Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)
© 2000 WIT Press, www.witpress.com, ISBN 1-85312-818-X

Numerical investigation of plate heat exchanger surfaces

G. Bigoin[1], P. Tochon[1], J.F. Fourmigue[1] & J.M. Grillot[1]
[1]*CEA-Grenoble, DTP/GRETh, Grenoble, France.*

Abstract

Plate heat exchangers are more and more used in a wide range of industrial processes due to their high level of heat transfer compared to the volume of the device. It concerns particularly plate-and-frame heat exchangers which have already replaced classical shell-and-tube heat exchangers in many industrial applications, such as air conditioning, chemical, petro-chemical and food industries. Numerous types of corrugation plates are currently available on the market ; but enhancement of heat transfer is still a main concern for manufacturers, whose final goal is to provide heat exchangers with a reduced number of plates ; and consequently, a reduced cost of production. At the moment, this research of new geometries is essentially achieved by means of experimental tools. So, the objective of this paper is to contribute to the determination of the turbulent flow in a complex industrial geometry, under the aspects of Computational Fluid Dynamics (CFD) method. The approach is based on the numerical simulation of the turbulent flow using various turbulence models (mixing length model , eddy viscosity model and large eddy simulation) and various meshes. Results are qualitatively and quantitatively compared with experimental data obtained in the laboratory and from open literature. Then, advantages and drawbacks of each model are discussed. With the most accurate mesh and scheme, the numerical simulation gives good results compared with experimental ones. At least, some advices for numerical modelisation of plate heat exchangers are proposed.

1 Introduction

Compact heat exchangers are now often used in a wide range of industrial processes (air conditioning, chemical, petro-chemical and food industries).

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)
© 2000 WIT Press, www.witpress.com, ISBN 1-85312-818-X

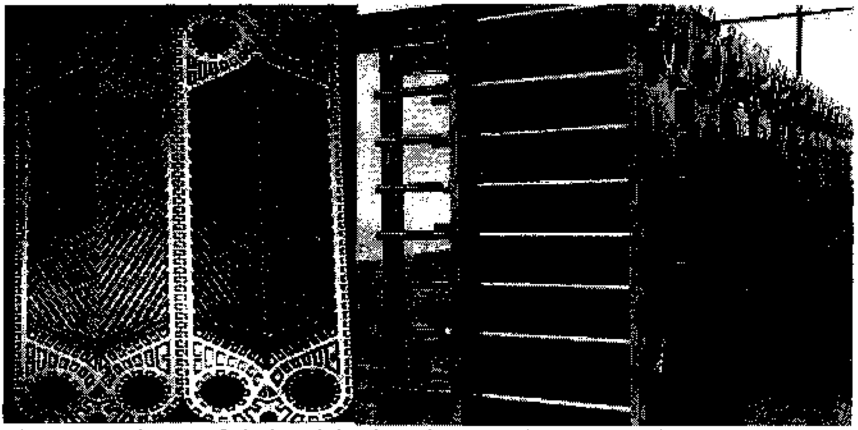

Figure 1: Views of industrial plate heat exchangers and corrugated plates (courtesy of Alfa-Laval Vicarb).

Plate Heat Exchangers (PHE) are made of pack of plates (figure 1) that are pressed together. These plates are corrugated so that a very high degree of turbulence is achieved, even for low Reynolds numbers, producing high film coefficient of heat transfer. The plate size ranges from 0.02 m^2 to over 3 m^2 with conventional pressing technology, but can reach up to 15 m^2 for explosion formed plates. The hydraulic diameter lies between 2 and 10 mm for most common plates, but free passages and wide gap plates exist for viscous fluid applications. Typically, the number of plates used is between 10 and 100, which gives 5 to 50 channels per fluid. To insure the tightness, three technologies are available: gasketed, semi-welded or totally welded and brazed plates. Gasketed PHE is the most common type, and the gasket material is selected according to the application (temperature, fluid nature...). Temperature up to 200°C and pressure up to 25 bars can be achieved by such heat exchangers. For applications where gaskets are undesirable (high pressure and temperature or very corrosive fluids), semi-welded or totally welded heat exchangers are available.
Because of the complexity of flow structure in plate heat exchangers, their performances are generally determined by means of experimental tools. In this article, determination of the turbulent flow in a complex industrial geometry is managed under the aspects of Computational Fluid Dynamics (CFD) method. The pertinent choice of turbulence models is examined and discussed.

Firstly, the geometry used for the flow description will be presented. Secondly, numerical simulations with various turbulent models inside the reference channel will be analysed and discussed. Finally, some highlights will be presented for the choice of the most accurate model.

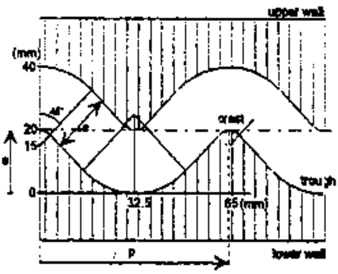

Figure 2: Description of the experimental channel.

2 Geometry and numerical method

2.1 Geometry and boundary conditions

Several experiments have been made in the past on a two dimensional corrugated channel (Hugonnot [1] and Kouidri [2]). The geometry consists in a succession of circles and straight lines (figure 2).
Tests were performed for water flow at $\mathrm{Re} = 1696$ (based on the channel height and the bulk velocity $u_{bulk} = 4.24$ cm/s) and $\mathrm{Re} = 8280$ ($u_{bulk} = 20.7$ cm/s).
Concerning the numerical computations, the modelled channel is reduced to 3 corrugations (figure 3). It allows to avoid problems with the boundary conditions influence, without generating high CPU costs. Indeed, according to Hugonnot [1], hydraulic steady regime is reached after less than 2 corrugation lengths. Comparisons were realized *a posteriori* between the section C and C' and the results showed that there was no significant difference. So the sections A and C could consequently be considered as references for our numerical simulations.
For all studies, a flat velocity u_{bulk} profile is set at the inlet of the computational domain. At the outlet, the pressure distribution is imposed to be uniform and equal to zero. At the solid boundary, no-slip condition is used for the velocity components.

2.2 Numerical method

In the present study, computations are carried out using various turbulence

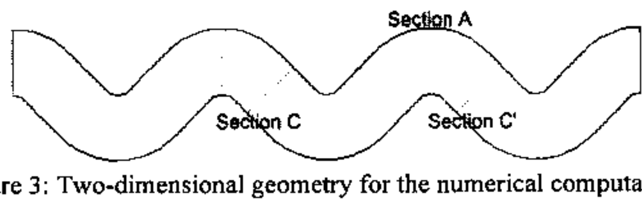

Figure 3: Two-dimensional geometry for the numerical computations.

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)
© 2000 WIT Press, www.witpress.com, ISBN 1-85312-818-X

modelling approaches : Reynolds Averaged Navier Stokes (RANS) and Large Eddy Simulation (LES). The flow field is governed by energy, momentum and continuity balance equations, with the assumption of constant fluid properties. These equations are solved by FLUENT software. Convective terms in the momentum, energy, turbulence kinetic energy and turbulence dissipation rate equations are computed with a third order QUICK scheme (Leonard [3]). The pressure-velocity coupling is performed using the SIMPLE algorithm.

3 Preliminary results : influence of the mesh

A preliminary study has been performed in order to determine the influence of the mesh structure on numerical results. Computations, for $\mathrm{Re}=1696$, have been realized with the Spalart-Allmaras [4], RNG (k,ε) (Yakhot [5]), Realizable (k,ε) (Shih [6]) models. Except the one-equation Spalart-Allmaras model which has its own set of wall functions, there are two ways of dealing with the wall treatment :

- semi-empirical wall functions called *standard wall functions* ;
- near wall modelling approach, where the turbulence models are modified to enable the viscosity-affected region to be solved all the way to the wall, called *two layers zonal model.*

These models have been tested on a quadrilateral (7200 faces, figure 4 a) and triangular mesh (14319 faces, figure 4 b) ; the velocity profiles at the location A (figure 3) obtained with the different turbulence models are compared. Using quadrilateral or triangular mesh leads to small difference (figure 5) between the profiles. The conclusion is similar concerning the comparison of mean pressure losses calculated on one corrugation. So the numerical results are not very sensitive to the element shape. In the next parts of this paper, the triangular mesh will be only used. Furthermore, the Spalart–Allmaras [4] model, based on

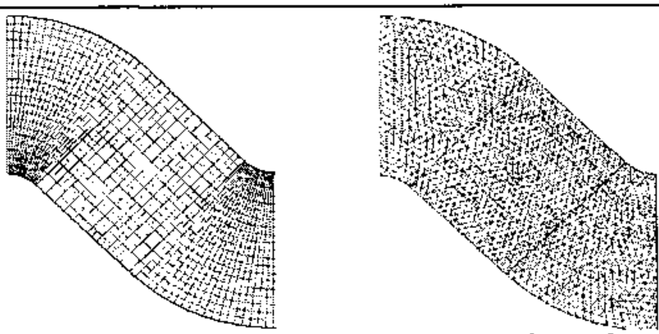
a) quadrilateral mesh *b) triangular mesh*

Figure 4: Description of the meshes.

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)

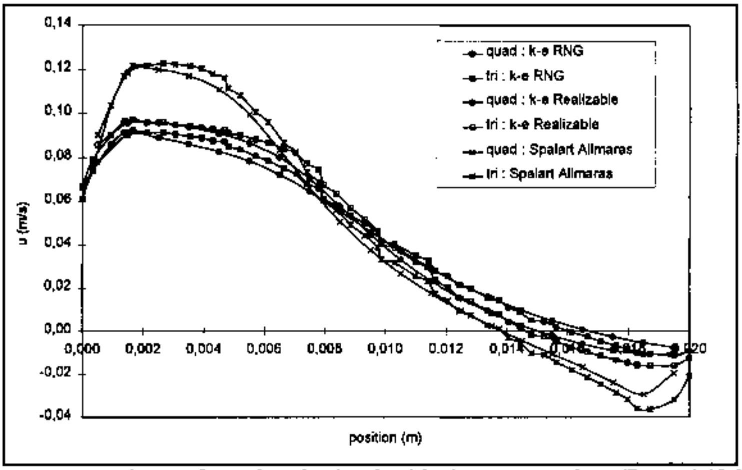

Figure 5: Comparison of results obtained with the two meshes (Re = 1696) with standard wall functions.

the mixing length concept, is not well suited to describe separated and reattached flows. Indeed, it leads here (figure 5) to an overestimation of the recirculating areas, which is not concordant with experimental observations (figure 6). So, this model will be no longer used in the present work.

4 Numerical results using RANS approach

In this section, a local approach of the flow behaviour inside an idealized compact heat exchanger is considered, through the comparison with experimental velocity profiles (Kouidri [2]).
Then, a global approach is presented by means of pressure losses comparison. The study is carried out for Reynolds number equal to 1696 (commonly used in industrial applications) and 8280.

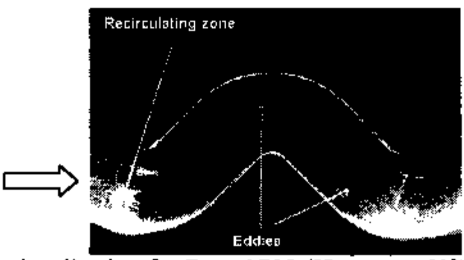

Figure 6: Flow visualisation for Re = 1700 (Hugonnot [1]).

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)

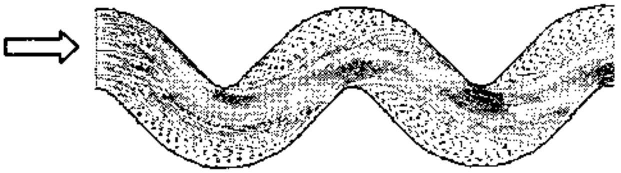

Figure 7: Velocity vectors with RNG (k,ε) model for $\mathrm{Re} = 1696$.

4.1 Local approach

Using RANS models yields to the a high velocity bulk flow in the centre part of the channel, and recirculating areas within the shear layer (figure 7). Despite RANS models are intrinsically not able to describe unsteady phenomena, global flow structure remains in good accordance with experimental one (figure 6).
In order to set the accuracy of various RANS models (RNG (k,ε), Realizable (k,ε)) velocity profiles at location A and C (figure 3) are presented and compared to experimental ones (Kouidri [2]). Concerning the wall treatment for the (k,ε) models, the choice is mainly lead by the non-dimensional wall distance y^+ of the mesh. For the considered Reynolds numbers, the computed wall distance of the mesh size is given as follow : $0.25 < y^+ < 5$ for $\mathrm{Re} = 1696$ and $0.5 < y^+ < 13$ for $\mathrm{Re} = 8280$. So, according to the recommendation for use of the various wall treatments, the *two layers zonal* model, generally useful for y^+ less than 4 – 5, should be used for $\mathrm{Re} = 1696$. For $\mathrm{Re} = 8280$, no particular conclusion comes from these results.

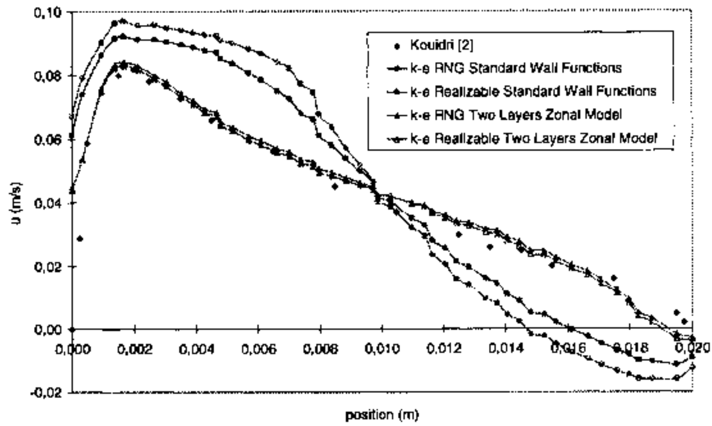

Figure 8: Velocity profiles at location A for $\mathrm{Re} = 1696$.

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)

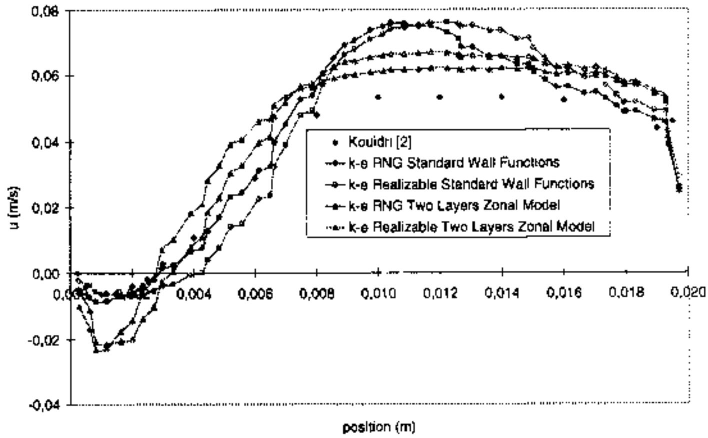

Figure 9: Velocity profiles at location C for $Re = 1696$.

Analysing the various velocity profiles (figure 8 and 9), the results are less sensitive to the turbulence model than to the wall treatment. Furthermore, the main difference lies in the location of the recirculating areas, which are more outstretched with standard wall functions. At location A (figure 8), the two (k,ε) models with the *two layers zonal* model are in very good accordance with experimental profile. In the same way, at location C (figure 9), the size of the recirculating areas are quite similar whatever the model may be. However, the fitting of the recirculating zones is better with standard wall functions.
The results are similar in all respects for Reynolds $Re = 8280$.

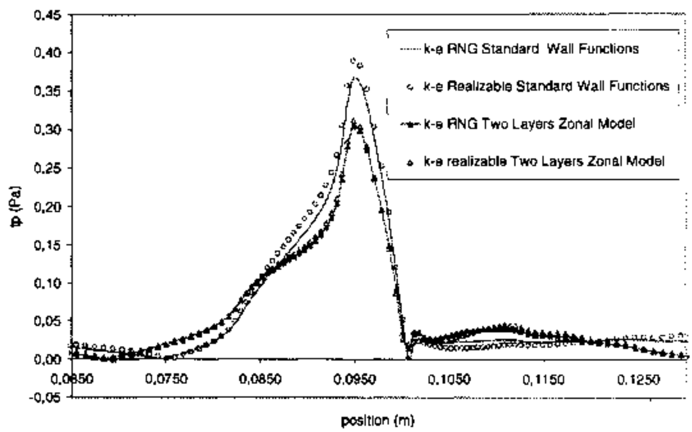

Figure 10: Wall shear stress profile at the top wall along one corrugation ($Re = 1696$).

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)

A complementary study has been performed based on the wall shear stress profile (figure 10). Models using the same wall treatment give logically quite the same profile. The RNG (k-ε) and Realizable (k-ε) models provide qualitatively the same profile : high shear stress after the reattachment point which is located upward the bump. Indeed, flow acceleration at the bump leads to a constriction of the stream lines and an increase of the wall shear rate. The shape of wall shear stress profiles is in good accordance with those obtained experimentally by Kouidri [2] and Gradeck [8]. As a conclusion, it seems difficult to do the right choice of turbulence models or wall treatment only from the analysis of velocity and wall shear stress profiles. But according to the flow visualisations of Hugonnot [1], recirculating zones have a mean size around 2/3 of the pitch which means particularly a recirculating zone at location A. Moreover, experimental measurements of velocity in this area should be taken with care due to the high fluctuating rates of the local velocity near the wall. So, one can finally assume that the use of standard wall functions is more appropriate here.

4.2 Global approach

In order to complement the qualification of this complex flow, computations of the mean values of pressure drops on the central corrugation have been performed (table 1).

Table 1: Pressure drops given by numerical simulation and experiments [2].

Re	Turbulence model	Wall treatment	ΔP_{num} (Pa)	ΔP_{exp} (Pa)
1696	(k,ε) RNG	standard wall functions	2.55	2.2
		two layers zonal model	2.05	
	(k,ε) Realizable	standard wall functions	3.15	
		two layers zonal model	2.15	
8280	(k,ε) RNG	standard wall functions	37.2	81.4
		two layers zonal model	27.8	
	(k,ε) Realizable	standard wall functions	44.4	
		two layers zonal model	30.2	

On one hand, for $\mathrm{Re} = 1696$, the RNG (k,ε) model is in best agreement with experimental results whatever the wall treatment is. But results with the Realizable (k,ε) model remains satisfactory. On the other hand, using any (k,ε) model for Re = 8280 is not valid due to the high unsteadiness of the flow. Indeed, this model underestimates the turbulent fluctuations part, and consequently the pressure drops in the channel.
According to the former results, a numerical simulation for high Reynolds

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)
© 2000 WIT Press, www.witpress.com, ISBN 1-85312-818-X

($0.5 < y^+ < 13$) has been carried out using a LES approach (Smagorinsky [7]).

5 Results using LES approach

At each corrugation, the flow separates and generates a recirculating zone. Due to turbulent instabilities, this area grows and then breaks suddenly in two parts. One part creates a vortex which is released in the flow in the downward direction as the other part remains in the protected region.

These mechanisms of eddy creation and motion are very similar to those observed experimentally (figure 6). The alternative generation of vortices on the lower and upper walls of the domain of calculation, liberates coherent eddies, with either clock or anti-clockwise motion (figure 10).
As a result, it is also apparent that this kind of geometry is able to insure good turbulent mixing after only few corrugations (2 corrugations according to Kouidri [2]). One can also notice that a steady regime is reached after 5 characteristic times (based on the developed length of the channel and the bulk velocity) : the pressure drops value stabilizes near the experimental one (table 2). Moreover, the resulting velocity profiles are in good agreement with those expected.

Table 2: Time evolution of the pressure drops with LES model for $Re = 8280$.

Characteristic time	ΔP_{num} (Pa)	ΔP_{exp} (Pa)
1	43	81.4
3	62	81.4
5	72	81.4
7	74	81.4
10	73	81.4

In conclusion, the Large Eddy Simulation, combined with a high order convective scheme, is more suited for high Reynolds flows. This model is able to display the main turbulence mechanisms for that geometry : fluid separation at the bump, reattachment on the face, recirculating zones and turbulent eddies.

Figure 11: Velocity vector with LES model for $Re = 8280$.

Advanced Computational Methods in Heat Transfer VI, C.A. Brebbia & B. Sunden (Editors)
© 2000 WIT Press, www.witpress.com, ISBN 1-85312-818-X

6. Concluding remarks

According to the present numerical simulations, the Spalart-Allmaras model is not suitable concerning flow modelling in a two-dimensional compact heat exchanger geometry. Concerning the mesh structure, numerical simulation have shown little difference between quadrilateral and triangular meshes. For low Reynolds numbers, the RNG (k, ε) model provides satisfactory local and global results, especially with the standard wall functions. But for high Reynolds numbers, this model is no more valid, because of an underestimation of the unsteady turbulent structures of the flow. For that reason, the LES model is more recommended in that case.

7. References

[1] Hugonnot, P. *Etude locale de l'écoulement et performances thermohydrauliques à faible nombre de Reynolds d'un canal corugué-Application aux échangeurs de chaleur à plaques.* PhD Thesis, Nancy I University, 1989.

[2] Kouidri, F. *Etude des écoulements turbulents chargés de particules. Application à l'encrassement particulaire des échangeurs à plaques corruguées.* PhD Thesis, Joseph Fourier University, 1995.

[3] Leonard, B.P. A stable and accurate convective modelling procedure base and quadratic upstream interpolation. *Comp. Meth. Appl. Mech. Engng*, **107**, pp. 467-476, 1979.

[4] Spalart, P. and Allmaras, S. A one-equation turbulence model for aerodynamic flows. Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics, 1992.

[5] Yakhot, V., Orzag, S. A., Thangman, S., Gatski, T. B. & Speziale, C. G. Development of turbulence models for shear flows by a double expansion technique. *Phys. Fluids*, **A4**, pp. 1510-1520, 1992.

[6] Shih, T.H., Liou, W.W., Shabbir, A. & Zhu, J. A new *k-ε* eddy-viscosity model for high Reynolds number turbulent flows - model development and validation. *Computer Fluids.*, **24**, 3, pp. 227-238, 1995.

[7] Smagorinsky, J.S. General circulation experiments with the primitive equations: I - The basic experiments. *Mon. Weather Rev.*, **91**, pp. 99-164, 1963.

[8] Gradeck, M. *Structure de l'écoulement diphasique gaz-liquide dans les échangeurs à plaques corruguées.* PhD Thesis, Henri Poincaré University, Nancy I, 1996.