

FIRST NUMERICAL RESULTS ON FORCED-CONVECTION HEAT TRANSFER INSIDE A WAVY CHANNEL

Riccardo Mereu, Emanuela Colombo, Fabio Inzoli, Alfonso Niro

Dipartimento di Energia, Politecnico di Milano

Campus Bovisa, via Lambruschini, 4 – I-20156 Milano, Italy

alfonso.niro@polimi.it

ABSTRACT

This paper presents the first results of a numerical investigation on fluid dynamics and heat transfer characteristics of a forced air-flow inside a rectangular channel with the lower and upper walls wavy configured at Reynolds numbers ranging between 500 and 10000. The duct cross-section is 120 mm wide and 12 mm height whereas waviness have in streamwise direction a triangular profile of 68-mm pitch and 161° apex angle; the channel is operated with the wavy walls at fixed temperature and the side ones adiabatic. The numerical analysis is performed by means of a finite volume, commercial CFD code considering a three-dimensional fluid domain over a single module with periodic conditions; analysis over 6 modules and a 400-mm length, adiabatic inlet-duct are also performed. Results are compared with the data obtained by an experimental facility with the same geometry and operating conditions as the numerical model.

Key Words: *Heat Transfer, wavy channel, experimental approach, computational fluid dynamic, forced convection.*

1. INTRODUCTION

In designing compact heat exchangers, high values of heat transfer area per unit volume are looked for; however, increasing this parameter over a given value, thermal performances start worsening. Indeed, the higher the surface-to-volume ratio, the narrower the passages, and hence the air velocity has to be lowered to maintain acceptable pressure drops; in turn, narrow passages and low air velocities induce a laminar or weakly turbulent flow that is characterized by a quite poor convective coefficient which eventually defeats completely the area increase benefits. To overcome this limit, designer enhances heat transfer using specially configured extended-surfaces, such as offset or louvered or wavy fins, which are an efficient and cost-effective solution. These structured extended-surfaces modify the fluid dynamics by various mechanisms, such as periodic interruption of the boundary layer growth or periodic streamline deflection, and in addition they promote turbulence development as their characteristic size is close to the turbulent microscales (the lower the Reynolds number, the larger the size of dissipative structures).

As the shape of these structured extended-surfaces depends on the enhancement mechanisms employed, there is a large variety of geometries and consequently a large number of studies in the literature (Webb [1] reports near 40 papers). More precisely, several correlations deal with the offset and louvered fins (Joshi and Webb [2], Wieting [3], Davenport [4]), whereas the wavy-fin geometry is suffering from a some lack of data as well as of relevant correlations yet. For this reason, the authors are currently involved in an investigation on fluid dynamics and heat transfer characteristics of forced convection inside wavy channels. This paper reports on the numerical results on the convective coefficients and pressure drops of an air-flow inside a wavy channel, which reproduces geometry and operating conditions of an experimental facility; comparisons between numerical and experimental results are also presented.

2. NUMERICAL MODELLING

The wavy channel here investigated has a rectangular cross section whereas the corrugations, schematically showed in Figure 1, have in streamwise direction a triangular cross-section, with the apex on one wall coinciding with the trough on the other wall; it is noteworthy that corrugation dimensions, listed in the table also reported in the figure, are a scale modelling of a typical geometry used in compact heat exchangers with this kind of extended surfaces.

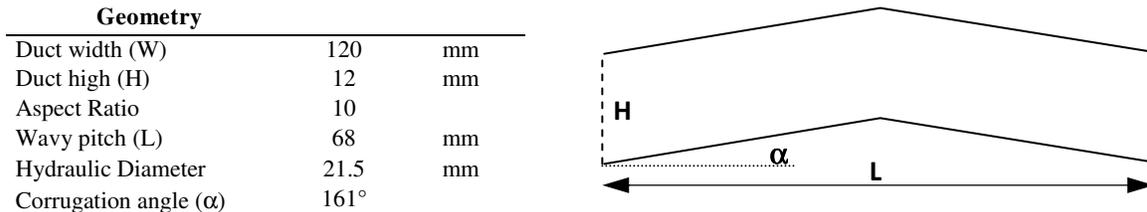


FIGURE 1. Geometrical characteristics of the wavy channel under analysis

The wavy channel experimentally investigated, which the present numerical analysis refers to, is 820 mm long, and hence there are 12 waviness, and it is operated with the wavy walls at fixed temperature while the side ones being adiabatic. Upstream the wavy channel, there is an entry-section, which consists of a 400-mm long rectangular channel with the same transverse dimensions as the test section but with flat and adiabatic walls, in order to attain developed velocity conditions at the test channel entry. Room air flows through the test section by means of a blower operating in the suction mode since it is mounted as the last stage of the facility. Finally, air flow rates have been varied in order to investigate Reynolds numbers, based on the duct hydraulic diameter, between 500 and 10000.

From the modelling point of view, a three-dimensional fluid domain over a single module is used and considered as periodic, referred to as MODEL1, in order to reduce grid size and calculation time. Anyhow, to check the relevancy of the assumption of periodic conditions without taking into account neither the actual velocity-profile development of the air flow entering the test section during the experiments nor the consequent effects on thermal development, a second modelling approach is used which considers the entry section and 6 full periodicities without periodic conditions (MODEL2).

In both cases, the mesh strategy is based on a structured-like approach, as it can be seen from Figure 2, and the map scheme is used because almost orthogonal and uniform elements help in reducing numerical diffusion far as discretization is concerned. Special attention is also given, by means of a grading on the wall, to the boundary layer in order to properly capture the velocity and thermal profiles at wall. The total number of elements is almost 397,000 for the laminar cases and the MODEL1. For the turbulent cases, a further adaption up to 600,000 elements in the wall region was done in order to avoid wall functions. For the MODEL2 the mesh is obtained by repeating the mesh of the single corrugation.

The models are implemented and solved in a finite volume, commercial CFD code, i.e., FLUENT vers. 6.3.26. Due to the flow characteristics and the Reynolds number investigated, the Pressure Based solver is used and the SIMPLE algorithm is applied in velocity-pressure coupling. A second order scheme is used for the spatial discretization. A steady and unsteady laminar flow model is used for Reynolds number up to 1000 and a comparison between different two-equations RANS models is studied for the other Reynolds while keeping the same mesh size. A sensitivity analysis is computed on the grid, and the Grid Convergence Index is evaluated. The asymptotic convergence criterion for the mesh used is confirmed, and a numerical error band not greater than 10% for the turbulent cases and 1% for the laminar one was found.

Convergence is checked from both the numerical and physical point of view by using residual criterion, mass unbalance and by monitoring meaningful quantities stability.

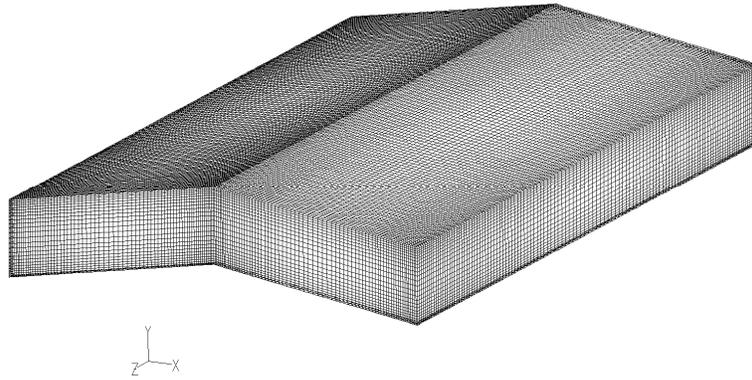


FIGURE 2. Model1: 3-D mesh

3. RESULT DISCUSSION AND CONCLUSIONS

Numerical results here reported are for Reynolds number ranging from 500 to 10000, and they are all grid independent; for $Re > 1000$ a RNG k- ϵ turbulence model is used. Figure 3 (left) shows the per-module-average values of the Nusselt number, based on the hydraulic diameter as well, plotted versus the Reynolds number; for comparison, the figure also reports the corresponding values experimentally found. It is quite evident the laminar-turbulent transition starts at Re slightly greater than 1000. In laminar flow, numerical evaluations seem to be a 10% lower than the experimental one, but we have to take in mind they are obtained with the MODEL1 and therefore they are the thermally fully-developed values whereas the experimental Nu are values averaged over the entire test section which thus benefit of the entry region. Indeed, the Nu value found at $Re=1000$ with the MODEL2, which accounts for the first 6 modules, is quite coincident with the experimental datum. In transitional flow-regime, instead, numerical evaluations tend to slightly overpredict experimental values, but differences are within 20% while experimental uncertainties being around 10%. Anyway as Re increases, numerical and experimental values are again in good agreement. Heat transfer enhancement with a respect to a flat rectangular channel ranges between 20% and 60%. In Figure 3 (right) the apparent Darcy-Weisbach friction factor is plotted versus the Reynolds number.

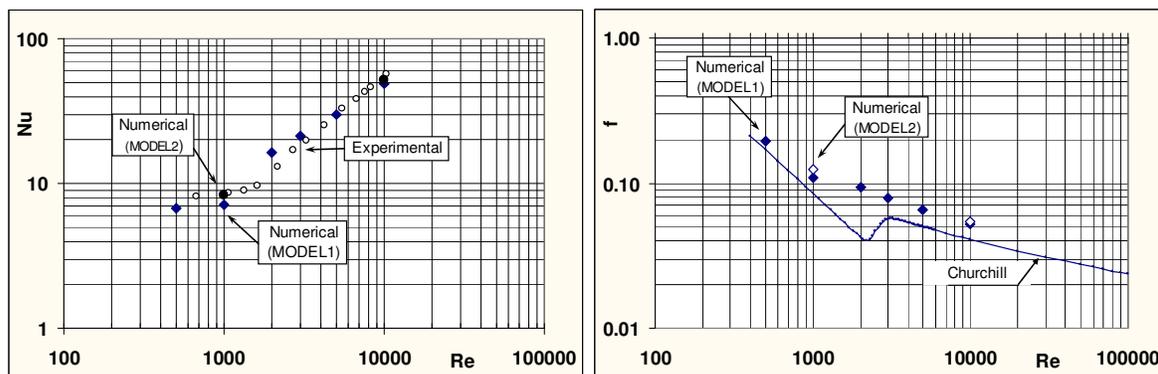


FIGURE 3. Per-module average Nusselt number (left) and Darcy-Weisbach friction factor (right) vs Reynolds number

Since at this time no experimental data are available yet, the figure reports as a light line the values calculated by means of the Churchill correlation adapted to a rectangular flat duct, with the same cross-section dimensions as the investigated channel, according to Shah and London. As it can be seen, there are no significant differences between the evaluations with the two models, and it is

confirmed that laminar-turbulent transition occurs for Re between 1000 and 2000. Penalization in pressure drops amounts to 30% on average.

Figures 4 and 5 report the velocity magnitude in the streamwise and spanwise directions, respectively, for Reynolds number of 1000 and 10000. Numerical results show the streamwise flow is deflected to the pressure-side wall and it separates on the peak corner while recirculation occurring inside the opposite corner, i.e., between the suction and pressure sides. The behaviour is more evident at low Reynolds numbers. Finally, it is noteworthy to highlight the wavy channel here investigated has some geometric characteristics, when compared to other wavy channels studied in the literature, that have to be taken into consideration. For instance, with respect to the channel considered by Hwang et al. [5] our channel has a wider cross section, a larger hydraulic diameter and, more important, a lower corrugation angle. This configuration, in some sense, is less critical than the Hwang's channel and it yields to a fluid dynamic behaviour more similar to the flat channel. This could explain why the Taylor-Görtler vortex pairs do not appear in neither the laminar and transitional cases. This consideration suggested to the authors a major parametric investigation with special attention to the corrugation angle which strongly effect fluid dynamic behaviour. The analysis is currently under processing.

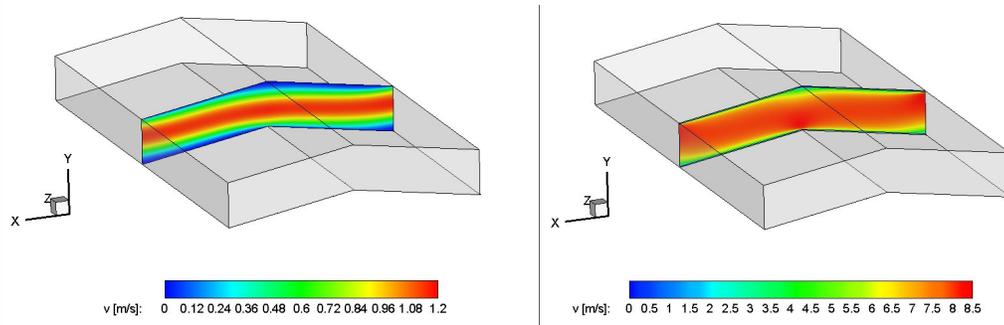


FIGURE 4. Velocity magnitude in the streamwise direction at Re 1000 (left) and 10000 (right)

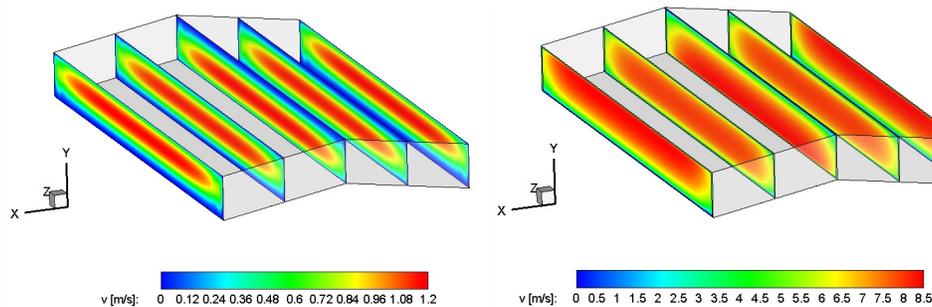


FIGURE 5. Velocity magnitude in the spanwise direction at Re 1000 (left) and 10000 (right)

ACKNOWLEDGMENTS

This work is supported by MURST (the Italian Ministry for the University and the Scientific and Technicaal Research) via 2007 PRIN grants.

REFERENCES

- [1] Webb, R.L., Principles of Enhanced Heat Transfer, John Wiley & Sons, cap. 9, pp. 228-284, 1994.
- [2] Joshi, H.M., Webb, R.L., Prediction of heat transfer and friction in the offset strip fin array, Int. J. Heat and Mass Transfer, Vol. 30, No. 1, pp. 69-84, 1987.
- [3] Wieting, A.R., Empirical correlations for heat transfer and flow friction characteristics of rectangular offset fin heat exchangers, J. Heat Transfer, Vol. 97, pp. 488-490, 1975.
- [4] Davenport, C.J., Correlations for heat transfer and flow friction characteristics of louvered fin, Heat transfer-Seattle 1983, AIChE Symp. Ser., No. 225, Vol. 79, pp. 19-27, 1984.
- [5] Hwang, S.D, Jang, I.H, Cho, H. H, Experimental study on flow and local heat/mass transfer characteristics inside corrugated duct, Int. J. Heat and fluid flow, 27 (2006) 21–32.