29th Congress of the International Council of the Aeronautical Sciences
St. Petersburg, Russia; September 7-12, 2014

HEAT EXCHANGER THERMAL STRATIFICATION MODEL

Celso Yukio Nakashima*, Luiz Tobaldini Neto*[,1]
***Embraer**

Keywords: *cross flow, heat exchanger, thermal stratification, modeling, bleed*

Abstract

Compact cross-flow plate-fin heat exchangers are largely used in aircraft systems for temperature control. Due to construction characteristics, the flow temperature profiles at the heat exchanger outlets are inherently stratified. Space constraints usually contribute to non-ideal installations directly influencing the inflow velocity distribution and the outflow development. These installation effects diminish the efficiency of the heat exchanger and may cause several operational problems. Therefore, it is important to be able to evaluate the influence of inlet flow velocity distribution, outflow development and thermal stratification in the heat exchanger efficiency and system operation.

This paper presents a method that couples models developed in Simulink with a commercial CFD solver (Fluent) to evaluate the outlet temperature profiles in air-to-air heat exchangers and how this profile evolves downstream the heat exchanger, considering the influence of installation effects (inflow velocity profiles) in heat exchanger effectiveness. The main goal of the present model is to be able to calculate 1) the global heat exchanger effectiveness and 2) the thermal stratification at the heat exchanger outlets including the effects of non-uniform inlet velocity profiles. Knowing the thermal stratification at the outlet allows for the study of the evolution of the temperature profile downstream the heat exchanger. Ultimately, this model can be applied to evaluate the influence of flow development in heat exchanger effectiveness and also to propose methods of improving the system operation.

1 Introduction

Cross-flow plate-fin heat exchangers (HX) are commonly used in aircraft systems due to its compactness and lower weight. In bleed systems they are usually employed to control the temperature of the air bled from engine compressors (hot air). As illustrated in Fig. 1, the hot air extracted from engine compressor is cooled with the engine fan air (cold air) before it is delivered to its clients (environmental control system – ECS – and anti-ice – AI – among others). The cold air flow is typically regulated with a valve that will operate in order to control a set-point temperature for the hot-air flow downstream the HX at the temperature sensor position.

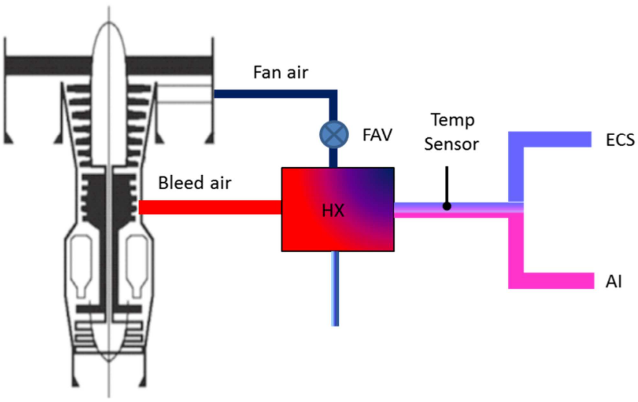


Fig.1. Bleed system temperature control.

However, the use of this type of HX brings some design challenges. One of them is the thermal stratification caused by the heat exchanged between the cross flow streams. Depending on the inlet conditions the outlet air stream may present a large difference between its lowest and highest temperature regions, as illustrated below.

1 PhD student at University of Toronto/Institute for Aerospace Studies

Even assuming uniform velocity and uniform temperature at the inlets, the hot and cold flow temperatures will continually change as the flow progresses inside the HX. This generates an outlet thermal stratification which is inherent to the HX design. For example, as seen in Fig. 2, the hot air outlet region closer to the cold inlet will experience the highest cooling effect from the incoming cold flow. The cold flow will then be continuously heated as it evolves inside the HX. This will make the hot flow incoming at the regions closer to the cold flow outlet to encounter a heated up cold flow, which will naturally have a lower cooling capacity. So, the hot air outlet region adjacent to the cold flow outlet will have higher temperatures than the hot air outlet region adjacent to the cold air inlet.

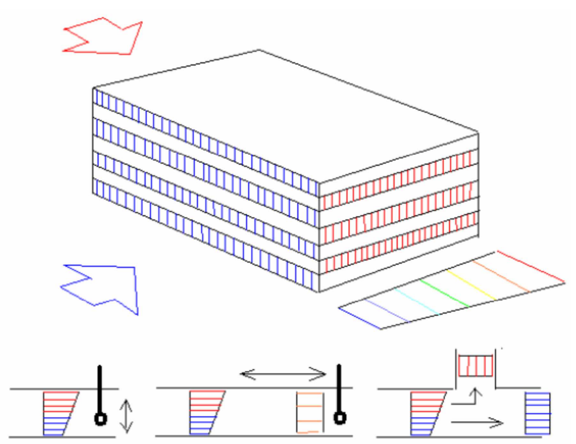


Fig. 2. Thermal stratification – cold inlet (blue) and hot inlet (red) in crossflow HX generate a temperature gradient at the outlet.

One important consequence of this stratification is that, if there is not enough length downstream the HX to allow enough mixing, the measured temperature can vary depending on the sensor position inside the duct. Another possible scenario that may be compromised by the outlet thermal stratification would be the need to split the flow downstream the HX. Depending on where this split is done it could lead to one colder stream and one warmer stream, causing potential operational and controlling issues to the systems served by these streams. Therefore the correct prediction of such phenomenon is important for the correct design of bleed, ECS and anti-ice systems.

Many papers studied heat transfer and temperature distributions inside ducts. Liberto and Ciofalo [1], for instance, studied numerically turbulent heat transfer in curved pipes with constant wall temperature. The curvature induces a thermal stratification in the flow. Lu et al. [2] used large eddy simulation to analyze temperature fluctuations in thermal stratified flows induced by a mixing tee with one cold branch injecting flow in a hot main duct. They calculated the temperature fields in two scenarios: with and without the use of a porous media in the mixing tee region. The porous media helps reducing temperature and velocity fluctuations downstream. The result is that in a given downstream position the mixing is better without the porous media since any upstream turbulence is attenuated by the porous media. In plate-fin HXs this effect can be even worse since the plates will avoid vertical mixing.

Another characteristic of HX installation in aircrafts is that, due to space constraints, it is practically impossible to have the equipment operating under an ideal inflow distribution condition. If the duct lengths are not enough to allow full development of flow, the flow distribution will be uneven as illustrated in Fig.3 and may influence the HX actual effectiveness. In general these equipments are usually designed and tested only in ideal operational conditions (uniform inflows).

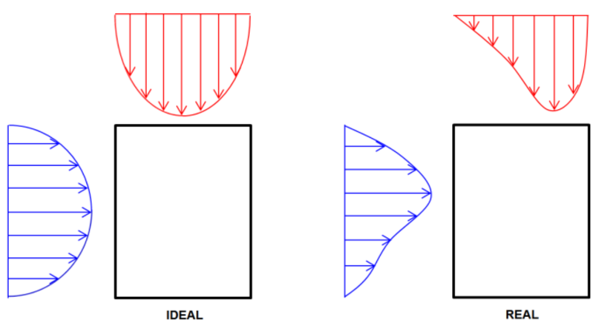


Fig. 3. Ideal and real flow profiles.

Zhang [3] carried out a CFD calculation of plate-fin cross flow HXs and showed that the velocity non-uniformity will cause thermal performance deterioration especially when pressure resistance is small. For high pressure drop, the HX core induces flow redistribution reducing the effect of inlet maldistribution. Yaïci, Ghorab and Entchev [4] did similar analysis with plate and tube cross flow HXs and proved that different velocity profiles will have a huge influence in thermal performance. The same conclusion achieved by Mao et al. [5] for louvered fin and tube HXs. In the work of Lalot et al. [6] it is indicated that crossflow HXs configurations are the most sensitive to flow maldistribution, with potential effectiveness losses of up to 25% depending on the level of velocity non-uniformity. Specific studies about outlet thermal stratification in cross flow HXs and downstream mixing were not found.

Laboratory tests of the equipment in different configurations can be very expensive and time-consuming. Therefore, it is important to develop a method to be able to simulate the effects of flow distribution and thermal stratification on the system's performance. This would allow for the rapid evaluation of different design alternatives without the risks and costs incurred with multiple laboratory tests. This paper describes a method developed to calculate the thermal stratification caused by cross-flow HX's in bleed systems. The method can also be used to evaluate the influence of velocity distribution in HX performance. The main goal is to identify impacts of HX installation in the HX performance, evaluate the thermal stratification downstream HX, and also to analyze methods to improve HX performance and reduce this stratification.

2 Methodology

2.1 General Description

In order to evaluate both flow development and thermal stratification it was necessary to use different tools for complete modeling. In Fig. 4 the flowchart describing the methodology is shown. A commercial CFD solver, Fluent® , was used to calculate velocity distribution upstream the HX considering all installation characteristics. In Matlab® it was developed a simple method based on the ε–NTU method to calculate the outlet temperature profile based on the effectiveness map of the HX. The upstream velocity distribution calculated in Fluent® and HX effectiveness map are used to calculate the temperature distribution at the HX outlet. Then, the hot outlet temperature profile obtained is used as input to the downstream flow model so that the evolution of the temperature distribution downstream the HX can be evaluated.

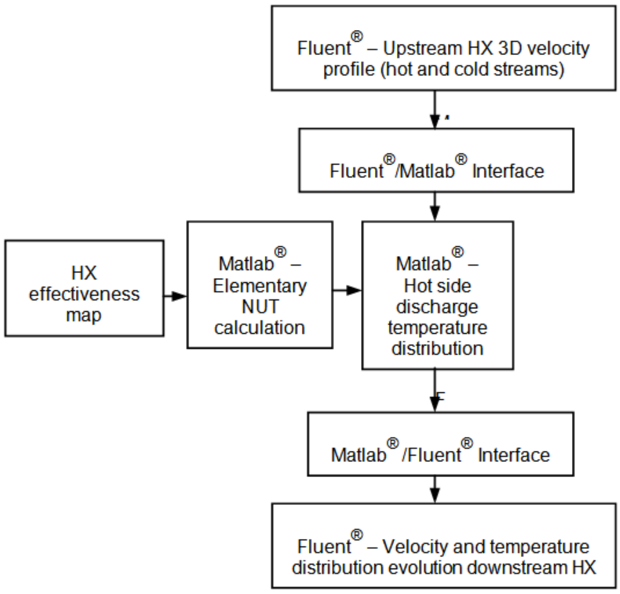


Fig. 4. Methodology flowchart.

2.2 Upstream flow modeling

The inlet velocity distribution is influenced by several parameters: presence of valves or orifices close to the inlet, bends or splitters, and headers geometry as illustrated in Fig. 5. The main goal of the upstream CFD analysis is to provide the velocity distribution at HX cold and hot inlets taking into account the actual installation characteristics defined by the system engineers. These distributions will then be used to define the hot and cold flow distributions entering each part of the HX in the ε-NTU method. Fig. 6 shows the result of the influence of the ducts and valves in the hot side inlet velocity distribution. This result shows that each portion of the HX inlet is subjected to different mass flows and therefore the heat transfer effectiveness will be different in each part of the

HX since it is directly related to hot and cold mass flows.

The CFD simulations of the inlet flows are done using a second-order finite volume [7], pressure-based scheme [8]. The domain is discretized using tetrahedral meshes for all the inlet ducts and uniform hexahedral meshes for the HX volume. Fluent's native HX model is active for these simulations because the known boundary conditions are downstream the HX. Simulating the proper pressure drop through the HX core is important to recover a realistic pressure distribution on the HX inlet face. A k-ω SST turbulence model is used [9].

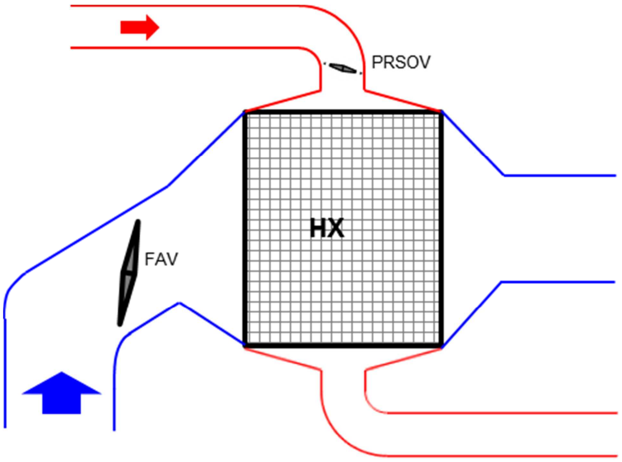


Fig. 5. Illustration of a HX installation constraints in aircrafts.

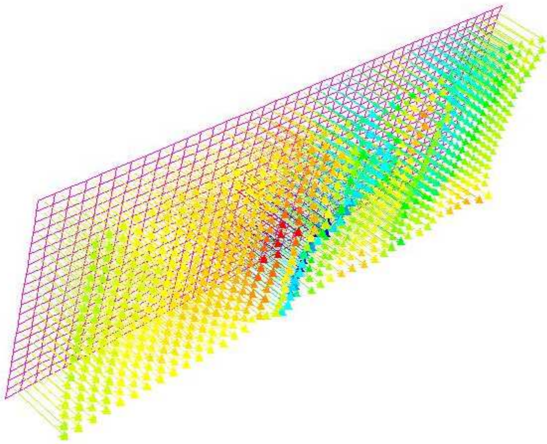


Fig. 6. Velocity distribution at hot side inlet.

The CFD model HX discretization may not necessarily match the HX discretization used in the Simulink model, the former being usually much finer than the latter. The cold and hot velocity distributions calculated in Fluent are then averaged and re-mapped into the coarser discretization of the Simulink model, as shown in Fig. 7 to be used as inputs to the discretized HX model as explained in next item.

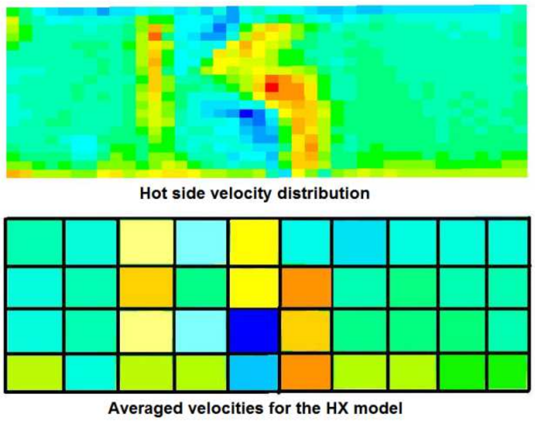


Fig. 7. CFD averaged velocities to be used in the discrete HX model.

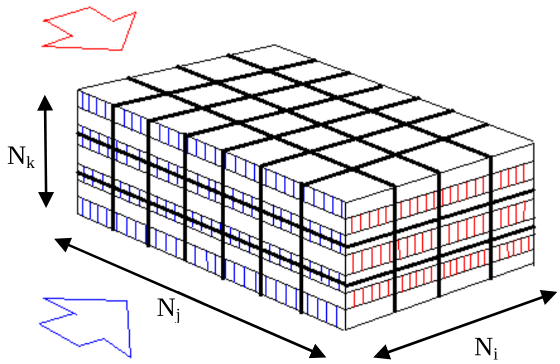


Fig. 8. Precooler discretization.

2.3 ε-NTU scaling method

The thermal and pressure drop performance of HXs are usually provided by the equipment suppliers as pressure drop curves and effectiveness maps. This is typically the only information available so it was decided to use it to create a method to extrapolate the performance to non-ideal installation conditions and evaluate the temperature distribution at the HX outlet. The effectiveness map gives only global behavior of the precooler, therefore, to obtain the temperature distribution, the approach used was to divide the precooler in small portions (Ni, Nj and Nk divisions for cold, hot and no-flow lengths respectively) and use the NTU definition and the ε-NTU relationship to scale down the effectiveness map for each discrete portion as illustrated in Figs. 8 and 9 below.

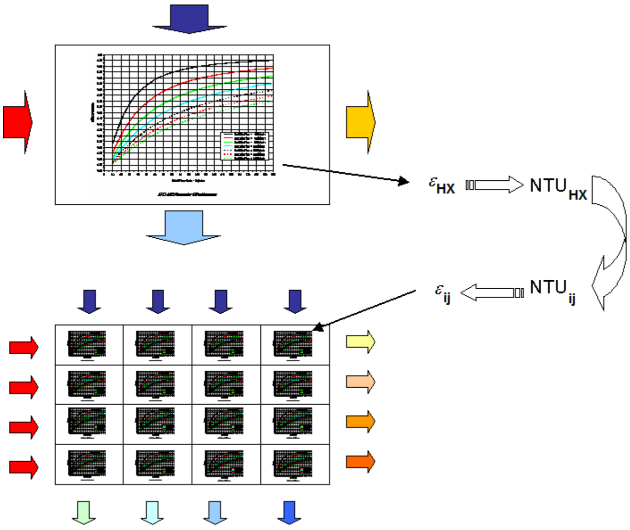


Fig. 9. Calculation of the effectiveness map for each discrete part of the divided HX.

The number of transfer units (*NTU*) is defined as the dimensionless capacity of the HX and it is given by:

$$NTU = \frac{UA}{\dot{m}_{min}\, c_p} \quad (1)$$

If the HX is divided in smaller portions (N_i x N_j x N_k) then the NTU_{ijk} of each portion will be given by:

$$A_{ijk} = \frac{A}{N_i N_j N_k} \quad (2)$$

$$\dot{m}_{H\,ijk} = \frac{\dot{m}_H}{N_i N_k},\ \dot{m}_{C\,ijk} = \frac{\dot{m}_C}{N_j N_k} \quad (3)$$

$$NTU_{ijk} = \frac{UA_{ijk}}{\dot{m}_{min,ijk}\, c_p} = \begin{cases} NTU / N_i & ,\dot{m}_{min,ijk} = \dot{m}_{C,ijk} \\ NTU / N_j & ,\dot{m}_{min,ijk} = \dot{m}_{H,ijk} \end{cases} \quad (4)$$

Where NTU of the HX can be obtained from the original effectiveness map according to the ε-NTU relation for cross flow HXs without mixture [10]:

$$\varepsilon = 1 - \exp\left\{ \left(\frac{1}{c_r} \right) NTU^{0.22} \left[\exp\left(c_r NTU^{0.78} \right) - 1 \right] \right\} \quad (5)$$

With the effectiveness defined as:

$$\varepsilon = \frac{\dot{m}_{hot} \left(T_{hot,in} - T_{hot,out} \right)}{\dot{m}_{min} \left(T_{hot,in} - T_{cold,in} \right)} \quad (6)$$

and is provided by HX suppliers in order to calculate the average hot and cold temperatures at HX outlets.

After calculating the NTU_{ijk} using Eq. (4), the Eq. (5) can be used again to calculate the effectiveness map ε_{ijk} for each discrete part of the HX.

With the effectiveness map for each discrete part it is possible to evaluate the heat transfer in each portion of the HX instead of the global heat transfer given by the original effectiveness values. Therefore the sequential calculation in each discrete part, from the inlet to the outlet, will give the temperature distribution at both the hot and cold outlets as shown in Fig. 10. Also, the division of the HX allows for the evaluation of non-homogeneous flows since it is possible to attribute different flow values ($\dot{m}_{H\,i,0,k}$ *and* $\dot{m}_{C\,0,j,k}$ in Fig. 10) in each element of the HX. With this approach, it is possible to impose different flow distributions in both hot and cold sides and evaluate their influence in HX effectiveness and outlet thermal stratification. Therefore the hot and cold flow distributions calculated in the upstream flow CFD analysis can be used instead of considering an evenly distributed flow.

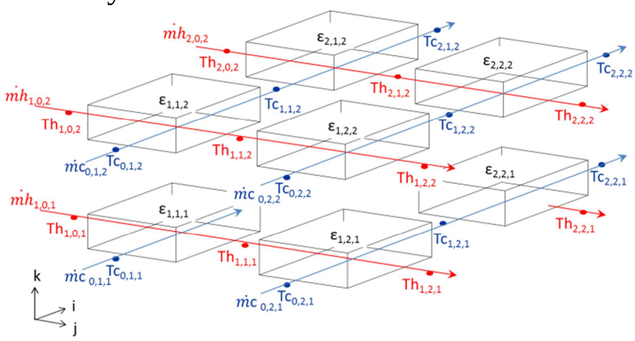


Fig. 10. Example of the division of a HX and the sequential calculation process.

Fluent has its own built-in methodology to model HXs. This methodology is somewhat similar to the described above in the sense that HX volume is also re-organized in small heat transfer units which include several finite volume cells inside the domain. However, in this model it is possible to simulate the inlet velocity distributions for only one side (hot or cold flow). The other side is assumed to have an uniform velocity distribution and could be composed of a multiple passes configuration. One of the major benefits of the present approach is that one can impose the velocities distribution on both hot and cold sides of the HX.

2.4 Downstream flow modeling

The outlet temperature profile calculated using the ε-NTU method is used as boundary condition in the inlet of the downstream duct. The velocity distribution is assumed to be homogeneous due to the high pressure drop in the hot side HX core that will homogenize momentum distribution in the flow. With these boundary conditions, the evolution in velocity and temperatures fields after the HX can be calculated.

3 Results

3.1 Validation of the ε-NTU method

The first step is checking whether the scaling method is consistent or not. Two different checks are conducted for this purpose. The first is intended to certify that the discretized HX has the same results of the original one. And the second is the validation of the method with experimental data.

In the first method, the HX is split as defined in item 2.3 and then each calculated outlet temperature are averaged:

$$\overline{T}_{H,out} = \frac{\sum_{i=1}^{N_i}\sum_{k=1}^{N_k} \dot{m}_{H\,i,0,k} T_{H\,i,N_j,k}}{\dot{m}_H} \quad (7)$$

$$\overline{T}_{C,out} = \frac{\sum_{j=1}^{N_j}\sum_{k=1}^{N_k} \dot{m}_{C\,0,j,k} T_{H\,N_i,j,k}}{\dot{m}_C} \quad (8)$$

and compared against the outlet temperature calculated with the original effectiveness map. The results obtained are almost the same with an average error of 0.9% and a maximum error of 1.6%.

For the second method, fractions (50%, 75% and 125% of the original size) of a HX core were tested. For these reduced cores, experimental effectiveness map are known. The scaling method was used to calculate the new effectiveness maps for each fraction using as input the original (100%) effectiveness map. The measured values obtained in the tests were then compared with the calculated values. Table 1 shows that the calculated values are close to the measured. The maximum error for the 50% core size is large but it occurs only when the effectiveness is very low (lower than 0.3) with a low influence in the outlet temperature.

Table 1: Comparison of the effectiveness values: experimental vs scaling method

Core size	Average error (%)	Maximum error (%)
50%	1.7	11.8
75%	1.0	4.9
125%	1.8	5.4

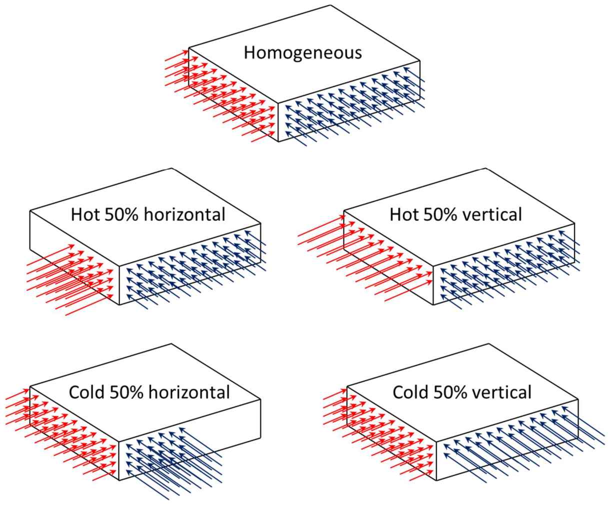


Fig. 11. Flow distribution scenarios.

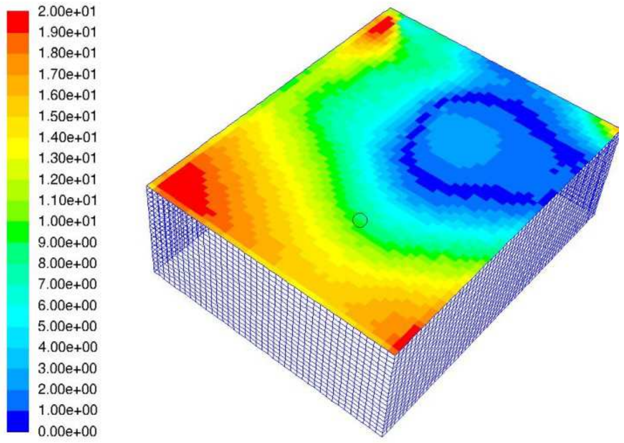
3.2 Influence of inlet flow distribution

After confirming the validity of the ε-NTU scaling method, it is applied to evaluate the possible influence of flow distribution on the effectiveness. For a given operational condition the scaling method is used to evaluate what would be the effectiveness if the flow is not evenly distributed but concentrated in one part of the HX. Fig. 11 shows the considered scenarios.

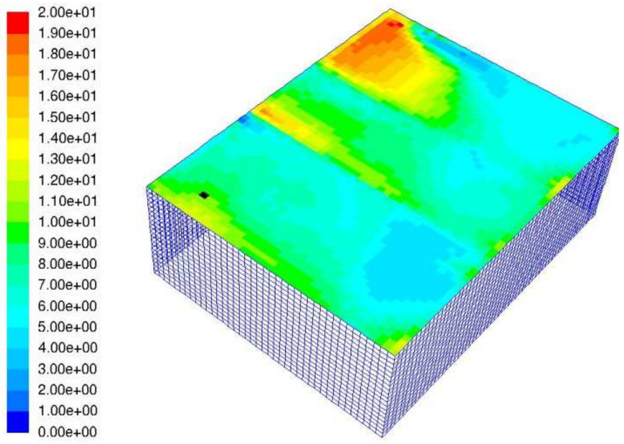
Table 2 shows that the effectiveness results can be reduced in critical cases where the flow is highly non-homogeneous. And both hot and cold velocity distributions can affect the results in a combined way.

Table 2: Influence of flow distribution in effectiveness

Flow scenario	Effectiveness
Homogeneous	0.436
Hot 50% horizontal	0.306
Hot 50% vertical	0.260
Cold 50% horizontal	0.321
Cold 50% vertical	0.270



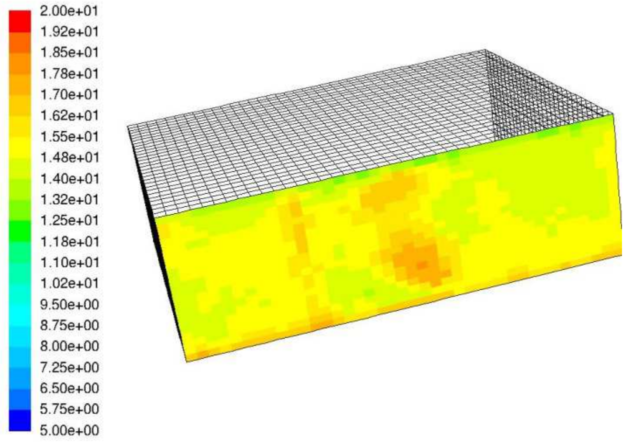
(a) No-vanes



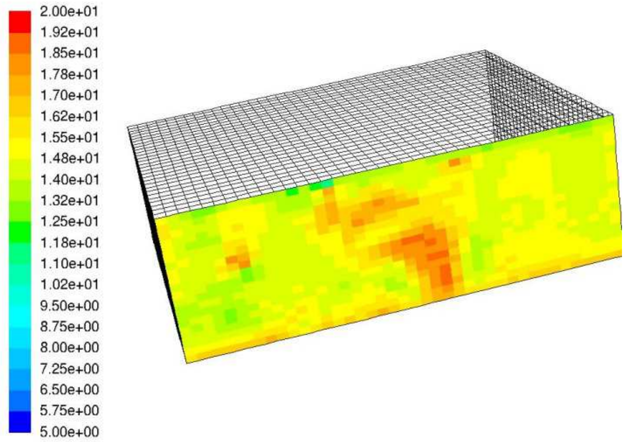
(b) 2-vanes

Fig. 12. Using vanes to modify cold side velocity distribution.

With the results showing the importance of having a velocity distribution as close to uniform as possible, it was decided to verify if it was possible to increase the installed HX performance with the use of devices (vanes) to improve flow distribution in both hot and cold sides. Two vanes were used in the cold side and 2 or 4 vanes were considered in the hot side. The velocity distribution for each option is shown in Figs. 12 and 13.



(a) 2-vanes



(b) 4-vanes

Fig. 13. Using vanes to modify hot side velocity distribution.

Table 3 summarizes the influence of different hot and cold vanes combinations. The effectiveness results are compared with the ideal homogeneous flow case. The results show that for the HX installation studied, the cold side vanes were effective to improve HX performance due to the better flow distribution as Fig. 12 shows. But the hot side vanes had no effect over effectiveness. The hot side vanes tested showed little or no influence in the performance. Due to the high pressure resistance, the hot side 4-vanes configuration is not improving the flow distribution significantly, as can be seen comparing the velocity contours at the HX hot inlet (Fig. 13). These results agrees with Zhang's [3] results since the hot air side has a much larger pressure resistance compared to the cold side.

Table 3: Influence of cold and hot sides flow distribution vanes in effectiveness.

Case	Effectiveness
Homogeneous	0.530
No-vanes cold/2-vanes hot	0.473
2-vanes cold/2-vanes hot	0.515
No-vanes cold/4-vanes hot	0.472
2-vanes cold/No-vanes hot	0.515

3.3 Downstream thermal stratification

The last study using the model is the evaluation of the thermal stratification. The purpose of this study was to verify if the flow temperature would be homogeneous at the downstream temperature sensor position.

The outlet temperature distribution calculated using the ε-NTU method is used as a boundary condition for the inlet of the downstream hot side duct. Fig. 14 shows one example of temperature profile for the duct inlet (HX' s hot outlet = downstream duct's inlet).

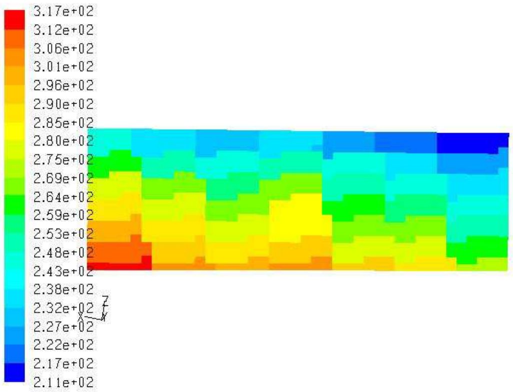
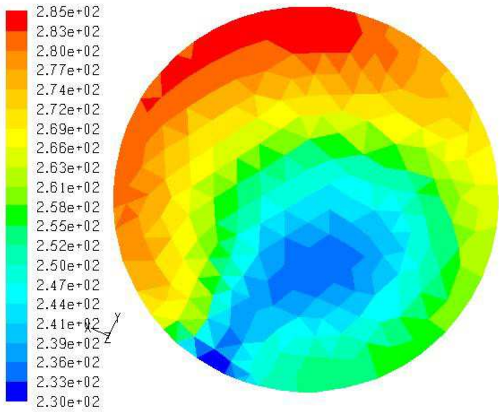
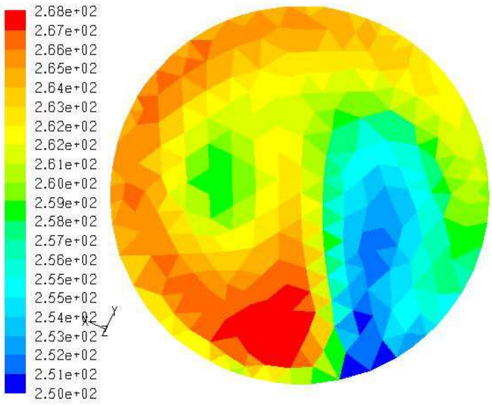


Fig. 14. Hot side thermal stratification at HX outlet/Downstream duct inlet.

Using this approach, the hot side downstream flow is simulated and the evolution of the temperature distribution until the temperature sensor position can be calculated. The same input was used to simulate the evolution of the flow until the temperature sensor position with the original duct and with a mixing enhancement device. Figs. 15 shows that the mixer is able to reduce thermal stratification from 55 to 18 oC at sensor position and the temperature distribution is less stratified.



(a) No mixer



(b) With mixer

Fig. 15. Temperature distribution at sensor position (cell center values with colormap ranges based on minimum and maximum for each case).

The results without mixer are also compared with experimental data. Fig. 16 shows the temperature distribution measured vertically and horizontally at the sensor position compared with two sets of calculated data: using Fluent's native HX model and using the scaling method. The red line is the set-point of the temperature control system. The results show that the scaling method developed in the present work provides a better prediction of the temperature distribution. The deviations verified closer to the duct walls may be related to uncertainties in the boundary conditions adopted (specified heat transfer coefficient and external temperature) and will be subject of future studies. Other uncertainties also arise regarding the exact positioning of the temperature reading sensors since the horizontal reading at 2.5in also shows a different temperature trend (similar to the CFD results) approaching the wall when compared to the reading at 0.0in horizontal.

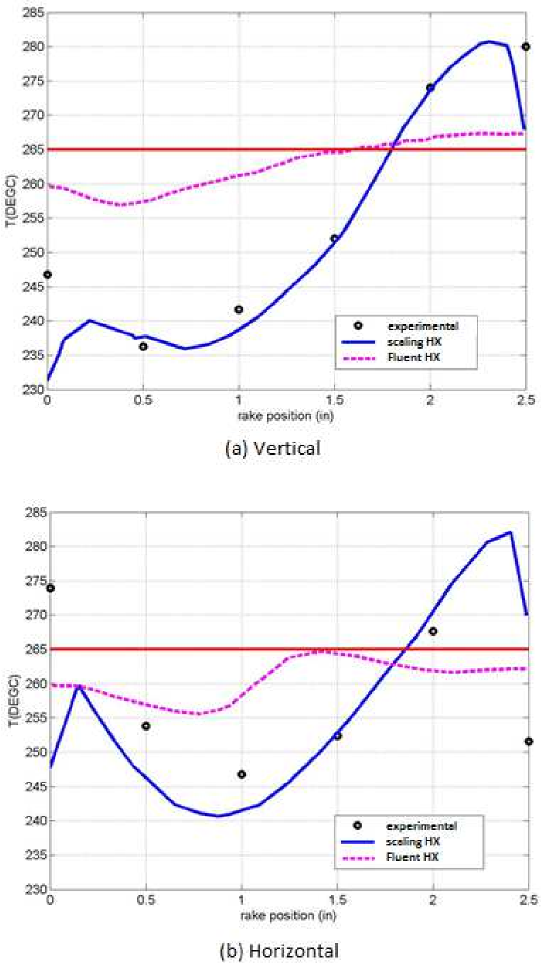


Fig. 16. Temperature distribution comparison at sensor position for the no-mixer case (Fig. 15a). - experimental (dots), present scaling method (full blue line), Fluent's native HX model (dashed magenta line), control set-point (full red line)

4 Conclusion

The proposed methodology proved to be a useful tool to evaluate thermal stratification and the influence of uneven inlet flow distribution in HX effectiveness. It allows the study of cross-flow HXs even without any geometrical data like plate thickness, fin length and others. Only with the effectiveness map, which is always provided by suppliers, it is possible to carry out the analysis. With the developed tool, different methods to reduce operational problems and improve HX performance can be rapidly simulated to find the best solution to be implemented in the aircraft and reducing the number of experiments. In the particular case of understanding the velocity distribution influence, further studies are still necessary for more robust conclusions about the results. The present model appears to be a more useful tool especially in cases with very non-uniform inlet flow velocities. The HX model can also be further improved. In the proposed method the flow distribution is considered to be the same in any cross-section of the HX. After setting up the flow in one element, this flow will be the same until the outlet section. In this type of HX core however, there is mass transfer perpendicularly to the main flow direction. This means that – as in any porous media – the flow will become more homogeneous as it flows across the HX core until the outlet. This effect could be taken into account by evaluating the velocity distribution with CFD not only in the inlet section but also in as many sections as the HX core is divided in the scaling method. Then the flow distribution could be updated at each step of the calculation sequence defined in item 2.3.

5 References

[1] Liberto, M., Ciofalo, M., “A study of turbulent heat transfer in curved pipes by numerical simulation,” International Journal of Heat and Mass Transfer. 59 (2013) 112–125.

[2] Lu, T., Jiang, P.X., Guo, Z.J., Li. H., “Large-eddy simulations (LES) of temperature fluctuations in a mixing tee with/without a porous medium,” International Journal of Heat and Mass Transfer 53 (2010) 4458–4466.

[3] Zhang, L., “Flow maldistribution and thermal performance deterioration in a cross-flow air to air HX with plate-fin cores,” International Journal of Heat and Mass Transfer. 52 (2009) 4500–4509.

[4] Yaïci, W., Ghorab, M., Entchev, E. “3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar HXs,” International

Journal of Heat and Mass Transfer. 74 (2014) 490–500.

[5] Mao, J.N., Chen, H.X., Jia, H., Wang, Y.Z., Wu, H.M. “Effect of air-side flow maldistribution on thermal-hydraulic performance of the multi-louvered fin and tube HX,” International Journal of Thermal Sciences. 73 (2013) 46-57.

[6] Lalot, S., Florent, P., Lang, S.K., Bergles, A.E., “Flow maldistribution in HXs,” Applied Thermal Engineering. 19 (1999) 847-863

[7] Barth, T., and Jespersen, D., 1989, "The Design and Application of Upwind Schemes on Unstructured Meshes," 27th AIAA Aerospace Sciences Meeting, 1989, AIAA-89-0366.

[8] Patankar, S., and Spalding, D., 1972, "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three- Dimensional Parabolic Flows," International Journal of Heat and Mass Transfer, 15 pp. 1787-1806.

[9] Menter, F.R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications”, AIAA Journal, 32(8) (1994) 1598-1605.

[10] Incropera, FP and DeWitt, DP; Fundamentals of Heat and Mass Transfer; John Wiley & Sons; 3rd edition

6 Contact Author Email Address

celso.nakashima@embraer.com.br

Copyright Statement