International Communications in Heat and Mass Transfer 58 (2014) 54–62

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Optimised mixing and flow resistance during shear flow over a rib roughened boundary☆

A. Arfaie [a], A.D. Burns [a,*], R.M. Dorrell [b], J.T. Eggenhuisen [c], D.B. Ingham [a], W.D. McCaffrey [b]

[a] Energy Technology and Innovation Initiative (ETII), University of Leeds, Leeds LS2 9JT, UK
[b] School of Earth and Environment, University of Leeds, Leeds LS2 9JT, UK
[c] Department of Earth Sciences, Utrecht University, PO Box 80021, 3508 TA Utrecht, Netherlands

ARTICLE INFO

Available online 13 August 2014

Keywords:
Turbulent flow
Roughness
CFD

ABSTRACT

A series of numerical investigations has been performed to study the effect of lower boundary roughness on turbulent flow in a two-dimensional channel. The roughness spacing to height ratio, w/k, has been investigated over the range 0.12 to 402 by varying the horizontal rib spacing. The square roughness elements each have a cross-sectional area of $(0.05\ H)^2$, where H is the full channel height. The Reynolds number, Re_τ is fixed based on the value of the imposed pressure gradient, dp/dx, and is in the range $6.3 \times 10^3 - 4.5 \times 10^4$. A Reynolds Averaged Navier–Stokes (RANS) based turbulence modelling approach is adopted using a commercial CFD code, ANSYS-CFX 14.0. Measurements of eddy viscosity and friction factor have been made over this range to establish the optimum spacings to produce maximum turbulence enhancement, mixing and resistance to flow. These occur when w/k is approximately 7. It is found that this value is only weakly dependent on Reynolds number, and the decay rate of turbulence enhancement as a function of w/k ratio beyond this optimum spacing is slow. The implications for heat transfer design optimisation and particle transport are considered.

1. Introduction

The study of turbulent flow over surface roughness is important in a variety of engineering and environmental applications. Surface roughness is used as a tool to enhance heat transfer in turbines [1], heat exchangers [2], micro-scale electric mechanical systems [3], the hypervapotron (high heat flux) heat transfer device employed in nuclear fusion [4], chemical reactors, refrigeration systems and air conditioners [5]. Examples of rough-wall flows include particle transport in pipes and channels with rough walls, supersonic flows inside cavities for aerospace applications, wind flow over urban-like surfaces and turbidity currents over rough substrates [1,6–9]. In recent decades, a wide range of experimental and computational studies has been performed to understand the effect of surface roughness on the structure of the turbulent flow. The computational domain and experimental configuration of these studies are typically a two-dimensional or three-dimensional rectangular channel flow with roughness on one or both walls [10–23]. The effect of surface roughness on the flow, as reviewed by [24] and more recently by [25], is often separated into three different regimes. Chow [26] was first to identify three flow regimes over beam-type roughness as quasi-smooth or skimming flow, wake-interference flow and isolated-roughness flow. Perry et al. [27] categorised two distinct types of roughness, namely, "d" and "k" denoting channel height and roughness height, respectively (see below), following from the earlier experimental work conducted by Nikuradse [28] on the turbulent flow of fluids in rough pipes.

The roughness type can be correlated to the spacing to height ratio of a roughness element, w/k. The roughness spacing is differently defined as either the distance between roughness faces w, or the distance between roughness-element centre-lines λ; values differ by unity for square ribs. Therefore one must be careful not to confuse the cavity width to height ratio w/k to the pitch to height ratio λ/k.

For a sufficiently low width to height ratio, $w/k \lesssim 2$, or d-type roughness, the flow undergoes a "skimming flow" regime and the effective height, y_l above the channel bed where the velocity profile begins to take a logarithmic shape becomes independent of the roughness height, k. In this flow regime there is minor shedding or interaction from the vicinity of the roughness element to the outer flow region [22,29,30]. The k-type roughness (isolated-roughness flow regime) is associated with $w/k \gtrsim 4$. The roughness height becomes a crucial parameter for $w/k \gtrsim 4$ when the flow in the roughness cavity begins to interact with the main body of the flow. For this roughness type, the origin of the logarithmic profile, y_l is proportional to the roughness height, k and the flow regime are characterised by separation occurring at the crest of the first roughness element followed by a reattachment within the distance away from the next adjacent element. The experimental study of Djenidi et al. [22] suggested a similarity in the quasi streamwise vortices and low-speed streaks of the roughened wall cases, to a flat turbulent boundary layer. Tani [31] found the demarcation line between

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: a.d.burns@leeds.ac.uk (A.D. Burns).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.08.005
0735-1933/

Nomenclature

$\frac{\partial p}{\partial x}$	mean pressure gradient
$\overline{u_i' u_j'}$	Reynolds stress
p_0	reference pressure point

Greek variables

ε	dissipation rate of K
μ	dynamic viscosity
μ_{eff}	effective viscosity
μ_t	turbulent viscosity
ν, ν_t	kinematic viscosity, eddy viscosity
ρ	density
τ_w	wall shear
ω	turbulent eddy frequency

Roman variables

C_f	skin friction coefficient $\left(=\frac{\tau_w}{\frac{1}{2}\rho \overline{U}_b^2}\right)$
C_p	pressure coefficient $\left(=\frac{p-p_0}{\frac{1}{2}\rho \overline{U}}\right)$
f	Darcy friction factor $\left(=\frac{(H/2)(dp/dx)}{0.5\rho \overline{U}^2}\right)$
H	full channel height
K	turbulence kinetic energy
k	roughness height
k^+	dimensionless roughness height
p	pressure
Re_τ	$u_\tau(H/2)/\nu$, shear Reynolds number
u_τ	shear velocity $\left(=\sqrt{\tau_w/\rho}\right)$
U_b	bulk velocity
w	width of the cavity
x	streamwise direction
y	wall-normal direction
y^+	non-dimensional distance to the wall $\left(=\frac{yu_\tau}{\nu}\right)$
y_l	the origin of the logarithmic profile

Subscripts

i, j	coordinate direction 1, 2 or 3
max	maximum of variable
min	minimum of variable
rms	root mean square value of the variable

the d-type and the k-type roughness occurs at $w/k = 4$. Cui et al. [16] observed a similar transition for $w/k = 4$ and named this roughness type as intermediate. This transition flow regime corresponds to with the wake interference flow regime classified by Chow [26]. In this regime a weak interaction between the inner and outer roughness layers occurs and the reattachment takes place at the crest of the next roughness element. The direct numerical simulation (DNS) of [29] showed that the intermediate regime appears within the range $3 < w/k < 7$.

In a fully rough flow, the ratio of the product of the roughness height and shear velocity to the kinematic viscosity of the fluid k^+ ($k^+ = ku_\tau/\nu$), is greater than ≈ 70 and the pressure drag component of the total drag dominates the viscous drag component. In this flow regime the flow characteristics are only dependent on the roughness spacing to height ratio w/k. Hence, the viscous length scale (ν/u_τ) near the wall scale becomes irrelevant [20,32].

Orlandi et al. [33] and Leonardi et al. [30] found similarity in the vortex shedding distribution between the intermediate and k-type roughness. Therefore, they suggested that classification of different roughness types should not be based on the state and intensity of vortex shedding. Instead, they related the transition between d-type and k-type to the magnitude of the viscous and pressure drags.

Both LES and DNS numerical modelling of rough-wall flows have proven to be highly accurate in predicting the turbulent kinetic energy and Reynolds stresses in the near-wall region. However, in order to capture most of the flow characteristics within the roughness sub-layer, a higher grid resolution and time step accuracy are required than in a normal smooth-wall case. This makes such approaches expensive, particularly for high Reynolds number flows. Leonardi et al. [29] used DNS to investigate the effect of the w/k ratio on the turbulence structure near the wall, and its overlying flow by considering two-point velocity correlations. They observed that in the fully rough regime, with the increase in the w/k ratio, the coherence structure becomes less elongated in the streamwise direction, and larger in the spanwise direction as a result of outward jets of fluid at the leading edge of the roughness element. Such coherence structure would appear to be less influenced by the rough wall in the transition regime ($k^+ \simeq 13$), as observed by [34]. The maximum strength of the outward jet and the minimum reduction of the coherence occurred at the critical value $w/k = 7$. They further found that the influence of roughness can extend up to 2 k above the roughness crest for $w/k = 3$ and up to 5 k for $w/k = 7$. The study conducted by [16], for a channel with transverse rib roughness on one wall, suggests a strong interaction between the inner and outer layer roughness for k-type roughness.

Numerous authors have performed numerical and experimental analyses to investigate the relationship between heat transfer and fluid flow behaviour by varying the w/k ratio [35–38]. However most of these investigations suffer from a lack of a detailed range of w/k ratio and Reynolds number. The most detailed study was performed experimentally by [39,40] for boundary layer fluid flow. [39] investigated the maximum resistance of the turbulent boundary layer in a plate roughened by equally spaced wires. They found that the maximum skin drag coefficient, c_f and pressure coefficient, c_p values appear at $w/k = 7$. However the DNS result of [29] suggests that minimum c_f occurs at $w/k = 7$, but agrees with the maximum pressure coefficient c_p occurring at this w/k ratio. The experimental study by [40] has shown that the maximum heat transfer occurs when the turbulence intensity is maximised. They have shown that the maximum flow resistance occurs between $w/k = 6$ and $w/k = 8$. This paper aims to explicitly identify where the optimum flow resistance occurs for a more detailed range of w/k ratio as a function of Reynolds number.

In the present study, we employ a RANS method to simulate turbulent flow in a two-dimensional channel with an asymmetric two-dimensional rough lower boundary for a wide range of ratio w/k and Reynolds numbers. In this paper, we attempt to accurately constrain the critical w/k ratio for an optimum turbulence enhancement, mixing and resistance to the flow. For this purpose, we evaluate the dependence of eddy viscosity and friction factor on Reynolds number for a series of w/k values. The aims of this research are to better constrain optimum conditions for heat transfer, and to assess lower boundary roughness effects on turbidity current turbulence generation, flow depletion and runout.

The paper is organised as follows. Sections 2 and 3 give brief description of the numerical procedure and flow configuration. In Section 4 we validate our model with previous experimental and numerical data. The results of the numerical modelling are given in Section 5 and discussed in Section 6.

2. Numerical method

2.1. Turbulence modelling

Steady state CFD simulations have been performed using the commercial code, ANSYS CFX 14.0. This code uses a finite volume method