DETERMINATION OF CONVECTION AVERAGE COEFFICIENTS IN FINS DUCTS TROUGH NUMERICAL SIMULATION

Cristina Autuori Tomazeti, crist@utfpr.br
Departamento de Engenharia Mecânica
Universidade Tecnologica Federal do Paraná
Av Alberto Carazzai, 1640
CEP 86300-000 – Cornelio Procopio – PR -Brazil

Abstract. *Heat exchangers are equipments with many applications in several industrial sectors. As a device that allows the heat change between two fluids, the study has a main goal find a suitable type and determine the superficial area of heat exchange. This area is calculate to get a specific rate heat transfer between the fluids or reach the suitable exit temperature. Fundamental parameters in thermal analysis of heat exchange is the Global Coefficient of Heat Transfer. This parameters is determined through of total thermal resistance between two fluids, hot and cold, that consider the convective resistance associated with the flow through of heat transfer and conductive resistance in plate that separate the fluids. In order to determine the global coefficient, is necessary know mainly the average convective coefficients of flows of the fluids and the heat exchange geometry. The average convective coefficients can be determined through experimental measurements, analytical methods available in literature or numerical methods. A complete experimental study can take many months, besides can restrict only of the types tested during the experiments. Parallel, the literature make available several empirical and semi-empirical correlations to few ducts types of heat exchanges. Thus, the numerical simulation arise as a important option. This study was developed to verify the feasibility to use the numerical simulation methods to determine average convective coefficients and consequently the global coefficient of heat transfer. Thus, a study of characteristic of laminar and turbulent tridimensional flow and heat transfer process in permanent flow in different models of ducts. The numerical results obtained was compared with experimental results.*

Keywords*: Heat Exchangers, Global Coefficient of Heat transfer, Numerical.*

1. INTRODUCTION

Heat Exchangers are equipment that many application in several industrial sectors. In study of heat exchanges the main goal is select the adequate type and determine the superficial area of heat transfer. The main parameter is global coefficient of heat transfer, that consider conductive (due the plate that separate the fluids) and convective (due the fluids flow) resistance. In order to calculate this parameter is necessary to know average convective coefficients based in characteristics of fluids flow and geometry of heat exchanges, as shown in Equation (1).

$$\frac{1}{UA} = \frac{1}{\eta_{oc} A hm_c} + \frac{t}{k} + \frac{1}{\eta_{oh} A hm_h} \tag{1}$$

where
- U Global coefficients of heat exchanges $[W/m^2.K]$
- A Area of heat exchange, $[m^2]$
- hm_c Average Coefficient convective of cold fluid $[W/m^2.K]$
- hm_h Average Coefficient convective of hot fluid $[W/m^2.K]$
- t thickness of wall that separate the fluids $[m]$
- k Thermal Conductivity of the wall $[W/m.K]$
- η_o Global efficiency of set fin

The average coefficients convective in ducts can be determined through experimental measurements (Kays and London, 1955), analytical methods available in literature or numerical methods. A complete experimental study can be made in several months and can be applicability in specific ducts. Parallel, the literature available some empirical or semi-empirical correlations, that can be used to specific type of channels. Thus, the numerical simulation rise like as good opportunity to available all types of channels used in heat exchangers. The current study was developed to validate the use of numerical simulations to determine the average coefficients convective used in the calculus of global coefficient of heat transfer. Thus, study characteristics of laminar and turbulent tridimensional flow and the process of heat transfer in permanent regime for different models shown by Kays and London (1955). The numerical results obtained were compared with experimental results obtained by the same authors.

2. Description of Geometry Analysed

The ducts testes are compose of fins not-interrupted inside of channel. Both, fins and channels are made by the same material. Three models were studies by Kays and London (1955), called models 6.2, 11.94T and 15.08. The model 6.2, shown in figure 1(a), has 6.2 fins per inch and is compose of a bended aluminum fins set in insert in the duct. The model 11.94T, Figure 1(b), has a triangular section and 11.94 fins per inch and is made in aluminum and composes of fins insert in the duct of the same material, length and thickness of fins. The Figure 1(c) showed the geometry of model 15.08. This model is made in aluminum and has 15.08 fins per inch. In order for the all models, fins and ducts have the same length (L).

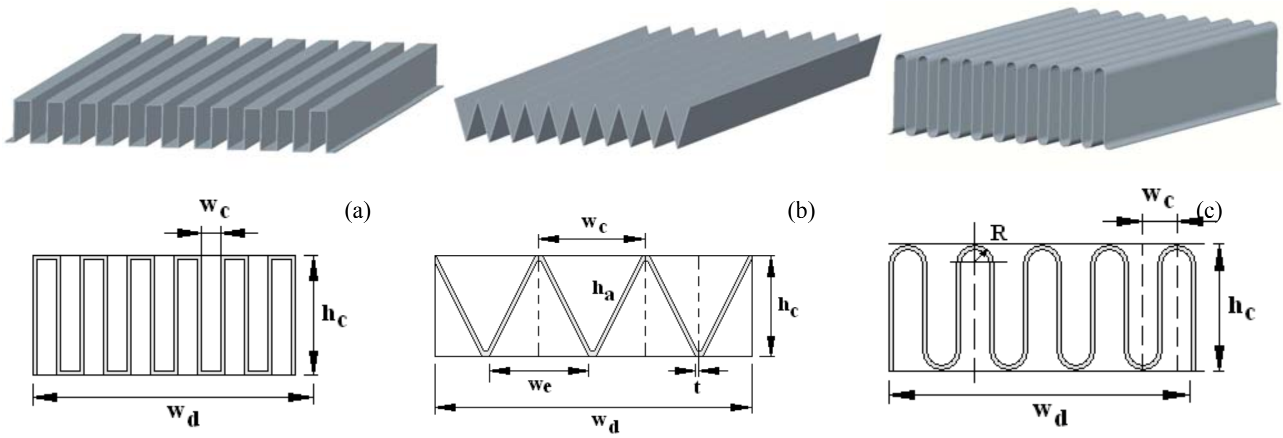

Figure 1. Ducts tested: (a) 6.2, (b) 11.94T e (c) 15.08.

In Fig 1, (w_c) show the width of each channel between fins and (h_c) the height. Table 1 show as determined the width of duct (w_d), transverse section area of channel (A_c), total area of fins (A_{ta}) and total heat transfer area per duct (A_d). In this table, (N_a) and (N_c) are the fins and channels numbers, respectively. The thickness of material is give by (t). The expressions used to determine the geometrical characteristic were obtained through approach from values available by Kays and London (1955).

Table 1. Geometrical Characteristics of Ducts Models Tested.

Duct	w_d	A_c	A_{ta}	A_d
6.2	$N_a t + N_c w_c$	$w_c h_c$	$L\left(2\,N_a h_a + N_c w_e\right)$	$2\,L\left(N_a h_c + N_c w_c\right)$
11.94T	$N_c w_c$	$\frac{w_e h_c}{2}$	$2\,h_a N_a L$	$L\,N_a\left(2\,h_a + w_e\right)$
15.08	$N_c w_c$	$w_c\left(h_c - R\right) + \frac{\pi R^2}{4}$	$2\,L\,N_a\left(h_c - R + \frac{\pi R}{2}\right)$	$2\,L\,N_a\left(h_c - R + \frac{\pi R}{2} + w_c\right)$

In model 11.94T, (h_a) represent the fin height, determined by Eq. (2).

$$h_a = \sqrt{\left(\frac{w_e}{2}\right)^2 + \left(h_c\right)^2} \tag{2}$$

The diameter hydraulic of ducts of the all models tested was calculated by Eq. (3).

$$Dh_d = \frac{4\,A_c L}{A_d} \tag{3}$$

The Table 2 show the dimensions to each duct. The dimensions showed referrer to the geometry described in Fig 1.

Table 2. Dimensions of models in ducts analysed.

Type	Fins per Inch	w_c $[mm]$	h_c $[mm]$	t $[mm]$	$\mathrm{Dh_c}$ $[mm]$	$\mathrm{A_c}$ $[mm^2]$
6.2	6.2	3.84	10.29	0.2540	5.60	39.54
11.94T	11.94	3.93	6.32	0.1524	2.88	148.5
15.08	15.08	1.68	10.62	0.1524	2.66	256.6

3. NUMERICAL MODELLING

The flows and convective heat transfer in each duct were numerically simulated through Control Volume Method, using computational code PHOENICS. All study cases, the mass, movement and energy conservation equations were solved through tridimensional simulations of flow. For the models 6.2 and 11.94T the flows were considered in laminar and turbulent conditions. In turbulent conditions the model used was LVEL model (Spalding, 1994) included in PHOENICS. The model 15.08 was simulated only in laminar conditions. The code computational is based in discretization of differential equations by the control volume method, that link the velocity and pressure using the SIMPLEST algorithm (SIMPLE-ShorTeden), Patankar (1980). To flows in permanent regime, the conservation equations to will be solved has a general formulation shows I by the Eq. (4).

$$\rho\frac{\partial \mathrm{w_i}\phi}{\partial \mathrm{x_i}}=\frac{\partial}{\partial \mathrm{x_i}}\left(\Gamma_\phi\frac{\partial\phi}{\partial \mathrm{x_i}}\right)+\mathrm{S}_\phi \tag{4}$$

In this equation, (ϕ) show the property to will be conserved, (Γ) the diffusion coefficient of property and (S) the source term. First and second term of Eq. (4) indicate the convective and diffusive terms, respectively.

3.1. Transport Equations

The fundamental conservation laws in fluids flows in a duct are expressed by the general transport equations: continuity, linear movement quantify and energy, represented by equations (4), (5) and (6). In these equations, the average velocity and floating turbulent velocity are indicate by (w) and $(w^{'})$, respectively. The variables (T) and $(T^{'})$ indicate the temperature and floating turbulent temperature, respectively. The cinematic viscosity are indicate by (ν) and thermal diffusivity by (α).

$$\frac{\partial \mathrm{w_i}}{\partial \mathrm{x_i}}=0 \tag{5}$$

$$\frac{\partial \mathrm{w_i w_j}}{\partial \mathrm{x_i}}=-\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial \mathrm{x_i}}+\frac{\partial}{\partial \mathrm{x_j}}\left[\nu\left(\frac{\partial \mathrm{w_i}}{\partial \mathrm{x_j}}+\frac{\partial \mathrm{w_j}}{\partial \mathrm{x_i}}\right)-\overline{w_j^{'}w_i^{'}}\right] \tag{6}$$

$$\frac{\partial(\mathrm{w_j T})}{\partial \mathrm{x_j}}=\frac{\partial}{\partial \mathrm{x_j}}\left(\alpha\frac{\partial T}{\partial \mathrm{x_j}}-\overline{w_j^{'}T^{'}}\right) \tag{7}$$

In energy and linear movement quantify equations, the product $(-\rho\overline{w_j^{'}w_i^{'}})$ represent the transport in direction “I”, movement quantify in direction “j” and your components act like as stress in fluid volume and, due are called of turbulent stress or Reynolds stress. The product $(-\rho cp\,\overline{w_j^{'}T^{'}})$ indicates the turbulent heat flux and quantify the flux density of heat in direction “j”.

3.2. Turbulence model used

In simulation of turbulent flow in the channels of crossflow heat exchange, firstly were tested the turbulence models LVEL e κ–ε. The comparison with experimental results shows that the better adjust were obtained by the model LVEL to the geometry used (presence of the walls in duct with small dimensions). The LVEL model, introduced in computational code PHOENICS in 1993, is used in situations where there are several walls in contact of fluids, when the traditional model with two equations hard to applied. The model can be used to a large range of Reynolds number and has an own method to calculate the distance form any part of flow to the walls. In the model is used a effective viscosity, that include the laminar and turbulent component. Thus, is possible to model the flow conditions inside of dominium. In this model, the turbulent viscosity is determined through of wall law showed by Spalding (1961). To $(0 \le y^+ \le 100)$, the author propose the expression showed by Eq. (8)

$$y^+ = u^+ + \frac{1}{E}\left[\exp\left(k\,u^+\right) - 1 - k\,u^+ - \frac{1}{2}\left(k\,u^+\right)^2 - \frac{1}{6}\left(k\,u^+\right)^3 - \frac{1}{24}\left(k\,u^+\right)^4\right] \tag{8}$$

The Eq. (8) is obtained from adjust of experimental results in region near a solid surface or laminar sub layer. The effective viscosity, Eq. (9), is obtained through of derive of (y^+) in relation of (u^+).

$$\nu_{ef}^+ = 1 + \frac{k}{E}\left[\exp\left(k\,u^+\right) - 1 - k\,u^+ - \frac{1}{2}\left(k\,u^+\right)^2 - \frac{1}{6}\left(k\,u^+\right)^3\right] \tag{9}$$

Where $(k = 0{,}417)$ indicate a Von Karman constant. In Eq. (9) in region together of the wall, the effective viscosity is equal of the unit, due an average velocity is near of zero. With the increase of distance from the wall, the viscosity value tends to:

$$\nu_{ef}^+ = k\,y^+ \tag{10}$$

3.3. Domain simulated

In order to assess numerically the average convective coefficients (hm) in separate fins ducts was considered that the walls were in a uniform temperature with the same value of inlet temperature of the other fluid. Due the symmetry of type of duct, the domain simulated is indicated by the traced line in figure 2. The contour conditions used are of the symmetry in the two lateral faces and isothermal conditions of walls (T_w) in upper and lower faces.

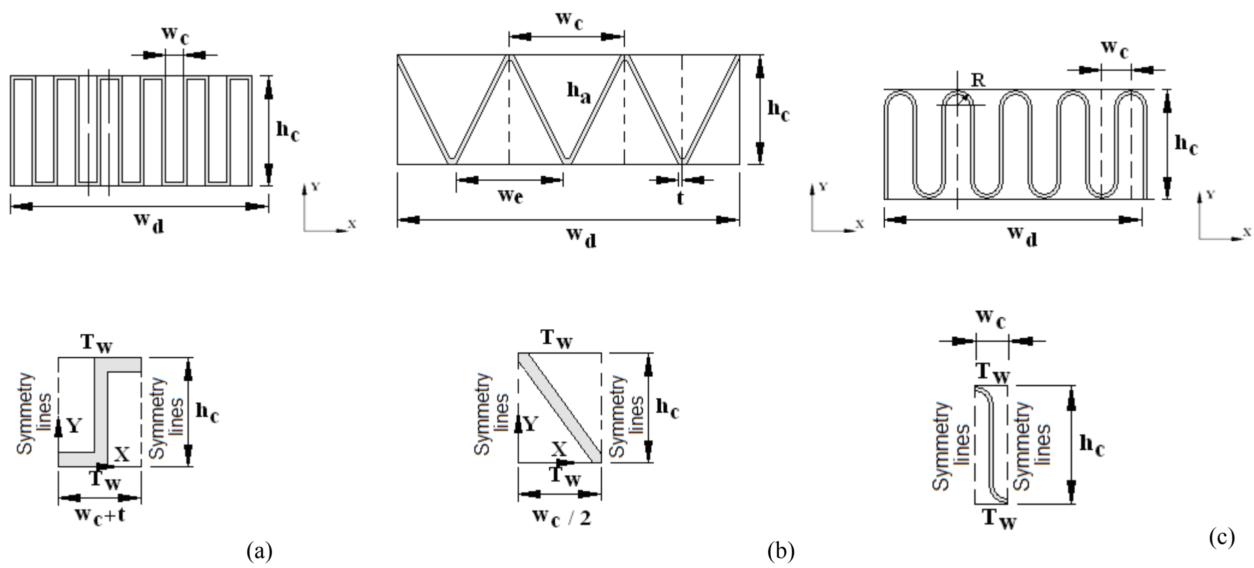

Figure 2. Domain simulated with numerical method: (a) 6.2, (b) 11.94T e (c) 15.08.

The simulation of the three models was made by the dimensions shown in Table 2. The values of average coefficients of heat transfer (hm) were obtained simulating the flows of cold and hot fluids in each channel. The computational grid used in simulation in terms of control volumes numbers used in each direction is indicated in Table 3. The coordinate aces are indicating in Figure 3. The numbers indicate in Table 3 corresponding to the control volumes used in calculate domain to obtain the results independent of the grid.

Table 3 Grid used in simulation of the ducts.

Type	Domain Dimensions [mm]			Laminar Control volume numbers			Turbulent Control volume numbers		
	X	Y	Z	NX	NY	NZ	NX	NY	NZ
6.2	4.1	10.3	304.8	23	56	100	23	29	77
11.94T	2.13	6.31	127.0	52	80	100	2.13	6.31	127.0
15.08	1.98	10.92	173.7	34	110	61	-	-	-

4. NUMERICAL AND EXPERIMENTAL RESULTS

The models of fins ducts shown were obtain from Kays and London (1955) that present experimental results obtained on the laboratories form Stanford University, California and on engineering experimental station of Navy of United States of America in Maryland. According to authors, the same techniques and equipment of measurements were used in both laboratories. Each model of fin duct was submitted in the extern side to saturated steam and the intern side a controlled air flux. Details of experiments and data treatment method are presented in London and Ferguson (1946, 1949), Kays and London (1950) and Kays et all (1955). The tables 4 and 5 show the results of simulation of the each separated channel with laminar and turbulent flow and a comparison of experimental results from Kays and London (1955).

The variable (v) indicate the air inlet uniform velocity in the channel, (Re) is the Reynolds number, (Nu) is the average Nusselt number of the flow, (fm) is the friction average coefficient. The inlet and outlet temperature in duct are indicated by (T_e) and (T_s), respectively. In tests were considered the same mixture temperature in inlet (T_e) to all models of ducts. The values of (Nu) e (Re) are based in hydraulic diameters in each channel between fins. The values of velocities and temperature in inlet of each channel were choices arbitrarily, however, assuring laminar or turbulent flows in fins channels. The properties of fluids were used in average value, by an interactive method, from average mixture temperature on inlet and outlet of the channels. The experimental results of London and Fergunson (1946 and 1949) too are shown.

Table 4. Experimental and Numerical Results of laminar flows in models of ducts studied.

Type	v $[m/s]$	T_e $[°C]$	Experimental						Numerical					
			Re	T_s $[°C]$	hm $\left[\frac{W}{m^2K}\right]$	Nu	q $[W]$	$fm\ .10^2$	Re	T_s $[°C]$	hm $\left[\frac{W}{m^2K}\right]$	Nu	q $[W]$	$fm\ .10^2$
6.2	2.5	20	870	35.4	19.8	4.3	11.0	1.98	868	36.2	22.5	4.9	11.6	2.49
11.94T	2.0	20	355	38.1	31.1	3.5	6.2	4.67	356	34.2	26.2	2.9	5.9	4.09
15.08	5.0	20	824	37.4	46.6	4.8	25.9	2.58	825	36.7	40.8	4.2	24.9	1.83

Table 5. Experimental and Numerical Results of turbulent flows in models of ducts 6.2 e 11.94T.

Type	v $[m/s]$	T_e $[°C]$	Experimental						Numerical					
			Re	T_s $[°C]$	hm $\left[\frac{W}{m^2K}\right]$	Nu	q $[W]$	$fm\ .10^3$	Re	T_s $[°C]$	hm $\left[\frac{W}{m^2K}\right]$	Nu	q $[W]$	$fm\ .10^3$
6.2	25	20	8810	31.2	113.8	24.9	80.6	7.55	8772	32.7	139.7	30.5	91.0	8.76
11.94T	30	20	5461	29.9	140.5	15.9	52.1	8.41	5460	29.9	141.5	15.9	52.3	9.00

The numerical and experimental results of laminar flow in the three models of ducts are show in Table 4. The values show that of values of numerical average convective coefficient to the type 11.94T is approximately 10% higher than experimental values, while to the model 6.2, the difference between numerical and experimental values is lower than 1%. The model 15.08 has only experimental results to laminar flow. For this ducts, the value of (hm) numerical is 10% lower than experimental. The comparison of average friction coefficient, in model 11.94T the numerical value is 12% lower than experimental. The numerical value of (fm) in duct model 6.2 is approximately 25% higher than (fm) experimental. For the model 15.08 the numerical value of (fm) is 29% lower than experimental value.

The Table 5 show the numerical and experimental results to the models 11.94T and 6.2. The average values of numerical convective coefficients are 23% and 1% higher than experimental values to the models 6.2 and 11.94T, respectively. While the numerical average friction coefficients are approximately 16% and 7% higher than experimental values in respective models.

The Figure 3 shows the velocity and temperature distribution to laminar flow ($Re \approx 870$), with uniform inlet velocity of 2,5 m/s and inlet and wall temperature of 20ºC and 40ºC, respectively.

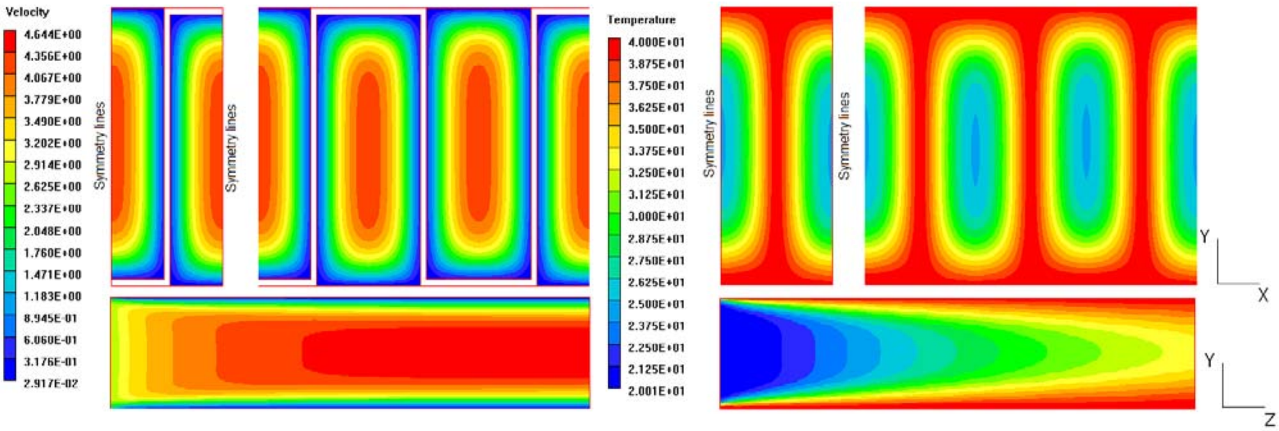

Figure 3. Profile of velocity and temperature to the laminar flow in model 6.2 ($X = 0$ and $Z = 0{,}1m$).

The figure 4 shows the temperature and velocity distribution indicating the flow symmetry. The simulation was performed in conditions of laminar flow ($Re \approx 360$), with uniform inlet velocity of 2 m/s and air inlet temperature of 20ºC and upper and lower temperature faces of 40ºC.

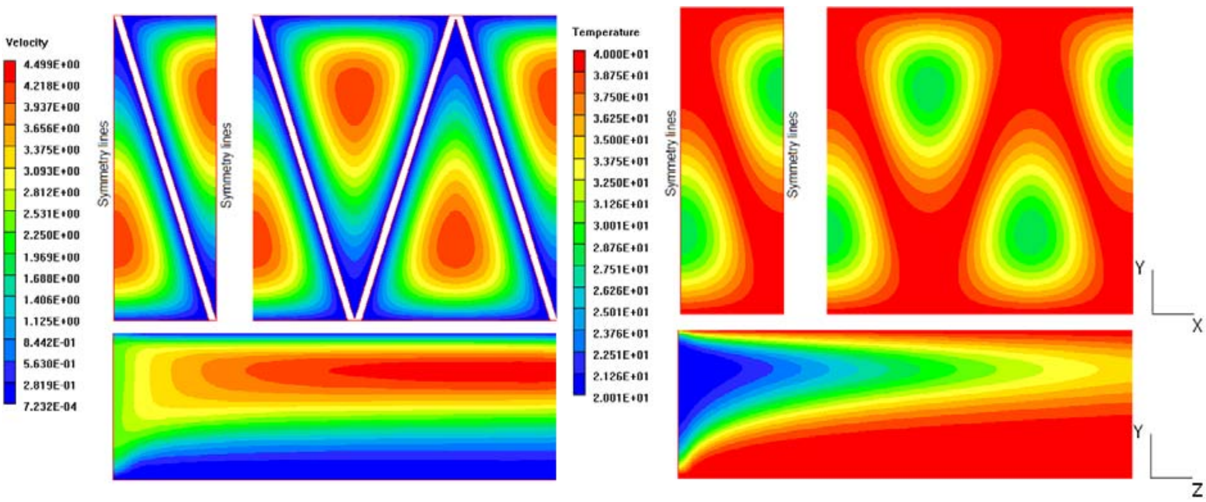

Figure 4. Profile of velocity and temperature to the laminar flow in model 11.94T ($X = 0$ and $Z = 0{,}05m$).

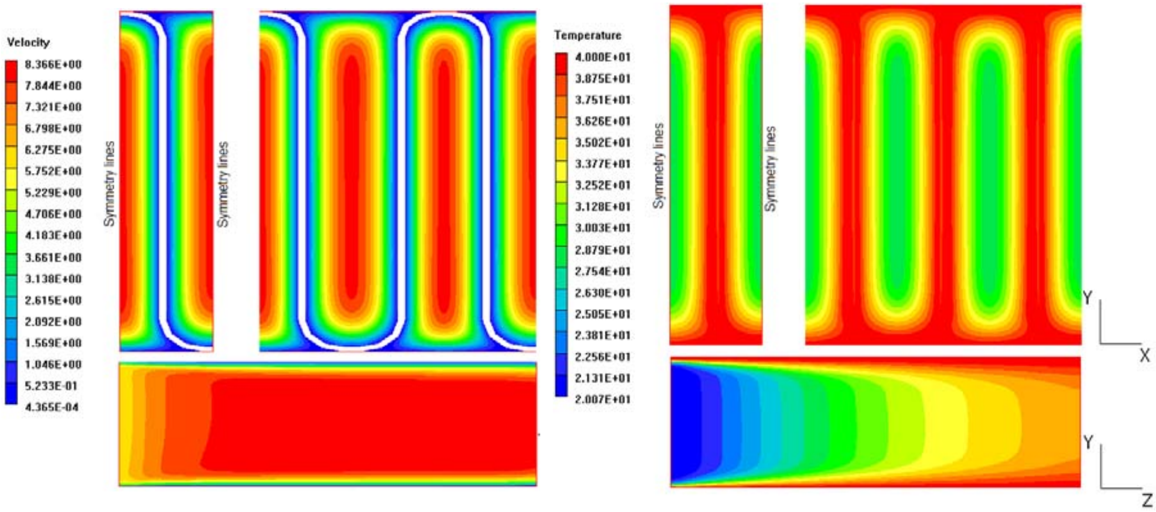

Figure 5. Profile of velocity and temperature to the laminar flow in model 15.08 ($X = 0$ and $Z = 0{,}05m$).

The velocity and temperature distribution to the model 15.08, indicating symmetry in flow is showed in Figure 5. This profiles referee to a laminar flow ($Re \approx 830$), with uniform inlet velocity of 5 m/s and fluid inlet and wall temperature of 20ºC e 40ºC, respectively. In order to this simulation, the geometry was generated in software PRO/ENGINEER and imported to the PHOENICS software.

The heat transfer average and friction coefficients are so influenced by inlet region or flow development. In Figures 3, 4 and 5 is possible visualize the inlet region, indicating the hydrodynamics inlet length and development region of flow. An internal flow is hydrodynamic developed when your velocity profile doesn't change in the flow direction. The flow became developed from a specific position in Z coordinate. Observe that significant part of the flow stay in development region and that the profile together the wall of the ducts are very close to developed profiles. The hydrodynamic inlet length, to laminar flow with uniform profile of velocity in the channel inlet, can be approximate by the Eq (11), where (Re_D) indicate the Reynolds number and (Dh) hydraulic diameters of channel.

$$z_{fd,h} \approx \frac{Re_D\, Dh}{20} \tag{11}$$

In figures 3, 4 and 5 too is possible visualize the thermal development of flow along of the duct that contain the fins. In laminar flow and Prandtl numbers lower than 0,7, the temperature profile develop before of velocity profiles, as can be observed in figures. In turbulent flow, both hydrodynamic and thermal lengths are independent of Reynolds and Prandtl

For a turbulent flow, hydrodynamic and thermal lengths are practically independent of Reynolds number. The hydrodynamic and thermal lengths in inlet region to turbulent flows can be obtained, as a fist approximation, by equations (12) and (13), respectively (Incropera and De Witt, 2002).

$$10\, Dh \le z_{fd,h} \le 60\, Dh \tag{12}$$

$$z_{fd,t} = 10\, Dh \tag{13}$$

The numerical simulation considers simultaneous developments of velocity and temperatures profiles. When occurs the simultaneous developments of velocity and temperature in laminar flows, the values of heat transfer and friction average coefficients are approximately 8% higher than compared with developed flow. In order to turbulent flows, the values of (hm) and (fm) are 25% higher than foresee to developed flows.

6. Conclusions

In this study were numerically assessed the characteristic of tridimensional flow in laminar and turbulent regimes and of heat transfer convective process in three models of fins ducts showed by Kays and London (1955). The main goal of the simulation was the determination of heat transfer average coefficient, obtained by numerical simulation of flow in each channel, to the calculus of the heat transfer global coefficient (U) of a heat exchanger. A comparison between the experimental results (Kays and London, 1955) and the numerical results obtained was performed. The heat exchanges can be composed by duct from different geometry and can there aren't data available in literature to determination of (U). The goal of the process studies is allow that the heat transfer global coefficient can be determined in geometry more complex, in cases where there aren't data of heat transfer average coefficient (hm).

7. Acknowledgments

The author would like to thanks to CAPES (Coordenação de Aperfeiçoamento de Pessoal de Nível Superior) by the Doctorate Scholarship, Prof. Dr. Carlos Alberto Carrasco Altemani and State University of Campinas (UNICAMP) by the support during the PhD.

8. References

Kays, W. M., 1993, "Convective Heat And Mass Transfer", Ed. MacGraw-Hill, 3ºEd.
Kays, W. M., London, A. L., 1955, "Compact Heat Exchangers", Ed. MacGraw-Hill, 2ºEd.
Incropera, F. P., Dewitt, D. P., 2002, "Fundamentos de Transferencia de Calor e de Massa", Ed. John Wiley & Sons.
Agonafer D., Gan-Li L., Spalding B., 1996, "The LVEL turbulence model for conjugate heat transfer at low Reynolds
Spalding, B.D.,1994, "The Phoenics Encyclopedia".
Turner, M., Rotron, C. "All You Need to Know About Fans". Electronics Cooling, vol. 1, No. 2, p. 16-19, May. 1996.
London, A. L., Fergunson, C. K. "Gas Turbine Plant Regenerator Surfaces". Bureau of Ships Research Memorandum 2-46, Navships (250-338-3), July, 1946.
London, A. L., Fergunson, C. K. "Test Results of High Performance Heat Exchanger Surfaces Used In Aircraft Intercoolers And Their Significance For Gas Turbine Regenerator Designer". Trans. ASME, v. 71, p.12, 1949.