HEFAT2014
10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
14 – 16 July 2014
Orlando, Florida

EVALUATION OF THE LIQUID CHANNEL INFLUENCE ON HEAT TRANSFER PERFORMANCE IN A PLATE-FIN HEAT EXCHANGER BY MEANS OF THREE-DIMENSIONAL CFD SIMULATIONS

N. Rossi, M. C. Barbato*, D. Montorfano
*Author for correspondence
Department of Innovative Technologies,
University of Applied Sciences of the Southern Switzerland,
Manno 6928, Switzerland
*E-mail: maurizio.barbato@supsi.ch

ABSTRACT

Compact Heat Exchangers (CHEs) play an important role in a wide range of applications, e.g. in windmill gear units, machine tools, mobile hydraulic systems and so on. For each field, specific requirements are needed (i.e. heat transfer rate, mass, size and pressure drop), which are achieved also through different types of configuration for the fin geometry in both the oil and the air channel.

By means of CFD simulations of a small part of the CHE core, it is possible to know how a certain coupling of air and oil channels performs. However, when the number of configurations to be analysed is consistent, it is preferable to choose the smallest meaningful computational domain, in order to reduce computational resources while keeping all important physical phenomena.

The purpose of this paper is to demonstrate that, despite oil flow can lead to convective heat transfer coefficient several times higher with respect to air flow, a change in the fin geometry affects significantly the “conjugate heat transfer” and the CHE performance. Besides, it is presented a simplified model for the heat exchanger element where the oil channel conductive and convective heat exchange are modelled by using a fixed temperature boundary condition and an effective thermal resistance. 3-D simulations were carried out considering a fixed fin geometry for the air channel and five different fin geometries for the oil channel. Two cases of operative conditions were taken into account. Furthermore, for each coupling of channels, the simplified model was applied.

Results demonstrate that a change in the fin geometry for the oil channel affects the overall heat transfer, and this influence is greater or smaller depending on the operative conditions. Secondly, the reduced model is shown to yield results with a reduction in accuracy that can not be neglected.

NOMENCLATURE

D_h	[m]	Hydraulic diameter
f	[-]	Average Fanning friction factor
h	[m]	Height of the offset strip fin channel
h_c	[W/(m^2·K)]	Heat transfer coefficient
j	[-]	Colburn factor [= $St \cdot Pr^{2/3}$ or $Nu/(Re \cdot Pr^{1/3})$]
k	[W/(m·K)]	Thermal conductivity
l	[m]	Length of fin
L	[m]	Length
$\dot{m}$	[kg/s]	Mass flow rate
N	[-]	Number of fins in the channel
Nu	[-]	Average overall Nusselt number based on hydraulic diameter
Δp	[Pa]	Pressure drop
Pr	[-]	Prandtl number
Re	[-]	Reynolds number based on hydraulic diameter
Re^*	[-]	Critical Reynolds number based on Joshi and Webb [4]
St	[-]	Stanton number based on hydraulic diameter
s	[m]	Fin spacing
t	[m]	Fin thickness
T	[K]	Temperature
ΔT	[K]	Temperature difference
U	[m/s]	Velocity
$\vec{u}$	[m/s]	Velocity vector
y^+	[-]	Wall distance for a wall-bounded flow

Special characters

α	[-]	Aspect ratio s/h
γ	[-]	Ratio t/s
δ	[-]	Ratio t/l
β	[°]	Fin angle

Subscripts

eq	Equivalent
w	Wall

INTRODUCTION

Compact heat exchangers, which usually present a high surface area/volume ratio, can be classified in either plate-fin or tube-fin heat exchangers. In the former, a variety of extended surfaces are used such as plain fins, wavy fins, offset strip fins, perforated fins, pin fins and louvered fins [1]. The work carried out in this paper will consider only offset strip fins, which among the aforementioned types of geometries are widely used.

In the last 60 years a lot of analytical, numerical and experimental investigations have been conducted for offset strip fins and the obtained results have provided important information about heat transfer rate, pressure drop and the complexity of flow structures. The work done by Manglik and Bergles [2] provided a thorough review of the extensive research and the literature on interrupted surfaces in an offset strip arrangement. In addition, they presented generalised correlations for the friction factor f and the Colburn factor j. However, generally focus has been on the description of the performance of the fin geometry in a single channel, whereas in this paper the overall performance of a coupling of channels was considered.

In a CFD model, that comprised two coupled channels with offset strip fins, two fluids were used, respectively air and oil. With the intent to reduce the model without compromising the results, two pieces of information were necessary. Firstly, since oil flow can lead to a convective heat transfer coefficient several times higher with respect to air flow, it was of interest to evaluate the influence that changes on the fin geometry of the oil channel have on the “conjugate heat transfer”. Secondly, in order to assess the feasibility, an attempt was made to substitute the oil channel with an imposed temperature and adding an esteem of the thermal resistance due to convection of the fluid and conduction through the fins.

A validation of the CFD model was made through a comparison with experimental data of fin geometries analysed by Kays and London [3]. The influence of changes in the fin geometry on “conjugate heat transfer” and the effectiveness of the reduced model were studied considering five different fin configurations and two operative conditions.

CFD MODEL DESCRIPTION

Computational domain

The model taken into consideration is a small part of the CHE core as shown in Figure 1.

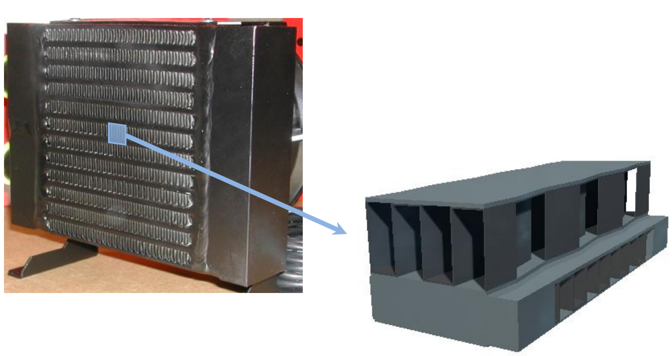

Figure 1 Compact Heat Exchanger core element

Starting from this small element, the computational domain was built. The two fluids, air and oil, go through the channels in cross-flow. The whole length of the air channel (L = 28.5 mm) is used and hence two additional zones were considered in the computational domain: an inlet zone to account of the incidence of the air before entering into the channel, and an outlet zone to avoid the backflow during the simulations. For both inlet and outlet air zones a length of 80 mm was adopted. For the oil channel, the whole width is considered and a supplementary fluid row is added before the outlet in case of backflow. Fluid rows before the inlet are not necessary because the input parameters are obtained from a simulation of a periodic part of the oil channel. In such way, the input parameters applied to the oil inlet correspond to a fully developed regime condition. In Figure 2 is illustrated the computational domain which is enclosed in space of approximately 12 mm x 15 mm x 190 mm.

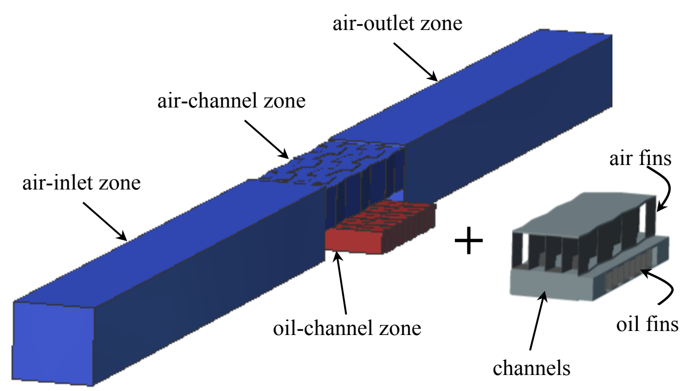

Figure 2 Computational domain

Computational grid

For all simulated cases the mesh was generated separately for the air channel and the oil channel. This entailed the use of non conformal interfaces which were set in the thin solid plates that separate the channels and on the outer walls of the oil channel. The decision to have the interfaces in the solid was taken because there is no mass flow there. Therefore, the unavoidable interpolation is not applied to fluid flow properties and affects less numerical results accuracy. In Figure 3 is shown where the interfaces were placed.

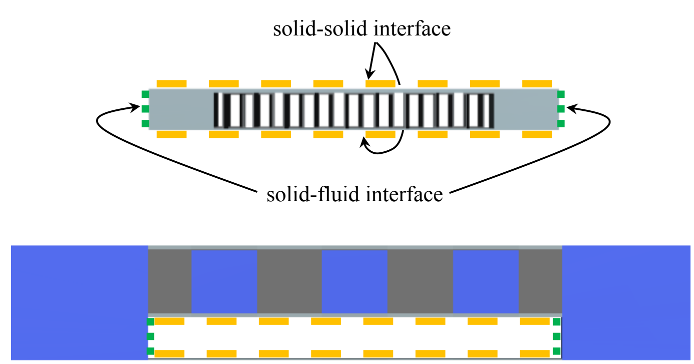

Figure 3 Non-conformal interfaces

Since the mesh was generated separately, in both channels was possible to achieve always an hexahedral mesh of very good quality. In the fluid zones the first element thickness was chosen, based on the operative conditions, in order to obtain a desired value of y^+ depending on the turbulence model. In average the mesh size for different cases was about 5 millions of elements.

Boundary conditions

The core of the heat exchanger can be represented as a matrix composed by a certain number of small elements, as the one that we used for our computational domain. Far from the borders, each element is surrounded by four nearly identical elements. Accordingly, lateral faces were linked with a periodic condition as well as top and bottom faces. At the air inlet was imposed a uniform velocity profile, whereas at the oil inlet a velocity profile obtained from the simulation of a periodic part of the oil channel was used. Regarding the outlets, for both was set a pressure-outlet condition. Figure 4 illustrates inlets, outlets and periodic faces location.

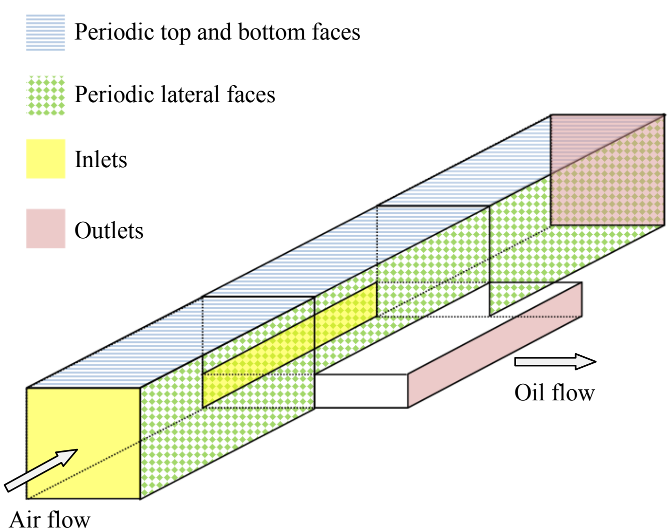

Figure 4 Locations of inlets, outlets and periodic faces

Modelling equations

Both flows were considered viscous, incompressible, with temperature dependant properties and, depending on operative conditions, in a laminar or turbulent regime. On the basis of these considerations, continuity, Navier-Stokes, energy, turbulent kinetic energy and turbulent dissipation rate transport equations were numerically solved with the finite volume method approach [5] by means of Fluent code from ANSYS.

Turbulence model

The steady state turbulent flow was solved with RANS equations, k-ε (Realizable) and k-ω (SST) turbulence models were tested. The latter was chosen given that it yielded better results for heat transfer, namely discrepancies for the Colburn *j* factor in the validation process were smaller than the other model, and that it can handle to some extent boundary layer transition. When using k-ω (SST) turbulence model, it is advised to have a value of y+ of about 1 or lower. As aforementioned, each mesh was generated taking account of this requirement.

Numerical schemes and convergence criteria

The numerical schemes used to solve the discretised model equations were all second order accurate; second order upwind schemes were applied. For the pressure velocity coupling, the SIMPLE algorithm was used. The iterative solution process was limited using values, reported in Table 1, for the numerical residuals:

Table 1 Residuals limits applied to simulations

Equation	Threshold
Continuity	$< 10^{-3}$
Navier-Stokes	$< 10^{-4}$
Energy	$< 10^{-6}$
Turbulent kinetic energy	$< 10^{-3}$
Turbulent dissipation rate	$< 10^{-3}$

To ensure the achievement of fully convergence, as further conditions, temperatures at the air outlet and the oil outlet were monitored. Therefore, once the residuals were under the fixed threshold, the computations were left to continue until the two quantities were stable.

OIL FINS REDUCED MODEL

The reduced model consisted in modelling, with an electrical analogy, the set of thermal resistances due to convection of the fluid and conduction through the fins in the oil channel. Then, to the inner walls of the oil channel was assigned a wall thickness to model a thin layer of material with an equivalent thermal conductivity calculated as:

$$k_{eq} = \frac{\Delta x}{A \cdot R_{eq}} \tag{1}$$

where A is the area of the wall with the thin layer. Figure 5 shows where the two thin layers were located in the model.

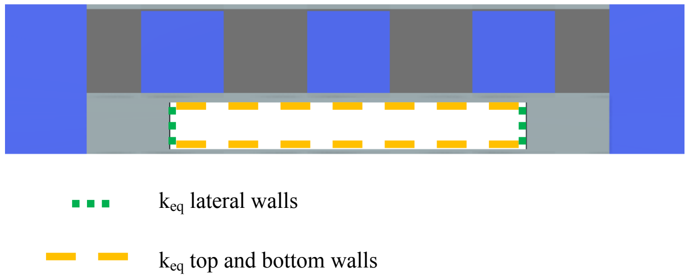

Figure 5 Thin layers of material with equivalent thermal conductivity

All considerations made for boundary conditions, modelling equations, turbulence model, numerical schemes and convergence criteria are valid also for the reduced model.

However, since fluid and fins are absent, a boundary condition of imposed temperature is applied to the inner walls of the oil channel.

Esteem of thermal resistance

The convective heat transfer coefficient is calculated starting from the Colburn modulus [7]:

$$j = \frac{Nu}{Re \cdot Pr^{1/3}} \tag{2}$$

and using the following dimensionless numbers:

$$Nu = \frac{h_c \cdot D_h}{k} \tag{3}$$

$$Re = \frac{\rho \cdot \overline{U} \cdot D_h}{\mu} \tag{4}$$

$$Pr = \frac{c_p \cdot \mu}{k} \tag{5}$$

In fact, equation (2) can be rearranged by using equation (3), equation (4) and equation (5) to isolate the heat transfer coefficient:

$$h_c = j \cdot Re \cdot \left(\frac{c_p \cdot \mu}{k} \right)^{1/3} \cdot \frac{k}{D_h} \tag{6}$$

The Colburn modulus for laminar and turbulent regimes is calculated by using the correlations of Manglik and Bergles [2].

For $Re \leq Re^*$, laminar flow region:

$$j = 0.6522 \cdot Re^{-0.5403} \cdot \alpha^{-0.1541} \cdot \delta^{0.1499} \cdot \gamma^{-0.0678} \tag{7}$$

For $Re \geq (Re^* + 1000)$, turbulent flow region:

$$j = 0.2435 \cdot Re^{-0.4063} \cdot \alpha^{-0.1037} \cdot \delta^{0.1955} \cdot \gamma^{-0.1733} \tag{8}$$

where the hydraulic diameter is defined as:

$$D_h = \frac{4 \cdot s \cdot h \cdot l}{2 \cdot (s \cdot l + h \cdot l + t \cdot h) + t \cdot s} \tag{9}$$

For fins and fluid, heat transfer occurs through four different types of resistances:

- R_1 convection from fluid to plate/fin base
- R_2 convection from fluid to fin projection
- R_3 conduction through fin base
- R_4 conduction through fin projection

Figure 6 depicts the aforementioned resistances.

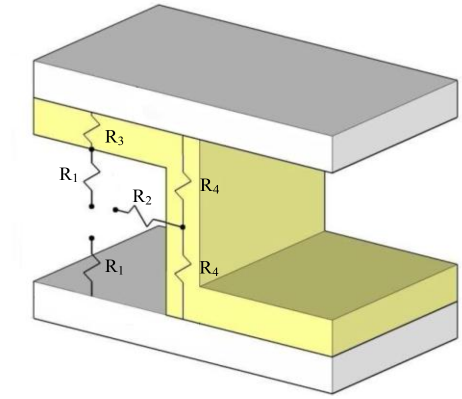

Figure 6 Fluid and fin thermal resistances (modified picture of ref. [7])

After defining the equations for each resistance, a thermal circuit was built, and an equivalent thermal resistance was calculated. The detailed procedure to obtain this data, which includes also a model proposed in [6], is presented in [7]. This resistance was used to determine the equivalent thermal conductivity to be assigned to the material of the thin layer for top and bottom walls of the channel.

As for lateral walls, the equivalent thermal conductivity was calculated using the following thermal resistance:

$$R_{eq} = \frac{1}{h_c \cdot A_w} \tag{10}$$

where A_w is the area of the lateral faces.

RESULTS AND DISCUSSION

In the phase of validation, f and j were compared with experimental data from Kays and London [3] for two fin geometries.

For the evaluation of both the influence of geometrical changes in the oil channel and the effectiveness of the reduced model, the heat flux of each case was determined and used as criterion of comparison.

Processing of results

From the numerical results, the Fanning friction factor was calculated as [8]:

$$f = \frac{\Delta p \cdot D_h}{2 \cdot \rho \cdot \overline{U}^2 \cdot L} \tag{11}$$

Where Δp is the pressure drop between inlet and outlet and L is the channel length. The Colburn factor was determined with equation (2).

Each fluid property and its temperature were obtained averaging on the flow cross section as:

$$\phi = \frac{\int \phi \cdot \rho \cdot \left| \vec{u} \cdot d\vec{A} \right|}{\int \rho \cdot \left| \vec{u} \cdot d\vec{A} \right|} \tag{12}$$

The mean value between inlet and outlet was then calculated and used. The heat transfer coefficient was deduced from the heat flux between the two fluids and the average temperature difference between walls and fluid:

$$h_c = \frac{Q}{A \cdot \left(\overline{T}_w - \overline{T}\right)}, \qquad \overline{T} = \frac{1}{A} \cdot \int T \cdot dA \tag{13}$$

For the hydraulic diameter equation (9) was always used.

Validation

To verify the validity of the model, in the coupled channels two different fin geometries (see Table 2) were simulated at six different Reynolds numbers. The CFD results for f and j of oil and air geometry were compared with published data of Kays and London and Manglik and Bergles [2] correlations respectively in Figure 7 and Figure 8. There was quite good agreement for both the friction factor, whose variations were found to be from 0.1% to 14% and the Colburn factor, which varied from 1.8% to 14%. However, in most cases the variation for both factors was around 7-8%.

Table 2 Fin geometries dimensions

	s [mm]	h [mm]	t [mm]	l [mm]
Oil side	1.267	2.553	0.102	3.175
Air side	2.085	4.483	0.102	4.521

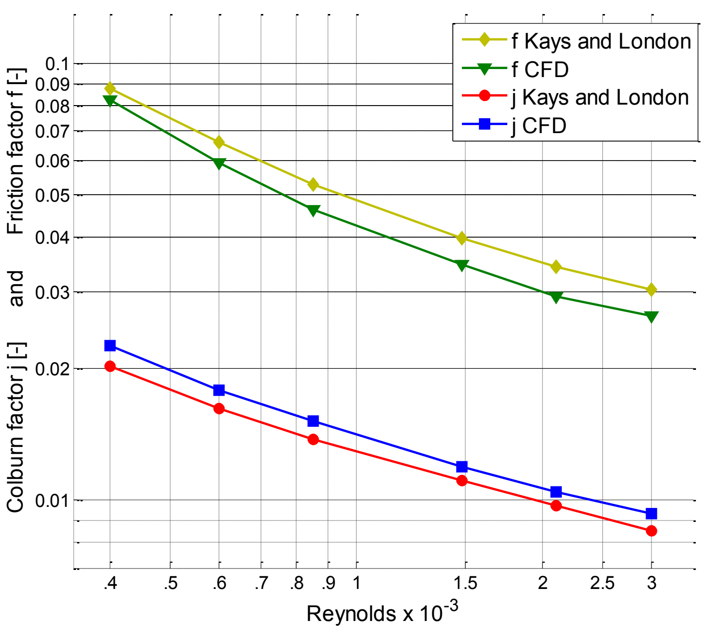

Figure 7 Comparison of results with Kays and London, *f and j* vs *Re* (oil side)

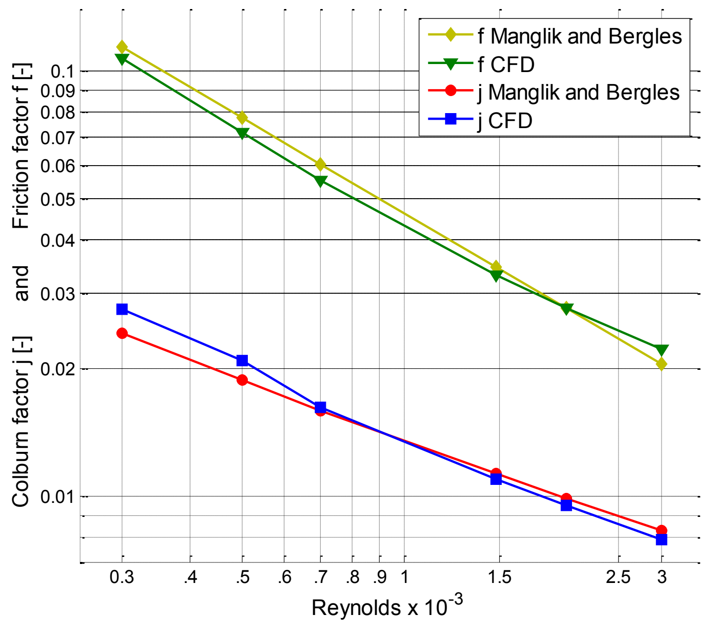

Figure 8 Comparison of results with Manglik and Bergles, *f and j* vs *Re* (air side)

Influence of geometrical changes

In Figure 9 are represented typical geometrical parameters, which all were changed, for an offset strip fin. The dimension that did not change was the height of the channel (h = 2.21 mm). In Table 3 are instead reported the ratios values and the angle of each case.

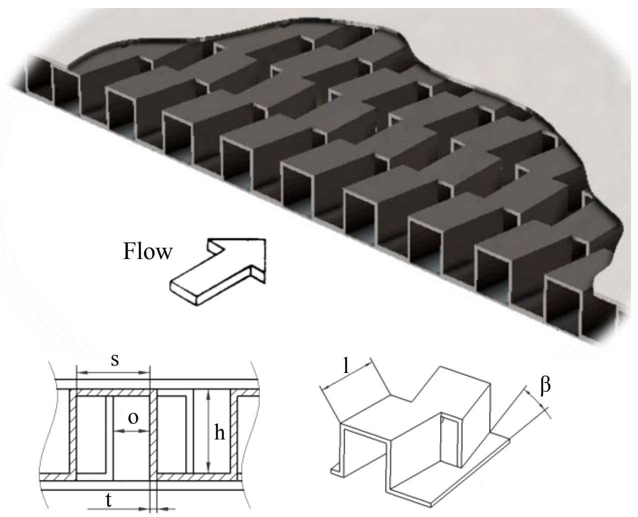

Figure 9 Geometrical description of offset strip fin analysed

Table 3 Dimensionless ratios and angle

	α (s/h)	δ (t/l)	γ (t/s)	β (°)
SMP-1	2.21	0.05	0.06	9.3
SMP-2	1.11	0.40	0.20	6.3
SMP-3	3.3	0.03	0.02	9.3
SMP-4	1.63	0.18	0.12	0
SMP-5	2.64	0.04	0.04	0

The influence of the five geometrical changes was studied in two diverse operative conditions:

a) considered turbulent flow for air (Re ≅ 2700) and laminar flow for oil ($74 \leq Re \leq 114$)

b) considered laminar flow for air (Re ≅ 630) and turbulent flow for oil ($1262 \leq Re \leq 1935$)

In this way it was possible to evaluate the influence when the air channel plays or does not play a significant role. In Figure 10 is illustrated the heat flux for each coupling operating in condition a). Percentage variations were calculated in respect to SMP-1. Changes brought to the fin geometry led to a maximum increase of 27.3% and a minimum decrease of -12.9% in the heat flux.

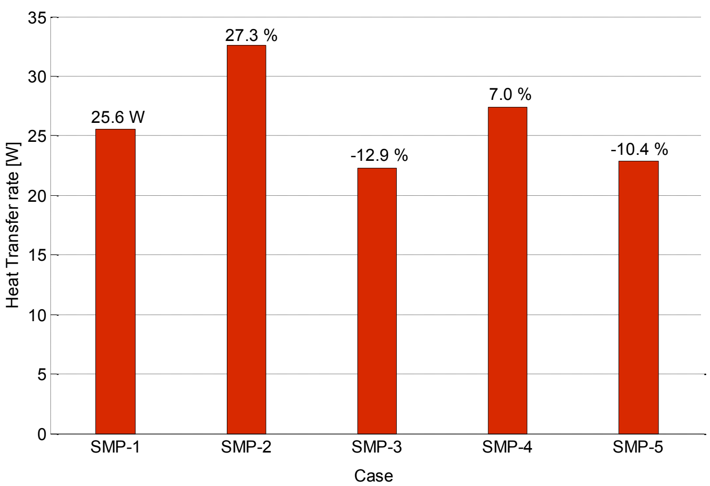

Figure 10 Variation of *heat flux* vs *fin geometry* turbulent air flow, laminar oil flow

A smaller range of variation resulted from the same fin geometries operating in condition b). Always in respect to SMP-1, variations went from -3.6% to 6.3%. These results are reported in Figure 11.

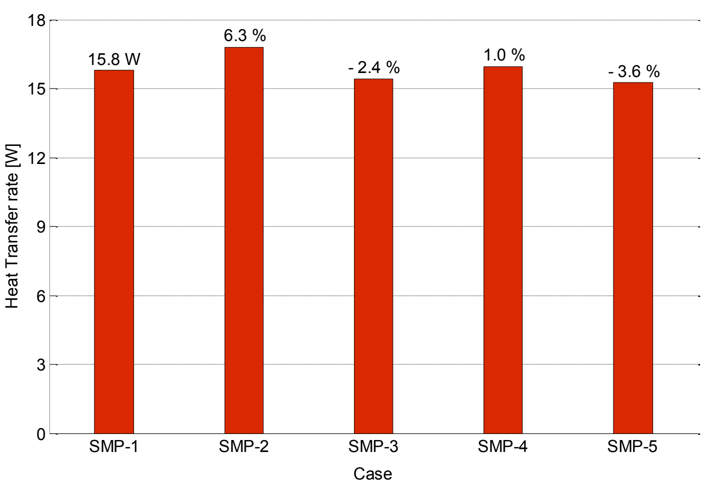

Figure 11 Variation of *heat flux* vs *fin geometry* laminar air flow, turbulent oil flow

Effectiveness of reduced model

The evaluation of the effectiveness was made by comparing the heat flux of the reduced model with the corresponding detailed model (see Figure 2). The percentage variation are reported for each case. It was found that, when oil flow was laminar, the reduced model underpredicted the heat flux. Besides, the discrepancy was higher when the fin geometry comprised an angle and for these cases variations were over -6.5%. For cases SMP-4 and SMP-5, where the angle value was zero, the heat flux was underpredicted at most of -2.3%.

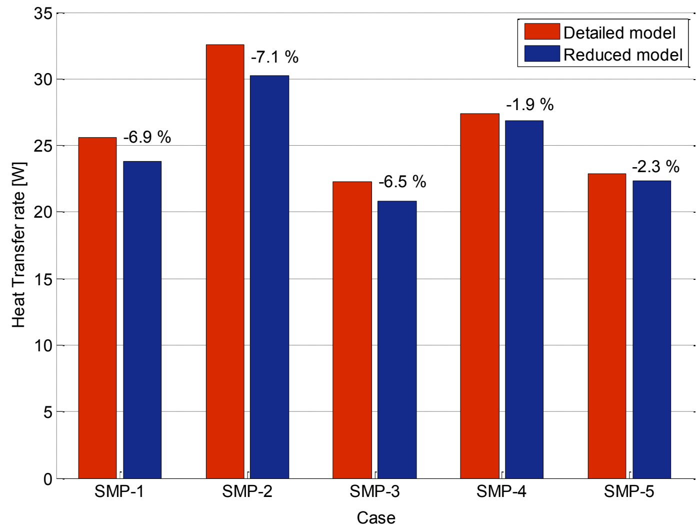

Figure 12 Comparison between detailed and reduced model turbulent air flow, laminar oil flow

In the operative condition where the oil flow was turbulent, the heat flux was also underpredicted, except for case SMP-5 in which there was a slight increase of 0.9%. For all cases, the prediction was more accurate in comparison with operative condition a). However, the greater difference (-5.2%) is again obtained in a fin geometry that comprised an angle.

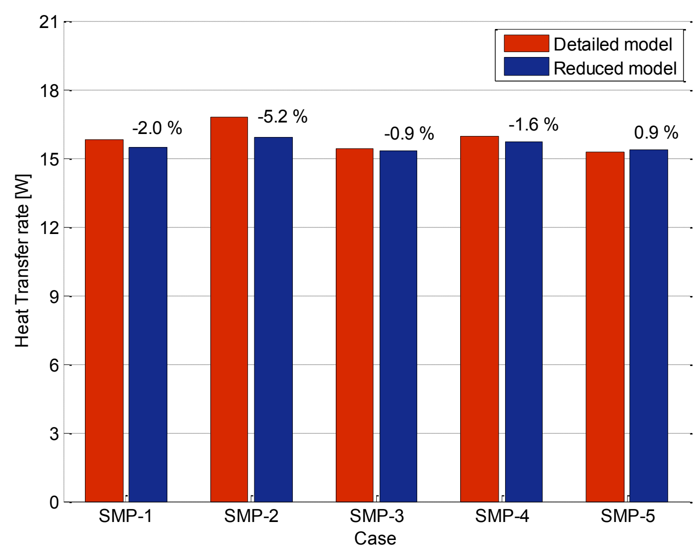

Figure 13 Comparison between detailed and reduced model laminar air flow, turbulent oil flow

CONCLUSION

The heat transfer for a coupling of two channels in cross-flow, with air and oil as fluids, was analysed by means of CFD. Offset strip fins were used inside both channels. One of the purposes of this paper was to demonstrate that a change in the fin geometry for the oil channel affects the overall heat transfer, and this influence is greater or smaller depending on the operative conditions. Indeed, numerical results showed that a geometrical modification can cause heat flux variations from -12.9% up to 27.3% in respect to a chosen sample case. These high differences were achieved in operative condition a). On the contrary, in operative condition b) the heat flux was less affected by geometrical changes and variations were smaller, from -3.6% to 6.3%, but not negligible.

The second purpose of this work, with the intent to reduce the computational effort, was to study the effectiveness of a reduced model for the coupled air-oil channels. On one hand, when the oil fin geometry comprised an angle, the model yielded results that were not acceptable in terms of accuracy. The heat flux was mostly underpredicted and variations up to -7.1% and -5.2% (with respect to detailed models) were obtained in operative conditions a) and b) respectively. This is due to the fact that correlations for offset strip fins used to prepare the reduced model do not take into account the fin angle. On the other hand, when the reduced model was applied to fin geometries without angle, the results were more accurate. The heat flux was anyway slightly underpredicted, but with variations up to -2.3% and -1.6% in operative conditions a) and b) respectively. Although the reduced model was able to describe quite well the heat transfer trend for the geometries analysed, as it can be seen in Figure 12, the difference in predicting the heat flux for fin geometries with or without the angle may lead to erroneous evaluation of the trend for other fin geometries. However, to extend the effectiveness of this approach, the Colburn j factor, needed to prepare the reduced model, could be computed by means of CFD from the smallest periodic element of the oil channel. This alternative though requires further investigations.

REFERENCES

[1] Ramesh K. Shah and Dušan P. Sekulic, *Fundamentals of Heat Exchanger Design*, John Wiley & Sons, 2003

[2] Manglik R. M and A. E. Bergles, Heat transfer and pressure drop correlations for the rectangular offset strip fin compact heat exchanger. *Experimental Thermal Fluid Science*, 1995 10, 171-180

[3] W. M. Kays and A. L. London, *Compact Heat Exchangers*, 2nd Edition, McGraw-Hill Series in Mechanical Engineering. New York, 1964

[4] H. M. Joshi and R. L. Webb. Heat Transfer and Friction in the Offset Strip-Fin Type Compact Heat Exchanger, *Int. J. Heat Mass Transfer* 30, 69-84, 1987

[5] H.K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method Approach*, England: Longman Scientific and Technical, Harlow, 1995

[6] Theodore L. Bergman, Adrienne S. Lavine, Frank P. Incropera, David P. DeWitt, *Fundamentals of Heat and Mass Transfer*, John Wiley & Sons, 7th edition 2011

[7] M.H. Teruel, C.Y. Nakashima and P. Paglione, *Rectangular offset strip-fin heat exchanger lumped parameters dynamic model*, 3rd CTA-DLR Workshop on Data Analysis & Flight Control, September 14-16, 2009, S. J. Campos, SP, Brazil

[8] Yunus A. Çengel and John M. Cimbala, *Fluid Mechanics Fundamentals and Applications*, McGraw-Hill, 2006