International Journal of Heat and Mass Transfer 77 (2014) 406–418

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Turbulent heat transfer in a two-pass cooling channel by several wall turbulence models

R.S. Amano [a,*], H. Arakawa [a,b], K. Suga [b]

[a] Department of Mechanical Engineering, University of Wisconsin-Milwaukee, Milwaukee, WI 53201, USA
[b] Department of Mechanical Engineering, Osaka Prefecture University, Sakai, Osaka 599-8531, Japan

ARTICLE INFO

Article history:
Received 12 January 2014
Received in revised form 27 April 2014
Accepted 28 April 2014

Keywords:
Turbulent heat transfer
Two pass channel
Turbulence model
Wall function

ABSTRACT

This study presents a computational study of turbulent heat transfer through two-pass square-duct channel flows with several different turbulence models that were developed to improve near-wall predictions. To evaluate the performance of the advanced wall function (the analytical wall-function: AWF), the results were compared with experimental data, those by a low-Reynolds-number k–ε model and a conventional wall function (the log-low based wall-function: LWF). Furthermore, the study extended to examining three more extended forms of the AWF. The duct used in this paper was a square duct of 50.8 mm side length (hydraulic diameter is 50.8 mm) and three Reynolds number cases were examined (30,000, 60,000 and 90,000). The LWF showed much lower Nusselt number levels in complex turbulent flow regions such as separation and reattachment zones, because the log-low employed in the model is impossible in such regions. On the other hand, the AWF proved its better performance over the LWF in the width of Reynolds number flow range. In addition, the extended forms of the AWF also showed improvements in the heat transfer predictions.

1. Introduction

It is well known that a higher turbine inlet temperature (TIT) is desirable to enhance the thermal efficiency of a gas turbine system although there are difficulties to set the TIT very high. When the designed TIT exceeds the melting temperature of the turbine material, to prevent the melting damage it is essential to cool the system under the melting temperature. The internal cooling flow of a turbine blade thus plays a key role to enhance the gas turbine efficiency and durability. Accordingly, elaborate multi-bend cooling channel systems have been developed to achieve a higher TIT. The flow inside such a cooling channel becomes complex three dimensional turbulence as a result of sharp bends of the channel and high pressure gradients. For designing a cooling channel, it is hence important to analyse turbulent heat transfer characteristics inside the channels.

There have been a number of experimental and numerical studies conducted for heat and mass transfer of blade cooling. Among them, many were related to flow-passages having two-pass channels with sharp180 degree turns (e.g., [1–6]). The University of Wisconsin, Milwaukee (UWM) group also performed such studies [7–10]. Through those studies, heat transfer performance was further improved by introducing various shaped ribs, e.g., V and inverted V-shaped ribs [10,11]. As far as the numerical studies for blade cooling passages are concerned, according to the recent advancement in computer technology, computation using high resolution computational grids has become popular in academic institutes [5,6].

For computations of turbulent heat transfer, it is important to treat turbulence physics correctly in near-wall regions because there are steep variations of turbulence quantities in such regions. Low-Reynolds-number (LRN) turbulence models, such as Launder–Sharma (LS) model [12], aim to capture satisfactorily this near-wall phenomenon. They solve turbulence quantities all through the boundary layer including the viscous sub-layer with near-wall corrections. Instead, they require a very fine mesh for the near-wall region, resulting in extremely expensive computational costs if one considers full three-dimensional (3D) computations. Indeed, it is known that the LRN k–ε model requires more than ten times higher computational costs compared with the k–ε model with a wall-function (WF) method for 3D computations. Therefore, it is still difficult for industrial engineers to optimise and design a

* Corresponding author. Tel.: +1 414 229 2345.
E-mail address: amano@uwm.edu (R.S. Amano).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.04.066
0017-9310/

Nomenclature

$A_T,\ \acute{A}_T,\ B_T,\ \acute{B}_T$	integration constants in the energy equation of the AWF
$A_U,\ \acute{A}_U,\ B_U,\ \acute{B}_U$	integration constants in the momentum equation of the AWF
$B,\ C$	model constants in the LWF
c_P	specific heat ratio at constant pressure
$c_\varepsilon,\ c_\mu$	model constants
D_h	hydraulic diameter
$k,\ k_P$	turbulent energy, turbulent energy at node P
$\mathrm{Nu},\ \mathrm{Nu}_0$	Nusselt number, reference Nusselt number
P	pressure or cell centre of the wall-adjacent cell
P_k	production term of turbulent energy
$\mathrm{Pr},\ \mathrm{Pr}_t$	Prandtl number, turbulent Prandtl number
$\dot{Q}_s$	supplied power
q_w	wall heat flux
$\dot{Q}_{cd},\ \dot{Q}_{Rd}$	conductive heat loss and thermal radiation
r_d	radius of the divider tip
$\mathrm{Re},\ \mathrm{R}_{tP}$	Reynolds number and turbulent Reynolds number
$\bar{S}$	strain parameter
S_{ij}	strain tensor, $\partial U_i/\partial x_j + \partial U_j/\partial x_i$
S_θ	source term of the energy equation
u_τ	friction velocity
$U,\ U^+,\ U_b$	mean, normalised and bulk velocities
W_d	divider thickness
$y^+,\ y^*$	normalised distances,$U_\tau y/\nu$ and $\sqrt{k_P}y/\nu$
α	growth ratio of eddy viscosity in the wall-adjacent cell for the AWF model
$\alpha_\lambda,\ \gamma_\lambda$	model constants
ε	emissivity of the stainless steel foil, or dissipation rate of turbulent energy
$\tilde{\varepsilon}$	isotropic dissipation rate
$\Theta,\ \Theta_a,\ \Theta_r,\ \Theta_w$	mean, atmospheric, reference and wall temperatures
$\Theta^+,\ \Theta_\tau$	Θ/Θ_τ and $q_w/(\rho c_p U_\tau)$
$\kappa,\ \kappa_t$	model constants in the LWF
λ	thermal conductivity of working fluid or τ_w/τ_v
λ_α	thermal conductivity of acrylic plate
$\mu,\ \mu_t$	molecular viscosity and eddy viscosity
$\nu,\ \nu_t$	kinematic viscosity and kinematic eddy viscosity
$\rho,\ \rho_R$	density and electrical resistivity
$\tau_w,\ \tau_v$	wall shear stress, shear stress

whole 3D flow system by an elaborate LRN model for their routine work.

Accordingly, to save the computational costs and time, a wall-function strategy is still frequently applied in industry. The conventional WF models such as the standard log-law based wall-function (LWF) [13], were developed based on logarithmic laws and thus allow us to use much coarser grids in the near-wall region. However, such log-laws are reasonable only for fully developed boundary layer type of flows in simple configurations. Therefore, complex flow geometry deteriorates the performance and there has been a strong demand to improve the model. Indeed, soon after the original proposal of the LWF, Chieng and Launder [14] improved the performance by allowing for a linear variation of both the shear stress and the turbulent kinetic energy across the wall-adjacent cell. Other researchers also attempted to improve the LWF (e.g., [15,16]). However, their attempts were still based on the log-laws. Since, as aforementioned, the empirical log-law formulas are valid only for fully developed turbulent boundary layer type flows and do not consider pressure gradient problems, it was difficult to obtain reasonable results in complex flows with high pressure gradients. To break through this issue, Barenblatt et al. [17] and Kader [18] (particularly for heat transfer) improved LWF models considering pressure gradients.

To ensure more reliable results than those of the LWF models, several research groups have developed new WF schemes. As Durbin [19] briefly summarised on the recent revisits to the wall function approach, the method of Craft et al. [20] differs from traditional approaches solving the turbulent energy k and its dissipation rate ε transport equations in the wall adjacent cells. It is called the analytical wall-function (AWF). Based on the similar concepts, Knopp et al. [21], Popovac and Hanjalić [22] and Utyuzhnikov [23] also developed new WF schemes. The original AWF [20] has two main assumptions inside the wall-adjacent cells. One is applying the boundary-layer-like equations; the AWF is constructed by the boundary layer approximated momentum and energy equations. Another is assuming a simple near-wall distribution profile for the eddy viscosity: the eddy viscosity is zero inside the viscous sub-layer and linearly increases above the sub-layer. These assumptions make it possible to integrate the momentum (that includes the pressure gradient term) and energy equations analytically over the wall-adjacent cells. Therefore, the AWF is expected to be more reliable in a complex flow where the pressure gradient is large and the log-laws are not valid.

Since it is relatively easy to introduce further refinements to the AWF scheme, Craft et al. [24] proposed an extended AWF re-activating the wall normal convection term of the simplified energy equation of the AWF. A laminarization effect was also introduced by Gerasimov [25]. Suga et al. [26] discussed a growth ratio of the eddy viscosity in the wall-adjacent cells. For the other flow boundaries, the AWF was extended to treat rough, porous and gas–liquid surfaces [27–30]. Several other studies on the AWF (e.g., [31]) showed encouraging results, however, many of them were two-dimensional (2D) computations, though there were a few 3D examples reported (e.g., [24,32]).

Therefore, to confirm whether such wall-function schemes are useful for the practical 3D computational fluid dynamics (CFD) for developing cooling passages of turbine blades, this study aims at evaluation of some modified versions of the AWF in a flow passage related to the turbine blade cooling systems. The test case focused on in this study is turbulent heat transfer in a 3D stationary two-pass channel with smooth walls as shown in Fig. 1. This channel configuration simulates an internal flow passage inside a gas turbine blade. Although this test configuration is relatively simple compared to the internal channels employed in real gas turbine blades because they have various shapes of rib turbulators and rotate at high speeds, it is 3D and still leads to complex flow characteristics such as separations and reattachments. Thus, the present test flow case is desirable to investigate the basic performance of the turbulence models in 3D CFD for blade cooling flows. To obtain the reference data for the thermal fields, heat transfer measurements are also performed in this study whilst the reference data for the flow fields are chosen in Liou et al. [4]. (Note that although the channel configuration and the Reynolds number are slightly different from the present thermal experiments, there seemed to be no fundamental difference in the flow physics.)