Direct numerical simulation (DNS) approach can simulate the turbulence at all scales, which indicates that very fine grids are needed. In practice, DNS is only limited to address the low Reynolds number flow problems [15]. Large-eddy simulation (LES) is a method between RANS and DNS. Nevertheless, LES also requires extremely fine grid-resolution in the turbulent boundary layer for the high Reynolds number flow. To reduce the requirement of computational grids, RANS/LES hybrid methodology has been proposed, which can combine the advantages of RANS and LES. Detached-eddy simulation (DES), a popular RANS/LES hybrid technique, is firstly suggested by Spalart et al. [16] in 1997. In this DES version, the calculated zones can be divided into two different parts through a grid-spacing limiter, i.e., the RANS and LES computational zones. This RANS/LES zone-divided method relies heavily on the grid setup, which can result in a gray-area problem [17]. To avoid this gray-area problem, Menter et al. [18] proposed a scale-adaptive simulation (SAS) technique. In the SAS method, the RANS switches to the LES through the resolved small scale, other than the grid setup. In our previous work [19], a modified SAS method based on one-equation turbulence model has been proposed and proved to be effective for the compressible turbulence simulation. Therefore, this SAS method is chosen as the numerical tool in the present study.

## 2. Simulation and numerical methodology

### 2.1. Governing equations

Governing equations employed for the present physical problems are the 3D Favre-averaged compressible Navier–Stokes equations, including one continuity equation, three momentum equations and one total energy equation. Some free-stream velocity variables are chosen as the characteristic parameters, i.e., velocity $U_\infty$, density $\rho_\infty$, pressure $p_\infty$, temperature $T_\infty$, and the mean diameter of wavy cylinder $D$. In the generalized coordinate system, the dimensionless governing equations can be given as

$$\partial_t\left(\boldsymbol{Q}/\boldsymbol{J}\right)+\partial_\xi\left(\boldsymbol{F}-\boldsymbol{F}_\nu\right)+\partial_\eta\left(\boldsymbol{G}-\boldsymbol{G}_\nu\right)+\partial_\zeta\left(\boldsymbol{H}-\boldsymbol{H}_\nu\right)=0; \tag{1}$$

where, $\boldsymbol{Q}$ is the vector of conservative flow variables including the density $\overline{\rho}$, three velocity components $(\widetilde{u}, \widetilde{v}, \widetilde{w})$, and specific total energy $\widetilde{E}$:

$$\boldsymbol{Q}=\left[\overline{\rho}, \overline{\rho}\widetilde{u}, \overline{\rho}\widetilde{v}, \overline{\rho}\widetilde{w}, \widetilde{E}\right]^T. \tag{2}$$

$\boldsymbol{J}$ denotes the transformation Jacobian of Cartesian-coordinate $(x, y, z)$ to generalized-coordinate $(\xi, \eta, \zeta)$,

$$\boldsymbol{J}=\frac{\partial\left(\xi,\eta,\zeta\right)}{\partial\left(x,y,z\right)}. \tag{3}$$

The convective flux terms $(\boldsymbol{F}, \boldsymbol{G}, \boldsymbol{H})$ and viscous flux terms $(\boldsymbol{F}_\nu, \boldsymbol{G}_\nu, \boldsymbol{H}_\nu)$ are

$$\boldsymbol{F}=\frac{1}{\boldsymbol{J}}\begin{bmatrix}\xi_{x_i}\overline{\rho}\widetilde{u}_i\\ \xi_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{u}+\xi_x\overline{p}\\ \xi_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{v}+\xi_y\overline{p}\\ \xi_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{w}+\xi_z\overline{p}\\ \xi_{x_i}\left(\widetilde{E}+\overline{p}\right)\widetilde{u}_i\end{bmatrix},\quad \boldsymbol{G}=\frac{1}{\boldsymbol{J}}\begin{bmatrix}\eta_{x_i}\overline{\rho}\widetilde{u}_i\\ \eta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{u}+\eta_x\overline{p}\\ \eta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{v}+\eta_y\overline{p}\\ \eta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{w}+\eta_z\overline{p}\\ \eta_{x_i}\left(\widetilde{E}+\overline{p}\right)\widetilde{u}_i\end{bmatrix},$$

$$\boldsymbol{H}=\frac{1}{\boldsymbol{J}}\begin{bmatrix}\zeta_{x_i}\overline{\rho}\widetilde{u}_i\\ \zeta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{u}+\zeta_x\overline{p}\\ \zeta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{v}+\zeta_y\overline{p}\\ \zeta_{x_i}\overline{\rho}\widetilde{u}_i\widetilde{w}+\zeta_z\overline{p}\\ \zeta_{x_i}\left(\widetilde{E}+\overline{p}\right)\widetilde{u}_i\end{bmatrix}; \tag{4}$$

$$\boldsymbol{F}_\nu=\frac{1}{\boldsymbol{J}}\begin{bmatrix}0\\ \xi_{x_i}\widetilde{\boldsymbol{\tau}}_{i1}\\ \xi_{x_i}\widetilde{\boldsymbol{\tau}}_{i2}\\ \xi_{x_i}\widetilde{\boldsymbol{\tau}}_{i3}\\ \xi_{x_i}\left(\widetilde{u}_j\widetilde{\boldsymbol{\tau}}_{ij}-\widetilde{q}_i\right)\end{bmatrix},\quad \boldsymbol{G}_\nu=\frac{1}{\boldsymbol{J}}\begin{bmatrix}0\\ \eta_{x_i}\widetilde{\boldsymbol{\tau}}_{i1}\\ \eta_{x_i}\widetilde{\boldsymbol{\tau}}_{i2}\\ \eta_{x_i}\widetilde{\boldsymbol{\tau}}_{i3}\\ \eta_{x_i}\left(\widetilde{u}_j\widetilde{\boldsymbol{\tau}}_{ij}-\widetilde{q}_i\right)\end{bmatrix},$$

$$\boldsymbol{H}_\nu=\frac{1}{\boldsymbol{J}}\begin{bmatrix}0\\ \zeta_{x_i}\widetilde{\boldsymbol{\tau}}_{i1}\\ \zeta_{x_i}\widetilde{\boldsymbol{\tau}}_{i2}\\ \zeta_{x_i}\widetilde{\boldsymbol{\tau}}_{i3}\\ \zeta_{x_i}\left(\widetilde{u}_j\widetilde{\boldsymbol{\tau}}_{ij}-\widetilde{q}_i\right)\end{bmatrix}. \tag{5}$$

Here, '$\overline{(\cdot)}$' and '$\widetilde{(\cdot)}$' represent the ensemble-averaged variables and Favre-averaged variables, respectively. Under the Stokes hypothesis for bulk viscosity, the stress terms $\widetilde{\boldsymbol{\tau}}_{ij}$ and heat flux terms $\widetilde{q}_i$ in tensor notations, including the resolved and unresolved parts, can be written as

$$\boldsymbol{\tau}_{ij}=\left(\widetilde{\mu}+\widetilde{\mu}^T\right)\left(2\widetilde{\boldsymbol{S}}_{ij}-\frac{2}{3}\delta_{ij}\widetilde{\boldsymbol{S}}_{kk}\right)/Re, \tag{6}$$

and

$$\widetilde{q}_i=-\frac{1}{Re\left(\gamma-1\right)}\left(\frac{\widetilde{\mu}}{Pr}+\frac{\widetilde{\mu}^T}{Pr^T}\right)\frac{\partial\widetilde{T}}{\partial x_i}, \tag{7}$$

where $Re$ is the Reynolds number defined as $Re=\rho_\infty U_\infty D/\mu$; $\widetilde{\mu}$ and $\widetilde{\mu}^T$ are the molecular and turbulent viscosity coefficients, respectively; $Pr$ and $Pr^T$ are the laminar and turbulent Prandtl numbers, respectively; $\widetilde{\boldsymbol{S}}_{ij}=\left(\partial\widetilde{u}_i/\partial x_j+\partial\widetilde{u}_j/\partial x_i\right)/2$ is the strain-rate tensor. The pressure can be obtained by the perfect gas relationship

$$\overline{p}=\left(\gamma-1\right)\left[\widetilde{E}-\frac{\overline{\rho}}{2}\left(\widetilde{u}^2+\widetilde{v}^2+\widetilde{w}^2\right)\right]. \tag{8}$$

For the governing equations closing, Sutherland's law for molecular viscosity coefficient $\widetilde{\mu}$ is also employed

$$\widetilde{\mu}=\left(\widetilde{T}\right)^{\frac{3}{2}}\left[\frac{1+\frac{T_0}{T_\infty}}{\widetilde{T}+\frac{T_0}{T_\infty}}\right], \tag{9}$$

where $T_0 = 110.4$K is Sutherland's constant.

### 2.2. Turbulence modeling for SAS method

Similarly to the DES technique, SAS is also developed on the RANS turbulence model. In the SAS approach, a key feature is the introduction of the von Karman length-scale $L_{vk}$ into the RANS turbulence model equation. Taking this basic concept, SAS has a pure RANS nature but achieves the LES-like behaviors, without