Proceedings of the 3rd IASME/WSEAS Int. Conf. on HEAT TRANSFER, THERMAL ENGINEERING AND ENVIRONMENT, Corfu, Greece, August 20-22, 2005 (pp311-315)

Turbulent Heat Transfer Modelling in a Vessel Stirred by a Pitched Blade Turbine Impeller

BARBARA ZAKRZEWSKA, ZDZISŁAW JAWORSKI
Chemical Engineering Faculty
Szczecin University of Technology
Al. Piastów 42, 70-065 Szczecin
POLAND
jaworski@ps.pl zakrzewska@ps.pl http://www.pmg.ps.pl

Abstract: - The CFD modelling of the turbulent heat transfer in a stirred vessel has been presented. Four turbulence models, i.e. the standard and optimized Chen-Kim k-ε and also the standard and shear-stress transport k-ω models along with two near-wall region models were used in the modelling. The predicted local values of the heat transfer coefficient favourably compared with the experimental data from literature.

Key-Words: - CFD modelling, heat transfer, turbulent flow, stirred tank, PBT impeller, turbulence models.

1 Introduction

Numerical modelling of heat transfer with the use of CFD codes has been widely applied in process engineering. However, only few papers were published on that modelling in stirred vessels. Kaminoyama et al. [1] and Kuncewicz et al. [2] obtained a good agreement between the predicted and the experimental results in the laminar flow regime. However, in the turbulent flow regime this agreement was poor [3, 4]. In the earlier publication [5], concerning the CFD modelling in the Rushton turbine stirred vessel, two possible reasons of the underprediction of the heat transfer coefficient were given. One of them was low accuracy of the turbulent momentum transfer simulation in the vessel, especially in terms of the turbulence characteristics in the wall jet. In this paper a heat transfer modelling methodology was proposed, which has led to better agreement between the CFD and experimental data, being close to the experimental error level. The same methodology was used in the present investigations.

2 Modelling methodology

The computational fluid dynamics (CFD) method was applied to the heat transfer modelling in a jacketed stirred tank. The vessel with a dished bottom had the diameter, T = 0.158m, and was equipped with a pitched blade turbine impeller and four standard baffles. The vessel was filled with stirred fluid to the height H = T and the fluid had physical properties similar to those of water. The four bladed impeller had the diameter D = 0.483T and was situated axially at the off-bottom distance, C = 0.333H, and had the rotational speed, N, resulting in Re = 3.4×10^4. The vessel geometry and operating conditions were identical to those used in the experiments published by Post [6].

The standard Reynolds averaged Navier-Stokes (RANS) equations (Eq. (1)) along with the continuity (Eq. (2)) and energy transport (Eq. (3)) equations were numerically solved with the help of the Fluent 6.0 code. The source term S_T on the left-hand side of Eq. (3) represents the viscous and turbulent energy dissipation function. It is usually omitted, except for the systems when the large viscosity and the large velocity gradients appear. The values of the turbulent thermal conductivity, λ_t, in Eq. (3) were calculated from turbulent Prandtl number (Eq. 4), which was assumed to be equal to 0.85.

$$\frac{\partial(\rho \bar{v}_i)}{\partial t} + \frac{\partial(\rho \bar{v}_j \bar{v}_i)}{\partial x_j} = \frac{\partial}{\partial x_j}\left((\mu + \mu_t)\frac{\partial \bar{v}_i}{\partial x_j}\right) + \rho F_i - \frac{\partial \bar{p}}{\partial x_i} \quad (1)$$

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \quad (2)$$

$$\frac{\partial(\rho c_p \bar{T})}{\partial t} + \frac{\partial}{\partial x_j}(\rho c_p \bar{v}_j \bar{T}) = \frac{\partial}{\partial x_j}\left((\lambda + \lambda_t)\frac{\partial \bar{T}}{\partial x_j}\right) + S_T \quad (3)$$

$$Pr_t = c_p \frac{\mu_t}{\lambda_t} \quad (4)$$

Based on the previous numerical investigations [5], four selected turbulence models were used in order to close the equation set. The following models were tested: the standard [7] and optimized Chen-Kim [8] k-ε models and also the standard k-ω [9] and the shear-stress transport k-ω (k-ω SST) [10]. The standard logarithmic wall functions were applied to describe the boundary flow at the vessel wall. In addition, the standard k-ε model along with the enhanced wall treatment was used in the CFD simulations.

Good prediction of the flow field and turbulence intensity, especially near the wall region is very important for the numerical heat transfer modelling. The effect of both the numerical grid density and also the turbulence model on the turbulent momentum transfer modelling in the wall jet region was reported elsewhere [11, 12] for the standard stirred tank equipped with pitched blade turbine impeller and four baffles. The effect of the turbulence models from the k-ω family for the same geometry was also investigated [13]. It was concluded that the flow field was predicted quite well, whereas the turbulence kinetic energy was underpredicted, when the 100K cells grid and the standard and optimized Chen-Kim k-ε models were used. The k-ω models gave a lower accuracy of the predicted mean velocity and turbulence kinetic energy than the standard k-ε model. The fine grid (340K cells) was rejected, because the values of the nondimensional distance from the tank wall, y^+, were below 20. This meant that recommendations for using wall functions were not fulfilled. In the present study two numerical grids of different density were employed. The number of grid cells was either about 100K or 340K. It was decided to use the fine grid, because the impeller investigated in this paper did not have standard diameter (D = 0.483T) and generated bigger turbulence than the standard impeller. The values of y^+ were checked after the simulations.

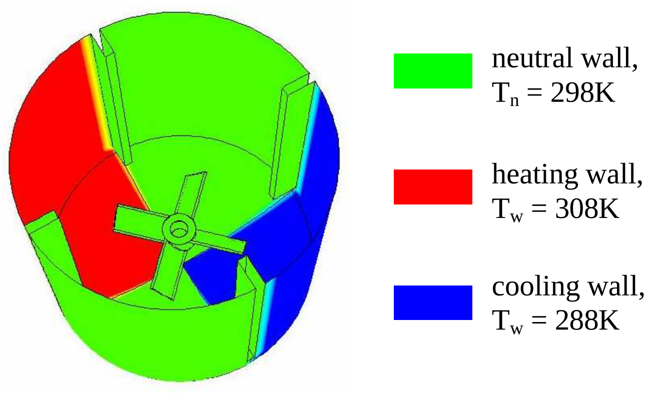

Fig. 1. Heat transfer boundary conditions

The MixSim™ 1.7 preprocessor was used to generate two unstructured, hexahedral grids of different density. In order to simulate the action of the pitched blade turbine impeller, the option of the multiple frames of reference was used. The no-slip conditions at all fluid-solid boundaries and the symmetry conditions at the free surface of the stirred liquid were chosen. All the heat transfer boundary conditions as well as the fluid properties were defined in the Fluent™ code. The tank wall, including the dished bottom, was divided into 4 quarters. One of them played the role of the heating wall. The opposite one was the cooling wall. The imposed wall temperatures were equal to 308 and 288 K for the heating and cooling walls, respectively. Two other quarters were assumed adiabatic and had the same temperature as the stirred fluid before the heat transfer process. Other boundary conditions were the same for all the tank wall quarters. The arrangement of the heating, cooling and neutral walls is shown in Fig. 1.

The segregated solver was employed to carry out the steady-state simulations. The numerical computation of the Reynolds equations (1) along with the continuity equation (2) and with the selected turbulence model was the first stage of the CFD simulations. The flow field and the turbulence quantities were obtained. Next, the energy transport equation (3) was solved, until the normalized sum of residuals decreased below 10^{-8}.

The local values of the heat transfer rate, $\dot{q}_w$, were obtained as the result of the heat transfer simulations and the local heat transfer coefficient, h, was calculated from Eq. (5). The difference between the temperature of the heating (cooling) wall, T_w, and the fluid bulk temperature, T_{bulk}, was the driving temperature difference.

$$h = \frac{\dot{q}_w}{(T_w - T_{bulk})} \tag{5}$$

3 Results

The contours of the computed heat transfer rate, $\dot{q}_w$, for all the turbulence models and grid densities used are shown in Fig. 2. The highest values of $\dot{q}_w$ were obtained for the standard k-ε model with enhanced wall treatment. Slightly lower values of $\dot{q}_w$ were achieved for the other turbulence models tested. These values were close to each other in the vertical part of the heating (cooling) walls. Large discrepancies between the predicted values of the heat transfer rate were obtained for the k-ε and the k-ω models in the bottom of the tank. The average $\dot{q}_w$ values were equal to 43000 and 33000 W/m^2, for the models from the k-ε and k-ω families, respectively. The local values of $\dot{q}_w$ higher than 39000 W/m^2 in the bottom of the tank were shown as a white places in the contours of $\dot{q}_w$ in Fig. 2.

The stirred fluid temperature, averaged in the whole tank volume was equal to 298.00 for all tested cases. The maximum differences between the local and the average temperature were equal to ± 0.3%. Therefore, in the calculations of the local heat transfer coefficient, the

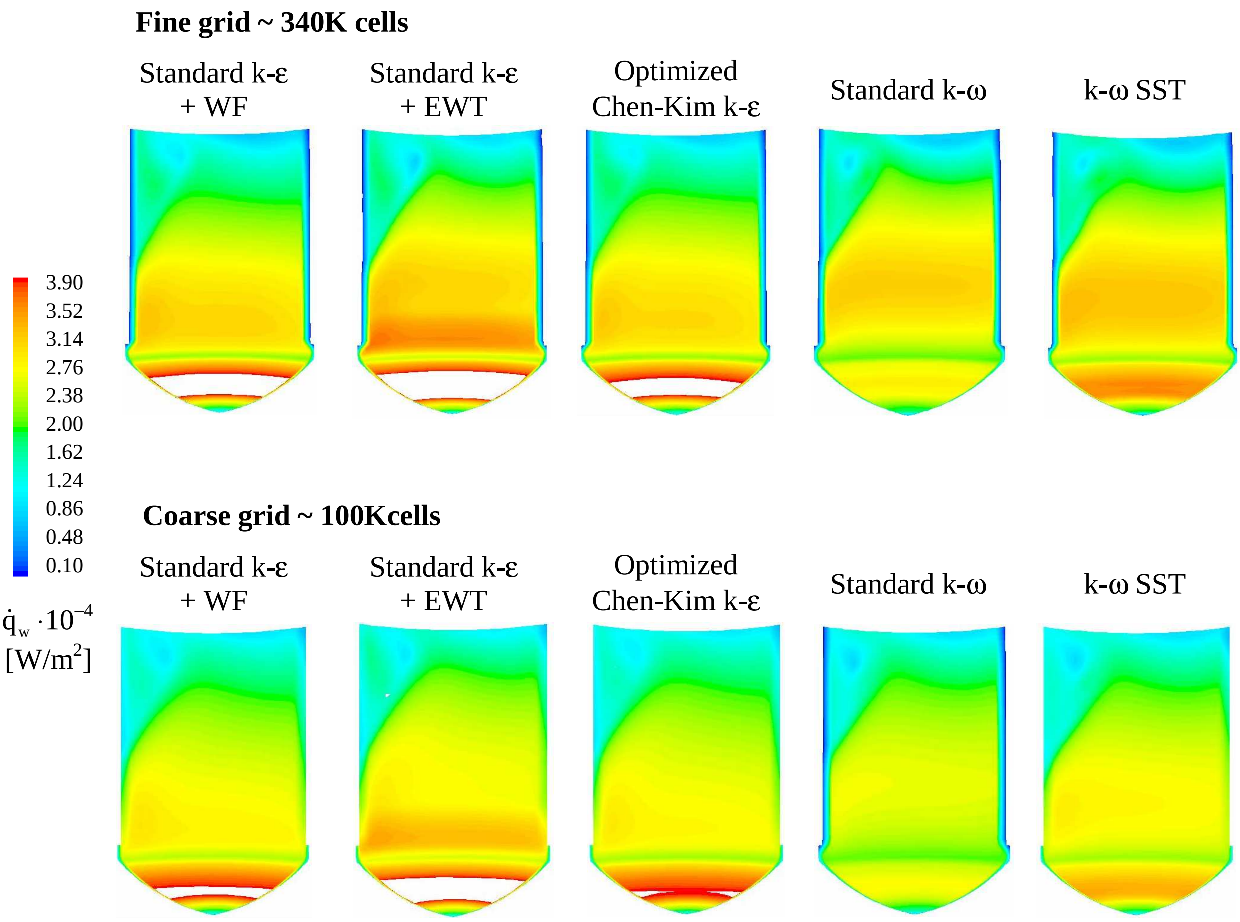

Fig. 2. Contours of the heat transfer rate at the heating wall for all turbulence models tested.
WF – wall functions, EWT – enhanced wall treatment

averaged value of temperature was used as the bulk fluid temperature, T_{bulk} in Eq. (5). The same results were obtained in previous investigations [5] when the heat transfer modelling in a standard Rushton turbine stirred tank was carried out using the same thermal boundary conditions.

The predicted local values of the heat transfer coefficient, h, calculated from Eq. (5) were compared with the experimental [6]. The comparison was performed at nine axial positions and for the angular position of $\theta = 45°$ between the adjacent baffles. The axial profiles of the h coefficient for all the turbulence models and grid densities are shown in Fig. 3.

A good agreement between the predicted and experimental values of h, with discrepancies of about 8% (fine grid) and 12% (coarse grid) was obtained for the axial positions z/H = 0.4 to 0.85. Near to the free surface of the stirred liquid the difference in the h values from CFD and experiment was about 35% for the fine grid and 28% for the coarse grid. Similarly, in the lower part of the vessel, below the impeller level, the h values were underpredicted, especially by the standard k-ω model when the discrepancies were equal to 44%. For the other cases tested, they were about 35% being roughly the same for the two tested grids.

4 Conclusions

CFD modelling of the turbulent heat transfer in stirred tank equipped with the non-standard pitch blade turbine impeller and four flat baffles was carried out. The following conclusions were drawn from the analysis of the modelling results:

1. The best agreement between the predicted and experimental [6] values of the local heat transfer coefficient, h, was obtained for the fine grid and standard k-ε turbulence model along with the enhanced wall treatment. The differences were equal to 12.2%.
2. The simulated values of h were underpredicted near to the free surface of the stirred liquid and to the tank bottom in all the tested cases.
3. The same methodology used in the present and previous [5] investigations resulted in a good

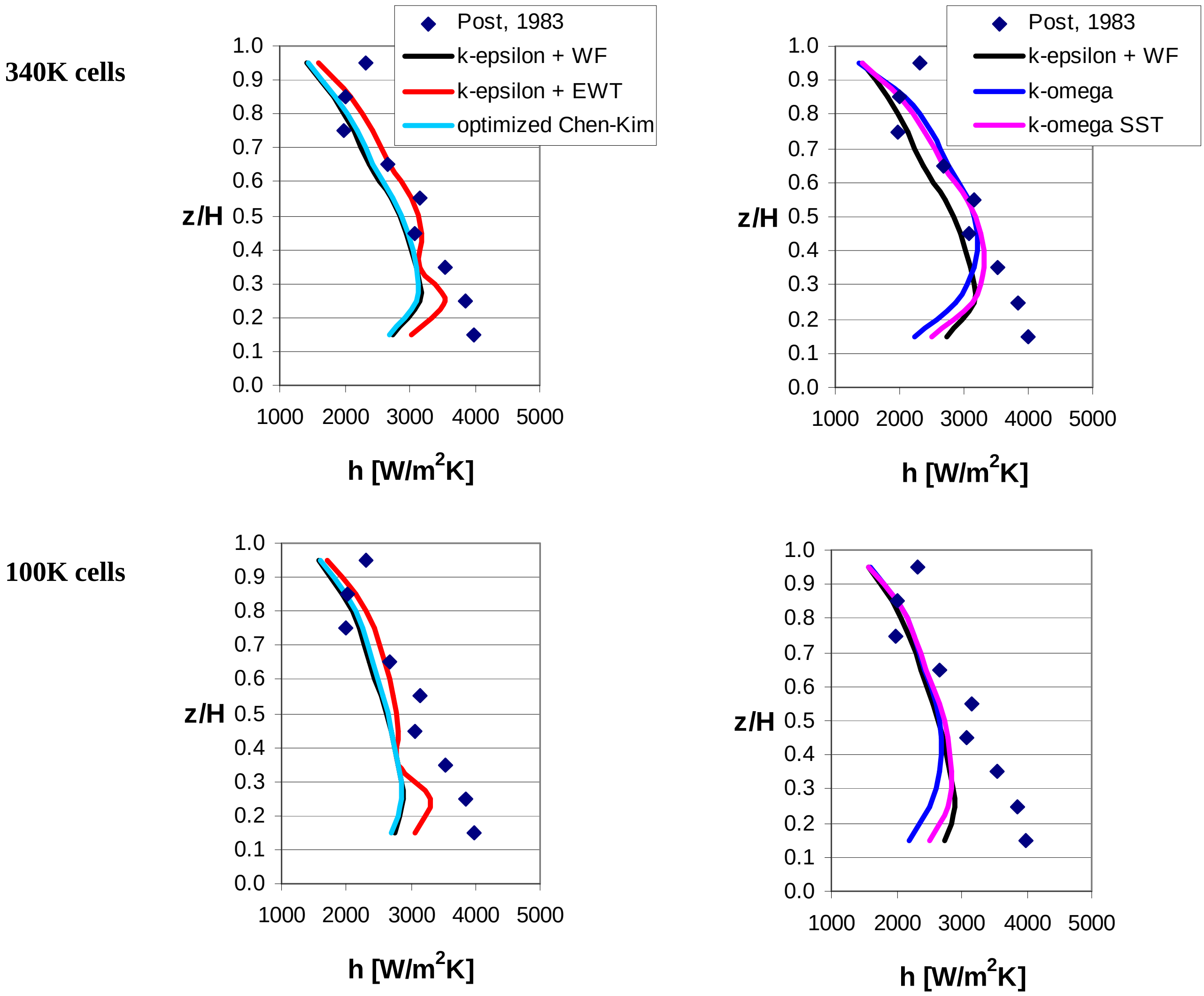

Fig. 3. Axial profiles of heat transfer coefficient for all turbulence models and grid densities.
WF – wall functions, EWT – enhanced wall treatment

agreement between the CFD and experimental values of h for the standard and non-standard geometry of the stirred tank.

ACKNOWLEDGEMENTS
The State Committee for Scientific Research is gratefully acknowledged for financial support within the grant No. 4 T09C 003 22 and thanks are also due to Fluent Inc. for using the MixSim software.

References:
[1] M. Kaminoyama, M. Watanabe, K. Nishi, M. Kamiwano, Numerical simulation of local heat transfer coefficients in stirred vessel with impeller for highly viscous fluids, *Journal of Chemical Engineering of Japan*, Vol. 32 No. 1, 1999, pp. 23-30.

[2] Cz. Kuncewicz., A. Niedzielska, M. Pietrzykowski, Formulation and solution of a heat transfer model for ribbon impellers (in Polish), *Inżynieria Chemiczna i Procesowa*, Vol. 25, No. 3/2, 2004, pp. 1225-1230.

[3] Z. Jaworski, Numerical studies on jacket heat transfer in stirred tanks, *CHISA Congress*, *Prague*, *31 August 2000*, G7 – Symposium on CFD, 2000.

[4] A.J. Manning, R. Mann, M. Colley, CFD model of the flow from a heated impeller in a stirred vessel, In: IChemE Symp. Ser., 140, *Fluid Mixing 5 Conference*, *4-5 July 1996*, *Bradford*, *UK*, pp. 95-105.

[5] B. Zakrzewska, Z. Jaworski, CFD modelling of turbulent jacket heat transfer in a Ruston turbine

stirred vessel, *Chemical Engineering and Technology*, Vol. 27, No. 3, 2004, pp. 237-242.

[6] T. A. Post, *Geometrical influences on local and total mass and heat transfer in an agitated tank*. PhD thesis. Swiss Federal Institute of Technology, Zurich, 1983.

[7] B. E. Launder, D. B. Spalding, The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, 1974, pp. 269-289.

[8] M. Jenne, M. Reuss, A critical assessment on the use of k-ε turbulence models for simulation of the turbulent liquid flow induced by a Rushton-turbine in baffled stirred-tank reactors, *Chemical Engineering Science*, Vol. 54, 1999, pp. 3921-3941.

[9] D.C. Wilcox, Reassessment of the scale-determining equation for advanced turbulence models, *American Institution of Aeronautics and Astronautics Journal*, Vol. 26, No. 11, 1988, pp. 1299-1310.

[10] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *American Institution of Aeronautics and Astronautics Journal*, Vol. 32, No. 8, 1999, pp. 1598-1605.

[11] Z. Jaworski Z., B. Zakrzewska, Modelling of the turbulent wall jet generated by a pitched blade turbine impeller. The effect of turbulence model, *Trans IChemE Part A*, Vol. 80, 2002, pp. 846-854.

[12] B. Zakrzewska, Z. Jaworski, Modelling of the turbulent wall jet generated by a pitched blade turbine impeller. The effect of numerical grid density (in Polish), *Inżynieria i Aparatura Chemiczna*, Vol. 4S, 2002, pp. 149-150.

[13] B. Zakrzewska, *Numerical heat transfer modelling in stirred tanks*. PhD thesis. Szczecin University of Technology, Poland, 2003.