International Journal of Thermal Sciences 135 (2019) 302–318

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

# Large-Eddy Simulations of turbulent flow through a heated square duct

A. Schindler[a], B.A. Younis[b,*], B. Weigand[a]

[a] Institut für Thermodynamik der Luft- und Raumfahrt, Universität Stuttgart, Stuttgart, Germany
[b] Department of Civil & Environmental Engineering, University of California, Davis, USA

ARTICLE INFO

*Keywords:*
LES
Square duct
Heat transfer
Thermal log-law
High-efficiency cooling systems

ABSTRACT

The behavior of the flow through a straight square duct is examined using Large-Eddy Simulations. Both isothermal and non-isothermal conditions were considered, the latter generated with either the wall temperature or the wall heat flux held constant. Fully-developed conditions were attained by initializing the velocity field with perturbed streamwise streaks, and by employing a method for efficiently applying cyclic boundary conditions to the velocity field while minimizing aliasing errors. Results were obtained for two values of bulk Reynolds numbers, $Re_b$ = 6,000 and 10,000. The numerical accuracy was checked via several alternative methods that included performing computations on grids with different resolutions, both with and without sub-grid scale models. The results were used to test some of the assumption underlying the use of RANS approaches to predict the flow and thermal fields. Of particular interest was the examination of the effects of the turbulence-driven secondary motions on the near-wall processes, especially on existence and extent of the thermal logarithmic law of the wall, and on departures from the Reynolds analogy. Their effect on the turbulent Prandtl number was quantified, and the implications on the use of Fourier's law to relate the turbulent heat fluxes to the gradients on mean temperature are discussed.

## 1. Introduction

The turbulent flow of a fluid in a square duct is of practical interest for its frequent occurrence in engineering practice such as in high performance heat exchangers and in the cooling passages in gas-turbine blades [1]. It is also of fundamental interest due to the presence of turbulence-driven secondary flows that impact the rates of transfer of heat and momentum leading to distortion of the mean flow field, and to non-uniform distributions of shear stresses and heat fluxes on the duct walls. The prediction of these flows has exposed the limitations of many of the most-widely used RANS approaches for the prediction of turbulent flows. These approaches, being based on Boussinesq's assumption of linear stress-strain relationship, predict, in fully-developed flow conditions, isotropy of the normal stresses. Since it is the anisotropy of these stresses that provides the sole mechanism for generating the secondary flows, these are not obtained leading to significant errors in the prediction of important parameters such as the pressure drop and the overall heat-transfer rate [2]. Models that are based on non-linear stress-strain relationships (e.g. Refs. [3] [4]), or that involve the solution of modeled differential transport equations for all six non-zero components of the Reynolds-stress tensor ([5]) are more successful in capturing the details of the secondary flows but uncertainties remain, especially in predicting the consequences of these motions on the thermal field.

In contrast to conventional turbulence closures, the use of Large-Eddy Simulations (LES) to predict the behavior of turbulent flows involves fewer modeling assumptions and can thus potentially provide more reliable means for engineering predictions. Several studies of flows in square ducts with LES have been reported in the literature. For isothermal flows, these include the studies of Madabhushi and Vanka [6] and Breuer and Rodi [7]. For non-isothermal flows, results have been reported by Pallares and Davidson [8], Vázquez and Métais [9], Qin and Pletcher [10] and Zhu et al. [11]. Recently, Direct Numerical Simulations (DNS) have been used to study the fundamental flow physics associated with turbulence-driven secondary motions in ducts of increasing aspect ratio ([12]). The computations, which were performed using a Spectral-Element Method with a number of nodes of up to $326 \times 10^6$, revealed a degree of complexity arising from the interaction of bursting mechanisms from horizontal and vertical walls that would be hard to capture with either LES or RANS. Within the constraints of computational resources, the LES approach will probably remain the advanced design tool of choice for the foreseeable future.

In this paper, we make comparisons with some of the previously-reported LES results and proceed further in investigating in far more

* Corresponding author.
*E-mail address:* bayounis@ucdavis.edu (B.A. Younis).

https://doi.org/10.1016/j.ijthermalsci.2018.09.018
Received 15 May 2018; Received in revised form 10 September 2018; Accepted 11 September 2018
1290-0729/ 

detail the sensitivity of the predictions to some of the assumptions and practices underlying the use of LES in the prediction of heat transfer in square ducts. Amongst the issues examined is the intersection between grid resolution and the models used to approximate the sub-grid scale fluxes of momentum and thermal energy. We also examine the efficacy of methods used to generate initial conditions that lead to the rapid establishment of a sustainable turbulence field, and of methods for incorporating cyclic boundary conditions that minimize aliasing errors. Thus one objective of this research was to contribute a reliable set of results to the rather limited literature related to the use of LES in the prediction of non-isothermal flows in non-circular ducts. Another objective stems from the recognition that the task of obtaining numerically-accurate LES results within reasonable turn-around times is not trivial and in many cases precludes the use of this methodology for routine engineering design. There is thus some benefit in using LES to obtain descriptions of the flow details that are difficult to obtain via measurements, and to use these to examine some of the assumptions underlying the more practical RANS approaches. Amongst these is the assumption of the existence of logarithmic laws of the wall for momentum and temperature to obtain wall boundary conditions for these parameters (the 'wall functions' approach), and the assumption that the Reynolds analogy connecting the rates of heat and momentum transport via a constant Prandtl number holds even in the presence of anisotropy-driven secondary motions. We use our results to test both of these assumptions.

## 2. Mathematical formulation and computational details

The methodology underlying the Large-Eddy Simulations approach to modeling turbulent flows is well know (see, for example [13,14]). Briefly, the instantaneous equations governing the conservation of mass, momentum and energy are filtered to yield:

$$\frac{\partial \overline{U_i}}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{U_i}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{U_i}\,\overline{U_j}) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\frac{\partial U_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_i} \tag{2}$$

$$\frac{\partial \overline{\Theta}}{\partial t} + \frac{\partial(\overline{U_j}\,\overline{\Theta})}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\Gamma\frac{\partial \overline{\Theta}}{\partial x_j}\right) - \frac{\partial \tau_{\Theta j}}{\partial x_j} \tag{3}$$

We use the Smagorinsky [15] model for the sub-grid scale stresses:

$$\tau_{ij} = -\nu_{sgs} 2\overline{S}_{ij} \tag{4}$$

where $\nu_{sgs}$ is the turbulent eddy viscosity and $\overline{S}_{ij}$ is the resolved strain rate tensor. The turbulent eddy viscosity is obtained from

$$\nu_{sgs} = (C_S\Delta)^2|\overline{\boldsymbol{S}}| \tag{5}$$

where $\Delta$ is a length scale obtained from

$$\Delta = (\Delta_x\Delta_y\Delta_z)^{\frac{1}{3}} \tag{6}$$

is used, where $\Delta_x$, $\Delta_y$, $\Delta_z$ is the grid spacing in $x$, $y$, $z$, respectively.

Several different values for the Smagorinsky constant $C_S$ have previously been used in this and similar flows. In this work, $C_S$ was set equal to 0.065 which is in line with the recommendations of [16,17] for wall-bounded flows.

The Smagorinsky model with a constant $C_S$ tends to overpredict the viscosity in close proximity to the wall [18]. Following the usual practice, we use van Driest's damping function to bring about the correct behavior:

$$\nu_{sgs} = (C_S\Delta)^2|\overline{\boldsymbol{S}}|\left(1 - e^{-y^+/A^+}\right) \tag{7}$$

where $y^+ = \frac{u_\tau y}{\nu}$ is the non-dimensional wall distance and $A^+ = 26$.

The unresolved heat fluxes are modeled according to [13,17,19] as

$$\tau_{\Theta j} = -\Gamma_{sgs}\frac{\partial \Theta}{\partial x_j} \tag{8}$$

where

$$\Gamma_{sgs} = \frac{\nu_{sgs}}{\mathrm{Pr}_{sgs}} \tag{9}$$

is the thermal sub-grid scale diffusivity, $\nu_{sgs}$ is the sub-grid scale eddy viscosity in Eq. (7) and $\mathrm{Pr}_{sgs}$ is the sub-grid scale Prandtl number which was set equal to 0.4 in line with the recommendations of [19,20].

The simulations were performed using the open source CFD software OpenFOAM. Fluid properties for dry air at 293.15 K were assumed to be constant for both isothermal and heated flow conditions thus the temperature variations did not influence the flow field. Turning to details of the discretization schemes, a second-order accurate backward differencing scheme is used for time integration. The time-step size was restricted according to the Courant - Friedrichs - Lewy number

$$CFL = \triangle t\ \max\left(\frac{|U|}{\triangle x} + \frac{|V|}{\triangle y} + \frac{|W|}{\triangle z}\right) \tag{10}$$

To ensure stability, $CFL$ was set equal to 0.6 to ensure time accurate and stable results. Concerning the spatial discretization, second-order linear schemes were used for both the diffusive and convective fluxes. It should be noted that a detailed study of the fundamental physical processes of turbulence-driven secondary flows would demand the use of a higher-order accurate discretization scheme such as the Spectral Element Method used by Marin et al. [2] to examine the nature of flows in hexagonal ducts. Coupling of the continuity and momentum equations was done iteratively using the PISO algorithm.

The boundary conditions employed were as follows. At the walls, the no-slip condition was applied for the velocities while the wall-normal pressure gradient was set equal to zero. At the outlet, the pressure and the streamwise gradients of the axial velocity were set to zero. At inlet, the internal mapping procedure of de Villiers [18] was used. An arbitrary plane close to the outlet was identified from where the computed velocities were extracted at each time step to be inserted at the inlet. Placing this arbitrary plane at some distance upstream of the outlet prevents spurious errors due to reflections from the outlet from being propagated back to the inlet. In the present work the mapping plane was located at a distance of $6.0D_h$ from the inlet plane; the total length of the channel being $6.4D_h$. Subsequent tests confirmed that the feedback effects were minimal. The mapping length of $6.0D_h$ is in-line with the recommendations of Pallares and Davidson [8] who performed calculations in the same $\mathrm{Re}_b$ regime. Other researchers (e.g. Refs. [21,22]) recommended longer domains but preliminary tests showed that this is not necessary.

Several methods for initializing the velocity field were considered. The one which proved most effective in producing a realistic and sustainable turbulence field was the method proposed by de Villiers [18] which is based on the initialization of the velocity field with a transitional-like state. A laminar mean profile ($U_0$) was superimposed by artificially created streamwise streaks to yield an instantaneous velocity ($U$) that defines parallel low- and high-speed streaks:

$$U = U_0 + u_\tau\frac{\triangle U^+}{2}\cos(\beta^+ z^+)\frac{y^+}{40}\exp(0.5 - (y^+)^2\sigma)C_{as} \tag{11}$$

where $\triangle U^+ = 0.25U_b/u_\tau$, $\sigma = 5.5 \times 10^{-4}$ and $\varepsilon = U_b/200$. The friction velocity $u_\tau$ was estimated using the formulation of Jones [23]. This leads to $\mathrm{Re}_\tau(=u_\tau D_h/2\nu)$ of 196.5 for the case of $\mathrm{Re}_b(=U_bD_h/\nu)$ of 6,000 and $\mathrm{Re}_\tau = 306$ for the case of $\mathrm{Re}_b = 10{,}000$. The value $\beta^+ = 2\pi/200$ determines the spanwise streak spacing. The constant $C_{as} = 1.1$ introduces a small random deviation in order to break the symmetry. Then streak waviness is introduced in order to perturb the streaks and produce streamwise vortices

$$W = \varepsilon\sin(\alpha^+ x^+)y^+\exp(-(y^+)^2\sigma)C_{as} \tag{12}$$