

HEAT TRANSFER AND FLUID FLOW IN A COUNTER FLOW MICRO HEAT EXCHANGER

Bogdan Alexandru Nichita¹, Florea Chiriac², Dan Gabriel Cacuci³

¹Technical University of Civil Engineering Bucharest, 66 Pache Protopopescu Blvd, Bucharest 021414, Romania,
Tel/Fax: +40 252 6545, E-mail: bogdan@eits.mediasat.ro

²Technical University of Civil Engineering Bucharest, 66 Pache Protopopescu Blvd, Bucharest 021414, Romania,
Tel/Fax: +40 252 6001, E-mail: fchiriac@eits.mediasat.ro

³Technical University Karlsruhe, Haid und Neu Str. 7 76131 Karlsruhe, Germany, Tel: +49 7216086740,
Fax: +49 7216086749, Email: cacuci@ikr.uni-karlsruhe.de

ABSTRACT

In this paper we analyze the experimental results for a counter flow micro heat exchanger built and tested at the Forschungszentrum Karlsruhe, by comparing them with the numerical results, using a “in house” computer code FLUTAN and the commercial code FLUENT. This counter flow micro heat exchanger consists of 6 stainless foils, four double side channels and two with channels only on one side. The thickness of the foil is 1mm and between these six foils we have five blind foils with a thickness of 0.1mm. In the cross section a micro channel is 200 μ m width and 100 μ m high. The distance between 2 channels is 100 μ m. These channels are 30mm long.

Numerical studies started considering two channels, one with hot water and another with cold water, and the flow as laminar one (in cold channel the Re number varies from 404 to 1169 and in the warm channel from 894 to 3174). And because the results weren't comparable with the experiments, we moved on and we considered the flow a turbulent one using the k- ϵ low Re numbers model and k- ϵ high Reynolds number model. The conclusion was that when we considered the flow as a turbulent one, we get results comparable with the experiments. This conclusion was also verified with the commercial code FLUENT.

1. INTRODUCTION

In recent years the research in the field of thermal-hydraulic at micro scale level has been constantly increasing due to the rapid grow of the technology applications which require transferring high heat rates in relatively small space and volume. Such applications spread from compact heat exchangers to cooling systems for computer CPU, to micro fluidic devices. The aim of this paper is to study the heat transfer and fluid flow in a micro channel heat exchanger. The study is done numerically, with the computer code FLUTAN and with the commercial CFD code FLUENT, by comparing the numerical results with the experimental ones.

2. HEAT EXCHANGER DESCRIPTION

The micro heat exchanger we are talking about, is built at the Forschungszentrum Karlsruhe, one of the world wide leading in building and testing such micro channel heat exchangers. It consists of 6 stainless foils, four double side channels and two with channels only on one side. The sequence of the fluid type is warm fluid – cold fluid – cold fluid – warm fluid – warm fluid and so on. The number of the micro channels on one side is 68. So we have 340 channels with warm water and 340 channels with cold water. The thickness of the foil is 1mm and between these six foils we have five blind foils with a thickness of 0.1mm. In cross section a micro channel is 200 μ m width and 100 μ m high. The distance between 2 channels is 100 μ m. These channels are 30mm long. In the figure 1 we present the counter flow micro heat exchanger and you can see the single and double side stainless foils. In the figure 2 we present a cross section through these foils.

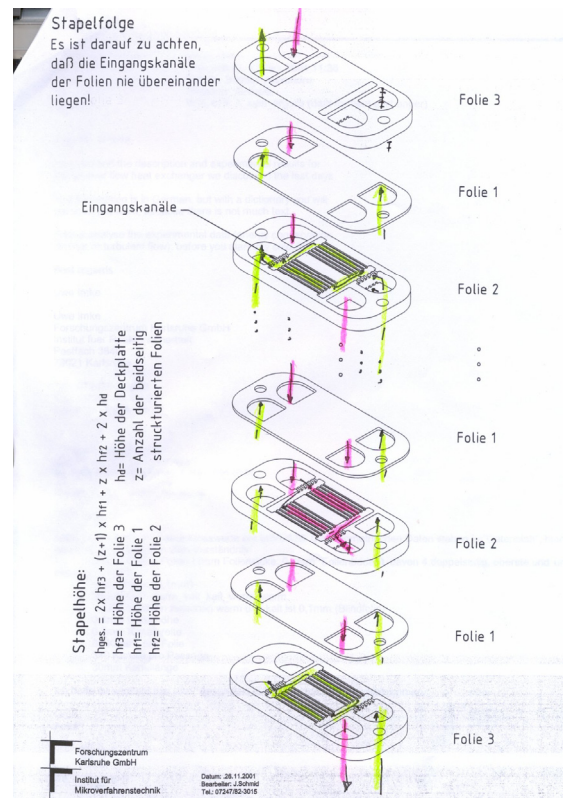


Figure 1. The counter flow micro heat exchanger

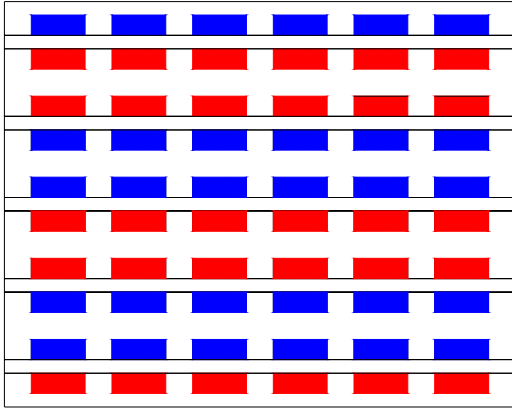


Figure 2. A cross section through the foils

For this counter flow micro heat exchanger we had experimental results. Experiments were carried using different flow rates of hot water (temperature approx. 94°C) and cold water (temperature approx. 10°C). At the maximum flow rate (approx 200kg/h) a thermal power of 6 kW was reached.

3. COMPUTER CODE FLUTAN – DESCRIPTION

The numerical studies were realized with the “in-house” computer code FLUTAN and with the commercial CFD code FLUENT. FLUTAN [1-2] is a computer code for 3D Fluid and Thermo-Dynamic Analysis related to the family of COMMIX codes, which were originally developed at the Argonne National Laboratory (ANL) USA. The FLUTAN code provides detailed local velocity and temperatures fields for the problem under consideration. The conservation equations of mass, momentum and energy, and transport equations of turbulence parameters are solved as a boundary value problem in space and an initial value problem in time. The discretization equations are obtained by integrating the conservation equations over a control volume. The code has a modular structure and permits analysis using either Cartesian or Cylindrical coordinate systems. A staggered grid is used for the velocities. The discretization of the diffusive terms is performed by a central difference scheme. A first order upwind or one of two second order upwind methods can be chosen for the convective terms; i.e. QUICK (Leonard 1979) and LECUSSO (Günther 1992).

The FLUTAN structure is:

For each time step:

- 1) pressure – momentum – mass loop:
 - compute coefficients for pressure equation;
 - solve the pressure equation;
 - use pressure to compute new velocities;
- 2) if turbulent flow then compute turbulent quantities;
- 3) energy loop:
 - compute coefficients for energy equation;
 - solve energy equation and compute temperatures;
- 4) if velocities and temperatures have not converged go to 1;
- 5) go to next time step.

4. EQUATIONS AND TURBULENCE MODELS

The conservation equations of mass momentum and energy have a common form. If we symbolize the general dependent variable as Φ the corresponding conservation equations have the following form in the Cartesian coordinate systems. [1]

$$\begin{aligned} & \frac{\partial}{\partial t}(\rho\Phi) + \frac{\partial}{\partial x}(\rho u\Phi) + \frac{\partial}{\partial y}(\rho v\Phi) + \frac{\partial}{\partial z}(\rho w\Phi) = \\ & = \frac{\partial}{\partial x}\left(\Gamma_{\Phi} \frac{\partial\Phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_{\Phi} \frac{\partial\Phi}{\partial y}\right) + \frac{\partial}{\partial z}\left(\Gamma_{\Phi} \frac{\partial\Phi}{\partial z}\right) + S_{\Phi} \end{aligned} \quad (1),$$

where u, v, w are the velocities in the x, y and z directions, respectively and Φ, Γ_{Φ} and S_{Φ} are given in the next table for each mass, momentum and energy equations.

Table 1

| Equation | Variable (Φ) | Direction | Diffusion coefficient (Γ_{Φ}) | Source term (S_{Φ}) | |
|--------------|---------------------|-------------|---|--|--|
| Continuity | 1 | Scalar | 0 | 0 | |
| Momentum (i) | u | x direction | μ | $\rho g_x - R_x - \frac{\partial p}{\partial x}$ | |
| (ii) | v | y direction | μ | | $\rho g_y - R_y - \frac{\partial p}{\partial y}$ |
| (iii) | w | z direction | μ | | $\rho g_z - R_z - \frac{\partial p}{\partial z}$ |
| Energy | h | Scalar | λ | $\frac{dp}{dt} + \dot{Q}$ | |

Now μ is dynamic viscosity λ is thermal conductivity and R_x, R_y and R_z are the friction factor due to solid structures in a momentum control volume.

For computing the heat transfer between a structure and a fluid, we need to specify a turbulence model. With the counter flow micro heat exchanger discussed here, we analyzed three turbulence model, namely the laminar model, $k-\varepsilon$ model for high Re numbers, $k-\varepsilon$ model for low Re numbers. Next we discuss briefly these models.

4.1 Laminar model

This is a very simplified turbulence model, in which the turbulent viscosity and the turbulent conductivity are assumed to be constant. The turbulent viscosity can be estimated using the following equation suggested by Sha and Launder:

$$\mu_{tur} = 0.007 c_{\mu} \rho U_{max} l \quad (1)$$

where

$$\begin{cases} c_{\mu} = 0.1 & \text{pentru } Re_{max} > 2000 \\ c_{\mu} = 0.1(0.001 Re_{max} - 1) & \text{pentru } 1000 < Re_{max} < 2000 \\ c_{\mu} = 0 & \text{pentru } Re_{max} < 1000 \end{cases} \quad (2)$$

$$U_{max} = \text{Max}(u, v, w) \quad (3)$$

$$Re_{max} = \text{Max}(Re_x, Re_y, Re_z)$$

The mixing length scale: $l = 0.4D_h$, D_h – hydraulic diameter.

We can approximate the turbulent conductivity with the following relation:

$$\lambda_{tur} = \frac{c_p \mu_{tur}}{Pr_{tur}} = \frac{c_p \mu_{tur}}{0.8 \left[1 - \exp(-6 \times 10^{-5} Re Pr^{1/3}) \right]^{-1}} \quad (4)$$

Regarding the heat transfer, in the laminar flow the heat transfer coefficient is computed with the following relation:

$$\alpha = \frac{\lambda}{\delta} \quad (5)$$

Where λ is the fluid thermal conductivity and δ is half of the mesh distance near the wall.

4.2 Turbulent k- ε model for high Re numbers

This is the most rigorous turbulence model. First we solve the transport equation for turbulence kinetic energy k , and the dissipation rate of turbulence kinetic energy ε . After obtaining the values of k and ε , we compute the turbulent viscosity μ_{tur} using the relation:

$$\mu_{tur} = \frac{C_D \rho k^2}{\varepsilon} \quad (6)$$

Here C_D is a constant with value of 0.09

$$k = \frac{1}{2} \left(u'^2 + v'^2 + w'^2 \right) \quad (7)$$

$$\varepsilon = \nu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_i}{\partial x_j} \quad (8)$$

After computing turbulent viscosity, we compute the thermal conductivity with the relation:

$$\lambda_{tur} = \frac{c_p \mu_{tur}}{\text{Pr}_{tur}} \quad (9)$$

The transport equations for k and ε have the following form [3]:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[(\mu + \mu_{tur} / \sigma_k) \frac{\partial k}{\partial x_j} \right] + \left(2\mu_{tur} S_{ij} - \frac{2}{3} \rho k \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} - \rho \varepsilon \quad (10)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[(\mu + \mu_{tur} / \sigma_\varepsilon) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \frac{\varepsilon}{k} \left(2\mu_{tur} S_{ij} - \frac{2}{3} \rho k \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} - C_2 \rho \frac{\varepsilon^2}{k} \quad (11)$$

Regarding the heat transfer, we use the wall functions to compute the heat transfer coefficient between a structure and the fluid. We start with the dimensionless velocity distribution in the turbulent boundary layer:

$$u^* = \frac{1}{k} \ln(Ey^*) \quad (12)$$

Where

$$u^* = \frac{U_p C_D^{1/4} k_P^{1/2}}{\tau_p / \rho} \quad (13)$$

$$y^* = \frac{\rho C_D^{1/4} k_P^{1/2} \delta}{\mu} \quad (14)$$

Dimensionless temperature distribution has the following form:

$$T^* \equiv \frac{(T_w - T_p) \rho c_p C_D^{1/4} k_P^{1/2}}{\dot{q}} = \text{Pr}_{tur} \left[\frac{1}{k} \ln(Ey^*) + P \right] \quad (15)$$

$$P = 9.24 \left[\left(\frac{\text{Pr}}{\text{Pr}_{tur}} \right)^{3/4} - 1 \right] \left[1 + 0.28 e^{-0.007 \text{Pr} / \text{Pr}_{tur}} \right] \quad (16)$$

It is clearly that $\frac{T_w - T_p}{\dot{q}}$ represent $\frac{1}{\alpha}$.

4.3 Turbulent k- ε model for low Re numbers

Here we are talking about turbulent Re number which is low.

$$\text{Re}_t = k^2 / \varepsilon \nu \quad (16)$$

The transport equations for k and ε has the following form [3]:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[(\mu + \mu_{tur} / \sigma_k) \frac{\partial k}{\partial x_j} \right] + \left(2\mu_{tur} S_{ij} - \frac{2}{3} \rho k \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} - \rho \varepsilon - 2\mu \frac{k}{y^2} \quad (17)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[(\mu + \mu_{tur} / \sigma_\varepsilon) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \frac{\varepsilon}{k} \left(2\mu_{tur} S_{ij} - \frac{2}{3} \rho k \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} - C_2 f_1 \rho \frac{\varepsilon^2}{k} - \frac{f_2}{y^2} \quad (18)$$

$$\mu_{tur} = C_D f_\mu \rho k^2 / \varepsilon \quad (19)$$

$$f_\mu = 1 - e^{-0.0115 y^*} \quad (20)$$

$$f_1 = 1 - 0.22 e^{(-\text{Re}_t / 6)^2} \quad (21)$$

$$f_2 = -2\mu \frac{\varepsilon}{y^2} e^{-y^* / 2} \quad (22)$$

Regarding the heat transfer, we use the same correlations as in the laminar flow.

5. NUMERICAL RESULTS

The counter flow micro heat exchanger described in section 2 is simulated numerically. We made some simplification suppositions and we choose two channels to simulate them, one with hot water and the other with cold water. First we use the computer code FLUTAN and then the commercial CFD code FLUENT. In figure 3 and in figure 4 we present the outlet temperature for cold and hot water for the counter flow micro heat exchanger, considering the flow as laminar one, turbulent flow k- ε for high Re numbers, k- ε for low Re numbers, and turbulent flow k- ε for high Re number with FLUENT. The numerical results are then compared with the experimental results, and the conclusion is that when we consider the flow as turbulent one (k- ε for high Re numbers, both with FLUTAN and FLUENT) we get comparable results with the experiments.

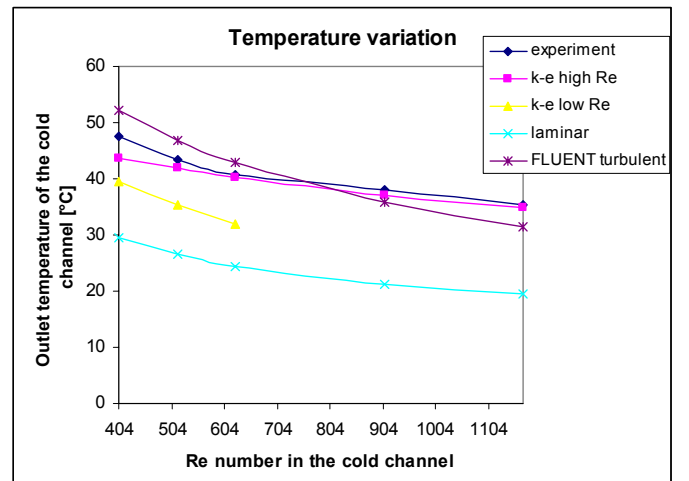


Fig.3. The temperature variation for the cold channel

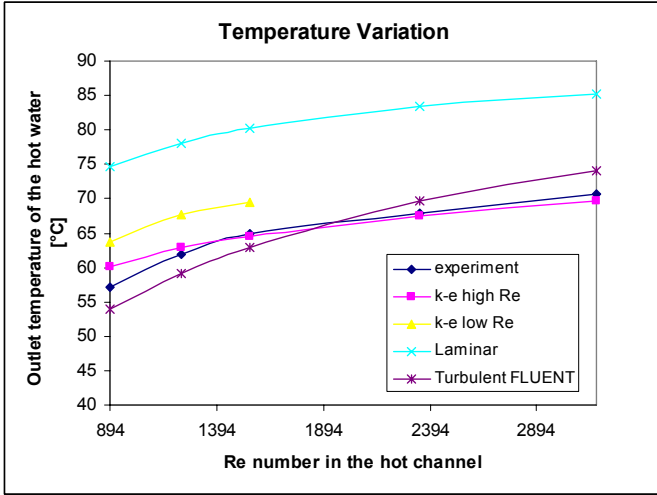


Fig. 4. The temperature variation for the hot channel.

6. CONCLUSIONS

A counter flow micro heat exchanger built at the Forschungszentrum Karlsruhe is tested using the computer code FLUTAN. Also we tested this micro heat exchanger with the commercial code FLUENT.

The computer code FLUTAN is in good agreement with the experimental results for the turbulent flow, considering the $k-\varepsilon$ model for high Re number. The commercial code is in good agreement with the experiments for the turbulent flow, also considering the standard $k-\varepsilon$ model (for high Re numbers).

Based on the Re number the flow seems to be laminar, just for the two first mass flow rate in warm channels we get a Re number greater than 2300, but the numerical results were in good agreement with the experiment just for $k-\varepsilon$ turbulence model, and $k-\varepsilon$ not for low Re numbers but for high Re numbers. That means that the transition from laminar to turbulent flow occurs at lower Re numbers than in the classical sized channels (2300).

NOMENCLATURE

- C_D – constant [-]
 C_1, C_2 – constants [-]
 c_p – specific heat capacity [$\frac{J}{kg K}$]
 c_μ – constant in the laminar model [-]
 k – turbulent kinetic energy [$\frac{m^2}{s^2}$]
 l – mixing length [m]
 \dot{q} – heat flux [$\frac{W}{m^2}$]
 Pr_{tur} – turbulent Prandtl number [-]

Re – the Reynolds number [-]

Re_t – turbulent Reynolds number [-]

S_{ij} – main strain tensor

T^* – dimensionless temperature distribution [-]

T_w – wall temperature [K]

T_p – fluid temperature [K]

u', v', w' – velocity fluctuations [$\frac{m}{s}$]

α – heat transfer coefficient [$\frac{W}{m^2 K}$]

δ – half of the mesh distance near the wall [m]

δ_{ij} – Kronecker symbol

λ – effective thermal conductivity [$\frac{W}{m K}$]

λ_{tur} – turbulent thermal conductivity [$\frac{W}{m K}$]

ε – dissipation rate of the turbulent kinetic energy [$\frac{m^2}{s^3}$]

ν – cinematic viscosity [$\frac{m^2}{s}$]

σ_k – Prandtl number for k equation [-]

σ_ε – Prandtl number for ε equation [-]

ρ – fluid density [$\frac{kg}{m^3}$]

μ_{tur} – turbulent viscosity [Pa s]

REFERENCES

1. H.M. Domanus, W.T. Sha, R.C. Schmitt, V.L. Shah, F.F. Chen, C.C. Miao *COMMIX – 1B: A three – dimensional transient single phase computer program for thermal hydraulic analysis of single and multi component systems*; Volume I: Equations and Numerics; NUREG/CR-4348, ANL-85-42 VOL I, September 1985;
2. H.M. Domanus, W.T. Sha, R.C. Schmitt, V.L. Shah, F.F. Chen, C.C. Miao *COMMIX – 1B: A three – dimensional transient single phase computer program for thermal hydraulic analysis of single and multi component systems*; Volume II: Users guide and Manual; NUREG/CR-4349, ANL-85-43 VOL II, September 1985;
3. John C. Tannehill, Dale A. Anderson, Richard H. Pletcher *Computational fluid mechanics and heat transfer* Second Edition.