International Journal of Heat and Mass Transfer 52 (2009) 1690–1700

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Three-dimensional simulation of turbulent forced convection in a duct with backward-facing step

H. Lan, B.F. Armaly *, J.A. Drallmeier

Department of Mechanical and Aerospace Engineering Missouri University of Science and Technology, 400 West 13th Street, Rolla, MO 65409, USA

ARTICLE INFO

Article history:
Received 11 April 2008
Received in revised form 30 September 2008
Available online 10 December 2008

ABSTRACT

Results from three-dimensional (3-D) simulations of turbulent forced convection adjacent to backward-facing step in a rectangular duct using a $k-\epsilon-\zeta-f$ turbulence model are reported. This turbulence model is numerically robust near the wall, and it has been shown to predict turbulent heat transfer in separated and wall-bounded flows better than commonly used two-equation turbulence models. FLUENT-CFD code is used as the platform for these simulations and User Defined Functions (UDF) are developed for incorporating this turbulence model into the code. The UDF implementation is validated by simulating several 2-D separated flow/heat transfer benchmark problems. The resulting excellent agreements between the simulated results and benchmark data for these 2-D problems justify the use of this resource for simulating 3-D convection problems. Three-dimensional backward-facing step geometry with an expansion ratio of 1.48 and with a step height of 4.8 mm is used in this study. Three aspect ratios of 3, 8 and infinity (2-D simulation) are considered for studying its effect on the flow and heat transfer, and similarly the effect of the Reynolds number was examined by varying its magnitude in the range of 20,000–50,000. Simulated results are presented for the general 3-D flow features, the reattachment lines, temperature and Nusselt number distributions that develop in this geometry.

1. Introduction

Flow separation and reattachment that are caused by a sudden change in geometry occur in many heat-exchanging devices, and 3-D effects are significant in these applications. The heat transfer rate varies greatly in the recirculation flow region due to the 3-D flow behaviors, and accurate heat transfer prediction could improve and optimize the design of such devices. The separated flow over a backward-facing step has received a great deal of attention, and lots of 2-D laminar [1–2], 2-D turbulent [3–6] and 3-D laminar [7–14] flow and heat transfer studies have been published for this geometry. Relatively few numerical simulations for 3-D turbulent flow and convective heat transfer [15–17] have been published, and the need for a more reliable turbulence model to accurately predict the turbulent heat transfer has been cited in the above references. To the authors' knowledge, simulations of turbulent forced convection in 3-D separated flow using recent and improved turbulence models have not appeared in the literature and that motivated this study.

The $k-\varepsilon$ turbulence model has been used extensively in simulating turbulent flow, but it has been shown to fail in accurately predicting heat transfer in separated flows. Nie and Armaly [16] have reported that simulations using the Low-Re $k-\varepsilon$ turbulence model resulted in reasonable agreement with flow measurements but poor agreement with heat transfer measurements. It was also reported in [4] that commonly used two-equation turbulence models generate inaccurate predictions for normal Reynolds stress due to the use of isotropic eddy viscosity. Recent turbulence models, which account for turbulence anisotropy, such as $\bar{v}^2-f$ model and its modified versions [18–19], have improved significantly the heat transfer prediction in separated flows. As a subset of that model, the $k-\varepsilon-\zeta-f$ turbulence model [20] offers the benefit of being numerically robust and has been shown to simulate well the heat transfer in a variety of 2-D turbulent separated flow problems; hence the turbulence model is utilized in this 3-D study.

2. Description of turbulence model

The $k-\varepsilon-\zeta-f$ turbulence model was developed from the original $\bar{v}^2-f$ model [18] by replacing the $\bar{v}^2$-equation with a ζ-equation where $\zeta=\bar{v}^2/k$, the ratio of wall-normal Reynolds stress to turbulent kinetic energy. It has been shown that the new equation for ζ is more robust numerically than the $\bar{v}^2$ equation and that results in improved numerical stability [20]. A quasi-linear pressure-strain model is also applied in the f-equation with additional improvements for simulating non-equilibrium wall-bounded flows [20].

The governing equations for turbulent forced convection together with the $k-\varepsilon-\zeta-f$ turbulence model [20] and the corresponding boundary conditions are listed below:

* Corresponding author. Tel.: +1 573 341 4601; fax: +1 573 341 4609.
E-mail address: armaly@mst.edu (B.F. Armaly).

0017-9310/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2008.09.022

Nomenclature

C_p	thermal specific heat (J kg^{-1} K^{-1})
AR	aspect ratio, $AR = 2L/h$
D	nozzle diameter (m)
ER	expansion ratio, $ER = H/h$
f	elliptic relaxation function
G_k	turbulent kinetic energy production (times density) (kg m^{-1} s^{-3})
h	duct height upstream of the step (m)
H	duct height downstream of the step (m)
k	turbulent kinetic energy (m^2 s^{-2})
k^+	non-dimensional turbulent kinetic energy, $k^+ = k/u_\tau^2$
L	half of the duct width (m)
L_t	turbulent length scale (m)
Nu	local Nusselt number, $Nu = q_{\mathrm{w}D}/\lambda(\bar{T}_{\mathrm{w}} - \bar{T}_{\mathrm{in}})$
Nu_b	bulk Nusselt number, $Nu_{\mathrm{b}} = q_{\mathrm{w}D}/\lambda(\bar{T}_{\mathrm{w,avg}} - \bar{T}_{\mathrm{b}})$
Pr_t	turbulent Prandtl number
q_w	heat flux (W m^{-2})
r	radius from the jet impingement center point (m) Re_τ Reynolds number, $u_\tau\delta/v$
Re_h	Reynolds number, $Re_h = U_b h/v$
S	step height (m)
$\bar{S}$	strain rate magnitude (s^{-1})
S_{ij}	strain rate tensor (s^{-1})
St	stanton number, $St = q_{\mathrm{w}}/[\rho U_{\mathrm{ref}} C_{\mathrm{p}}(\bar{T}_{\mathrm{w}} - \bar{T}_{\mathrm{in}})]$
T	turbulent time scale (s)
$\bar{T}$	local temperature (K)
$\bar{T}_b$	bulk flow temperature $\bar{T}_b = \int\int U\bar{T}\mathrm{d}y\mathrm{d}z/\int\int U\mathrm{d}y\mathrm{d}z$ (K)
$\bar{T}_{w,avg.}$	averaged bottom wall temperature, $\bar{T}_{\mathrm{w,avg.}} = \int_0^L \bar{T}_{\mathrm{w}}\mathrm{d}z/L$ (K)
U_b	bulk average velocity (m s^{-1})
U_i	velocity component (m s^{-1}) (U_i is U, V, W in X-, Y-, Z-direction, respectively)
U_{ref}	reference velocity (m s^{-1})
u_τ	friction velocity, $u_\tau = \sqrt{\tau_w/\rho}$ (m s^{-1})
$\bar{v}^2$	wall-normal Reynolds stress component (m^2 s^{-2})
x_i	coordinate vector component (m)
X_r	reattachment point (m)
X_s	separation point (m)
y	distance from the wall (m)
y^+	non-dimensional wall coordinate, $y^+ = yu_\tau/v$

Greek symbols

δ	half channel height (m)
ε	turbulent dissipation rate (m^2 s^{-3})
λ	thermal conductivity (W m^{-1} K^{-1})
μ_t	turbulent viscosity (kg m^{-1} s^{-1})
v	kinematics viscosity (m^2 s^{-1})
ρ	density (kg m^{-3})
ζ	$\bar{v}^2/k$

Subscripts

in	inlet of the computational domain
w	wall
avg.	averaged value

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

$$\frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] \tag{2}$$

$$\frac{\partial}{\partial x_j}(\rho U_j \bar{T}) = \frac{\partial}{\partial x_j}\left(\left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr}\right)\frac{\partial \bar{T}}{\partial x_j}\right) \tag{3}$$

$$\frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\epsilon \tag{4}$$

$$\frac{\partial}{\partial x_j}(\rho U_j \epsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{1.3}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \frac{C_{\epsilon 1}G_K + 1.9\rho\epsilon}{T} \tag{5}$$

$$\frac{\partial}{\partial x_j}(\rho U_j \zeta) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{1.2}\right)\frac{\partial \zeta}{\partial x_j}\right] + \rho f - \frac{\zeta G_k}{k} \tag{6}$$

$$L_t^2\nabla^2 f - f - \frac{1}{T}\left(0.4 + \frac{0.65 G_k}{\rho\epsilon}\right)\left(\zeta - \frac{2}{3}\right) = 0 \tag{7}$$

where

$$\mu_t = 0.22\zeta k T \tag{8}$$

$$C_{\epsilon 1} = 1.4(1.0 + 0.012/\zeta) \tag{9}$$

$$T = \max\left[\min\left(\frac{k}{\epsilon}, \frac{0.6}{0.22\sqrt{3}\zeta\bar{S}}, 85\frac{v^3}{\epsilon}\right)^{0.25}\right] \tag{10}$$

$$L_t = 0.36\max\left[\min\frac{k^{1.5}}{\epsilon}, \frac{k^{0.5}}{0.22\sqrt{3}\zeta\bar{S}}, 85\left(\frac{v^3}{\epsilon}\right)^{0.25}\right] \tag{11}$$

$$\bar{S} = \sqrt{2_{ij}S_{ij}} \tag{12}$$

$$S_{ij} = \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \tag{13}$$

$$G_k = \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} \tag{14}$$

where T and L_t are turbulent time scale and turbulent length scale respectively. In this model, a constant turbulent Prandtl number $Pr_t = 0.9$ is used when solving the energy equation. The wall boundary condition is fairly simple. All the variables vanish to zero when a wall is approached except for ε and f:

$$\epsilon_w = \lim_{y\to 0}\frac{2vk}{y^2}, \quad f_w = \lim_{y\to 0}\frac{-2v\zeta}{y^2} \tag{15}$$

User Defined Functions (UDFs) were developed and added to the FLUENT-CFD code for solving the four turbulence model equations (Eqs. (4)–(7)) together with other governing equations for the flow and thermal fields. Although the source term in Eq. (4) is theoretically exact, some treatment is needed in the numerical realization stage of the simulation. The turbulent dissipation rate, ε which appears as a source term in that equation, is numerically realized by $\epsilon_{i,j}^n = k_{i,j}^n/T_{i,j}^{n-1}$ instead of using its value from the previous iteration as $\epsilon_{i,j}^n = \epsilon_{i,j}^{n-1}$. The use of either one of these expressions produces similar flow results; however, using the former numerical expression in the simulation produces much better heat transfer results that agree well with reported measurements. This treatment is similar to the one that has been utilized in the process of developing Eq. (5) from the ε-equation that appears in the standard $k - \varepsilon$ turbulence model.

The governing equations are solved by a segregated solver, and the SIMPLEC algorithm is used to deal with the coupling between the flow field and pressure field. The PRESTO! scheme is used for pressure correction equation, and the QUICK scheme is used for all other equations (see the FLUENT manual for details [21]).

3. Model validation

The developed UDF code for this turbulence model is tested and validated by comparing simulated results with available 2-D benchmark data. The results for four of these benchmark cases are presented below. Properties for air that are used in the numerical validation are evaluated at a static temperature of 293 K and are listed as: density $\rho = 1.225$ kg/m^3; molecular