Numerical Heat Transfer, Part A: Applications

Publication details, including instructions for authors and subscription information:
http://www.tandfonline.com/loi/unht20

Turbulence Models for Fluid Flow and Heat Transfer Between Cross-Corrugated Plates

Lei Zhang \(^a\) & Defu Che \(^a\)

\(^a\) State Key Laboratory of Multiphase Flow in Power Engineering, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, P. R. China

Available online: 08 Sep 2011


To link to this article: http://dx.doi.org/10.1080/10407782.2011.600583

PLEASE SCROLL DOWN FOR ARTICLE

Full terms and conditions of use: http://www.tandfonline.com/page/terms-and-conditions

This article may be used for research, teaching, and private study purposes. Any substantial or systematic reproduction, redistribution, reselling, loan, sub-licensing, systematic supply, or distribution in any form to anyone is expressly forbidden.

The publisher does not give any warranty express or implied or make any representation that the contents will be complete or accurate or up to date. The accuracy of any instructions, formulae, and drug doses should be independently verified with primary sources. The publisher shall not be liable for any loss, actions, claims, proceedings, demand, or costs or damages whatsoever or howsoever caused arising directly or indirectly in connection with or arising out of the use of this material.
TURBULENCE MODELS FOR FLUID FLOW AND HEAT TRANSFER BETWEEN CROSS-CORRUGATED PLATES

Lei Zhang and Defu Che
State Key Laboratory of Multiphase Flow in Power Engineering, School of Energy and Power Engineering, Xi’an Jiaotong University, Xi’an, P. R. China

Three-dimensional numerical predictions of fluid flow and heat transfer between cross-corrugated plates were obtained for the same geometry and grid using eight turbulence models, i.e., LBKE, SKE, RKE, RNGKE, RSM, KW, SST and LES, for the purpose of model performance evaluation. The average Colburn factor $j$, friction factor $f$, and local Nusselt number distribution were presented and compared with available experimental data. The velocity, temperature, and turbulent viscosity ratio distributions were recorded and discussed. It has been found that all models can predict practically satisfactory $j$ and acceptable $f$ within the current Reynolds number range. LBKE and SST provide the best overall agreement with experimental data and thus are highly recommended for application. Taking into account its robustness and economy, SKE with enhanced wall treatment is also recommended for CC channel flow and heat transfer simulation. In addition, near wall treatment approach seems to be significant for the current wall-bounded flow simulation. Since some models predict similar $j$ and $f$ values but very different velocity and temperature distributions, it seems not quite sufficient to only compare the overall heat transfer and pressure drop data between simulation and experiment for comprehensive model evaluations.

1. INTRODUCTION

The cross-corrugated (CC) plate, also called the chevron plate or the herringbone type plate, has been classified traditionally as a plate-type heat transfer surface. Since it fulfills all requirements of a primary surface, it may be considered as a primary surface. It has passive enhancement of the heat transfer process where secondary flow structures are created by means of curved surfaces. The secondary flow structures in heat exchangers disturb the insulating near wall layers and thus improve the thermal exchange process.

The CC plate is often used to enhance heat transfer in the equipment such as plate heat exchangers, microturbine recuperators and rotary air preheaters. In real heat exchangers, a number of CC plates are closely packed with corrugations inclined at a certain angle to the main flow direction (MFD). Such arrangement produces a complex three-dimensional flow passage and a high density of contact points.
The plate surface geometry is characterized by corrugation profile, corrugation inclination angle (θ), corrugation pitch (P) and corrugation height (H), see Figure 1a. The enhancement of heat transfer between CC plates is directly related to these features, which provide increased effective heat transfer area, disruption and reattachment of boundary layers, swirl or vortex flow generation, and small hydraulic diameter flow channels [1].

The CC plate is one of the most commonly used patterns among the many available heat transfer element types because of its relatively high thermal performance, low pressure drop, simple structure and high rigidity. The rapidly growing demand for CC plate application promotes related investigations.

Many experimental measurements have been carried out to study the flow and heat transfer mechanism between CC plates. Some typical ones are reviewed as follows. Muley and Manglik [1] and some other authors [2–5] used the steady-state

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ L \]

turbulence energy

\[ N \]

unitary cell length along the main flow direction

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]

turbulence energy

\[ q \]

pitch of corrugations

\[ r \]

heat flux

\[ T \]

temperature

\[ \Delta T_{\text{log}} \]

logarithmic mean temperature difference

\[ u_i, u_j \]

velocity components

\[ A \]

cross-section of unitary cell

\[ c \]

cross-section of unitary cell

\[ D \]

specific heat

\[ D_{\text{ef}} \]

Darcy friction factor

\[ f \]

hydraulic diameter

\[ G \]

mass flow rate in unitary cell

\[ H \]

height of corrugations

\[ j \]

average Colburn j-factor

\[ k \]

turbulence energy

\[ L \]

unitary cell length along the main flow direction

\[ \text{Nu} \]

average Nusselt number

\[ \Delta \]

pressure drop

\[ P \]
heat transfer test method to study the thermal-hydraulic performance of the CC plate. Focke and Knibbe [6] and Dović et al. [7] tried to reveal the complex flow pattern between CC plates by means of visualization techniques. Focke et al. [8] investigated geometry effect on the heat transfer and pressure drop of CC plates by mass transfer measurement technique and \( j \)-factor analogy method. Stasiek et al. [9] resorted to liquid crystal thermography to measure the local Nusselt number \( \text{Nu}_L \) on CC plate surface. Generally speaking, most of the experimental results are either qualitative conclusions or overall averages because measurement of detailed flow or temperature field in the CC channel is very difficult and expensive considering the complex geometry and small dimensions. However, knowledge of detailed flow and temperature field is of no less importance than the knowledge of the overall averages for better understanding of the fundamental mechanisms. To achieve both overall and local information, more and more researchers turn to computational fluid dynamics (CFD). Compared with experimental measurement, CFD is an effective, convenient and economical tool for obtaining the complete picture of the flow structures between CC plates. Furthermore, the flexibility of CFD methods in handling changes in geometric parameters, Reynolds number or boundary conditions helps to understand the mechanisms affecting the flow and temperature fields.

The choice of an appropriate numerical model is one foundation for accurate CFD study. Such a choice depends on the physics of flow, level of accuracy, established practice and available computational resources, etc. Many researchers have reported utilization of different models for predicting CC channel flow. However, selection of an appropriate numerical model remains an open issue in the literature [10].

A few researchers tried to identify the appropriate model by comparing different models. Ciofalo et al. [11] used a variety of approaches ranging from laminar (LAM) flow assumptions to standard (SKE) and low Reynolds number \( k-\varepsilon \) (LBKE) models, large eddy simulation (LES), and direct simulation (DS) to simulate the heat transfer and fluid flow within the Reynolds number (Re) range \( 1000 < \text{Re} < 10,000 \). Simulating the entire plate is impossible because of the huge computing power requirements due to the large number of elements required for the geometry. The authors therefore simulated a unitary cell of the periodic plate structure. They stated that a CC plate matrix in a real heat exchanger is composed of a very large number of nominally identical, small geometrical elements repeating themselves modularly and defined such small element as a unitary cell (see Figure 1b). By comparing numerical predictions and well-controlled experimental data, they concluded that the best overall agreement with measured friction factors \( f \) and average \( \text{Nu} \) or local heat transfer coefficients \( \text{Nu}_L \) is obtained by using either LBKE or LES. SKE with standard wall functions is only acceptable at high Re and fails to reproduce the correct values of \( f \) and \( \text{Nu} \), and gives poor predictions of entry effects and of \( \text{Nu}_L \) distribution; also, it is completely inapplicable to \( \text{Re} \) lower than 2000. On the other hand, LAM yields acceptable results only for \( \text{Re} < 3000 \). Furthermore, the velocity fields predicted with different models were compared and discussed by the authors. Zhang [12, 13, 14] simulated air flow and heat transfer in 10 consecutive triangular unitary cells from the intake of the channel. The LAM and four turbulence models, SKE, the renormalized group \( k-\varepsilon \) (RNGKE), the low Reynolds \( k-\omega \) (LKW), and the Reynolds stress models (RSM) were used, with non-equilibrium wall functions approach (if applicable). Numerical results were compared with
available experimental Nu for cross-corrugated membrane modules. The author found that LAM is acceptable for $100 < \text{Re} < 500$, SKE and RNGKE are only applicable for $6000 < \text{Re} < 20000$. RSM gives the best prediction for $2000 < \text{Re} < 20,000$ and LKW prediction is in best agreement with experimental data for $2000 < \text{Re} < 6000$. However, the author did not provide any comparison of local information obtained with different models. Sunden [15] briefly reviewed CFD methods for single-phase flows and provided examples of CFD applications in a variety of heat exchanger problems including CC channel. Comparison between numerical and experimental data for $\text{Re} = 5000$ showed that all involved models, i.e., realized $k-\varepsilon$ (RKE), standard $k-\omega$ (SKW), shear-stress transport $k-\omega$ (SST), SKE, RNGKE, RSM, severely over-predict Nu, and all except SST give reasonable agreement of $f$ with experimental data. No more comparison was provided concerning the distribution results with various models. Etemad and Sunden [16] investigated the complex three dimensional flow and heat transfer for $\text{Re} = 4930$ in a unitary cell using two computational grids and four turbulence models. The obtained results were normalized with the data obtained using the low-Reynolds $k-\varepsilon$ model (LKE) due to lack of relevant experimental data. It was observed that the V2F $k-\varepsilon$ model (V2F) predicts stronger shear process, higher Nu and $f$ than the others. SKE and RSM with standard wall functions predict higher Nu and lower $f$ than LKE. The authors argued that RSM appears to be instable and gave results similar to SKE, therefore it is not motivated to use RSM for such applications. Comparisons of velocity, temperature and Nusselt number distributions obtained with different models were also presented by the authors. Freund and Kabelac [17] compared their simulation results with measured distribution of convection coefficients and found that EASM-RSM yields better results than SST but under-predicts the measured values by 25%.

Different from the above authors, many other researchers simply chose one specific model for their numerical investigations. Kanaris et al. [10, 18, 19] and Babu et al. [20] used SST for CC channel flow simulation. The authors considered SST as a combination of $k-\varepsilon$ and $k-\omega$ model. They believed that SST benefits from the advantages of both $k-\varepsilon$ and $k-\omega$ models by employing specific “blending functions” which activates the $k-\omega$ model near the wall and the $k-\varepsilon$ model for the rest of the flow. The fairly good agreement between simulation results and the limited experimental results published in the literature indicates the validity of SST. Since RKE is reported in Fluent [21] to provide better performance for flows involving boundary layer under strong adverse pressure gradient, separation, and recirculation, for several validations of separated flows and flows with complex secondary flow features, Jain et al. [22], Tsai et al. [23], and Liu et al. [24] chose RKE with nonequilibrium wall functions to simulate thermalhydraulic performance of the CC plate. Jain reported average underprediction of Nu and $f$ by 13%, and 10%, respectively, while Tsai and Liu reported average under-prediction of overall pressure drop by 20% and 7%, respectively. Han et al. [25] and Zhang et al. [26] resorted to RNGKE in their numerical study for its efficiency in solving the near-wall flow and high calculating precision of the turbulent swirl. They all claimed good agreement between their calculating results and experimental ones. Zhang and Che [27] selected LBKE with enhanced wall treatment method to study the influence of corrugation profile on CC plate performance and claimed satisfactory numerical results. Blomerius et al. [28, 29] used the numerical code FIVO, which corresponds to the direct simulation
of Navier-Stokes equations without turbulence model to solve the basic equations. Comparisons of their numerical results with available experimental data show good agreement. Zhang [30], Sawyers et al. [31], Ko [32] and Fernandes et al. [33] investigated laminar flow in CC channels numerically. Zhang studied performance of heat and mass transfer intensification in triangular CC ducts for $\theta = 90^\circ$ and $10 < Re < 1000$. Numerical results reveal that the triangular CC ducts brings about considerable effectiveness increasement in comparison with a traditional total heat exchanger of parallel plates. Sawyers et al. used numerical techniques to study the enhancement laminar heat transfer in CC channels for $\theta = 180^\circ$ and $0 < Re < 250$. They concluded that heat transfer enhancement in the CC channel compared to flat plates is due to the presence of asymmetrical recirculation. Ko investigated numerically the entropy generation in a CC duct for $\theta = 0^\circ$ and $86 < Re < 2000$. An optimal $Re$ is found to exist for every case with specific aspect ratio and wall heat flux to induce the minimal entropy generation. Fernandes et al. conducted numerical simulations in several unitary cells to predict the hydraulic performance, in fully developed laminar flow, with different geometrical properties of CC passages. With the help of geometrical considerations, the numerical values of the tortuosity coefficient and the coefficient from the $(f/Re)$ correlations were modeled by the authors.

Despite all previous efforts, to the authors’ knowledge, model evaluation studies for CC channel flow are few, and systematic investigation of both overall and local data with all involved models has not been reported in the literature. The aim of this work is therefore to provide a comprehensive performance evaluation of various numerical models for CC channel flow. Eight popular models, including LBKE, SKE, RKE, RNGKE, RSM, KW, SST, and LES are used to predict air flow and heat transfer in sinusoidal CC channel. Numerical results predicted by the eight models are presented and compared with available experimental data. DS, although possibly providing superior results, is computationally too expensive with sufficiently high spatial resolution for turbulent heat transfer and thus is not included here. The hydraulic diameter $D_{eq}$ of the CC plate in heat exchangers usually ranges from 1 to 10 mm, and the velocity of the air from 1 to 10 m/s, then $Re$ between 1000 and 10,000. Various investigators have documented critical $Re$ values ranging from 10 to 800 [8, 28, 34–36] for different CC geometries. In any event, as concluded by Muley and Manglik [37], $Re > 1000$ essentially characterizes turbulent flow. Thus, within the current range of $Re$ (1000–10,000), LAM is not used.

### 2. PLATE GEOMETRY AND PERFORMANCE PARAMETERS

To provide comparison only of the model performance possible, the same CC plate geometry was used for all models. Sinusoidal CC plate with $P = 10$ mm, $H = 5$ mm, and $\theta = 90^\circ$ was chosen for the current calculation.

Figure 2 shows the top view of a unitary cell between the chosen sinusoidal CC plates, where the vertices of the rhombus represent the four contact points of adjacent CC plates. The MFD is also illustrated by the arrow in Figure 2. In this figure, sections M-M and N-N, which are normal to the upper corrugation and the MFD, respectively, stand for two slices of the unitary cell. Most of the computational results have been presented for these sections.
The hydraulic diameter is traditionally defined as

$$D_{eq} = \frac{4V}{S_d}$$

where $V$ and $S_d$ are the volume and the developed area of a unitary cell, respectively.

Note that the above definition is not used here instead, $D_{eq}$ is defined as follows and the reason will be given in section 4.2.

$$D_{eq} = 2H$$

The reference velocity is defined as

$$U = \frac{G}{\rho A_n}$$

where $G$ is the mass flow rate across the unitary cell, $\rho$ is the density of the fluid, and $A_n$ is the cross-sectional area normal to the MFD, i.e., the area of section N-N.

Based on $D_{eq}$ and $U$, Re can be defined as

$$Re = \frac{UD_{eq}}{\nu}$$

where $\nu$ is the kinematic viscosity of the fluid.

The pressure drop can be normalized by defining the friction factor

$$f = \frac{2\Delta p}{\rho U^2 L}$$

where $\Delta p$ is the pressure drop between the entrance and the exit of the unitary cell, and $L$ is the extent of the unitary cell along the MFD.

As for heat transfer, the local Nusselt number can be defined as

$$Nu_L = \frac{q_w D_{eq}}{\lambda \Delta T_{log}}$$

where $q_w$ is the local wall heat flux, $\lambda$ is the thermal conductivity of the fluid, and $\Delta T_{log}$ is the logarithmic temperature difference between the wall and the fluid, which is defined as

$$\Delta T_{log} = \frac{(T_w - T_{in}) - (T_w - T_{out})}{\ln[(T_w - T_{in})/(T_w - T_{out})]}$$
where $T_w$ is the wall temperature, $T_{in}$ is the inlet temperature of the fluid, and $T_{out}$ is the outlet one.

The mean Nusselt number over the unitary cell surface can be defined as

$$ Nu = \frac{1}{S_d} \int_{S_d} N_u L dS_d $$

(8)

The average Colburn $j$-factor is then defined as

$$ j = \frac{Nu}{Re^{0.5}} $$

(9)

where $\sigma$ is the Prandtl number of the fluid.

The turbulent viscosity ratio is the ratio of turbulent viscosity to the laminar viscosity. It reflects the intensity of turbulence and is defined as

$$ r = \frac{\mu_t}{\mu} $$

(10)

3. MODEL AND METHOD

Three-dimensional numerical simulations of fluid flow and heat transfer within a unitary cell are obtained using a finite volume based commercial CFD package. GAMBIT is used as the preprocessor for geometry creation and mesh generation, while FLUENT 6.0 is used as the solver and postprocessor. Three basic aspects of the simulations have been presented, namely, the generation of computational grids; the introduction of numerical equations; and the solution methodology of the problem.

3.1. Grid Generation

Unstructured Tet/Hybrid grids were chosen for the current unitary cell. Three main steps were taken to construct the primary grids in GAMBIT: creation of the unitary cell geometry, grid generation, and smoothing of the generated grid. The obtained grid is then refined in FLUENT so that higher computational resolution is obtained near the wall surfaces, the inlet surfaces and the outlet surfaces than in the central core region. The external part of the resulting grids is demonstrated in Figure 3.

A grid independency check has been conducted to avoid the numerical error due to dependence of prediction results on the grid size. The primary calculations are carried out for night grid numbers. Typical results of $Nu$ calculated with different grid numbers for $Re \approx 2600$ are depicted in Figure 4. The deviation of $Nu$ between the chosen grid number (69,875) and the finer one (104,192) is 1.96%.

3.2. Numerical Equations

3.2.1. Governing equations. A complete time-dependent solution of the exact governing equations for high Re turbulent flows in complex geometries is
unlikely to be attainable for some time to come. Two alternative methods can be employed to transform these equations in such a way that the small-scale turbulent fluctuations do not have to be directly simulated: Reynolds averaging and filtering. Both methods introduce additional terms in the governing equations that need to be modeled in order to achieve closure.

The governing equations describing the fluid flow and heat transfer are conservation equations for mass, momentum, and energy, which are developed from conservation laws of physics.

**Reynolds-averaged equations.** In the Reynolds averaging method, the solution variables in the instantaneous equations are decomposed into the time-averaged and fluctuating components:

\[ \phi = \bar{\phi} + \phi' \]  

(11)

where \( \bar{\phi} \) and \( \phi' \) denote the mean and fluctuating components, respectively.
Substituting expressions of this form for the flow variables into the instantaneous governing equations and taking a time-average (and dropping the over-bar on the mean quantities) yield the time-averaged governing equations. The Reynolds-averaged approach represents transport equations for the mean flow quantities only, with all the scales of the turbulence being modeled. Such approach of permitting a solution only for the mean flow variables greatly reduces the computational effort. This approach is generally adopted for practical engineering calculations, and involves models such as k-ε and its variants, k-ω and its variants, and RSM. The Reynolds-averaged governing equations can be summarized as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \tag{12}
\]

\[
\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial u_j}{\partial x_j} \right] \tag{13}
\]

\[
\frac{\partial (\rho c_p T)}{\partial t} + \frac{\partial (\rho u_j c_p T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial (c_p T)}{\partial x_j} \right] \tag{14}
\]

The above governing equations have the same general form as the instantaneous equations, with the solution variables now representing the time-averaged values. They are written in conservation form and tensor notation, with Einstein’s convention of implied summation over repeated indices. More than five unknown variables exist in these governing equations, thus additional turbulence model is necessary to close the equations.

**Filtered equations.** LES provides an alternative approach in which large eddies are computed in a time-dependent simulation using a set of filtered equations. Filtering is essentially a manipulation of the exact governing equations to only remove eddies that are smaller than the size of the filter, which is usually taken as the mesh size.

A filtered variable (denoted by an over-bar) is defined by

\[
\bar{\phi}(x) = \int_D \phi(x') G(x, x') dx' \tag{15}
\]

where \(D\) is the fluid domain, and \(G\) is the filter function that determines the scale of the resolved eddies.

Filtering the instantaneous governing equations, one obtains

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_j)}{\partial x_j} = 0 \tag{16}
\]

\[
\frac{\partial (\rho \bar{u}_j)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \bar{p} + \frac{\tau_{kk}}{3} \right) + \frac{\partial}{\partial x_j} \left[ (\bar{u} + u_i) \frac{\partial \bar{u}_j}{\partial x_j} \right] \tag{17}
\]
Statistics of the mean flow quantities, which are generally of interest in engineering, are gathered during the time-dependent simulation.

The similarity between the filtered and Reynolds averaged equations is obvious. The major difference is that the dependent variables are now filtered quantities rather than mean quantities, and the expressions for additional turbulence terms differ.

**3.2.2. Turbulence models. The standard \( k-\epsilon \) model.** SKE is a semi-empirical model based on model transport equations for the turbulence kinetic energy \( k \) and its dissipation rate \( \epsilon \). The model transport equation for \( k \) is derived from the exact equation, while that for \( \epsilon \) is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. It is the most popular two-equation model and has produced qualitatively satisfactory results for a number of complex flows [21].

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k
\]

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients. \( G_b \) is the generation of turbulence kinetic energy due to buoyancy, and \( Y_M \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. Both of them are neglected in this study. \( \sigma_k \) and \( \sigma_\epsilon \) are the turbulent Prandtl numbers for \( k \) and \( \epsilon \), respectively. \( S_k \) and \( S_\epsilon \) are user defined source terms, which are not used in the present study. See reference [21] for more details.

The turbulent viscosity, \( \mu_t \), is computed by combining \( k \) and \( \epsilon \) as follows.

\[
\mu_t = \rho C\mu \frac{k^2}{\epsilon}
\]

where \( C\mu \) is a constant.

**The realizable \( k-\epsilon \) model.** RKE is a relatively recent development and differs from SKE in two important ways: RKE contains a new formulation for \( \mu_t \); a new transport equation for \( \epsilon \) has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The term realizable means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. An immediate benefit of RKE is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients,
separation, and recirculation. One limitation is that it produces non-physical turbulent viscosities in situations when the computational domain contains both rotating and stationary fluid zones. However, initial studies have shown that RKE provides the best performance of all the k-ε model versions for several validations of separated flows and flows with complex secondary flow features [21].

The modeled transport equation for $k$ in RKE is the same as that in SKE, except for the model constants. The transport equation for $ε$ in RKE is

\[
\frac{∂(ρε)}{∂t} + \frac{∂(ρu_iε)}{∂x_i} = \frac{∂}{∂x_j} \left[ \left( \frac{μ + μ_r}{σ_ε} \right) \frac{∂ε}{∂x_j} \right] + ρC_1Sε - ρC_2\frac{ε^2}{k + \sqrt{νε}} + C_1ε \frac{ε}{k} C_3ε G_β + S_ε \tag{22}
\]

where

\[
C_1 = \max \left[ 0.43, \frac{η}{η + 5} \right] \tag{23}
\]

The eddy viscosity is computed from

\[
μ_r = ρC_μ \frac{k^2}{ε} \tag{24}
\]

where $C_μ$ is no longer constant and it is a function of the mean strain and rotation rates, the angular velocity of the system rotation, and the turbulence fields. It is computed from

\[
C_μ = \frac{1}{A_0 + A_3 \frac{kU_ε}{ε}} \tag{25}
\]


**The renormalization group k-ε model.** RNGKE is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called renormalization group methods. It is similar in form to SKE, but includes the following refinements: RNGKE has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows; the effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows; the RNG theory provides an analytical formula for turbulent Prandtl numbers, while SKE uses user-specified, constant values; while SKE is a high-Reynolds-number model, the RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low-Reynolds-number effects. These features make RNGKE more accurate and reliable for a wider class of flows than SKE [21].

The transport equations for $k$ and $ε$ in RNGKE are

\[
\frac{∂(ρk)}{∂t} + \frac{∂(ρu_i k)}{∂x_i} = \frac{∂}{∂x_j} \left( \mu_k \mu_{eff} \frac{∂k}{∂x_j} \right) + G_k + G_β - ρε - Y_M + S_k \tag{26}
\]
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu_i \frac{\partial k}{\partial x_j} \right) + \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \varepsilon \]  
\]  
(32)

where the quantities \( \alpha_k \) and \( \alpha_\varepsilon \) are the inverse effective Prandtl numbers for \( k \) and \( \varepsilon \), respectively. They are computed using the following formula derived analytically by the RNG theory.

\[
\begin{align*}
\alpha_k - 1.3929 & = 0.6321 \left( \frac{\alpha_0 - 1.3929}{\alpha_0 + 2.3929} \right)^{0.3679} \\
\alpha_\varepsilon - 1.3929 & = \frac{\mu_{mol}}{\mu_{eff}}
\end{align*}
\]  
(28)

The main difference between RNGKE and SKE lies in the additional term in the \( \varepsilon \) equation given by

\[
R_\varepsilon = \frac{C_\mu \rho \eta^3 (1 - \eta / \eta_0) \varepsilon^2}{1 + \beta \eta^3} \]  
(29)

The scale elimination procedure in RNG theory results in a differential equation for turbulent viscosity.

\[
d \left( \frac{\rho^2 k}{\sqrt{\varepsilon \mu_t}} \right) = f \nabla \left( \frac{\rho^2 k}{\sqrt{\varepsilon \mu_t}} \right) + \frac{\dot{\nu}}{\sqrt{\dot{\nu}^2 - 1 + C_\nu}} d \dot{\nu}
\]  
(30)

The above equation is integrated to obtain an accurate description of how the effective turbulent transport varies with the effective Reynolds number (or eddy scale), allowing the model to better handle low-Reynolds number and near-wall flows. See reference [21] for more details.

**The low Reynolds number k-\( \varepsilon \) model by Lam and Bremhorst.** Lam and Bremhorst [38] extended SKE to the low Reynolds number form, i.e., LBKE, which allows calculations right to the wall. The authors stated that LBKE is valid throughout the fully turbulent, semi-laminar, and laminar regions of pipe flow according to their test. LBKE does not have to be used in conjunction with empirical wall functions and does not include additional terms in the \( k \) and \( \varepsilon \) equations. The expression used for \( \mu_t \), and the transport equations for \( k \) and \( \varepsilon \) can be summarized as follows.

\[
\mu_t = C_{\mu f} \frac{\rho k^2}{\varepsilon}
\]  
(31)

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \varepsilon
\]  
(32)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} + C_{1f} \mu_t \frac{\varepsilon}{k} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - C_2 f_2 \frac{\rho k^2}{\varepsilon}
\]  
(33)

where \( f_\mu, f_1, \) and \( f_2 \) can be defined as

\[
f_\mu = \left[ 1 - \exp(-0.0165 \text{Re}_t) \right]^2 \left( 1 + \frac{20.5}{\text{Re}_t} \right)
\]  
(34)
\[ f_1 = 1 + (0.05/f_\mu)^3 \]

\[ f_2 = 1 - \exp(-\text{Re})^2 \]

See reference [38] for more details.

The Reynolds stress model. Abandoning the isotropic eddy-viscosity hypothesis, RSM closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for \( \varepsilon \). Since RSM accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and two-equation models, it has greater potential to give accurate predictions for complex flows. However, the fidelity of RSM predictions is still limited by the closure assumptions employed to model various terms in the exact transport equations for the Reynolds stresses. The modeling of the pressure-strain and dissipation-rate terms is particularly challenging, and often considered to be responsible for compromising the accuracy of RSM predictions [21].

The exact transport equations for the transport of the Reynolds stresses \( \rho \bar{u}_i' \bar{u}_j' \) may be written as follows:

\[
\frac{\partial}{\partial t}(\rho \bar{u}_i' \bar{u}_j') + \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial \bar{u}_i'}{\partial x_k} \right] - \rho \left( \bar{u}_i' \frac{\partial \bar{u}_j'}{\partial x_k} + \bar{u}_j' \frac{\partial \bar{u}_i'}{\partial x_k} \right) - \rho \beta \left( g_1 \bar{u}_j' \bar{g} + g_2 \bar{u}_j' \bar{g} \right) + \rho \left( \frac{\partial \bar{u}_i'}{\partial x_j} + \frac{\partial \bar{u}_j'}{\partial x_i} \right) = - \frac{\partial}{\partial x_k} \left[ \rho \bar{u}_i' \bar{u}_j' \bar{u}_k' + p \left( \delta_{ij} \bar{u}_k' + \delta_{ik} \bar{u}_j' \right) \right] + \frac{\partial}{\partial x_k} \left( \mu \frac{\partial \bar{u}_j'}{\partial x_k} \right) - 2 \mu \left( \frac{\partial \bar{u}_i' \bar{u}_j'}{\partial x_k} \frac{\partial \bar{u}_i'}{\partial x_k} \right) - 2 \rho \Omega_k ( \bar{u}_i' \bar{u}_j' \bar{e}_k, km + \bar{u}_i' \bar{u}_j' \bar{e}_j, km ) + S
\]

Of the various terms in the above equation, \( C, D_L, P, \) and \( F \) do not require any modeling. However, \( D_T, G, \varepsilon, \) and \( \phi \) need to be modeled to close the equations. In this article, \( G, F, \) and \( S \) are neglected. See reference [21] for more details.

When RSM is applied to near-wall flows using the enhanced wall treatment in FLUENT, the linear pressure-strain model needs to be modified. FLUENT modifies the pressure strain model by specifying the model constants values of \( C_1, C_2, C_1', \) and \( C_2' \) as functions of the Reynolds stress invariants and the turbulent Reynolds number.

\[
C_1 = 1 + 2.58A \sqrt{A2} \left\{ 1 - \exp \left[ -(0.0067 \text{Re})^2 \right] \right\}
\]

\[
C_2 = 0.75 \sqrt{A}
\]
The standard $k$-$\omega$ model. The standard $k$-$\omega$ model (KW) is an empirical model based on model transport equations for the turbulence kinetic energy ($k$) and the specific dissipation rate ($\omega$), which can also be thought of as the ratio of $\varepsilon$ to $k$ [39]. KW in FLUENT is based on the Wilcox $k$-$\omega$ model [39], which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The Wilcox model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets and is thus applicable to wall-bounded flows and free shear flows [21].

The turbulence kinetic energy $k$ and the specific dissipation rate $\omega$ are obtained from the following transport equations:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \tag{42}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \tag{43}
\]

In these equations, $G_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients. $G_\omega$ represents the generation of $\omega$. $\Gamma_k$ and $\Gamma_\omega$ represent the effective diffusivity of $k$ and $\omega$, respectively. $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$ due to turbulence. $S_k$ and $S_\omega$ are user-defined source terms.

The effective diffusivities for KW are given by

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \tag{44}
\]

\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \tag{45}
\]

where the turbulent viscosity $\mu_t$ is computed by combining $k$ and $\omega$ as follows:

\[
\mu_t = \alpha^* \frac{\rho k}{\omega} \tag{46}
\]

The coefficient $\alpha^*$ damps the turbulent viscosity causing a low-Reynolds-number correction. It is given by

\[
\alpha^* = \alpha^*_{\infty} \left( \frac{\alpha_0 + \text{Re}_t/R_k}{1 + \text{Re}_t/R_k} \right) \tag{47}
\]

**The shear-stress transport k-ω model.** SST is developed by Menter [40] to effectively blend the robust and accurate formulation of the k-ω model in the near-wall region with the free-stream independence of the k-ε model in the far field. To achieve this, the k-ε model is converted into a k-ω formulation. SST model is similar to KW, but includes the following refinements: KW and the transformed k-ε model are both multiplied by a blending function and both models are added together; the blending function is designed to be one in the near-wall region, which activates the standard k-ω model, and far away from the surface, which activates the transformed k-ε model; SST incorporates a damped cross-diffusion derivative term in the ω equation; the definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress; and the modeling constants are different. These features make SST more accurate and reliable for a wider class of flows (e.g., adverse pressure gradient flows, airfoils) than both the standard k-ε model and the standard k-ω model [21].

SST equations have a similar form to KW except for an additional term $D_ω$ in the ω equation:

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_ω \frac{\partial \omega}{\partial x_j} \right) + G_ω - Y_ω + D_ω + S_ω$$

(48)

The effective diffusivities for SST are the same as that for KW and thus are not shown here.

The turbulent viscosity $\mu_t$ is computed as follows:

$$\mu_t = \frac{\rho k}{\omega} \max[1/\alpha^3, \Omega F_2/(a_1 \omega)]$$

(49)

The blending functions $F_1$ and $F_2$ are given by

$$F_1 = \tanh(\Phi_1)$$

(50)

$$F_2 = \tanh(\Phi_2)$$

(51)


**The large eddy simulation model.** Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy. It is theoretically possible to directly resolve the whole spectrum of turbulent scales using an approach known as direct simulation (DS). DS is not, however, feasible for practical engineering problems for its excessive computational cost. On the other hand, all the turbulent motions are modeled, resulting in a significant savings in computational effort in the conventional Reynolds-averaged (RA) approach. Conceptually, LES is situated somewhere between DS and the RA approach.

Basically large eddies are resolved directly in LES, while small eddies are modeled. The rationale behind LES can be summarized as follows: momentum, mass, energy, and other passive scalars are transported mostly by large eddies; large eddies
are more problem-dependent; small eddies are less dependent on the geometry, tend to be more isotropic, and are consequently more universal; the chance of finding a universal model is much higher when only small eddies are modeled [21].

The most basic of sub-grid scale (small eddies) models was proposed by Smagorinsky [41] and further developed by Lilly [42]. In the Smagorinsky-Lilly model, the eddy viscosity is modeled by

$$\mu_r = \rho L_S^3 |\mathbf{S}|$$  \hspace{1cm} (52)

where $L_S$ is the mixing length for sub-grid scales and is computed using

$$L_S = \text{min}(\kappa d, C_S V^{1/3})$$  \hspace{1cm} (53)

and

$$|\mathbf{S}| = \sqrt{2S_{ij}S_{ij}}$$  \hspace{1cm} (54)


### 3.3. Solution Methodology

#### 3.3.1. Simplifications and fluid properties.

The governing equations were solved with the following simplifications.

1. The conductive thermal resistance of the plates was neglected, compared to the film thermal resistance.
2. Heat transfer surface was assumed to be free from fouling.
3. There is no mal-distribution of flow in the small computational domain of a unitary cell.

Air is the working fluid in this article. Its density, specific heat, thermal conductivity, and viscosity were taken as second order polynomial functions of its temperature.

#### 3.3.2. Boundary and initial conditions.

Dirichlet boundary conditions are adopted in the inlet faces and Neumann conditions are imposed for the outlet faces of the unitary cell. More specifically, for the first cell of any path, uniform inlet velocity profiles and uniform inlet temperature distributions are assumed while zero-normal derivative conditions are imposed on the outlet faces. Rational values of $k$, $\varepsilon$, and $\omega$ representative of the inlet flow conditions, also need to be specified for SKE, RKE, RNGKE, LBKE, RSM, KW, and SST. They are estimated from empirical correlations provided in Fluent [21] for complex duct flows. For LES, the stochastic components of the flow at the velocity-specified inlet boundaries are accounted for by superposing random perturbations on individual velocity components [21]. Once convergence is achieved for the first cell, its outlet distributions can be used as inlet conditions for the next cell. As more and more consecutive cells are involved, results are expected to approach those obtained under fully developed situations.
Ordinary no-slip conditions are used as hydrodynamic wall boundary conditions. Uniform wall temperature conditions are used as thermal boundary conditions. For simplicity and better convergence, zero is assumed for all the concerning variables everywhere in the unitary cell as the initial conditions.

### 3.3.3. Further computational details. Discretization scheme.

In the present study, the Navier-Stokes equations are solved by the SIMPLEC scheme, the convective terms in the governing equations are solved by three-order QUICK scheme, and the diffusive terms by second-order central difference scheme. In addition, the standard scheme is used for pressure discretization and second-order implicit formulation is chosen for LES to improved accuracy.

In a numerical study for inertial flow, Ciofalo [43] found that the noniterative PISO algorithm leads to a loss of accuracy, which results in under-predicting cross-stream velocity fluctuations and resolved turbulent heat flux while SIMPLEC gives a much higher accuracy while requiring only 2-3 times more computational effort. The author also stated that the Smagorinsky model has performed remarkably well in a wide range of Reynolds numbers from transitional to highly turbulent and both for forced-convection and free-convection flows. Thus, SIMPLEC scheme and Smagorinsky-Lilly model are used for LES in the current study. Furthermore, the result of time step independence test, together with guidelines in FLUENT are used to decide the proper time step value for LES calculation.

**Near-wall treatments.** In the present wall-bounded CC channel flow, turbulent flows are significantly affected by the presence of walls. The near-wall modeling significantly impacts the fidelity of numerical solutions, inasmuch as walls are the main source of mean vorticity and turbulence. Therefore, accurate representation of the flow in the near-wall region determines successful predictions of wall-bounded turbulent flows. Traditionally, there are two approaches to model the near-wall region. In one approach, semi-empirical formulas called wall functions are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. In another near-wall model approach, the turbulence models are modified to enable the viscosity-affected region to be resolved with a mesh all the way to the wall, including the viscous sub-layer. The wall function approach, however, is inadequate in situations where the low-Reynolds-number effects are pervasive in the flow domain in question. Thus enhanced wall treatment [21] is used for SKE, RKE, RNGKE, and RSM in this study.

Enhanced wall treatment is a near-wall model approach that combines a two-layer model with enhanced wall functions. The two-layer model is an integral part of the enhanced wall treatment and is used to specify both $\varepsilon$ and the turbulent viscosity in the near-wall cells. In this approach, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region. In the fully turbulent region, the $k$-$\varepsilon$ models or RSM are employed. In the viscosity-affected near-wall region, the one-equation model of Wolfstein [44] is employed.

In the enhanced wall treatment approach, the two-layer definition is smoothly blended with the high-Reynolds-number $\mu_t$ definition from the outer region, as proposed by Jongen [45].

$$
\mu_{t, \text{enh}} = \lambda_{c} \mu_t + (1 - \lambda_{c}) \mu_{t, \text{2layer}}
$$
The $\varepsilon$ field is computed from

$$\varepsilon = \frac{k^{3/2}}{l_\varepsilon} \quad (56)$$

The enhanced wall functions in FLUENT were developed by blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall using a function suggested by Kader [46].

$$u^+ = C u^+_{\text{lam}} + C u^+_{\text{turb}} \quad (57)$$

Enhanced thermal wall functions follow the same approach developed for the profile of $u^+$. See reference [21] for more details.

The near wall conditions for the $k$ equation in KW and SST are treated in the same way as the $k$ equation is treated when enhanced wall treatments are used with the $k$-$\varepsilon$ models.

In FLUENT the value of $\omega$ at the wall is specified as

$$\omega_w = \frac{\rho(u^+)^2}{\mu} \omega^+ \quad (58)$$

The asymptotic value of $\omega^+$ in the laminar sub-layer is given by

$$\omega^+ = \min \left( \omega_w^+, \frac{6}{\beta_\infty^+ (u^+)^2} \right) \quad (59)$$

In the logarithmic (or turbulent) region, the value of $\omega^+$ is

$$\omega^+ = \frac{1}{\sqrt{\beta_\infty^{+2}}} \frac{du^+_{\text{turb}}}{dy^+} \quad (60)$$

As for LES, when the mesh is fine enough to resolve the laminar sub-layer, the wall shear stress is obtained from the laminar stress-strain relationship:

$$\frac{\bar{u}}{u_t} = \frac{\rho u_t y}{\mu} \quad (61)$$

If the mesh is too coarse to resolve the laminar sub-layer, it is assumed that the centroid of the wall-adjacent cell falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

$$\frac{\bar{u}}{u_t} = \ln \left( \frac{\rho u_t y}{\mu} \right) \quad (62)$$

**Convergence and computational cost.** The residuals, together with outlet mass-weighted average pressure, velocity and temperature were monitored to
identify convergence. Convergence is achieved once all those variables reach certain asymptotic values. At least 450 iterations are required before a satisfactory convergence is achieved in this paper.

Under the same grid density, SKE, KW and SST are the least expensive models among the models used here. RKE, LBKE and LES require only slightly more computational effort than SKE. Due to the extra terms and functions in the governing equations and a greater degree of non-linearity, computations with RNGKE tend to take 10–15% more CPU time than with SKE. Compared with the $k$-$\varepsilon$ and $k$-$\omega$ models, RSM requires additional memory and CPU time due to the increased number of the transport equations for Reynolds stresses. However, efficient programming in FLUENT has reduced the CPU time per iteration significantly. On average, the RSM in FLUENT requires 50–60% more CPU time per iteration and 15–20% more memory compared to the $k$-$\varepsilon$ and $k$-$\omega$ models [21].

4. RESULTS AND DISCUSSION

Typical simulation results of velocity, temperature, local Nusselt number, turbulent viscosity ratio distributions, mean Colburn $j$-factor, and friction factor are presented. Some of them are compared with available experimental data to evaluate the performance of numerical models and to shed light on the flow and heat transfer mechanism in CC channels as well.

4.1. Entry Effects

Entry effects on Nu and $f$ are shown typically in Figure 5. Predicted values of Nu and $f$ with different models are reported as a function of the unitary cell number from the intake for $Re \approx 2700$. The results are normalized by dividing Nu and $f$ with corresponding values in the fifth unitary cell. The figure shows that all Nu and $f$ decrease remarkably right after the first cell and reach asymptotic values up to the fifth cell from the entrance, implying fully developed flow and heat transfer, which is also reflected in the experimental results of reference [9]. Since all the concerning models correctly reflect entry effects in CC channel, predicting data of the fifth unitary cell are used for the following discussion. It should be noted that Ciofalo et al. [11] found that SKE with standard wall functions gave poor predictions of entry effects, which is different from the current observation. Different near wall treatment methods seem to account for such contradiction.

4.2. Heat Transfer and Pressure Drop

Heat transfer is characterized by Colburn factor $j$ or Nusselt number Nu and pressure drop is characterized by friction factor $f$ in the current study.

4.2.1. Average $j$ and $f$. Among the many researchers, Focke et al. [8] have reported the most detailed and comprehensive experimental measurements of $j$ and $f$ for CC channel flows. These results are the most commonly referenced experimental data in literature. Therefore, in an effort to evaluate the present numerical models, experimental data from reference [8] are used here. The CC plate with
identical geometries to Focke’s (see section 2) is simulated numerically. It is worth mentioning that the equivalent diameter was defined as the maximum passage gap in reference [8]. Thus the same definition is used for $D_{eq}$ to maintain consistency in the current study.

Figures 6a and 6b illustrate the fully developed numerical data in the fifth unitary cell and the corresponding experimental data for $j$ and $f$. On the whole, the deviations of predicted $j$ and $f$ from their corresponding experimental data are between $-14.53\%$ and $16.22\%$ and between $-33.21\%$ and $-12.41\%$, respectively. The numerical results of $j$ agree better with experimental data than that of $f$. Detailed comparisons between numerical and experimental data are summarized in Table 1.
The above table shows that practically good agreements between numerical and experimental data of $j$ are obtained with all the concerning models while practically good agreements of $f$ data are only achieved with LBKE, SST, LES, and SKE for $1000 < Re < 10,000$. The maximum absolute deviation values for $f$ with RSM, KW, RKE, and RNGKE all exceed 20%. Statement therefore can be made that LBKE, SST, LES, and SKE predict better $j$ and $f$ values for flow and heat transfer between CC plates under transitional and turbulent conditions than the rest of the models here. Nevertheless, Zhang [12, 13] provided different conclusion that RSM gives better predictions for $2000 < Re < 6000$ than that of SKE and RNGKE. Different near wall treatment approaches for RSM, SKE and RNGKE in Zhang’s study.
and in this study, i.e., nonequilibrium wall functions versus near wall modeling, are believed to account for the above discrepancy.

The most accurate predictions are made by both LBKE and SST within the whole Re range, indicating the best performance of LBKE and SST for simulating overall thermalhydraulic performance of CC plates among the present models.

Table 1. Computation deviations for different models

<table>
<thead>
<tr>
<th>Deviation (%)</th>
<th>LBKE</th>
<th>SST</th>
<th>LES</th>
<th>RSM</th>
<th>KW</th>
<th>SKE</th>
<th>RKE</th>
<th>RNGKE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$j_{\min}$</td>
<td>-13.54</td>
<td>-10.86</td>
<td>-9.67</td>
<td>-11.41</td>
<td>-14.53</td>
<td>-7.29</td>
<td>-8.87</td>
<td>-7.64</td>
</tr>
<tr>
<td>$j_{\max}$</td>
<td>5.22</td>
<td>16.22</td>
<td>14.31</td>
<td>13.48</td>
<td>15.28</td>
<td>-3.76</td>
<td>-3.21</td>
<td>1.08</td>
</tr>
<tr>
<td>$f_{\min}$</td>
<td>-14.83</td>
<td>-12.8</td>
<td>-18.89</td>
<td>-27.95</td>
<td>-33.21</td>
<td>-19.31</td>
<td>-24.07</td>
<td>-32.84</td>
</tr>
<tr>
<td>$f_{\max}$</td>
<td>-2.72</td>
<td>-1.12</td>
<td>4.86</td>
<td>5.75</td>
<td>-6.97</td>
<td>12.41</td>
<td>11.7</td>
<td>7.87</td>
</tr>
</tbody>
</table>

Figure 7. Numerical versus experimental data; (a) $j$-Re trends; and (b) $f$-Re trends.
Similar conclusion was also made by Ciofalo et al. [11] who claimed better prediction with LBKE than that of SKE. As seen from Figure 6b, absolute $f$ deviation values do not exceed 20% with RKE for $1000 < \text{Re} < 3900$, with RNGKE for $1000 < \text{Re} < 3300$, with RSM for $1000 < \text{Re} < 6300$ and with KW for $1000 < \text{Re} < 7500$. This finding evidently shows that acceptable results of $j$ and $f$ can be obtained with RKE, RNGKE, RSM, and KW for certain Re ranges. Moreover, it is worth mentioning that SKE with enhanced wall treatment under-predicts $j$ slightly ($-7.29\% \sim -3.76\%$) in this study while SKE with standard wall functions over-predicts $j$ seriously (about 20%) in Ciofalo’s study [11], which further confirms the significant effect of near wall treatment approach on the current wall-bounded turbulence flow simulation. Another discovery is that RSM predicts similar results to SKE, thus the more instable and computation-expensive RSM model appears to be inapplicable for the present simulation, which was also reflected in reference [16].

Further discussion of predicted and experimental $j$-Re and $f$-Re results are illustrated in Figure 7. Best-fitted lines of numerical results are compared with experimental data. From Figure 7a can be concluded that LBKE, SKE, RKE, and RNGKE provide curved $j$-Re lines that are close to the experimental data while LES, SST, RSM, and KW give poor linear $j$-Re lines. Figure 7b demonstrates that LBKE, LES, SST, and KW predict satisfactory $f$-Re trends for $1000 < \text{Re} < 10,000$ while SKE, RKE, RNGKE, and RSM offer good $f$-Re trends for $4500 < \text{Re} < 10,000$. Generally speaking, LBKE predicts the best $j$-Re and $f$-Re trends over the entire Re range in question.

In summary, all modes give satisfactory $j$ values and acceptable $f$ values over the whole Re range, from the engineering point of view. LBKE and SST predict the best $j$, $f$ values and quite satisfactory $j$-Re, $f$-Re trends. LES and SKE provide practically good $j$, $f$ values and satisfactory $j$-Re, $f$-Re trends. KW, RKE, RNGKE, and RSM offer acceptable $j$, $f$ values and poor $j$-Re, $f$-Re trends.

### 4.2.2. Local Nu distribution

The predicted $\text{Nu}_L$ distributions on the bottom wall for $\text{Re} \approx 2700$ are presented in Figure 8, where the deep color denotes low $\text{Nu}_L$.
while the light one denotes high $\text{Nu}_L$. The flow direction in the lower channel is illustrated by arrows. In Figure 8, basically two intense heat transfer regions are recognizable for all the cases: a large one near the trailing edge (the upper left edge) of the bottom wall, which is in correspondence with impingement on the bottom wall by the fluid stream flowing in the upper furrow, the other one occurs in the narrow strip of the leading (lower right) edge, close to the midplane ($y = 0$, see Figure 1b) where interaction between the upper and lower fluid streams generates high levels of turbulence and mixing. As expected, low $\text{Nu}_L$ is obtained in the central corrugated (furrow) region and around the four contact points due to almost undisturbed flow. Similar $\text{Nu}_L$ distributions have also been observed by Stasiek et al. [9] in their experimental study for sinusoidal CC plates when $\theta = 60^\circ$ and $Re = 2400$, which verifies the capability of all the present models in predicting approximately correct $\text{Nu}_L$ distributions for CC channel flow. Furthermore, $\text{Nu}_L$ distributions predicted by KW, SST, and LES are more inhomogeneous than that by the other models in Figure 8.

### 4.3. Velocity and Temperature Fields

Focke et al. [6, 8] argued that at intermediate angles, the swirling motions induced by the streams criss-crossing along the furrows are the main determinant of heat transfer. In order to study the swirling motion and its effect on heat transfer, velocity fields in the section M-M and temperature fields in the section N-N (see Figure 2) are selected for discussion.

Simulated velocity and temperature distributions with various models are summarized in Figures 9 and 10. In Figure 9 the mean flow direction is from left to right. The velocity scale is enhanced and some velocity vectors are skipped for better show of the swirling flow patterns, and only part of the (high and uninteresting) velocities in the lower duct are shown. Note that the velocities are not drawn to the same scale. As seen in Figure 9, swirling motions are clearly identified with all the models and swirling flow patterns change with models. One large vortex, centered roughly on the axis of the duct, is predicted by all concerning models except for LES which provides two vortices, with the main one located approximately on the axis of the duct and the secondary one shifted upstream of the main flow direction.

![Figure 9](image-url) **Figure 9.** In-plane velocity fields in the section M-M for $Re \approx 4500$. 
More specifically, KW and SST present an irregularly shaped vortex while SKE, RKE, RNGKE, LBKE, and RSM identify one regularly circular vortex. In Figure 10, symmetrical temperature distributions are easily recognizable, as expected, with all the models. Yet the difference in distribution detail is quite evident. For SKE, RKE, RNGKE, LBKE, and RSM, the similar two low temperature areas denoted by deep color and located in the upper and lower furrow are caused by almost stagnant flow around the large vortex cores. While the irregularly shaped vortex and the two vortices (see Figure 9) account for more complicated temperature distributions for KW, SST, and LES. Not surprisingly, similar flow fields lead to similar temperature distributions for KW and SST.

On the whole, it can be summarized from Figures 9 and 10 that LBKE, SKE, RKE, RNGKE, and RSM offer almost the same velocity and temperature fields, and that KW and SST present almost coincident distributions. Therefore, numerical data with RSM, SST, and LES are used for further discussion in the following section.

![Figure 10. Temperature fields in the section N-N for Re≈4500.](image)

![Figure 11. In-plane velocity fields in the section M-M predicted by various models.](image)
4.4. Further Computational Results

Typical calculation results with RSM, SST and LES are presented in Figures 11–14. The contrast of flow fields with different models is impressive in Figure 11. One single large vortex located on the duct axis, two small vortices, one midsize vortex shifting downstream of the MFD, are provided by RSM, SST and LES, respectively, when Re ≈ 1000. For Re ≈ 4500, the two small vortices change into an irregular large vortex in SST, while the midsize vortex in LES turns into two vortices. As Re increases to about 6300, at least three vortices are clearly...
visible in LES. Finally when Re reaches about 10,000, a single large vortex centered roughly on the axis of the duct is easily recognizable for all the three models. The velocity and consequently the driving force that produces swirl are high at large Re. It is assumed that strong interaction between crossing streams in CC channel becomes the predominant factor that controls flow pattern, thus resulting in the aforementioned similar velocity distribution for Re \( \approx 10,000 \).

Furthermore, as seen in Figure 11, RSM forecasts almost independent vortex of Re, while SST and LES predict changing vortex intensity and complexity with Re. Anyhow, conclusion can be drawn that various models predict dramatically different velocity distributions for CC channel flow and LES predicts the most complex one. Unfortunately, experimental distribution data is not available at the present time, thus no more comparison between numerical and experimental data can be made for further model evaluation.

Figure 12 depicts the temperature distributions for section N-N with various models. Symmetrical temperature distributions are predicted by all models once again. The large vortex in RSM leads to a considerable mixing of near-wall fluid with the fluid in the core of the channel which results in nearly uniform core temperature distributions, while the irregular vortex in SST and the complex vortices in LES are responsible for the more stratified temperature distributions. At small Re, the two low temperature areas are caused by almost stagnant flow in the re-circulation region cores. As Re increases, RSM offers similar temperature distributions while SST and LES provide more complicated ones. One common feature is captured by all the three models that the thermal boundary layers become thinner and thus the wall temperature gradients become higher as Re increases, implying better heat transfer.
On the whole, from Figure 12 can be concluded that different models provide very
different temperature distributions.

The predicted turbulent viscosity ratio distributions for section N-N are
illustrated in Figure 13. RSM provides nearly uniform distributions over the whole
region as Re increases. SST presents two small regions with high viscosity for
Re ≈ 1000, located in the upper and lower furrow, respectively. For larger Re a third
high viscosity strip region appears near the shear layer where the upper wall meets the
lower one and the interaction between the fluid streams takes place. LES forecasts
more complicated distributions than RSM and SST. Generally speaking, symmetrical
turbulent viscosity ratio distributions, high viscosity strip near the shear layer and
increasing turbulent viscosity ratio with Re are predicted by all the models. The above
high viscosity ratio strip near the shear layer is coincident with the midplane
high-shear region predicted by Ciofalo [11]. In such region, interaction between the
upper and lower fluid streams results in high levels of turbulence and shear stress.

Comparative heat transfer predictions with RSM, SST, and LES are reported
in Figure 14. The main features of Nu_L distribution, i.e., two regions of intense heat
transfer near the trailing edge and the leading edge of the lower wall, low heat trans-
fer region along the upper furrow, and very low heat transfer region surrounding the
four contact points are predicted by all models for various Re, which is also reflected

5. CONCLUSION

Three-dimensional numerical simulation has been carried out to investigate
air flow and heat transfer under transitional and weakly turbulent conditions in
sinusoidal CC passage. Predictions with eight turbulence models are compared with
available experimental results. Same geometry and grid are used to provide
comparison only of the model performance.

Compared to available heat transfer and pressure drop experimental data, all
models predict correct entry effects, practically satisfactory \( j \) values and acceptable \( f \)
values. They all capture the major characteristics in the Nu_L map over the whole
Re range here. LBKE, SST, LES, and SKE perform better than the other models.
The best predictions are provided by LBKE and SST and thus they are highly recom-
ended for application. Taking into account its robustness and economy, SKE with
enhanced wall treatment is also recommended for CC channel flow simulation from
the engineering point of view. Moreover, it is found that apart from proper turbulence
model, appropriate near wall treatment approach is also significant for successful
wall-bounded turbulent flow simulation. Swirling flow pattern is identified by all the
models. LBKE, SKE, RKE, RNGKE, and RSM predict very close velocity and tem-
perature fields, while KW and SST present similar ones. However, velocity and tem-
perature distributions predicted by RSM, SST, and LES differ a lot from each other.

All the concerning models provide practically similar \( j \) and \( f \) values, while some
of them predict much different velocity and temperature fields. The conclusion can
consequently be drawn that comparison only of the heat transfer and pressure drop
data between simulation and experiment seems not quite sufficient for comprehen-
sive model evaluations, and more detailed experimental data are therefore required
for further research.
REFERENCES


