INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 20, 459–468 (1995)

# A REVISED NUMERICAL MODEL TO PREDICT HEAT TRANSFER IN TURBULENT FLOW

J. A. VISSER AND J. P. CILLIERS

*Department of Mechanical Engineering, University of Pretoria, Pretoria, South Africa*

## SUMMARY

The accurate modelling of heat transfer to turbulent flow and the prediction of the temperature distribution in the flow remain one of the problem areas of numerical simulations. Traditional turbulence closure models, like the $k$–$\varepsilon$ model, effectively only increase the viscosity of the fluid and introduce wall functions close to boundaries to obtain the correct velocity distribution. These turbulence models do not model the small-scale mixing that occurs in turbulent flow. When solving the energy equation these small-scale mixings dominate the heat transfer rate at the boundaries as well as the temperature distribution in the flow. This paper outlines a revised method, based on the $k$–$\varepsilon$ turbulence model, that can be used to predict heat transfer in turbulent flow. A single turbulent conductivity term is introduced that can be used over the complete flow field including the boundaries. A detailed description of the mathematical model and boundary conditions used for the turbulence model are included in the paper. The effective turbulent conductivity method was evaluated in several finite difference simulations of water flowing through a smooth pipe while being heated. Simulation and verification were performed over a range of Reynolds numbers. Verification of the model is accomplished by comparing the numerically predicted centre temperature of the fluid as well as the heat flux to the fluid to measured temperatures in a similar pipe. From these results it is concluded that the revised turbulent conductivity model holds great potential to obtain accurate simulated heat transfer rates for general applications.

KEY WORDS: heat transfer; turbulent flow

## INTRODUCTION

Several large industrial installations rely on effective heat transfer processes to optimize their efficiency while minimizing the use of energy resources. This requires a detailed knowledge of the flow and temperature distribution in the fluids taking part in the heat transfer process. One-dimensional heat transfer calculations have therefore become insufficient for the design of complex modern heat transfer processes. During the development phase, more experiments are thus required to reach specified standards which increases the development cost and time of new heat exchangers.

An attractive alternative to this problem is the use of numerical methods to simulate the heat transfer in turbulent flow. Although several numerical simulations of heat transfer in turbulent flow were successful in the past, inaccurate results also occur frequently, the reason being that when the laminar energy equation is solved for turbulent flow, the heat transfer rates at the walls are overpredicted while centre temperatures are underpredicted. This is caused by a very low Peclet number close to the walls which increases sharply towards the centre of the tube preventing heat to penetrate into the fluid. An accurate model is therefore required to modify the method of heat transfer in the turbulent flow.

CCC 0271–2091/95/060459–10


*Received 15 February 1994*
*Revised 2 November 1994*

Through the years a fair amount of work has been done by several authors on the prediction of heat transfer in turbulent flow. One such model was introduced by Kadle and Sparrow[1] in which they investigated turbulent heat transfer in longitudinal fin arrays and used an effective turbulent conductivity ($k_t$) of the fluid in their numerical model. In this paper the numerical results agreed well with experimental measurements. The disadvantage of this numerical model is that it requires a wall function at the boundaries to model the rate of heat transfer accurately. The formulation was also used by Prakash[2] to predict the heat transfer in a ribbed rectangular duct. Devalba and Rispoli[3] used the numerical model to solve the oscillating fluid flow in the tubes. Their study showed that boundary layer effects and abrupt cross-sectional variations were found to influence the temperature distribution in the tube material. In compact heat exchangers extensive work has been done to numerically predict the heat transfer. Sparrow and Ohadi[4] investigated turbulent heat transfer in a tube and verified the numerical model experimentally, for Reynolds numbers between 5400 and 83 500. Results deviated from the measurements by a maximum of 6 per cent. Faghri and Rao[5] used a numerical model to solve heat transfer and fluid flow in-line finned and plain tube banks. Their experimental measurements corresponded well with the results of the numerical model.

The method of defining an effective turbulent viscosity can therefore be used with any turbulenece model. In several other simulations this $k$–$\varepsilon$ based turbulence model was found to be the cause of inaccurate results.[6,7] It can therefore be concluded that, although this turbulent heat transfer model has been applied successfully to several applications for which proper wall functions have been developed, several engineering applications do, however, exist where wall functions are difficult to apply or for which wall functions have not been developed. One such application is the mixing of hot and cold fluids where no formal boundary exists on the interface between these streams.

In this paper an alternative method, based on the $k$–$\varepsilon$ turbulence model, is presented to numerically predict the heat transfer to turbulent flow in a tube and the temperature distribution in the flow. Although the method still does not account for the small-scale turbulence which dominates the heat transfer in turbulent flow, no wall function is required by the new model. The model is evaluated by simulating water flowing through a smooth tube and comparing the results to measurements for the same configuration. The model will be outlined in the following paragraphs.

## NUMERICAL MODEL

### *The governing equations*

Simulations in this study were carried out by solving the full Navier–Stokes equations, using a finite difference solution technique and the $k$–$\varepsilon$ turbulence model. The partial differential equations for the conservation of momentum, mass and energy in a fluid are widely published.[8,9] These equations can be written in the following general form:

*momentum equation*

$$\frac{\partial}{\partial t}(\rho v) + \nabla \cdot (\rho v v) = \nabla \cdot \tau - \nabla p \tag{1}$$

*continuity equation*

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \tag{2}$$

*energy equation*

$$\rho \frac{DH}{Dt} = -[\nabla \cdot q] - [\tau : \nabla v] + \frac{Dp}{Dt} \tag{3}$$

*turbulence equations*

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla \cdot (\rho v \varepsilon) = \nabla \cdot \left( \frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon \right) + C_1 \Gamma \frac{\mu_t \varepsilon}{k} - C_2 \frac{\rho \varepsilon^2}{k} \tag{4}$$

$$\frac{\partial}{\partial t}(\rho k) + \nabla \cdot (\rho v k) = \nabla \cdot \left( \frac{\mu_t}{\sigma_k} \nabla k \right) + \mu_t \Gamma - \rho \varepsilon \tag{5}$$

where

$$\Gamma = 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] + \left[\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right]^2 + \left[\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right]^2 + \left[\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right]^2 \tag{6}$$

The finite difference equations used in the numerical model are obtained by integrating the partial differential equations over control volumes surrounding grid points in the flow field.[10] The general finite difference equation can be written in the following form:

$$a_p \phi_p = \sum a_{nb} \phi_{np} + b \tag{7}$$

where $a_p$ is the coefficient at the central point and $a_{nb}$ are coefficients at neighbouring grid points in the flow field.

### *Boundary conditions*

(a) *Momentum equations*: At the solid boundaries the no-slip condition is used. The near-wall values are corrected by introducing the generalized wall function treatment of Rosten and Worrel.[11] The required shear stress is obtained as

$$\tau_w = s \rho v_t^2 \tag{8}$$

where $s$ is the friction coefficient given by

$$s = \text{greater of} \left[ \frac{\kappa C_\mu^{0\cdot25} k_p^{0\cdot5}}{\ln(E \delta \rho C_\mu^{0\cdot25} k_p^{0\cdot5}/\mu) v_t}; \frac{\mu}{\rho \delta v_t} \right] \tag{9}$$

A zero gradient boundary condition is used along the centre line for the mainstream flow. At the outlet of the section a zero gradient velocity profile is used for all velocity components.

(b) *Turbulence equations*: The general boundary conditions for the turbulent transport equations differ greatly from those for the momentum equations. The inlet values for $k$ and $\varepsilon$ in these simulations are taken as 0·25 J/kg and 50 W/kg respectively.[12] Zero gradient boundaries for both $k$ and $\varepsilon$ are used along the centre line and outflow boundary. Near the solid boundaries where transitional and laminar flow occurs, the turbulent kinetic energy $k_p$, is obtained by solving the differential equation for $k$ with a zero-gradient condition imposed on the wall. When solving this equation the generation rate of $k$ is computed as

$$\Gamma = \tau_w \frac{v_t}{\delta} \tag{10}$$

while the rate of dissipation ($\varepsilon$) is calculated as

$$\varepsilon = \frac{C_\mu^{0\cdot75} k^{1\cdot5} \ln(E(\rho/\mu)\delta C_\mu^{0\cdot25} k^{0\cdot5})}{k\delta} \tag{11}$$

The following fixed value for $\varepsilon$ is adopted near the walls:

$$\varepsilon = \frac{C_\mu^{0\cdot75} k^{1\cdot5}}{\kappa\delta} \tag{12}$$

(c) *Energy equation*: All simulations and measurements were carried out for steady-state conditions for which the temperature of the walls was assumed to remain constant. A constant temperature boundary condition was therefore used when solving the energy equation. As an inlet boundary condition a constant temperature equal to the measured temperature was used. At the outflow boundary and along the centre of the tube a zero gradient temperature boundary condition was used. On the interface between the tube and fluid no boundary condition is required as a result of the turbulent conductivity model developed and outlined in the next paragraph.

## THE REVISED TURBULENT CONDUCTIVITY MODEL

In turbulent flow small-scale mixing occurs which greatly influences the effective heat transfer in the fluid. The turbulent viscosity used in the momentum equations to modify the flow distribution does not account for the effect of the small-scale mixing on the heat transfer rate in the fluid. When solving the energy equation, a turbulence model is required to account for the effect of turbulence on the temperature distribution in a fluid. Current models were found to underpredict the centre temperature of a tube on which experiments were carried out while complicated wall functions are required to correct the heat flux at the walls of the tube. In an attempt to develop a revised turbulent conductivity model a complete numerical simulation including the energy equation was conducted for water flowing through a 3·4 mm stainless steel tube while the temperature of the surroundings remains constant. Without an effective turbulent conductivity model a normalized temperature profile comparing the outlet temperature to the inlet fluid temperature, as shown in Figure 1, is obtained.

The results show a too high temperature close to the wall due to the inability of the fluid to conduct heat from the wall, while the centre temperature of the fluid remains virtually unchanged at the inlet temperature of the fluid. This can be explained by Figure 2 in which the conductivity of the fluid is compared to the convection in the fluid in the form of the Peclet number. For relative low turbulent Reynolds numbers the conductivity close to the wall is able to conduct the heat effectively into the fluid. Further from the wall the convection to conduction ratio becomes large that the fluid does not conduct the heat towards the centre of the fluid.

The purpose of the turbulent conductivity model is then to effectively increase the conduction rate in turbulent flow as a result of the turbulence while at the same time the wall function must ensure the correct heat transfer rate at the boundaries. Most of the existing turbulent conductivity models are based on the turbulent viscosity used in the momentum equations which are dominated by the wall functions. These models can be written in the following general form:

$$k_t = \left(\frac{C_P}{Pr_t}\right)\mu_t \tag{13}$$

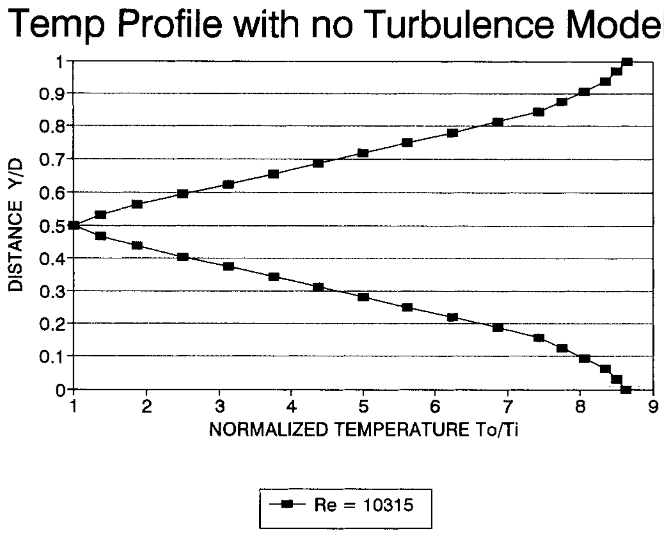


Figure 1. Predicted temperature profile in the tube with no turbulence model

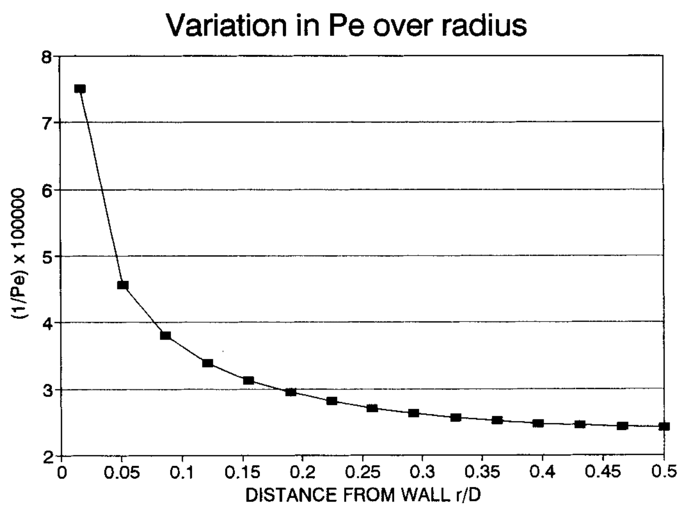


Figure 2. Ratio of conduction to convection without a turbulent conductivity model

In this formulation $C_P$ and $Pr_t$ are close to constants and, based on the $k$–$\varepsilon$ turbulence model, $\mu_t$ is calculated from the following equation:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{14}$$

For a constant temperature boundary a wall function is used at the boundary to calculate the rate of heat transfer to the fluid. The heat transfer at the wall is calculated as follows:

$$q_w = (St\,\rho v_t) C_P (T_w - T_p) \tag{15}$$

where $T_w$ and $T_p$ designate the temperature at the wall and the point p respectively, and $St$ is the Stanton number given by

$$St = \text{greater of}\left[\frac{S_{tur}}{Pr_t(1 + S_{tur}P/C_\mu^{0\cdot 25}k_p^{0\cdot 5})}; \frac{\mu}{\delta Pr\,\rho}\right] \tag{16}$$

From experimental results it was found that this model underpredicts the centre temperature of the water for a given heat flux at the wall. Figure 3 shows the change in $k^2/\varepsilon$, which represents the turbulent conductivity, from the wall to the centre of a typical tube. Close to the wall the turbulent conductivity will therefore be low and increase sharply in the near-wall region to virtually a constant value through the centre part of the tube. At very high Reynolds numbers these effects become even more severe. This implies that apart from having to use a wall function to predict the heat transfer rate at the walls, the grid formulation in the near-wall region will have an important influence on the predicted centre temperature. Together with the fact that larger-scale mixing of fluid occurs at the centre of a tube, effectively increasing the turbulent conductivity in that region, these problems have led to the formation of a somewhat different turbulent conductivity model.

The question that remains unsolved is how to calculate the turbulent conductivity to obtain an accurate predicted temperature distribution in the fluid. From several attempts it was found that the turbulent conductivity must be formulated with great care. If the turbulent conductivity values become too large, the combined conduction and convection terms have the effect that the inlet temperature is conducted along the fluid and very little heat transfer is predicted from the boundaries to the fluid.

The aim with the revised formulation of the turbulent conductivity is to use different combinations of the calculated $k$ and $\varepsilon$ values to obtain an improved method to predict the heat transfer in the turbulent flow. One alternative is to use the ratio of $k/\varepsilon$ instead of $k^2/\varepsilon$. The changes of $k/\varepsilon$ over the flow domain is as shown in Figure 4. Close to the boundary, the values are low and it increases gradually towards the centre of the tube. If these values are superimposed on the laminar conductivity values of the fluid an effective turbulent conductivity value is found with a better balance between the modifications close to the walls and to the centre of the tube. This leads to

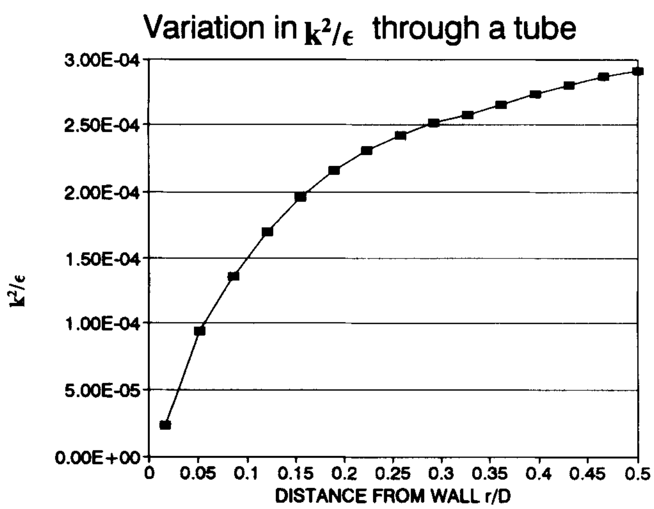


Figure 3. Variation in $k^2/\varepsilon$ through the tube

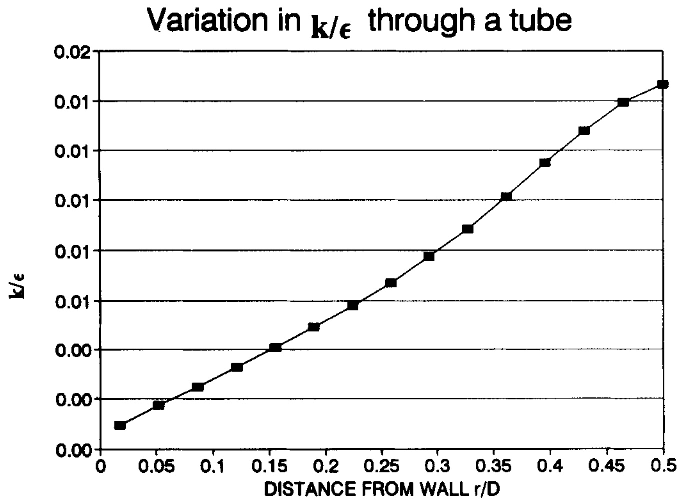


Figure 4. Variation in $k/\varepsilon$ through the tube

the following formulation of the turbulent conductivity value:

$$k_t = \left( C Re_d^n \left( \frac{k}{\varepsilon} \right) \left( \frac{\varepsilon}{k} \right)_{CL} + 1 \right) k_L \tag{17}$$

where $C$ and $n$ are constants with values of $1{\cdot}04393 \times 10^{-5}$ and 1·968 respectively.

From several numerical simulations it was found that with this formulation to turbulent conductivity, accurate results were obtained when predicting the centre temperature in a tube. The results also proved that with this formulation one can remove the previously required wall function and still obtain the same accuracy in predicted temperatures.

## EXPERIMENTAL WORK

Experiments were carried out to calibrate the external heat transfer coefficient as well as the numerical predicted results. The experimental work consisted of measuring the heat transfer to water flowing in a smooth tube. The tube was made of Inconel 625 with an outer diameter of 4·5 mm and an inner diameter of 3·4 mm. The tube was placed in a continuous diesel combustion chamber and on the surface of the tube operating at atmospheric pressure. Temperatures in the combustion chamber were measured with a rhenium–tungsten (B-type) thermocouple for a constant preset mass flow rate of air and diesel into the combustion chamber.

Water was pumped through the pipe at different flow rates resulting in a range of Reynolds numbers from 2580 to 10 315. Chromel–alumel (K-type) thermocouples were placed in the tube at different distances from the wall. Water temperatures were only measured over a flow range in which accurate readings could be obtained and at a distance of at least 40 tube diameters from the inlet so that entrance effects did not influence the readings. Figure 5 illustrates the positions of the thermocouple in the experimental model. The flow rate was measured with a calibrated rotameter. The experiment was repeated many times to ensure a good statistical sample of the readings.

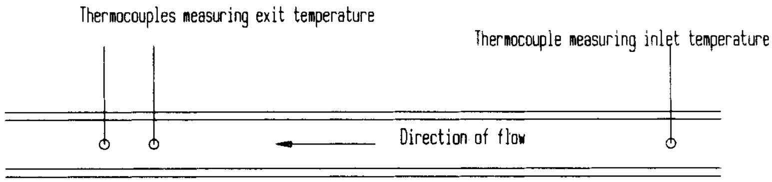


Figure 5. Experimental configuration

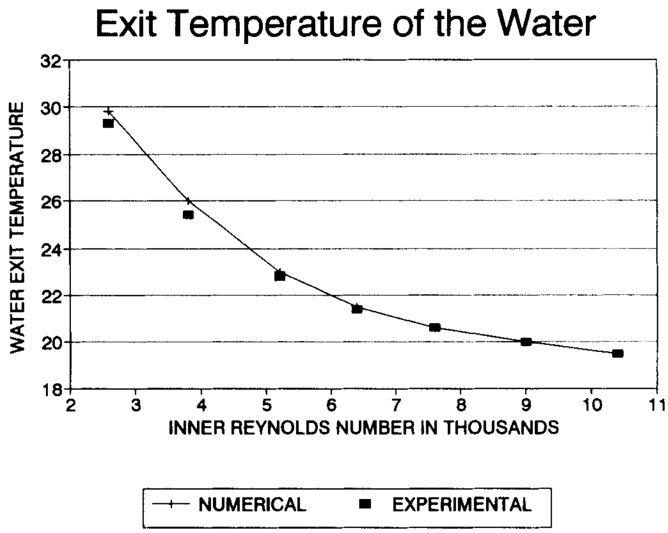


Figure 6. Predicted and measured centre temperatures

## RESULTS

The success of any turbulent conductivity method lies in the accuracy of results obtained with the model. The revised turbulent conductivity model was implemented in a computer code and used to predict the centre temperature in water flowing in the experimental tube. Simulations were carried out for various flow rates and compared to measurements for the same flow conditions. These measured and predicted centre temperatures for different flow rates are shown in Figure 6. In general, predicted and measured temperatures agreed well for Reynolds numbers greater than 5000, based on the inner diameter of the tube. In the transitional area the numerical model slightly overpredicted the centre temperature. This deviation can be caused by the fact that the $k$–$\varepsilon$ turbulence model is only valid for full turbulent flow or the inability of the revised turbulent conductivity model to account for the effect of the turbulence without a wall function being applied.

Numerical predicted heat transfer rates at the boundary of the experimental tube were also compared to the measured heat transfer to the water for different flow rates. Both numerically predicted and measured heat fluxes are presented in Figure 7. For low flow rates the numerical model overpredicted the heat transfer rate slightly, but the inaccuracies were within acceptable limits. For full turbulent flow predicted and measured heat fluxes agreed well for the flow rates evaluated.

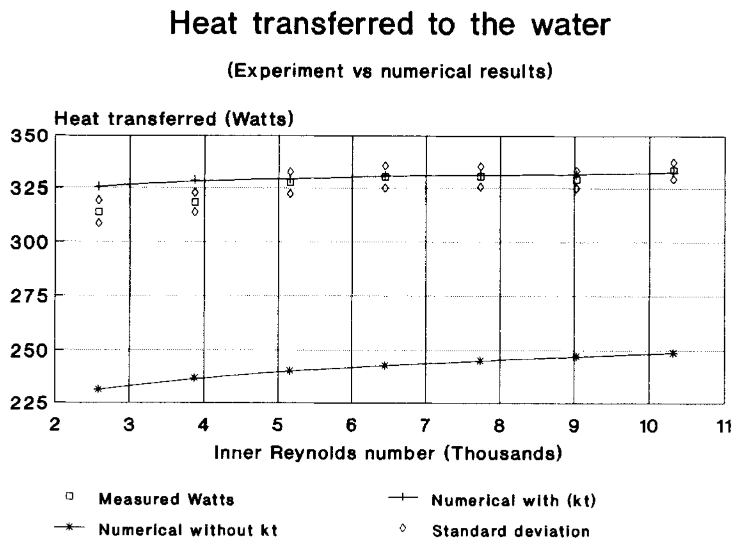


Figure 7. Predicted and measured heat transferred to the water

## CONCLUSION

From this study it can be concluded that the accurate modelling of small-scale turbulence is required for the accurate prediction of the heat transfer and temperature distribution in turbulent flow. Existing models used to calculate the turbulent conductivity are limited in application since wall functions of the energy equation are necessary to obtain accurate predicted values.

In this study a revised effective turbulent conductivity model was introduced to improve the accuracy for heat transfer simulations in turbulent flow. The model is still based on the $k$–$\varepsilon$ turbulence model but does not require a wall function when a temperature boundary condition is prescribed. This is an interesting and attractive alternative to simulate the turbulence conductivity over the complete flow field without requiring a prescribed wall function at the solid boundaries. The revised model also opens up the opportunity to simulate heat transfer across thermal boundaries in one flow domain accurately.

At this stage the model has only been tested and verified for tube flow in a particular flow range. Although the revised model holds great potential for complicated engineering problems where wall functions are difficult to be prescribed, care must be taken not to implement the model in general application without further investigations to validate the results for different geometries and over a wider range of Reynolds numbers.

## APPENDIX

*Nomenclature*

| | |
|---|---|
| $C$ | Constant |
| $C_P$ | specific heat (kJ/kg °C) |
| $C_\mu$ | turbulence constant |
| $C_1, C_2$ | turbulence constant |

$E$ turbulence constant
$H$ enthalpy (J)
$k$ turbulent kinetic energy (J)
$k_{subs}$ thermal conductivity of a fluid/material (W/m °C)
$p$ pressure (Pa)
$Pe$ Peclet number
$Pr$ Prandtl number
$q$ heat flux (W)
$Re$ Reynolds number based on tube diameter
$S_\phi$ source term
$t$ time (s)
$v$ velocity (m/s)
$v_t$ tangential velocity (m/s)
$y^+$ local Reynolds number

*Greek letters*

$\delta$ distance to the wall (m)
$\varepsilon$ rate of dissipation ($s^{-1}$)
$\phi$ general dependent variable
$\rho$ density (kg/m$^3$)
$\sigma_\varepsilon, \sigma_k$ turbulence constants
$\tau$ shear stress (Pa)
$\mu$ viscosity (kg/m s)

*Subscripts*

CL centre line
L laminar
nb general neighbour grid point
p central grid point under consideration
t turbulent

## REFERENCES


1. D. S. Kadle and E. M. Sparrow, 'Numerical and experimental study of turbulent heat transfer and fluid flow in longitudinal fin arrays', *J. Heat Transfer Trans. ASME*, **108,** 16–23 (1986).
2. C. Prakash and R. Zerkle, 'Prediction of turbulent flow and heat transfer in a ribbed rectangular duct with and without rotation', *Int. Gas Turbine and Aero-engine Congress*, Cincinnati, Ohio, *Paper 93-GT-206,* 1993.
3. M. Devalba and F. Rispoli, 'The oscillating fluid flow effects on the Stirling engine heat exchanger design', *ISEC 5th Int. Stirling Engine Conf.*, Dubrovnik, Yugoslavia, 1991, pp. 195–210.
4. E. M. Sparrow and M. M. Ohadi, 'Numerical and experimental studies of turbulent heat transfer in a tube', *Numer. Heat Transfer*, **11,** 461–476 (1987).
5. M. Faghri and N. Rao, 'Numerical computation of flow and heat transfer in finned and unfinned tube banks', *Int. J. Heat Mass Transfer*, **30,** 363–372 (1987).
6. J. J. Wormeck, 'Computation of turbulent reacting flows', Dynamics Technology Inc; California; 90505. 1983.
7. M. A. Habib and J. H. Whitelaw, 'Velocity characteristics of confined coaxial jets with and without swirl', *J. Fluids Eng.*, **102,** 47–53 (1980).
8. D. A. Anderson, J. C. Tannehill and R. H. Pletcher, *Computational Fluid Mechanics and Heat Transfer*, Hemisphere, USA, 1984.
9. J. F. Thompson, Z. U. A. Warsi and C. W. Mastin, *Numerical Grid Generation Foundations and Applications*; Elsevier, New York, 1985.
10. S. V. Patankar, *Numerical Heat Transfer and Fluid Flow*. Hemisphere, New York, 1980.
11. H. I. Rosten, J. K. Worrel, 'Generalised wall functions for turbulent flow', *PHOENICS J.*, **1,** 81–109 1988.
12. M. R. Marlyn, D. G. Tatchell and Ying-Shi, 'Calculations of steady three-dimensional turbulent reacting flow in a combustion chamber', *CHAM Paper, PDR/CHAM UK/19*, 1982.