VERIFICATION AND VALIDATION OF THREE DIFFERENT CFD CODES IN SIMULATING MIXED CONVECTION FLOWS USING TWO ADVANCED EDDY-VISCOSITY MODELS

Amir Keshmiri 1,2,*

1Advanced Design and Simulation Group, School of Engineering, Manchester Metropolitan University, Manchester, M1 5GD, U.K.

2School of Mechanical, Aerospace and Civil Engineering (MACE), The University of Manchester, Manchester, M13 9PL, U.K.

ABSTRACT

Verification and Validation (V&V) of CFD codes and turbulence models has been an on-going research area for decades. One of the sectors where CFD V&V plays an important role is the nuclear industry where simulation tools must have a sufficient degree of maturity and reliability. The aim of the present work is to carry out an assessment of three different CFD codes in simulating ascending mixed convection flows, a representative flow in the core of gas-cooled nuclear reactors under ‘post-trip’ conditions. For validation purposes, comparison is made against the Direct Numerical Simulation (DNS) data. The CFD codes used in the present study are ‘CONVERT’, ‘STAR-CD’, and ‘Code_Saturne’, which are respectively in-house, commercial, and industrial packages. Computations are carried out using two advanced Reynolds-Averaged Navier-Stokes (RANS) models, namely the $k$-$\omega$-SST model and the non-linear eddy-viscosity model of Suga. The results of CONVERT and STAR-CD are compared using the Suga model, while the $k$-$\omega$-SST model is used when comparing the computations of Code_Saturne and STAR-CD. In general the two models were found to perform poorly in capturing the laminarization phenomenon present at certain heat loading. However, mean flow profiles showed that there is good agreement between the codes despite using different numerical procedures. Some discrepancies were found when comparing the turbulence profiles which are mainly associated with the implementation of the turbulence models.

KEYWORDS: Convection, Turbulent Transport, Computational Methods, Verification and Validation, Mixed Convection, $k$-$\omega$-SST Model, Suga Model, RANS, Nuclear Safety.

1. INTRODUCTION

1.1 Background

- CFD Codes

In the past few decades, computational techniques have been used in simulating the operation and the safety of nuclear reactor systems. During this time, several computational codes have been developed by different institutions for better estimation of uncertainties and to improve the basis for regulatory and design decisions. Amongst different simulation techniques, Computational Fluid Dynamics (CFD) plays one of the most important roles in this field, particularly in the safety analysis of the current and future reactor designs. However, application of CFD to problems related to nuclear reactor safety (NRS) requires these simulation tools to have reached a sufficient degree of maturity and reliability.

At present there are several commercial CFD packages available which are currently being used for a wide range of applications including the nuclear industry. Amongst these packages, codes such as CFX (Ansys),

*Corresponding Author: a.keshmiri@mmu.ac.uk; a.keshmiri@manchester.ac.uk
Fluent (Ansys), STAR-CD (CD-Adapco) and STAR-CCM+ (CD-Adapco) are widely and increasingly used in NRS applications. Other industrial codes such as ‘Code_Saturne’ developed by EdF and ‘TRIO-U’ developed by French ‘Atomic Energy and Alternative Energies Commission’ (or CEA) are also widely used in the nuclear reactor modelling. In addition to these codes, there are several less popular in-house CFD codes which have been developed for the nuclear industry at various research institutions and universities in different countries. Therefore, there is certainly a need to benchmark the various simulations being undertaken, and validate these code predictions against experimental data, where available [1].

- Mixed Convection

Mixed convection occurs in a variety of engineering applications including nuclear reactors. In the currently-operating UK fleet of Advanced Gas-cooled Reactor stations, and also in proposed ‘Generation IV’ Very High Temperature Reactor (VHTR) designs, the core coolant flows vertically. The coolant in AGRs is CO₂ and the principal flow ascends through the core; in VHTRs the coolant is helium and the flow descends. An important NRS application of mixed convection flows is related to ‘post-trip’ decay heat removal in nuclear reactor cores, where the heat loading from the fuel elements is relatively large in relation to the low primary coolant flow rate. Under such conditions buoyancy effects have the potential to cause wholesale modifications to the turbulence structure.

The review papers of Jackson et al. [2] and Jackson [3] provide extended discussions of heat transfer performance under mixed convection conditions. The most popular CFD technique adopted in simulating mixed convection flows have been based on the solution of the Reynolds-Averaged Navier-Stokes (RANS) equations. Amongst all the RANS models, the Eddy-Viscosity Models (EVMs) have been the most common and thus have been used by the majority of the researchers. Reader interested in further information on the performance of RANS models as well as Large Eddy Simulation in mixed convection flows, is referred to Keshmiri et al. [4, 5].

- Turbulence Models

In general, the selection of code is often made on the basis of experience, user-friendliness, tradition or cost, or a combination. However, the most important consideration is the correct use of the chosen CFD code [1]. For example, usually there are several turbulence models available in CFD codes and according to previous benchmarking exercises, the resolved structure of the flow field are significantly affected by the choice of the turbulence model [6].

In the Best Practice Guidelines (BPGs) issued by Smith et al. [7], the $k-\omega$-SST model of Menter [8] was found to perform the best in all nine test cases and was therefore recommended for use in NRS applications. In the present study therefore, computations are carried out using the $k-\omega$-SST model as well as another advanced Reynolds-Averaged Navier-Stokes (RANS) model, the non-linear eddy-viscosity model of Suga [9]. These turbulence models are further explained in Section 3.

1.2 Aims and Objectives

The aim of the present work is to carry out a meticulous assessment of three different CFD codes in simulating a mixed convection flow problem which has direct relevance to nuclear reactors. In addition, this test case has been chosen because reliable Direct Numerical Simulation (DNS) and experimental data are available for validation purposes.

The case studied here consists of an ascending mixed convection flow in a vertical pipe, a representative flow in the core of gas-cooled nuclear reactors under ‘post-trip’ conditions and the results are validated against the DNS data of You et al. [10].

Although the present test case is geometrically simple, it would be viewed as challenging from turbulence modelling point of view. On the other hand, its relative simplicity allows a better control over the numerical procedures involved in the simulations.

The CFD codes used in the present study are ‘CONVERT’, ‘STAR-CD’, and ‘Code_Saturne’, which are respectively in-house, commercial, and industrial packages. They codes are briefly in the next section.
The results of CONVERT and STAR-CD are compared using the Suga model, while both turbulence models are used to compare the computations of Code_Saturne against STAR-CD. Table 1 provides an overview of the tests undertaken in the present study.

Table 1. Overview of the tests undertaken in the present work.

<table>
<thead>
<tr>
<th>Codes</th>
<th>Turbulence Models</th>
<th>No. of Grids</th>
<th>Ref. data</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONVERT</td>
<td>Non-linear k-ε</td>
<td>2</td>
<td>DNS of You et al. [10]</td>
</tr>
<tr>
<td>STAR-CD</td>
<td>k-ω-SST</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Code_Saturne</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

2. COMPUTATIONAL CODES

2.1 CONVERT

One of the CFD codes used in the present study is an in-house code, known as ‘CONVERT’ (for Convection in Vertical Tubes). CONVERT was originally developed by Cotton [11] and later extended by a number of researchers at the University of Manchester; the latest version which is used here is due to Keshmiri et al. [4]. The code has been mainly developed and used for nuclear applications and the results have been reported in Cotton and Jackson [12], Cotton and Kirwin [13], Cotton et al. [14], Keshmiri [5] and Keshmiri et al. [4], amongst others.

The code is written in the ‘thin shear’ (or ‘boundary layer’) approximation. The thin shear equations are of parabolic form and therefore the programme is able to ‘march’ in the streamwise direction. More information about the computational procedure used in CONVERT are given in Section 4.2.1, below. The reader interested in more detailed information about CONVERT is referred to Cotton [11] and Keshmiri [15].

2.2 Code_Saturne

Code_Saturne [16] is the Electricité de France (EDF) in-house CFD tool for incompressible flows. This open-source code has been used in several nuclear reactor projects including Addad et al. [17], Péniguel et al. [18], Fournier et al. [19], Keshmiri et al. [4], Ndombo and Howard [20] and Rolfo et al. [21], amongst others [20].

Code_Saturne is based on an unstructured collocated finite-volume approach for cells of any shape with Rhie and Chow interpolation and a SIMPLEC algorithm for pressure correction. For convection, Code_Saturne has a slope test based on the product of the gradients at the cell centres to dynamically switch from second order centred scheme to first order upwind scheme. The slope test aims at preventing some physical values from converging beyond the bounds imposed by the boundary conditions.

The time scheme is second order based on Crank-Nicolson/Adams Bashforth scheme (the diffusion is totally implicit whereas the convection is semi-implicit). While using RANS or URANS, a centred scheme with a slope test is used for the velocity components and the temperature and a first order upwind scheme for the turbulent quantities.

2.3 STAR-CD

The third code to be used here is ‘STAR-CD’ [22], a commercial unstructured CFD package developed by CD-Adapco. STAR-CD has widely been used by both the academia and the nuclear industry and some of the recent examples include Shang, [23], Simoneau et al. [24], Keshmiri [25], Keshmiri and Gotts [26] and Simoneau et al. [27], amongst others.

In common with Code_Saturne, the programme solves the governing equations using a collocated finite volume approach. Similar to Code_Saturne, STAR-CD is a co-localized cell centred incompressible Navier-Stokes solver. In the present STAR-CD computations, the momentum and turbulence transport equations are discretized using second order central differencing and first order upwind differencing schemes, respectively.
The SIMPLE algorithm is adopted for pressure-velocity correction. The energy equation is discretized using the ‘Monotone Advection and Reconstruction Scheme’ (MARS) [22].

3. TURBULENCE MODELS

3.1 Standard k-ω-SST Model (STAR-CD & Code_Saturne)

Advantages of both the k-ε and k-ω models are combined in the Shear Stress Transport (SST) model of Menter [8]. Through a blending function this model effectively uses the Low Reynolds Number (LRN) formulation of the k-ω model in the boundary layer and a version of the k-ε model (usually the ’standard k-ε model’) in the free shear layer. This is based on the observations that the k-ε model is much less sensitive to the free-stream value of \( \varepsilon \) than the k-ω model is to \( \omega \). Apart from this unique feature, the main differences between the k-ω model and the SST model are the following:

- The SST model includes a damped cross-diffusion derivative term, as well as a blending function, in the \( \omega \)-transport equation.
- The definition of the turbulent viscosity in the SST was modified to improve the prediction of the turbulent shear stress.
- The coefficients were modified to improve the overall performance of the model.

Note that the functions and coefficients of the k-ω-SST model may be found in several text books and are not included here for the sake of brevity. However, it is worth noting that the k-ω-SST models implemented in STAR-CD uses the original version of k-ω-SST model proposed by Menter [8] (except the value of \( \sigma_\varepsilon \) in STAR-CD is set to 0.85 instead of 0.5).

3.2 Suga Non-Linear k-ε Model (CONVERT & STAR-CD)

Non-linear eddy viscosity models are generally viewed as potential candidates to replace the well-known linear k-ε model [28]. Craft et al. [9] developed a non-linear two-equation model (this model was originally developed by Suga [29], thus it is usually known as the ‘Suga’ model) in which quadratic and cubic mean strain and vorticity terms were introduced into the constitutive equation (i.e. the general stress-strain relationship, \( -\vec{\nu}\vec{\nu}/k = C_\nu (k/\varepsilon) \partial U / \partial y \)).

Similar to the Launder-Sharma (LS) model [30], the transport equations for \( k \) and \( \tilde{\varepsilon} \) in the Suga model take the following forms:

\[
\frac{Dk}{Dt} = P_k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \tilde{\varepsilon} + 2\nu \left( \frac{\partial (k^{1/2})}{\partial x_j} \right)^2
\]

\[
\frac{D\tilde{\varepsilon}}{Dt} = C_\omega \frac{\tilde{\varepsilon}}{k} P_t + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \tilde{\varepsilon}}{\partial x_j} \right] - C_\omega f_\varepsilon \tilde{\varepsilon} + E_\varepsilon
\]

where \( P_k = -\bar{u}_j \bar{u}_j \left( \partial U_i / \partial x_j \right) \). One of the unique features of the Suga model was in the use of the new ‘strain-sensitive’ \( C_\nu \) coefficient instead of the typical \( C_\nu = 0.09 \) adopted by a wide range of eddy-viscosity models including the LS formulation.

More detailed descriptions of the three turbulence models may be found in the original papers and Keshmiri et al. [4].

4. CASE DESCRIPTION AND MESH

4.1 Introduction

The case studied here consists of a long vertical pipe for which the thermal boundary condition is one of uniform wall heat flux. The working fluid is assumed to be standard air and the Reynolds number based on
the pipe diameter is set to $Re = 5,300$. The Prandtl number of standard air ($Pr = 0.71$) is used throughout calculations. In addition, all fluid properties are assumed to be constant and buoyancy is accounted for within the Boussinesq approximation.

In computing mixed convection flows, You et al. [10] retained the same Reynolds and Prandtl numbers and varied buoyancy influence via the Grashof number. A total of four simulations were performed and these are detailed in Table 2. In each case a brief description of the thermal-hydraulic regime is included in the table. The mean flow and turbulence profiles presented in Section 5 are reported for the four thermal-hydraulic regimes indicated in Table 2.

### Table 2. DNS cases of You et al. [10].

<table>
<thead>
<tr>
<th>Case</th>
<th>$Gr/Re^2$</th>
<th>Bo</th>
<th>Thermal-Hydraulic Regime</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0</td>
<td>0</td>
<td>Forced convection</td>
</tr>
<tr>
<td>B</td>
<td>0.252</td>
<td>0.13</td>
<td>Early-onset mixed convection</td>
</tr>
<tr>
<td>C</td>
<td>0.348</td>
<td>0.18</td>
<td>Laminarization</td>
</tr>
<tr>
<td>D</td>
<td>0.964</td>
<td>0.50</td>
<td>Recovery</td>
</tr>
</tbody>
</table>

### 4.2 Grids

#### 4.2.1 CONVERT

The mesh used for the present CONVERT computations consists of 100 control volumes and a double expansion technique is employed to ensure good resolution of the near-wall flow (the wall-adjacent node is typically located at $y^+ = 0.5$ and half the nodes are located between the wall and $y^+ \approx 60$). Small steps (of 0.001$R$) are taken in the axial direction and at-station iteration is applied to ensure a converged solution before the computation is advanced to the next location downstream.

The solution technique used in CONVERT consists of 2 separate ‘RUNS’: first, an initial isothermal run (‘RUN 1’) is undertaken in which the dynamic field is allowed to develop from approximate initial profiles to a fully-developed state. Next, a mixed convection run, ‘RUN 2’ reads the fully-developed mean flow and turbulence profiles from RUN 1 at $x = 0$. A uniform wall heat flux is applied and the buoyancy force term is activated. In the present work RUN2 is typically extended 50 diameters downstream of $x = 0$. Interested readers are referred to Keshmiri et al. [4] for further details on CONVERT solution sequence.

#### 4.2.2 Code_Saturne and STAR-CD

The mesh used for the present Code_Saturne and STAR-CD computations consists of a 2-degree sector of the pipe cross-section. Periodic boundary conditions are applied in the streamwise ($x$) direction, while symmetry boundary conditions are applied at the two azimuthal ($\theta$) faces. The face at $r = R$ represents the pipe wall. In principle, only one cell is necessary in the streamwise direction, however, five are used to promote convergence. There are 120 cells in the radial direction and the wall-adjacent cell is positioned at $0.15 < y^+ < 0.2$.

### 5. RESULTS

#### 5.1 Verification of the Suga Model in CONVERT

As part of the present study, the Suga non-linear $k$-$\varepsilon$ model [9] was implemented in CONVERT by the author. The implementation of the model was then verified and validated against the results presented in Suga [29] for a channel flow at $Re = 5,600$ and 14,000, based on the DNS data of Kim et al. [31]. Some of the results of the validation tests for $Re = 5,600$ are shown in Fig. 1. It can be seen that the present results match those reported in [9] with excellent agreement.
Fig. 1. Verification and validation tests for the Suga model implemented in CONVERT for \( R_e = 5,600 \).

5.2 Overview of Heat Transfer Impairment and Enhancement

Fig. 2 provides an overview of heat transfer performance in ascending flow. Nusselt number in mixed convection, \( Nu \) is normalized by the corresponding forced convection value evaluated at the same Reynolds and Prandtl numbers using a re-optimized form of the Dittus-Boelter equation:

\[
Nu_0 = 0.022 \, R_e^{0.8} \, P_f^{0.5}\]

In Fig. 2 \( Nu/Nu_0 \) is plotted against the buoyancy parameter defined as

\[
Bo = 8 \times 10^4 \frac{Gr}{R_e^{1.425} P_f^{0.8}}
\]

Present turbulence model results are shown together with the ascending flow simulations of Launder-Sharma (LS) \( k-\varepsilon \) model [30] (taken from Keshmiri et al. [4]). For validation purposes, the DNS data of You et al. [10] and the experimental results of Steiner [32] and Carr et al. [33] are also included in the figure.

A notable feature of Fig. 2 is the sudden onset of heat transfer impairment that occurs at \( Bo \approx 0.2 \). Examining the EVM results, it is seen that the LS model is in closest agreement with the direct simulation data. Significantly lower levels of heat transfer impairment are returned by the Suga model and the onset of
impairment is delayed considerably. In the ‘recovery’ region \((Bo \geq 0.5)\) the LS model is in close agreement, but the Suga model yields lower \(Nu/Nu_0\). For the Suga model, in cases with relatively high \(Bo\), converged solutions could not be obtained, generally due to stability problems associated with this model which are believed to be associated with the dependence of \(C_\mu\) on strain rate [34, 35].

The \(k-\omega\)-SST model as implemented in both STAR-CD and Code_Saturne performs particularly poorly, although there is fairly close agreement between the two codes. The present \(k-\omega\)-SST results are similar to those obtained by Cotton and Kirwin [13] who used the original Wilcox \(k-\omega\) model [36]. The disappointing performance of \(k-\omega\)-based models is thought to be attributable, at least in part, to the omission of damping functions in the formulations [4].

Fig. 2. Heat transfer impairment and enhancement in ascending mixed convection flows.

5.3 CONVERT vs. STAR-CD

In this section, the results of STAR-CD and CONVERT are compared against each other using the cubic non-linear model of Suga [9]. Before discussing the results of this section, it is worth mentioning that in the course of the present study, numerical instability problems were encountered with the Suga model for cases with high buoyancy influence in both CONVERT and STAR-CD. These numerical issues were more significant in STAR-CD and consequently no solution could be obtained for \(Bo = 0.5\) (case D) using this code. Further information on the numerical stability problems of the Suga model can be found in Craft et al. [35].

In Fig. 3 to Fig. 5, comparison is made for the velocity, temperature, and turbulent kinetic energy for cases (A)-(C).

In the forced convection case in Fig. 3, both codes return nearly identical velocity and temperature profiles, while STAR-CD returns a turbulent kinetic energy profile that is lower than that returned by CONVERT.
Fig. 3. Comparison of the results for Case (A) obtained using the Suga model in CONVERT and STAR-CD.

For case (B), the results of both codes are somewhat different as shown in Fig. 4; the velocity and temperature profiles obtained using CONVERT return lower values, especially near the pipe centre-line. As can be seen in Fig. 4 (c) the prediction of the Suga model in CONVERT for $k$ is closer to the DNS data which is consistent with the predictions shown in Fig. 3 (c).

For the laminarized regime (case C) in Fig. 5, the discrepancies in the results of both codes are again negligible for the velocity and temperature profiles, while for the turbulent kinetic energy, STAR-CD returns lower values with the maximum difference of about 12%.

It is worth noting that as was mentioned earlier, the governing equations in CONVERT are parabolic and therefore, the code marches in the streamwise direction, while in STAR-CD a cyclic boundary condition with constant mass flow rate has been employed at inlet and outlet. This implies that, while in CONVERT the pipe length is set to $50D$, in STAR-CD the pipe length can be assumed to be infinite as a result of using cyclic boundary condition. This difference along with the differences associated with the mesh size and type used in both codes could perhaps be blamed for the discrepancies found between the results.
Fig. 4. Comparison of the results for Case (B) obtained using the Suga model in CONVERT and STAR-CD.
In this section, comparison is made for the results obtained using the \( k-\omega \)-SST model in STAR-CD and \textit{Code_Saturne}. Profiles of the normalized velocity, temperature, turbulent kinetic energy and Reynolds shear stress for cases (A) and (D) are compared in Fig. 6 and Fig. 7.

In the forced convection condition (case A) in Fig. 6, the profiles obtained from both codes are acceptably close, except for the turbulence kinetic energy (Fig. 6 c) where the \( k-\omega \)-SST model in STAR-CD returns levels that are lower by about 15% (at \( y^+ \approx 50 \)).

The largest discrepancies between the results of STAR-CD and \textit{Code_Saturne} occur in the recovery regime (case D; \( Bo = 0.5 \)), as shown in Fig. 7. While the velocity profiles are reasonably close, the temperature profiles returned by STAR-CD seem to be more turbulent that that found by \textit{Code_Saturne} (in spite of both codes returning very close values of \( Nu/Nu_0 \) at \( Bo = 0.5 \) – see Fig. 2, above).

In Fig. 7 (c) similar to case (A), STAR-CD returns somewhat lower levels of turbulent kinetic energy up to \( y^+ \approx 50 \). However, the turbulent kinetic energy profile returned by \textit{Code_Saturne} has a slightly steeper gradient thus, returning lower values of \( k \) beyond \( y^+ \approx 50 \). For the Reynolds shear stress shown in Fig. 7 (d), STAR-CD returns values that are higher by approximately 20% (at \( y^+ \approx 50 \)).
Fig. 6. Comparison of mean flow and turbulence profiles for Case (A) obtained using the standard \( k-\omega-SST \) model in STAR-CD vs. Code_Saturne.

Fig. 7. Comparison of mean flow and turbulence profiles for Case (D) obtained using the standard \( k-\omega-SST \) model in STAR-CD vs. Code_Saturne.
6. CONCLUSIONS

Numerical simulations were carried out for an ascending forced and mixed convection flows in a vertical pipe, a representative flow in the core of gas-cooled nuclear reactors under ‘post-trip’ conditions. The aim of this work was to benchmark the results of three different CFD codes which are used by the nuclear industry. The CFD codes tested in the present study were ‘CONVERT’, ‘STAR-CD’, and ‘Code_Saturne’, which are respectively in-house, commercial, and industrial packages. The latter is an opensource CFD code developed and maintained by Electricité de France (EdF), the world’s largest nuclear power generator.

Two advanced eddy-viscosity-based turbulence models, namely the non-linear k-ε model of Suga [9] and the standard k-ω-SST model of Menter [8] were tested. The numerical results were validated against the forced and mixed convection DNS data of You et al. [10]. In spite of some major differences in the numerical procedures used by each code, in all cross-code comparison tests, good agreement was obtained for velocity and temperature profiles. However, some discrepancies were present in the Reynolds shear stress and turbulent kinetic energy profiles which were found to be mainly associated with the implementation of the turbulence models in each code.

Despite their success in a wide range of flow problems, the Suga and k-ω-SST models were found to perform poorly in capturing heat transfer impairment in ascending mixed convection flows. Further work is required to identify the exact reasons behind their failure in capturing the extent of the laminarization phenomenon.

To facilitate future cross-code and turbulence model comparisons, all the results presented in this paper have been uploaded to ‘www.CFDtm.org’.

ACKNOWLEDGEMENTS

This work was partially funded by the UK Engineering and Physical Sciences Research Council ‘Towards a Sustainable Energy Economy’ (TSEC) programme ‘Keeping the Nuclear Option Open’ (KNOO). The author is also pleased to acknowledge the value of discussions with his colleagues at the School of Mechanical, Aerospace and Civil Engineering at the University of Manchester, especially Dr. M.A. Cotton, Dr. Y. Addad and Professor D. Laurence.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bo</td>
<td>Buoyancy parameter, $8 \times 10^4 , Gr \left( \frac{Re^{1/2 \times 10^4}}{Pr^{0.8}} \right)$</td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>Constant in Eddy-Viscosity Models</td>
</tr>
<tr>
<td>$D$</td>
<td>Pipe diameter</td>
</tr>
<tr>
<td>$Gr$</td>
<td>Grashof number, $\beta g D^4 q / (\lambda V^2)$</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy, $\overline{u_i u_i}/2$</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number, $q D / \lambda (T_w - T_p)$</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$P_k$</td>
<td>Rate of shear-production of $k$, $-\overline{u_i u_j (\partial U_j/\partial x_i)}$</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Wall heat flux</td>
</tr>
<tr>
<td>$R$</td>
<td>Pipe radius</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number, $U_j D / \nu$</td>
</tr>
<tr>
<td>$Re_t$</td>
<td>Turbulent Reynolds number, $k^2 / (\lambda \nu)$</td>
</tr>
<tr>
<td>$U_i, u_i$</td>
<td>Mean, fluctuating velocity components in Cartesian tensors</td>
</tr>
<tr>
<td>$\overline{u_i u_j}$</td>
<td>Reynolds stress tensor</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Dimensionless distance from the wall, $y U_j / \nu$</td>
</tr>
</tbody>
</table>

Greek Symbols:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon$</td>
<td>Rate of dissipation of $k$</td>
</tr>
<tr>
<td>$\tilde{\varepsilon}$</td>
<td>Modified dissipation rate, $\varepsilon - 2\nu (\partial k^{1/2}/\partial x_i)^2$</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Thermal conductivity</td>
</tr>
</tbody>
</table>
v_t \text{ Turbulent viscosity}

Additional symbols are defined in the text.

REFERENCES


