Supersonic turbulent channel flows over two and three dimensional sinusoidal rough walls

Mostafa Aghaei Jouybari¹, Junlin Yuan¹, Giles J. Brereton¹ and Farhad A. Jaberi†

¹Department of Mechanical Engineering, Michigan State University, East Lansing, MI 48824, USA

(Received xx; revised xx; accepted xx)

Direct numerical simulations were performed to study supersonic turbulent channel flows over isothermal rough walls. The effect of roughness was incorporated using a newly proposed immersed boundary method. The method uses a level-set/volume-of-fluid field to impose appropriate boundary conditions at the fluid-solid interface. Turbulence statistics of five channel flows (at a Mach number of 1.5 and a bulk Reynolds number of 3000) are compared, including one reference case with both walls smooth and four cases with smooth top walls and rough bottom walls. The four cases differ in the geometry of the roughness, including two 2-dimensional (2D) and two 3-dimensional (3D) sinusoidal waves. Results reveal a strong dependence of the turbulence on the roughness topography and the associated shock patterns. Specifically, the 2D geometries generate strong oblique shock waves that propagate across the channel height and are reflected back to the rough-wall side. These strong shocks are absent for cases with 3D roughness geometries, replaced by weak shocklets. At the impingement locations of the shocks on the top wall in the 2D cases, localized augmentations of turbulence shear production are observed. Such regions of augmented production also exist for the 3D cases, though the augmentation is much weaker. The oblique shock waves are thought to be responsible for a higher entropy generation for cases with 2D surfaces than those with 3D ones, which results in a higher irreversible heat generation and consequently higher temperature profiles for 2D cases compared to the 3D ones. In the present supersonic channels, the effects of roughness extend beyond the near-wall layer due to the shocks. In the 2D cases, for example, the roughness effects extend to the buffer layer on the smooth-wall side. This suggests that Townsend similarity (Townsend 1976) may not apply to a supersonic turbulent flow over rough wall.

1. Introduction

The effects of wall roughness on physics, control, and modeling of compressible flows (subsonic, sonic, super- and hypersonic) are not well understood today. In high speed flow studies, roughness is typically considered as an isolated (e.g. steps, joints, gaps, etc.), or a distributed (e.g. screw threads, surface finishing, and ablation) effect. The main effects of roughness on supersonic flight vehicles are to increase the drag coefficient and to promote boundary-layer transition to turbulence, which increases the heat transfer coefficient. An understanding of these effects is important for flight control and thermal management (through thermal coatings), especially for reentry applications and reusable launch vehicles. Reda (2002) and Schneider (2008) have reviewed the effects of roughness on boundary layer transition, based on experimental wind-tunnel and in-flight test data.

† Email address for correspondence: jaberi@egr.msu.edu
of flows in supersonic and hypersonic conditions. [Radeztsky et al. (1999)] analyzed the effects of roughness of a characteristic size of 1-µm (a typical surface finish) on transitions in swept-wing flows, and [Latin (1998)] investigated effects of roughness on supersonic boundary layers using rough surfaces with $k_s = O(1mm)$ ($100 < k_s^+ < 600$, superscript $+$ shows normalization in wall units). Experimental studies of distributed roughness effects on compressible flows, boundary layer transition, and heat transfer include those of [Braslow & Knox (1958); Reshotko & Tumin (2004); Ji et al. (2006) and Reda et al. (2008)].

Most numerical studies have focused on isolated roughness (see e.g. Bernardini et al. 2012) or ideal distributed roughness such as wavy walls (see e.g. Tyson & Sandham 2013), due to the simplicity in mesh generation and numerical procedures. However, complex distributed roughness is of primary importance and more relevant to flight vehicles, since in high-speed flows “even the most well-controlled surface will appear rough as the viscous scale becomes sufficiently small” (Marusic et al. 2010). Also, according to Schneider (2008), real vehicles may develop surface roughness during the flight which is not present before launch. This flight-induced roughness may be discrete steps and gaps on surfaces from thermal expansion, or distributed roughness induced by ablation or the impact of dust, water, or ice droplets. Studies of this kind demonstrate the need for a compressible solver that can handle complex distributed rough surfaces.

1.1. Literature review on immersed boundary (IB) methods

According to the review by [Mittal & Iaccarino (2005)], IB methods can be divided in the following two major categories for compressible or incompressible flows.

The first one is the continuous forcing approach. An example is the penalty IB method of [Kim & Peskin (2007)], among many others. In this approach, the effects of solid boundaries are accounted for by inserting an additional forcing term in the NS equations before discretization is performed. This method is well suited for elastic boundaries, and rigid bodies; see, for example, the method of [Goldstein et al. (1993)] based on feedback forcing. The fluid-solid interface is diffused in this approach. The method also requires a Lagrangian description of the interface location. In order to prevent stiffness of numerical solver for rigid boundaries, a very low CFL number of $O(10^{-3} - 10^{-2})$ is required. For 3D problems the implementation might be cumbersome (Fadlun et al. 2000).

The second one is the discretized forcing approach (for example, see Fadlun et al. 2000). In this approach, the effect of the wall is imposed based on an additional forcing term in the discretized NS equation. The discretized force term is determined based on the following,

$$\frac{u^{l+1} - u^l}{\Delta t} = \text{RHS}^{l+1/2} + f^{l+1/2}, \quad (1.1)$$

$$f^{l+1/2} = -\text{RHS}^{l+1/2} + \frac{V_B^{l+1} - u^l}{\Delta t}, \quad (1.2)$$

where $V_B$ is the desired velocity at the boundary and RHS contains the convective, pressure and viscous terms. This method does not suffer from numerical stiffness and is appropriate for flows with moving 3D solid surfaces.

The discretized forcing approach is subdivided into two types depending on the formulation of the forcing term. They are summarized as follows. The first is the direct imposition of boundary conditions (or sharp-interface method). An example is the ghost cell IB method (for examples, see Tseng & Ferziger (2003) and Mittal et al. (2008)), where the exact values of $V_B$ are imposed through an interpolation procedure using ghost points. Dirichlet and Neumann boundary conditions (BCs) can be imposed exactly.
However, 3D flows with complex interface geometries (especially with moving interfaces) cause difficulties and require special considerations. Specifically, issues arise when there are multiple image points for a ghost cell, or when there is none. Luo et al. (2017) addressed some of these issues in 2D domains. In addition, the interpolation schemes are dependent on the ghost point locations in the solid domain; the situation becomes complicated for 3D domains. To account for these difficulties in 3D flows with complex interface geometries, an indirect BC imposition can be employed instead of the direct imposition. In this approach, $V_B$ values are not imposed precisely at the interface but through a prescribed distribution of velocity across the interface. Examples include the approaches based on fluid-volume-fraction weighting proposed by Fadlun et al. (2000) and Scotti (2006).

The following comprises a brief description of the use of IB methods in the compressible flow literature. Ghias et al. (2007) used ghost cell method to simulate 2D viscous subsonic compressible flows. They imposed Dirichlet BC for velocity ($u$) and temperature ($T$); the pressure ($P$) at the boundary was obtained using the equation of state and the value of density ($\rho$) was obtained through extrapolation. Their method was second-order accurate, both locally and globally. Chaudhuri et al. (2011) used the ghost cell method to simulate 2D inviscid, sub- and supersonic compressible flows. They applied direct forcing for $\rho$, $u$ and total energy ($E$) equations, while $P$ was determined based on the equation of state. They used a fifth-order-accurate WENO shock-capturing scheme by using two layers of ghost cells. Yuan & Zhong (2018) also used ghost cell method to simulate 2D (sub- and supersonic) compressible flows around moving bodies. de’ Michieli Vitturi et al. (2007) used a discretized forcing approach for a finite volume solver to simulate 2D/3D viscous multiphase compressible flows; the forcing term was determined based on an interpolation procedure. They imposed Dirichlet BC for $u$ and $T$; the equation of state was used for $P$ and flux correction for $\rho$ and $E$. Wang et al. (2017) used continuous forcing (penalty IB method) to simulate fluid-structure interaction with 2D compressible (sub, super, and hyper sonic) multiphase flows.

1.2. Literature review: supersonic flows over roughness

Tyson & Sandham (2013) analyzed supersonic channel flows over 2D sinusoidal roughness at Mach number ($M$) of $M = 0.3, 1.5$ and $3$ to understand compressibility effects on mean and turbulence properties across the channel. They used body-fitted grids to perform the simulations and found that the values of velocity deficit decrease with increasing Mach number. Their results suggest strong alternation of mean and turbulence statistics due to the shock patterns associated with the roughness.

Ekoto et al. (2008) experimentally investigated the effects of square and diamond roughness elements on the supersonic turbulent boundary layers. The objective of their study was to understand how roughness topography alters the local strain-rate distortion, $d_{max}$, which has a direct effect on turbulence production. Their results indicate that the surface with d-type square roughness generated weak bow shocks upstream of the cube elements, causing a small value of $d_{max}$ ($\approx -0.01$), and the surface with diamond elements generated strong oblique shocks and expansion waves near the elements, causing a large variation in $d_{max}$ (ranging from $-0.3$ to $0.4$ across the elements). These values of $d_{max}$ led to a canonical rough-wall boundary layer trend for the square roughness and regions with localized extra turbulence production for the diamond surface.

Studies of Latin (1998), Latin & W. Bowersox (2000) and Latin & W. Bowersox (2002) include a comprehensive investigation on supersonic turbulent boundary layers over rough walls. Five rough surfaces (including 2D bar, 3D cube, and three different sandgrain roughness) have been analyzed at $M = 2.9$. Effects of wall roughness on mean flow,
turbulence, energy spectra and flow structures are studied to understand the physics of flow, to expand experimental database, and to evaluate algebraic numerical models for flows in this regime. Their results show strong linear dependence of turbulence statistics on the surface roughness, and also, strong dependence of turbulent structures length scales and inclination angles on the roughness topographies. Muppidi & Mahesh (2012) analyzed the role of ideal distributed roughness on transition to turbulence in supersonic boundary layers. They have found that counter-rotating vortices, generated by the roughness elements, break the overhead shear layer, leading to an earlier transition to turbulence. A similar study was conducted by Bernardini et al. (2012), who investigated the role of isolated cubical roughness on boundary layer transition. Their results suggest that the interaction between hairpin structures, shed by the roughness element, and the shear layer expedites transition to turbulence, regardless of the Mach number.

1.3. Objectives

In this study we first introduce a level-set based immersed boundary method and validate it by comparing mean and turbulence statistics with a baseline simulation using a body-fitted mesh. Then we analyze the flow physics in supersonic channel flows at \( M = 1.5 \) and a bulk Reynolds number of 3000 (based on the channel half height) over two 2D and two 3D sinusoidal surfaces. Various mean-flow, turbulence and energy quantities are analyzed. Finally, terms in the transport equations of turbulence kinetic energy (TKE) and normal Reynolds stress in the streamwise direction are examined.

2. Problem formulation

2.1. Governing equations

The non-dimensional form of compressible Navier-Stokes equation are

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0, \tag{2.1a}
\]
\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j + p \delta_{ij} - \frac{1}{Re} \tau_{ij}) = f_1 \delta_{i1}, \tag{2.1b}
\]
\[
\frac{\partial E}{\partial t} + \frac{\partial}{\partial x_i} \left[ u_i (E + p) - \frac{1}{Re} u_j \tau_{ij} + \frac{1}{(\gamma - 1)PrReM^2} q_i \right] = f_1 u_1, \tag{2.1c}
\]

where \( x_1, x_2, x_3 \) (or \( x, y, z \)) are coordinates in the streamwise, wall-normal and spanwise directions, with corresponding velocities of \( u_1, u_2 \) and \( u_3 \) (or \( u, v \) and \( w \)). Density, pressure, temperature and dynamic viscosity are denoted by \( \rho, p, T \) and \( \mu \), respectively. \( E = p/\gamma - 1 \) + \( \rho u_i u_i/2 \) is the total energy, \( \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \) is the viscous stress tensor, and \( q_i = -\mu \frac{\partial T}{\partial x_i} \) is the thermal heat flux. \( f_1 \) is a body force that drives the flow in the streamwise direction, analogous to the pressure gradient. The reference Reynolds, Mach and Prandtl numbers are, respectively, \( Re \equiv \rho U_r L_r/\mu_r, \) \( M \equiv U_r/\sqrt{\gamma R T_r}, \) and \( Pr \equiv \mu_r C_p/\kappa, \) where subscript \( r \) stands for reference values (to be defined in section 2.3). The gas constant \( R \) and the specific heats \( C_p \) and \( C_v \) are assumed to be constant throughout the domain (calorically perfect gas). They are related by \( R = C_p - C_v, \) and the ratio of specific heats \( \gamma \equiv C_p/C_v \) is assumed to be 1.4. The heat conductivity coefficient is denoted by \( \kappa. \)
The set of equations in (2.1) is closed through the equation of state, which for a perfect gas is
\[ p = \frac{\rho T}{\gamma M^2}. \]  

Equations (2.1) and (2.2) are solved using a finite-difference method in a conservative format and a generalized coordinate system. A fifth-order monotonicity-preserving (MP) shock-capturing scheme and a sixth order compact scheme are utilized for calculating the inviscid and viscous fluxes respectively. The solver uses local Lax-Friedriches (LLF) flux-splitting method and employs an explicit third-order Runge-Kutta scheme for time advancement. Readers are referred to Li & Jaberi (2012) for extensive details and explanations about the compressible solver.

2.2. Details of the present IB method

The present IB method is a combination of level-set (Sussman et al. 1994; Gibou et al. 2018) and volume-of-fluid (VOF) (Scotti 2006) methods. It is designed for stationary interfaces only. The level-set field \( \psi(x, y, z) \) is defined as the signed distance from the fluid-solid interface. Based on the prescribed roughness geometry, the \( \psi \) field is obtained by diffusing an initial discontinuous marker function,

\[
\psi_0(x, y, z) = \begin{cases} 
1 & \text{in fluid cells,} \\
0 & \text{in interface cells,} \\
-1 & \text{in solid cells,}
\end{cases}
\]  

(2.3)

in the interface-normal direction until a narrow band along the interface, within which \( \psi \) is sign-distanced, is generated; this is similar to the reinitialization process conducted by Gibou et al. (2018). This is done by solving

\[
\frac{\partial \psi}{\partial \tau} = \text{sign}(\psi)(1 - |\nabla \psi|),
\]  

(2.4)

where \( \tau \) is a fictitious time controlling the width of the interface band. It is sufficient to march in (fictitious) time until a band width of up to 2-3 grid size is obtained.

Based on the level-set field, the VOF field, \( \phi(x, y, z) \), is constructed as

\[
\phi \equiv (1 + \psi)/2,
\]  

(2.5)

such that \( \phi = 0 \), \( 0 < \phi < 1 \), and \( \phi = 1 \) correspond to the solid, interface and fluid cells, respectively.

To impose the desired boundary condition for a test variable \( \theta(x, y, z, t) \), we correct the values of the variable before each time step, i.e.,

\[
\theta \rightarrow \phi \theta + (1 - \phi)\theta_b,
\]  

(2.6)

for Dirichlet BC and

\[
\frac{\partial \theta}{\partial n} = \nabla \theta \cdot \hat{n} = \frac{\partial \theta}{\partial n} \bigg|_b
\]  

(2.7)

for Neumann BC, where the subscript \( b \) denotes boundary values and \( \hat{n} \) is the unit normal vector pointing into the fluid region at the interface. \( \hat{n} \) is obtained as

\[
\hat{n} = \nabla \psi = \nabla \phi / |\nabla \phi|.
\]  

(2.8)

Note that \( \phi(x, y, z) \) does not represent exactly the fluid volume fraction in each grid cell. Instead, \( \phi \) is termed the VOF field because of the analogy between the BC imposition in
As will be shown in Section 2.4, the accuracy of the IB method herein is sufficient to produce matching single-point statistics compared to a simulation using body-fitted grid.

2.3. Surface roughnesses and simulation parameters

Fully developed, periodic compressible channel flows are simulated using four roughness topographies. The channels are roughened only at one surface (bottom wall) and the other surface is smooth. A reference smooth-wall channel is also simulated for validation and comparison purposes. The channel dimensions in streamwise, wall-normal and spanwise directions are, respectively, $L_x = 12\delta$, $L_y = 2\delta$ and $L_z = 6\delta$, where $\delta$ is the channel half-height.

Figure 1 shows four roughness topographies used for the present simulations. All cases share the same crest height, $k_c = 0.1\delta$. The trough location is set at $y = 0$. Case C1 and C2 are 2D sinusoidal surfaces with streamwise wavelengths of $\lambda_x = 2\delta$ and $\lambda_x = \delta$, respectively. The roughness heights, $k(x,z)$, for these surfaces are prescribed as

$$k(x,z)/\delta = 0.05[1 + \cos(2\pi x/\lambda_x)]. \quad (2.9)$$

Case C3 and C4 are 3D sinusoidal surfaces with equal streamwise and spanwise wavelengths of $(\lambda_x, \lambda_z) = (2\delta, 2\delta)$ for C3, and $(\lambda_x, \lambda_z) = (\delta, \delta)$ for C4. The roughness heights for them are prescribed as

$$k(x,z)/\delta = 0.05[1 + \cos(2\pi x/\lambda_x) \cos(2\pi z/\lambda_z)]. \quad (2.10)$$

Table 1 summarizes some statistical properties of the surface geometries. These statistics are various moments of surface height, surface effective slopes and porosity.

For a test variable $\theta$, the time, Favre and spatial averaging operators are denoted respectively by $\bar{\theta}$, $\bar{\theta} = \rho \bar{\theta}/\rho$ and $\langle \theta \rangle$ (intrinsic average (Raupach & Shaw 1982) over homogeneous directions $x$ and $z$), with corresponding fluctuation components of $\theta'$, $\theta''$ and $\theta'''$. In other words,

$$\theta = \bar{\theta} + \theta'$$

$$= \tilde{\theta} + \theta''$$

$$= \langle \theta \rangle + \theta'''.$$ 

(2.11)
is the height skewness, and $K_{rms}$ is the height kurtosis; where $k(x, z)$ is the roughness height distribution and $A_{f}(y)$ and $A_{i}$ are the fluid and total planar areas. Values of $k_{c}$, $k_{avg}$, $k_{rms}$ and $R_{a}$ are normalized by $\delta$.

| Case | $k_{c}$ | $k_{avg}$ | $k_{rms}$ | $R_{a}$ | $E_{x}$ | $E_{z}$ | $S_{k}$ | $K_{u}$ |
|------|---------|-----------|-----------|--------|--------|--------|--------|--------|
| C1   | 0.1     | 0.05      | 0.035     | 0.032  | 0.100  | 0.000  | 0.0    | 1.50   |
| C2   | 0.1     | 0.05      | 0.035     | 0.032  | 0.200  | 0.000  | 0.0    | 1.50   |
| C3   | 0.1     | 0.05      | 0.025     | 0.020  | 0.064  | 0.064  | 0.0    | 2.25   |
| C4   | 0.1     | 0.05      | 0.025     | 0.020  | 0.127  | 0.127  | 0.0    | 2.25   |

Table 1. Statistical parameters of roughness topography. $k_{avg} = \frac{1}{A_{t}} \int_{x,z} k dA$ is the average height, $k_{rms} = \sqrt{\frac{1}{A_{t}} \int_{x,z} (k - k_{avg})^{2}dA}$ is the root-mean-square (RMS) of roughness height fluctuation, $R_{a} = \frac{1}{A_{t}} \int_{x,z} |k - k_{avg}|dA$ is the first-order moment of height fluctuations, $E_{x_{i}} = \frac{1}{A_{t}} \int_{x,z} \| \frac{dk}{\partial x_{i}} \| dA$ is the effective slope in the $x_{i}$ direction, $S_{k} = \frac{1}{A_{t}} \int_{x,z} (k - k_{avg})^{3}dA / k_{rms}^{3}$ is the height skewness, and $K_{u} = \frac{1}{A_{t}} \int_{x,z} (k - k_{avg})^{4}dA / k_{rms}^{4}$ is the height kurtosis; where $k(x, z)$ is the roughness height distribution and $A_{f}(y)$ and $A_{i}$ are the fluid and total planar areas. Values of $k_{c}$, $k_{avg}$, $k_{rms}$ and $R_{a}$ are normalized by $\delta$.

Periodic BCs are used in the streamwise and spanwise directions. A no-slip iso-thermal wall BC is imposed at both top and bottom walls. The values of velocity and temperature on both walls (denoted by subscript $w$) are $u_{w} = 0$ and $T_{w} = 1$ (i.e. the temperature at the wall is used as the reference temperature). There is no need to impose a BC for density, and equation (2.1a) can be solved using one-sided differentiation to update the density values at the boundaries. This approach is similar to other wall-bounded compressible flow studies (see e.g. Coleman et al. 1995; Tyson & Sandham 2013). The pressure at the boundaries is calculated through the equation of state.

The reference density and velocity used in this work are those of bulk values, defined as $\rho_{r} = \frac{1}{2\delta} \int_{0}^{2\delta} \langle \rho \rangle dy$ and $U_{r} = \frac{1}{2\delta} \int_{0}^{2\delta} \langle u \rangle dy$. The reference length and temperature scales are $\delta$ and $T_{r} = T_{w}$, respectively. The time-dependent body force $f_{1}$ in NS equation (2.1) is adjusted automatically in each time step to yield the constant bulk velocity under the prescribed Reynolds number. The simulations are conducted at $Re = 3000$ and $M = 1.5$, assuming $Pr = 0.7$ and the dimensionless viscosity and temperature satisfy $\mu = T^{0.7}$.

The respective numbers of grid points in the $x$, $y$ and $z$ directions are $n_{x} = 800$, $n_{y} = 200$ and $n_{z} = 400$. For the present channel size and Reynolds number of the simulations, the spatial resolution yield $\Delta x$, $\Delta y_{max}$ and $\Delta z$ less than 3.0, which is fine enough for DNS. The first 3 grid points in the wall-normal direction are in the $y^{+} < 1.0$ region. The simulations are run in a sufficient amount of simulation time to reach the steady state. Thereafter the statistics are averaged over approximately 20 large eddy turn over time ($\delta/u_{r,avg}$, where $u_{r,avg}$ is an average value of the friction velocities on both walls (see table 2 for definition).

2.4. Validation of the numerical method and the IB method

The numerical method is validated by simulating supersonic turbulent channel flow over a smooth wall at $M = 1.5$ and $Re = 3000$. The same setup was employed by Coleman et al. (1995), which is used here as the benchmark study.

Figure 2 compares mean and turbulence statistics of the present simulation with those of Coleman et al. (1995). As the figure shows, the two simulations are in a good agreement for mean velocity, density and temperature, as well as for Reynolds stresses (no summation over Greek indices) and density variance. This verifies the numerical solver.

To validate the proposed IB method, we simulated case C1 in two ways: one using
Figure 2. Profiles of mean and turbulence variables for the smooth-wall flow at Re = 3000 and $M = 1.5$. Present simulation (solid lines), (Coleman et al. 1995) (dash line). $\tau_{w,s} = \frac{1}{Re} \langle \frac{\partial p}{\partial y} \rangle_w$.

Figure 3. Contour of level set $\phi$ for the IB method (a), mesh grid for the conformal setup (b). Case C1.

The IB method and the other solving the conventional NS equations on a body-fitted mesh setup. The contour of level set function $\psi$ for the IB method and the mesh of the conformal setup are compared in figure 3. The contour line corresponding to $\psi = 0$ represents well the fluid-solid interface.

Figure 4 compares the results obtained using IB and those using body-fitted mesh, in terms of various mean and turbulence variables, including mean velocity, temperature and density as well as Reynolds stresses and variance of temperature. All plots show a very good match between the two simulations. These results validate the use of the
Supersonic flows over sinusoidal rough walls

Figure 4. Mean and turbulence variables for case C1, simulated using the IB method (solid lines) and the conformal mesh (dash lines): Double-averaged velocity, temperature and density (a), RMS of velocity components in plus units (roughness side, b), time and spanwise average of velocity and temperature at the roughness crest and valley locations (c), and RMS of temperature (d). The vertical dot-dash lines show $y = k_c$.

3. Results

Contours of the instantaneous streamwise velocity field are plotted in figure 5 for all cases. Strong modifications of the near wall turbulence, especially on the rough-wall side, are noticeable from the figures. The main difference between the effects of different geometries is the shock patterns generated by 2D and 3D roughnesses, which are visible in contours of $\nabla \cdot \mathbf{u}$, shown in figure 6. As the figure shows, 2D surfaces (cases C1 and C2) impart strong shock patterns that go all the way up to the upper surface and reflect from this wall to the domain. These shock patterns exhibit the same wavelength of the roughness geometries, and influence the flow properties in the whole channel. This is obvious in the contours of instantaneous temperature fields in figure 7, where temperature periodically changes in the compression and expansion regions associated with roughness geometries in C1 and C2. For 3D cases the embedded shock patterns are broken and, consequently, replaced by the small-scale shocklets, which are distributed over the whole domain with an apparent lack of correlation with the roughness wavelengths.

3.1. Mean and turbulence variables

Figure 8 compares profiles of the mean and turbulence quantities between different test cases. The mean streamwise velocity (figure 8a) and density (not shown) show a similar trend for all cases, which, other than the near wall regions, are weakly dependent present IB.
on the roughness geometry across the channel height. The temperature profiles (figure 8c), on the other hand, depend on the roughness topographies, and are higher for 2D cases (C1 and C2) compared to the 3D ones (C3 and C4). As explained in section (2.1), the velocity and density are normalized by the bulk values $U_r$ and $\rho_r$, respectively; such normalization absorbs major differences in velocity and density profiles in the bulk part of the channel between different roughness cases. In comparison, the mean temperature in figure 8(c) is normalized by the wall value, $T_w$, which does not absorb the differences in the core region. Since the strong shocks in 2D cases involve more entropy in the domain than the 3D cases, the irreversible heat generation is higher for these cases and therefore the temperature is higher for them than for the 3D surfaces.

The values of frictional velocities on the smooth and rough sides as well as the frictional Reynolds number $Re_\tau$ and drag coefficient $C_f$ are tabulated in table 2. Both $Re_\tau$ and $C_f$ show similar trends to the temperature profiles: i) they are higher for 2D cases than 3D ones, ii) they decrease with decreasing roughness wavelength in 2D surfaces, and iii) they increase with decreasing roughness wavelengths in 3D surfaces. The associated shock patterns are believed to also be responsible for these trends as explained above.
Supersonic flows over sinusoidal rough walls

Figure 6. Contours of instantaneous $\nabla \cdot \mathbf{u}$.

| Case | $u_{\tau,s}/U_r$ | $u_{\tau,r}/U_r$ | $u_{\tau,avg}/U_r$ | $\text{Re}_\tau$ | $C_f \times 10^3$ |
|------|-----------------|-----------------|------------------|-----------------|-----------------|
| C1   | 0.0652          | 0.0721          | 0.0687           | 206             | 9.4             |
| C2   | 0.0660          | 0.0675          | 0.0668           | 200             | 8.9             |
| C3   | 0.0650          | 0.0577          | 0.0615           | 184             | 7.6             |
| C4   | 0.0657          | 0.0620          | 0.0639           | 191             | 8.2             |

Table 2. Post processing data. $u_{\tau,s} = \sqrt{\tau_{w,s}/\rho_r}$ and $u_{\tau,r} = \sqrt{\tau_{w,r}/\rho_r}$, where $\tau_{w,s} = -\mu_w \frac{d\bar{\mathbf{u}}}{dy} \bigg|_{y=2\delta}$ and $\tau_{w,r} = -\int_0^{\delta} \langle F_{1,ibm} \rangle_T dy$. Here $F_{1,ibm} = \rho \frac{\Delta u_i}{\Delta t}$ is the corresponding body force due to IBM ($\Delta u_i$ is the velocity difference of $u_i$ after and before the IBM correction step). $\langle \cdot \rangle_T$ is a simple planar averaging operator that includes all the solid and fluid cells. $\text{Re}_\tau = \rho_r u_{\tau,avg} \delta / \mu_w$, $C_f = 2(u_{\tau,avg}/U_r)^2$ and $u_{\tau,avg}^2 = (u_{\tau,s}^2 + u_{\tau,r}^2)/2$. 


The RMS of velocity components are plotted in figure 8(b) in viscous units, where they are normalized by $u_{r,r}$ (see table 2 for definition) and $\delta_{r,r} = \mu_w/(\rho u_{r,r})$. The plots show that roughness effects are confined to a near-wall region and outside this region the profiles almost collapse for all velocity components; this is similar to the roughness sublayer in an incompressible turbulence bounded by rough wall. Near the wall, the $v$ and $w$ components exhibit almost the same trend among all cases. However, the $u$ components for 3D cases display a peak closer to the wall than their 2D counterparts. Similar phenomenon was observed for incompressible flow also; it was explained as a result of more significant roughness effect exerted by the 2D roughness (due to a large length scale it imparts to the flow (Volino et al. 2011)), which leads to a peak farther from the smooth-wall peak elevation ($y^+ \approx 15$).

The RMS of temperature, figure 8(d), depends strongly on the roughness geometry in the outer layer. For 2D cases, the variations of curve shape in the bulk of the channel are associated with the shock patterns in the domain. Temperature varies significantly near the locations where the shock waves coincide and form nodes of shock diamonds (i.e. the nodes away from walls). These shock diamonds are also visible in figure 7 (C1 and
C2). For 3D cases the shock diamonds are weak or nonexistent. Therefore, the curves of temperature RMS in 3D cases are smooth in the core region.

3.2. Budgets of the Reynolds stresses

The transport equation for different components of the Reynolds stress tensor reads as (Vyas et al. 2019)

\[
\frac{\partial}{\partial t} \left( \rho u_i u_j \right) = C_{ij} + P_{ij} + D_{ij}^M + D_{ij}^T + D_{ij}^P + \Pi_{ij} + \epsilon_{ij} + M_{ij}, \tag{3.1}
\]

where \( i, j = \{1, 2, 3\} \) and \( C, P, D^M, D^T, D^P, \Pi, \epsilon \) and \( M \), are, respectively, mean convection, production, molecular diffusion, turbulent diffusion, pressure diffusion, pressure-
Figure 9. Budgets of TKE. All terms are normalized by $U_r$ and $\delta$, and are double-averaged in time and the $x$-$z$ plane.

The budget terms are calculated for all non-zero components of the Reynolds stress tensor and for TKE. The budget balance of the transport equation of $\langle u'_i u'_j \rangle$ is denoted as $B_{ij}$. Figures 9 and 10 show the wall-normal profiles of spatial averages of selective terms. Only the terms for TKE and $\langle u'_i u'_j \rangle$ budgets are shown for brevity. The figures are
normalized by the reference units for comparison purposes. In the figures, $\sigma$ is the residual of the budgets, and is less than 1% of the maximum value of the shear production $P$ in all cases. This verifies that budgets are calculated correctly and confirms that the numerical dissipation (as a result of the solver’s flux-splitting procedure and the IB method) is small.

The budget terms are strongly modified by the roughness elements in all figures. The contours of $P_{11}$ in an $(x, y)$ plane are shown in figure 11 to compare the spatial distribution of this term. The production terms in TKE and $B_{11}$ budgets are higher for 2D cases than for the 3D ones. There are several reasons for this. First, 2D roughness elements lead to more organized recirculation regions that are aligned in $z$. This is expected to generate stronger shear layers around the recirculation regions than in the wake of 3D elements. Also, the 2D roughness elements impose large-scale blockage spanning the width of the channel, augmenting the roughness effects on Reynolds stresses than 3D elements. The effects penetrate farther into the boundary layer also, as observed by (Volino et al. 2011) for incompressible boundary layers.

In addition, enhanced turbulence production for 2D surfaces is also due to the mutual interaction between shock waves. As figure 11 shows, for 2D cases, the regions where 2 oblique shock waves impinge together have enhanced turbulence production, whether it is on the rough or smooth wall side.

This is an important phenomenon and represents a fundamental difference between supersonic and subsonic turbulent flows over rough walls – for subsonic flows most of the roughness effects are confined to near wall regions and the outer layer is expected to be independent of the wall condition, also known as outer layer similarity (Townsend 1976). This has been verified in numerous studies in the field (Krogstad & Antonia 1999; Jiménez...
But for the supersonic cases here, as is obvious in figure 11, the effects of wall roughness, via the generated oblique shocks, propagate across the channel and modify turbulence production in the upper wall region. The same process occurs on the rough wall side, where the reflected shocks from the smooth side impinge back to the rough-wall side and enhance the turbulence production in these regions. In other words, turbulence processes on both sides depend on the interaction of shocks, which, are themselves dependent on the roughness topography. This shows that Townsend’s outer layer similarity hypothesis does not apply to such supersonic channel flows. The far-reaching effect of surface details may be of potential use in flow and turbulence control.

Similar to $\mathcal{P}$, other turbulence processes are also affected by wall roughness. This is reflected by the budget terms shown in figures 9 and 10. In particular, the pressure-strain term $\Pi$ in $B_{11}$ reaches its maximum magnitude near the roughness crest location, which indicates a peak of energy production in this region for $v$ and $w$ fluctuations, as the negative of this term acts as a prominent source term in the $B_{22}$ and $B_{33}$ budgets.
4. Concluding remarks

In this study we simulate supersonic turbulent flows over rough walls using a new level-set method. At the fluid-solid boundary, the velocity and temperature fields were corrected to impose the Dirichlet and Neumann boundary conditions, while density and pressure were calculated using the continuity and state equations. The method was validated by comparing results of two DNS simulations of channel flow with a sinusoidal wavy wall simulated using the IB method and a body-fitted mesh. Excellent agreement was found for mean and turbulence quantities.

Then, we simulated supersonic flows at $M = 1.5$, over four roughness topographies, two of which were 2D sinusoidal surfaces and two were 3D sinusoidal surfaces. The surfaces shared the same roughness height, but they differed in the surface wavelengths. Our results indicate strong modifications of turbulence field as well as the mean and RMS of the temperature fields by the roughness geometries. Specifically, 2D surfaces generate strong oblique shock patterns throughout the channel, which predominantly act to modify the turbulence production term, $\mathcal{P}$. In comparison, the 3D rough walls generate less coherent, randomly oriented weak shocklets. Contour plots of $\mathcal{P}$ show that roughness enhances shear production not only on the roughness side, but also in the inner-layer on the smooth wall side. In addition, the strong shocks generated by the 2D roughnesses impart higher entropy into the flow field, causing higher irreversible heat generation and higher temperature in the bulk of the channel. These observations are not consistent with the Townsend’s outer layer similarity hypothesis, which was found to apply to incompressible turbulent flows.

REFERENCES

Aghaei Jouybari, M., Brereton, G. J. & Yuan, J. 2019 Turbulence structures over realistic and synthetic wall roughness in open channel flow at $Re_\tau = 1000$. J. Turbul. 20, 723–749.

Bernardini, M., Pirozzoli, S. & P., Orlandi. 2012 Compressibility effects on roughness-induced boundary layer transition. Int. J. Heat Fluid Flow 35, 45 – 51.

Braslow, A. L. & Knox, E. C. 1958 Simplified method for determination of critical height of distributed roughness particles for boundary-layer transition at Mach numbers from 0 to 5. Tech. Rept.. National Advisory Committee for Aeronautics.

Chaudhuri, A., Hadjadj, A. & Chinnayya, A. 2011 On the use of immersed boundary methods for shock/obstacle interactions. J. Comput. Phys. 230 (5), 1731 – 1748.

Coleman, G. N., Kim, J. & Moser, R. D. 1995 A numerical study of turbulent supersonic isothermal-wall channel flow. J. Fluid Mech. 305, 159–183.

Ekoto, I. W., W. Bowersox, R. D., Beutner, T. & Goss, L. P. 2008 Supersonic boundary layers with periodic surface roughness. AIAA Journal 46, 486 – 497.

Fadlun, E. A., Verzicco, R., Orlandi, P. & Mohd-Yusof, J. 2000 Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. J. Comput. Phys. 161, 35–60.

Ghias, R., Mittal, R. & Dong, H. 2007 A sharp interface immersed boundary method for compressible viscous flows. J. Comput. Phys. 225, 528 – 553.

Gibou, F., Fedkiw, R. & Osher, S. 2018 A review of level-set methods and some recent applications. J. Comput. Phys. 353, 82 – 109.

Goldstein, D., Handler, R. & Shirovich, L. 1993 Modeling a no-slip flow boundary with an external force field. J. Comput. Phys. 105, 354–366.

Ji, Y., Yuan, K. & Chung, J. N. 2006 Numerical simulation of wall roughness on gaseous flow and heat transfer in a microchannel. Int. J. Heat Mass Transf. 49 (7), 1329 – 1339.

Jiménez, J. 2004 Turbulent flows over rough walls. Annu. Rev. Fluid Mech. 36, 173–196.

Kim, Y. & Peskin, C. S. 2007 Penalty immersed boundary method for an elastic boundary with mass. Phys. Fluids 19, 053103.
Krogstad, P.-Å. & Antonia, R. A. 1999 Surface roughness effects in turbulent boundary layers. *Exp. Fluids* **27**, 450–460.

Latin, R. M. 1998 The influence of surface roughness on supersonic high Reynolds number turbulent boundary layer flow. PhD thesis, School of Engineering of the Air Force Institute of Technology Air University.

Latin, R. M. & W. Bowersox, R. D. 2000 Flow properties of a supersonic turbulent boundary layer with wall roughness. *AIAA Journal* **38**, 1804 – 1821.

Latin, R. M. & W. Bowersox, R. D. 2002 Temporal turbulent flow structure for supersonic rough-wall boundary layers. *AIAA Journal* **40**, 832 – 841.

Li, Z. & Jaberi, F. A. 2012 A high-order finite difference method for numerical simulations of supersonic turbulent flows. *Int. J. Numer. Meth. Fl.* **68** (6), 740–766.

Luo, K., Mao, C., Zhuang, Z., Fan, J. & Haugen, N.E.L 2017 A ghost-cell immersed boundary method for the simulations of heat transfer in compressible flows under different boundary conditions part-ii: Complex geometries. *Int. J. Heat Mass Tran.* **104**, 98 – 111.

Marusic, I., McKeon, B. J., Monkewitz, P. A., Nagib, H. M., Smits, A. J. & Sreenivasan, K. R. 2010 Wall-bounded turbulent flows at high Reynolds numbers: Recent advances and key issues. *Phys. Fluids* **22** (6), 065103.

de’ Michieli Vitturi, M., Esposti Ongaro, T., Neri, A., Salvetti, M. V. & Beux, F. 2007 An immersed boundary method for compressible multiphase flows: application to the dynamics of pyroclastic density currents. *Comput. Geosci.* **11**, 183–198.

Mittal, R., Dong, H., Bozkurttas, M., Najjar, F. M., Vargas, A. & Von Loebbecke, A. 2008 A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries. *J. Comput. Phys.* **227** (10), 4825–4852.

Mittal, R. & Iaccarino, G. 2005 Immersed boundary methods. *Annu. Rev. Fluid Mech.* **37**, 239–261.

Muppidi, S. & Mahesh, K. 2012 Direct numerical simulations of roughness-induced transition in supersonic boundary layers. *J. Fluid Mech.* **693**, 28–56.

Radeztsky, R. H., Reibert, M. S. & Saric, W. S. 1999 Effect of isolated micron-sized roughness on transition in swept-wing flows. *AIAA Journal* **37**, 1370 – 1377.

Raupach, M. R. & Shaw, R. H. 1982 Averaging procedures for flow within vegetation canopies. *Bound.-Lay. Meteorol.* **22**, 79–90.

Reda, D. C. 2002 Review and synthesis of roughness-dominated transition correlations for reentry applications. *J. Spacecr. Rockets* **39**, 161–167.

Reda, D. C., Wilder, M. C., Bogdanoff, D. W. & Prabhu, D. K. 2008 Transition experiments on blunt bodies with distributed roughness in hypersonic free flight. *J. Spacecr. Rockets* **45**, 210 – 215.

Reshotko, E. & Tumin, A. 2004 Role of transient growth in roughness-induced transition. *AIAA Journal* **42**, 766 – 770.

Schneider, S. P. 2008 Effects of roughness on hypersonic boundary-layer transition. *J. Spacecr. Rockets* **45**, 193–209.

Scotti, A. 2006 Direct numerical simulation of turbulent channel flows with boundary roughened with virtual sandpaper. *Phys. Fluids* **18**, 031701–1–4.

Sussman, M., Smereka, P. & Osher, S. 1994 A level set approach for computing solutions to incompressible two-phase flow. *J. Comput. Phys.* **114**(1), 146 – 159.

Townsend, A. A. 1976 *The structure of turbulent shear flow*. Cambridge University Press.

Tseng, Y. H. & Ferziger, J. H. 2003 A ghost-cell immersed boundary method for flow in complex geometry. *J. Comput. Phys.* **192**, 593 – 623.

Tyson, C.J. & Sandham, N.D. 2013 Numerical simulation of fully-developed compressible flows over wavy surfaces. *Int. J. Heat Fluid Flow* **41**, 2 – 15.

Volino, R. J., Schultz, M. P. & Flack, K. A. 2011 Turbulence structure in boundary layers over periodic two- and three-dimensional roughness. *J. Fluid Mech.* **676**, 172–190.

Vyas, M.A., Yoder, D.A. & Gaitonde, D.V. 2019 Reynolds-stress budgets in an impinging shock-wave/boundary-layer interaction. *AIAA Journal* **57**(11), 4698–4714.

Wang, L., Currao, G.M.D., Han, F., J., Neely. A., Young, J. & Tian, F.B. 2017 An immersed boundary method for fluid-structure interaction with compressible multiphase flows. *J. Comput. Phys.* **346**, 131 – 151.
YUAN, R. & ZHONG, C. 2018 An immersed-boundary method for compressible viscous flows and its application in the gas-kinetic BGK scheme. *Appl. Math. Model.* **55**, 417 – 446.