Heat transfer modelling in double pipes for domestic hot water systems

I. Gabrielaitiene, B. Sunden & J. Wollerstrand
Department of Energy Sciences, Lund University, Sweden

Abstract

The purpose of this work is to numerically investigate the heat transfer in a pipeline layout for domestic hot water systems. This layout, called double pipe, includes two adjacent counter-flow pipes, placed one above another in a common insulation (i.e., there is no insulation layer between the two pipes). However, when using this layout, the prediction of fluid temperatures in the pipes becomes complicated as thermal coupling occurs between the two pipes resulting from the presence of a temperature difference and physical contact of the two (copper) pipes. This coupling phenomenon is difficult to account for using analytical approaches and a numerical method, namely a finite volume method, was therefore applied in this study. It was found that a realizable k-epsilon, two-equation, turbulence model with non-equilibrium wall functions showed the best performance in terms of heat transfer prediction. The validation was carried out against the empirical Nu number correlation developed at uniform heat flux conditions. Since this condition is not relevant for the flows in the double pipe, these were simulated as being placed in separately insulated pipes. The results from modelling the double pipe layout showed that the heat flux increases compared to a single pipe arrangement.
Keywords: double pipe, domestic hot water systems, coupled thermal regime, low Re number turbulent regime, two-equation turbulent models.

1 Introduction

In domestic water systems, where hot water is distributed around a building from central supply, temperature control becomes an important issue due to restrictions associated with various health risks. The risk is connected with the pathogenic bacterium Legionella, which grows at temperatures below 46°C [1].

WIT Transactions on Engineering Sciences, Vol 53, © 2006 WIT Press
www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/HT060371

Another risk is associated with scolding of skin, which typically limits the upper water temperature to 65°C [2]. Based on the above-mentioned restrictions, the requirements for hot water systems [3] state that the outgoing service temperature should be of 55–60°C and a return temperature not lower than 50°C.

To meet the lower temperature bound, hot water recirculation is used. Since water recirculation system requires a closed loop between hot water generation and consumption points, return pipelines are added to existing supply lines. They are connected directly and are covered with common insulation forming the double pipe arrangement.

In the study described in this paper, no insulation layer was provided between the supply and return pipes which distinguishes our layout from those previously investigated [4]. Our considered layout provides several advantages over layouts in [4], such as compactness and simple installation, which are achieved at a competitive price. It also offers the flexibility in selecting different pipe combinations, since the simple assembling of the pipes can occur locally. Using this layout however adds complication to fluid temperature predictions as thermal coupling occurs between the two pipes resulting from the presence of a temperature difference and the physical contact of the two (copper) pipes. This coupling phenomenon is difficult to account for using analytical approaches and a numerical method, namely the finite volume method, was therefore applied in this study.

The double pipe was represented by a 3D numerical model and analysed with the general-purpose code Fluent. Reynolds averaged Navier-Stokes and energy transport equations were employed for the fluid flow and heat transfer solution. For the turbulence modelling, the main emphasis has been placed upon high Reynolds number, two-equation models, augmented by the wall-functions approach. Two-layer-zonal models, which do not utilize the wall functions approach, have also been considered for comparison purposes. It was found that realizable k-epsilon turbulence model with non-equilibrium wall function showed the best performance in terms of heat transfer prediction.

In the present study, consideration was given to the heat transfer analysis including the heat transfer coefficient and temperature distribution in the pipe. Attention was focused on the low Reynolds number turbulent flow (Re=$8\cdot10^3-11\cdot10^3$), because high water velocity can cause local erosion in copper pipes and is therefore generally avoided.

2 Problem statement

In this work, a new layout was analysed, which included two adjacent counter-flow pipes, placed in a common insulation, as seen in Fig. 1. The copper pipes were covered with fibreglass insulation and a layer of aluminium foil on the exterior [5]. The supply pipe (with external diameter 18 mm) was located above the return pipe (external diameter 12 mm) and Reynolds numbers were 8350 and 11453, respectively. Flow velocities were 0.25 m/s and 0.64 m/s for the 18 mm and 12 mm pipes, respectively, which satisfied the condition for velocity being below 0.7 m/s to avoid local erosion. A recirculation regime was only considered, implying that the flow volume was the same in both pipes.

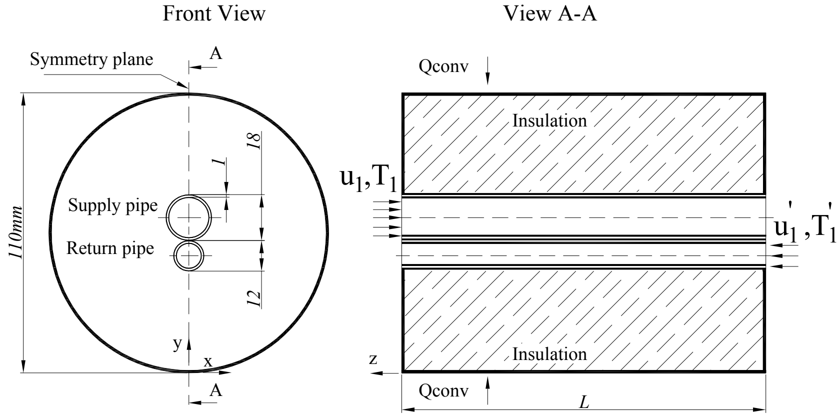

Figure 1: Schematic diagram of the double pipe layout

The two thermal regimes selected for the study accounted for the most typical regimes present in domestic hot water systems: i) The inlet temperatures were T_1=60°C and T_1'=50°C (according to Fig.1) ii) T_1=55°C and T_1'=50°C. The pipe was assumed to be placed in indoors with temperature 20°C.

The contact area between the supply and return pipes was modelled by assuming a small piece of copper material inserted between the pipes (dimensions x=0.5 mm, y=0.01 mm), because from the modelling viewpoint it was not possible to represent the contact area as a single point, which would be the case when two circles are adjacent.

To simplify the analysis, the flow was assumed to be fully developed. In defining the material properties of the water, constant values were prescribed, which corresponded to an inlet temperature according to the selected thermal regime. As the fluid temperature changed along the pipe by less than 2°C (pipe length L=1 m), neglecting temperature-dependent water properties did not distort the results of this study. The thermal insulation properties were assumed to be independent of temperature and were selected according to the average temperature. The variation of the thermal conductivity corresponding to the maximum and minimum insulation temperatures is less than 10% ($k_{(30°C)}$=0.0335 W/mK, $k_{(60°C)}$=0.0366 W/mK [5]), and could therefore be neglected without introducing significant errors.

3 Mathematical and numerical modelling

For the numerical investigations, the general-purpose CFD code Fluent [6] has been used, which applies a finite volume method to discretize the governing equations. Ensemble-averaged continuity, Navier-Stokes and energy transport equations have been solved for incompressible 3D turbulent pipe flow. 3D heat transfer by conduction has been also solved for the copper pipe walls and insulation.

Different two-equation turbulence models, namely the standard k-epsilon model [7], the RNG k-epsilon [8], and the realizable k-epsilon model [9] have been used. The modelling of the near-wall region is especially important in convective heat transfer problems. Several formulations including standard [10] or non-equilibrium wall-function [11] (for bypassing this region adoption logarithmic wall-functions) and two-layer zonal methods (adoption low Reynolds number amendments [12] to accurately resolve the near-wall regions) have been considered. The high Reynolds number models offer the advantage over two-layer-zonal model in reduced computational overhead. Two-layer-zonal methods on the contrary require a very fine grid resolution of the near-wall region, which increases the computational overhead considerably, especially in transient problems. As the transient conditions will be considered in the future research, modelling by high Reynolds number models augmented by the wall-functions approach were the focus in this work, whereas two-layer-zonal method was only considered for comparison purposes.

The solution domain represents the 3D double pipe shown in Fig. 1, where only the left part from the symmetry plane is considered for minimizing computational efforts. At the inlet, constant velocities (u_1, $u_1^{'}$) and temperatures (T_1, $T_1^{'}$) were applied assuming spatially constant profiles. Inaccuracies due to an uncertainty in the shape of inlet velocity and temperature profiles do not play an important role, since the pipe length was more than 60-pipe diameters. The conditions for the turbulent quantities were derived assuming a turbulence intensity of 5%. Convective boundary conditions were applied on the external surface of the double pipe, where the heat transfer coefficient includes convective and radiative terms. The convective term was estimated from the empirical relation for Nu number developed by Churchill and Chu for free convection from horizontal cylinders [13]. The radiative term was determined from an equation developed for a hot convex object in large enclosures, such as a room [13].

The solution domain was discretized by an unstructured grid. The number of computational cells was adjusted in an optimal way according to the Reynolds number for achieving grid independent results. Figure 2 shows the used computational mesh in the double pipe cross-section.

For the standard k-epsilon model, the non-dimensional wall distance describing the location of the near-wall cell is recommended to be $y^+ \geq 30$. However, the investigation in [14] indicates that the accuracy of the wall-function starts to show a remarkable deterioration only beyond $y^+ < approx.12$. Therefore, the values of y+ were allowed to be 20 for the extreme case of low Re number in the pipe with 12 mm diameter.

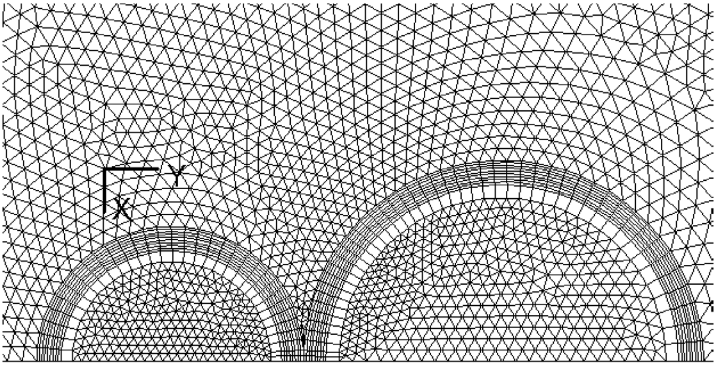

Figure 2: Example of grid in the double pipe cross-section.

In order to investigate the effect of grid size, the simulations were carried out with grid size variation between 1.5 million cells and 3 million cells. The grid was refined near the inlets of supply and return pipes to capture the locally high axial gradients in the entry region of the developing flow. This was followed by a smooth grid expansion towards the middle of the pipe. It was found that there is practically no change in the outlet fluid temperature when grid size was increased beyond 1.5 million cells. To attain a high numerical accuracy, second order schemes were used for the spatial discretization of the governing equations.

4 Numerical results

4.1 Single pipe arrangement

Before beginning the double pipe analysis, validation was carried out against the empirical Nu number correlation developed at a uniform heat flux condition around the wall circumference. Since this condition is not relevant for the flows in the double pipe arrangement, these were simulated as being placed in separately insulated pipes. The flow characteristic and inlet temperatures (T_1=60°C and $T_1^{'}$=50°C, for the supply and return pipes, respectively) were the same as for the flows in the double pipe. Both pipes were simulated with different turbulence models, as described in the previous section, to find a model with the best agreement with empirical values.

For empirical Nu numbers, the standard Dittus-Boelter correlation summarized in [13] was considered:

$$Nu = 0.023 \cdot \mathrm{Re}^{0.8} \cdot \mathrm{Pr}^{0.4} \tag{1}$$

The empirical heat transfer coefficient was estimated from the following equation:

$$h = Nu \cdot k / D_{hyd} \tag{2}$$

The predicted heat transfer coefficients using different turbulence models, for Re=8351 and Re=11453 are presented in Table 1, and the percentage deviations from the empirical values are indicated in brackets.

It can be observed that all models over-predict the empirical values of the heat transfer coefficient, however with different magnitudes. For the Re=8351, the RNG model (with standard wall function) showed highest deviations from empirical values, while predictions obtained for Re=11453 indicate that the standard k-epsilon model produces the highest deviation. Furthermore, it can be observed that the use of "standard" or "non-equilibrium" wall functions has a substantial influence on the results.

Table 1: Predicted and empirical heat transfer coefficients.

	Re=8351 (D18 mm)	Re=11453 (D12 mm)
Empirical heat transfer coefficient:	2004	4360
Predicted heat transfer coefficient according to the turbulence model:		
Standard k-ε, standard wall f.	2199 (9.7%)	4754 (9%)
Standard k-ε, non-equilibrium wall f.	2188 (8.1%)	4599 (5.5%)
Standard k-ε, two-layer zonal near-wall model	(9.9%)	(8%)
RNG k-ε, standard wall funct.	2210 (10.3%)	4751.1 (8.9%)
RNG k-ε, non-equilibrium wall f.	2163 (7.9%)	4569 (4.8%)
Realizable k-ε, standard wall f.	2126 (6.1%)	4640 (6.4%)
Realizable k-ε, non-equilibrium wall f.	2092 (4.4%)	4431 (1.7%)

Based on the comparison presented in Table 1 it may generally be concluded that the realizable k-epsilon turbulence model with non-equilibrium wall function performs better than others, with a deviation of approximately 5% from the empirical value.

4.2 Double pipe arrangement

The effect of thermal coupling between two copper pipes is presented in Figs. 3 and 4 in terms of heat flux and temperature distribution. They are shown at the double pipe longitudinal location z=-0.5 m for two thermal regimes i) inlet temperatures were T_1=60°C and $T_1^{'}$=50°C (for the 18 mm and 12 mm pipes, respectively), and ii) T_1=55°C and $T_1^{'}$=50°C. It was found, that non-uniform heat flux prevailed around the perimeter of the supply and return pipes, and was considerably greater at the contact interface of the two pipes than their average value. It is increased in the above-mentioned area by approximately 60% for the first thermal regime, where the positive heat flux is assumed for the pipe gaining heat. The difference in heat transfer rates between the supply and return pipes is depicted in Fig. 3 as a function of pipe perimeter length. The total heat transfer rate for the 18 mm pipe differs from the 12 mm pipe by 2%, which is due to the heat losses to the surroundings.

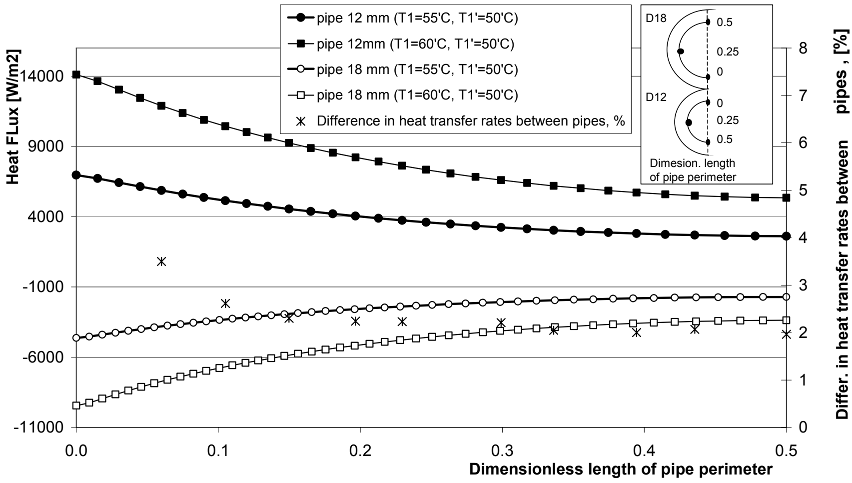

Figure 3: Heat flux distribution in the double pipe cross-section for two thermal regimes: i) T_1=60°C, $T_1^{'}$=50°C and ii) T_1=55°C, $T_1^{'}$=50°C.

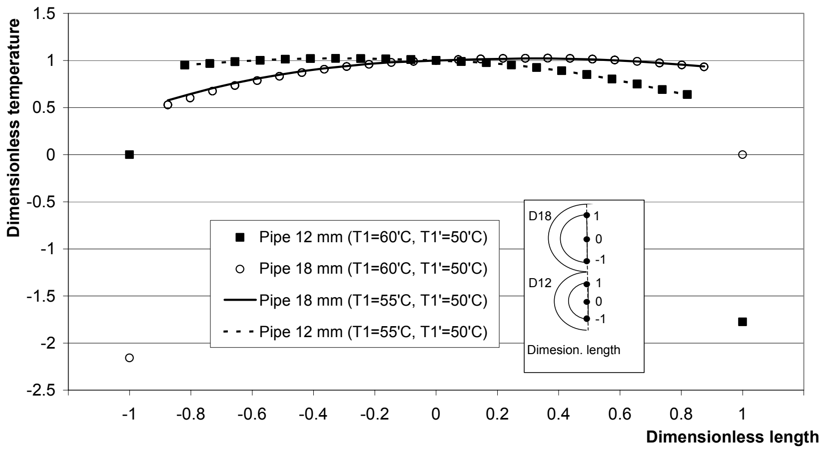

Figure 4: Temperature distribution in the double pipe cross-section for two thermal regimes: i) T_1=60°C, $T_1^{'}$=50°C and ii) T_1=55°C, $T_1^{'}$=50°C.

Due to non-uniform heat flux around the pipe circumference, the anisotropy of turbulence in the wall region might be more pronounced, than in case with uniform heat flux. Since the turbulence models employed in the study neglect that phenomenon, the more advanced turbulence model, which account for the anisotropy of turbulence, can be recommended for future research.

The temperature profile of the fluid in the double pipe is presented in Fig. 4 in dimensionless form, which is defined from eq. (3):

$$T_{dim} = \frac{T - T_{wall}}{T_{centre} - T_{wall}} \tag{3}$$

where T_{centre} – temperature at the pipe centre; T_{wall} – wall temperature, taken at the location, where the dimensionless length equals 1 (see Fig. 4) for the 18 mm pipe, while for the 12 mm pipe the wall temperature is taken at the location, where the dimensionless length is equal to –1.

As can be seen from Fig. 4, the dimensionless temperature profile, which is independent of pipe length (except for the entrance effect), has practically the same form irrespective of considered thermal regime. The temperature profile is non-symmetric and the temperature peak is shifted from the pipe centre towards the outer pipe surface (where dimensionless length is equal to 1, Fig. 4). Furthermore, for the 12 mm pipe the wall temperature is greater than the fluid temperature and the cooling regime prevails.

The heat transfer through the contact interface causes the fluid temperature in the supply line to drop more than in a case of single pipe arrangement. In the return pipe, on the contrary, the pipe is gaining heat and the fluid temperature increases with length, as can be seen in Fig. 5.

4.3 Simplified estimation of fluid temperatures in double pipe arrangement

The temperatures of the warm and cold fluids (the 18 mm and 12 mm pipes, respectively) in the inlet and outlet of the double pipe can be estimated on the basis of equations developed for counter flow in recuperators [15]. A conduction shape factor, which is a prerequisite for solution of these equations, was determined on the basis of the results obtained from the numerical modelling. The defining equation for conduction shape factor is recalled as:

$$S = q / k \cdot \Delta T_{overall} \tag{4}$$

where k – thermal conductivity [W/mK]; q – heat flow [W].

The outlet temperatures (T_2 and $T_2^{'}$, for warm and cold fluid, respectively) are estimated from equations (5) and (6):

$$T_1 - T_2^{'} = (T_1 - T_2^{'}) / \left[1 - 1/\exp(kS/C)\right] \tag{5}$$

$$T_1 - T_2 = (T_1 - T_1^{'}) \cdot \left[1 - 1/\exp\left(\frac{kS/C}{1 + kS/C}\right)\right] \tag{6}$$

where C – product of fluid specific heat and mass flow rate [W/K]; S – conduction shape factor [m], T and T' - temperature of warm fluid (18 mm pipe) and cold fluid (12 mm pipe), respectively. Subscripts $_1$ and $_2$ indicate inlet and outlet temperatures, respectively.

After the outlet temperatures were estimated, the longitudinal temperature distribution is obtained from equations (7) and (8):

$$T_1 - T = (T_1 - T_2^{'}) \cdot \left[1 - 1/\exp(ks/C)\right] \tag{7}$$

$$T_2^{'} - T^{'} = (T_1 - T_2^{'}) \cdot \left[1 - 1/\exp(ks/C)\right] \tag{8}$$

where s – conduction shape factor [m] is presented as a function of longitudinal coordinate and was estimated based on the numerical results.

Longitudinal temperature distributions obtained from the numerical solution and from eqs. (7) and (8) are presented in Fig. 5 for the supply and return pipes (diameters 18 mm and 12 mm, respectively). The difference between the numerical and simplified solutions is larger in the flow entrance area.

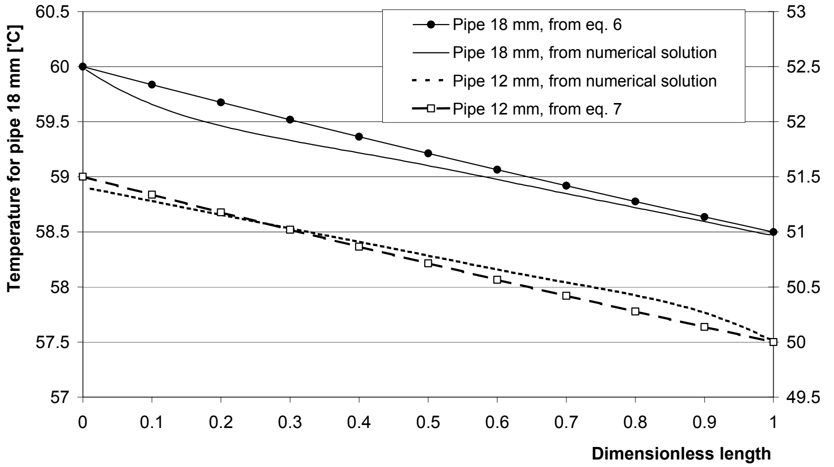

Figure 5: Longitudinal temperature distribution in the supply and return pipes (diameters 18 mm and 12 mm, respectively).

From the practical viewpoint the described methodology is useful for estimating fluid temperature in the double pipe arrangement. The presented results are applicable for the considered thermal regimes and Reynolds numbers.

5 Conclusions

In the present work, a numerical study has been performed to investigate conjugated forced convection in two adjacent counter-flow pipes, placed one above another in a common insulation – double pipe layout.

For the considered thermal regimes (temperatures 60°C and 55°C in supply pipe, and temperature 50°C in return pipe) the effect of thermal coupling is significant. The enhanced heat transfer at the contact interface between two pipes modified greatly the temperature profile and caused non-uniform heat flux distribution in both pipes.

It was found, that a realizable k-epsilon, two-equation, turbulence model with non-equilibrium wall function showed the best performance in terms of heat transfer prediction. This prediction is highly sensitive to the turbulence model selection for the low Reynolds numbers turbulent regimes (Re=8351 and Re=11453).

The simplified equations developed for counter flow in recuperators were applied to estimate the fluid temperatures in the double pipe arrangement. A conduction shape factor, which is a prerequisite for solution of these equations, was determined based on the results obtained from numerical modelling.

References

[1] Brundrett, G.W., *Legionella and building services*, Butterworht-Heineman: Oxford, 1992.

[2] Standards and regulations for engineering systems, BFS 1993:57, 2002:19 *(in Swedish, Boverkets byggregler BBR)*

[3] District heating systems: installation and design. FVF F:101, Svenska Fjärrvärmeföreningens Service Ab, 101 52 Stockholm. *(in Swedish, Fjärrvärmecentralen – Utförande och installation, Svenska Fjärrvärmeföreningen).*

[4] Jonson, E., *Heat losses from district heating network – influence of pipe geometry* (in Swedish), Thesis of Licentiate of Engineering, Lund University of Technology, 2001.

[5] Properties of fibreglass insulation, http://sg-isover.dk

[6] Fluent 6.1, User's Guide, Fluent: Lebanon, 2003.

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

[8] Yakhot, V., Orszag S.A., Renormalisation group analysis of turbulence: I. Basic theory, *Journal of Scientific Computing*, 1, pp. 1-51, 1986.

[9] Shih, T.H., Liou, W.W., Shabbir, A., Zhu, J., A new k-epsilon eddy-viscosity model for high Reynolds number turbulent flow, *Computational Fluids,* 24, pp. 227-238, 1995.

[10] Launder, B.E., Spalding, D.B., The numerical computation of turbulent flows, *Computational Methods in Applied Mechanical Engineering*, 3, pp. 269-289, 1974.

[11] Kim, S.E., Choudhury, D., A near-wall treatment using wall functions sensitized to pressure gradient, *in: Separated and Complex Flows, in ASME FED*, vol. 217, ASME, 1995.

[12] Wolfstein, M., The velocity and temperature distribution of one-dimensional flow with turbulence augmentation and pressure gradient, *International Journal of Heat Mass Transfer*, 12, pp. 301-318, 1969.

[13] Holman, J. P., *Heat transfer*, 9th ed., pp. 268-269, McGraw-Hill, 2002.

[14] Benim, A.C., Arnal, M., A numerical analysis of the labyrinth seal flow, *Proc. of the 2nd European Fluid Dynamics Conf.*, eds. S. Wagner, J. Hirschel, J. Periauz, R. Pive, Wiley: Chichester, pp. 839-846, 1994.

[15] Hausen, H., *Heat Transfer in counterflow, parallel flow and cross flow*, McGraw-Hill: New York, 1983.