IL NUOVO CIMENTO Vol. 32 C, N. 2 Marzo-Aprile 2009
DOI 10.1393/ncc/i2009-10375-x

Colloquia: CSFI 2008

Fluid dynamic analysis of a Ranque-Hilsch vortex tube

A. Secchiaroli, R. Ricci, S. Montelpare and V. D'Alessandro

Dipartimento di Energetica, Università Politecnica delle Marche
Via Brecce Bianche 1, Ancona, Italy

(ricevuto il 15 Maggio 2009; pubblicato online il 6 Agosto 2009)

Summary. — Fluid dynamic analysis of a commercial, counter-flow Ranque-Hilsch Vortex Tube (RHVT), Exair 25 scfm, has been performed in this work both experimentally and numerically; in particular RHVT cooling power and temperature separation performances have been tested in both direct cooling employment (jet impingement) and indirect cooling employment (supplying cold plates). Experimental techniques, used in this work, revealed several difficulties to produce detailed information about velocity and temperature fields inside the tube and at both the exits. Hence numerical simulation of the flow inside the tube has been conducted using the commercial CFD code FLUENT 6.3.26. Compressible, turbulent, high swirling flow inside RHVT has been simulated by using both RANS and LES approaches. In particular several turbulence closures have been used in the RANS simulations and results have been compared with LES ones. Large Eddy Simulations have been performed by using Smagorinsky sub-grid model.

PACS `47.27.E-` – Turbulence simulation and modeling.
PACS `47.27.em` – Eddy-viscosity closures; Reynolds stress modeling.
PACS `47.27.ep` – Large-eddy simulations.

1. – Introduction

The RHVT is a simple device without moving parts that, fed with compressed gas, is able to split the fluid flow into two low-pressure streams. These streams, leaving the device from the two opposite sides of the tube (counter-flow), have temperatures higher and lower than the inlet gas, respectively. This effect, called Ranque-Hilsch effect (or thermal separation effect), is due to the fluid dynamic features of the internal flow only (compressibility, high swirl number, turbulence). Although the RHVT is a simple device with several industrial applications, the fluid dynamic effect that produces thermal separation is extremely complex and not completely understood. The aim of this work is to perform the fluid dynamic analysis on a commercial model of the RHVT in order to test its potential as back-up cooling device for electronic devices. Potential new applications of the RHVT are micro-electronic equipments (chipset, GPU, CPU, etc.) with low total, but high specific, heat flux and power-electronic equipments (RF transmitter, receiver, etc.) with high total and specific heat flux. In this research both direct (jet-impingement)

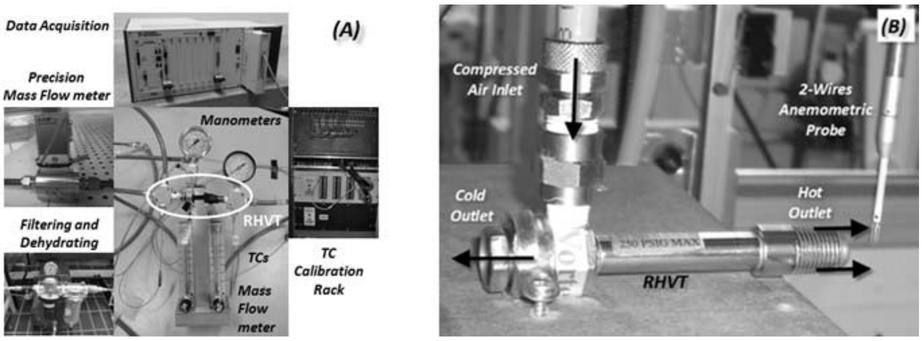

Fig. 1. – RHVT experimental analysis: (A) power performances test circuit; (B) outflow jets analysis.

and indirect (cold plate supplying) cooling employment are considered. As a consequence, performances of the RHVT, inserted in a closed circuit and free jet characteristics leaving the tube, have been both investigated. Finally numerical simulation has been used to obtain the requested details about the velocity and temperature fields inside the tube that experimental tests were unable to produce. Results showed a good qualitative agreement with previous literature ones [1-3].

2. – Experimental analysis

An experimental test-bed able to measure temperature, pressure and mass flow rate, at the inlet and at both the outlets, has been realized. By means of this apparatus fig. 1(A) it was possible to measure both the maximum cooling power and the thermal separation, varying the cold fraction, *i.e.* the ratio between mass flow rate leaving RHVT from cold exit and supplying mass flow rate. Performance curve measurement can provide useful information about using RHVT in a cold plate, anyway thermal separation values are influenced by the characteristics of the circuit in which the tube is inserted; the pipeline length and the presence of valves modifies the effective length of the RHVT and, as a consequence, its performances. Hence a measuring campaign has been conducted to investigate cooling performances of RHVT, in jet-impingement employment, *i.e.* evaluating velocity and temperature fields in the free jets leaving the device. Hot-wire two-components anemometric moving probe and a matrix of K-type thermocouples have been used to measure velocity and temperature fields in a plane normal to RHVT axis at both the exits; fig. 1(B). Experimental results, not reported in this paper, can be found in [4]. Anyway, by these techniques, it was impossible to obtain a suitable description of the flow field inside the tube. Moreover conventional techniques like inserting Pitot tubes or hot-wire anemometry probe are useless due the high blockage factor, and the high swirl velocity makes the use of optical techniques inappropriate (like LDA and PIV) that require seeding particles. In these conditions an accurate numerical analysis is the only way to obtain information about internal flow field in the whole domain and at both the outlets. Hence CFD analysis of the internal flow has been performed in this work.

3. – Numerical simulations

Numerical simulation of the flow inside a RHVT deals with the prediction of a high swirling turbulent compressible flow feature; in this research both RANS and LES

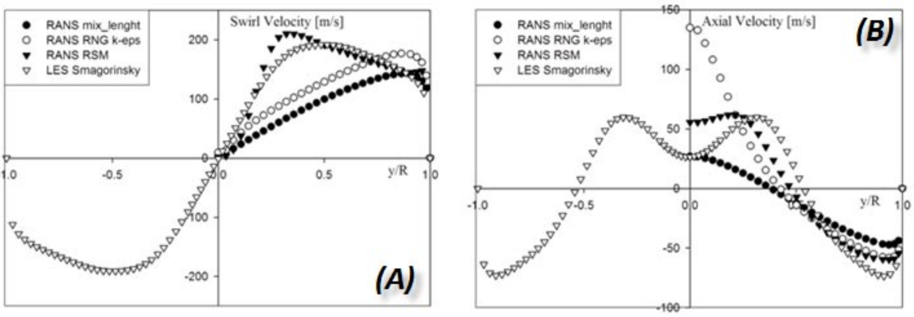

Fig. 2. – Swirl (A) and axial (B) velocity radial profiles in a central section of the tube.

approaches to turbulence simulation have been used. Flow government equations have been solved by means of the commercial CFD code FLUENT 6.3.26. In RANS simulations (steady axialsymmetric) several turbulence closure models have been tested: in particular, a linear RSM (Reynolds Stress differential Model) [5, 6], more useful to describe turbulent motion in swirling flows has been used successfully for the first time and results have been compared with those obtained by using RNG k-ϵ and mixing length models [5, 6]. In addition, LESs of an RHVT flow were for the first time performed, on a full three-dimensional computational model with a hot axial outflow, closer to real geometry of the vortex tube [7]. Smagorinsky model has been used as sub-grid model. Grid independence procedure, for all simulations, has been done by means of Richardson extrapolation technique, ensuring a negligible influence of grid spacing on the results. Grids used in this simulations are described in [7]. Boundary conditions have been set as close as possible to the experimental conditions: pressure, mass flow rate, velocity vector direction and total temperature have been imposed at the inlet; pressure has been fixed at both the exits; adiabathic and no-slip conditions have been used for external walls. Finally thermo-physical properties of the fluid have been expressed as a third-order polynomial function. More details can be found in [7]. Flow equations have been expressed by using Favre's filtering both in the time and in the space domain, eliminating density turbulent fluctuations from mean motion equations. Discretization schemes for convective terms used in RANS were a SOU scheme [6] for mass, momentum and energy conservation equations, and a QUICK scheme [6] for the other fluid dynamics variables. In LESs discretization of convective terms has been performed by SOU schemes [6] and an implicit second-order accurate scheme has been used for time integration. Simulations have been performed on a PC equipped with an Intel Core2 Quad processor at 2.4 GHz.

4. – Results and conclusions

Experimental investigations performed on the RHVT in normal operation conditions were able to determine heat transfer and temperature separation performances in both the employment of the tube as direct or indirect cooling system but seemed to be unable to produce detailed information about its internal fluid dynamic behavior, useful to improve the performance of this device for different applications. Computational results are shown in figs. 2 and 3 in which swirl, axial velocity and temperature radial profile are represented for a middle section of the tube. Results produced by different turbulence closures in RANS simulations are compared with LES time-averaged results. In RANS

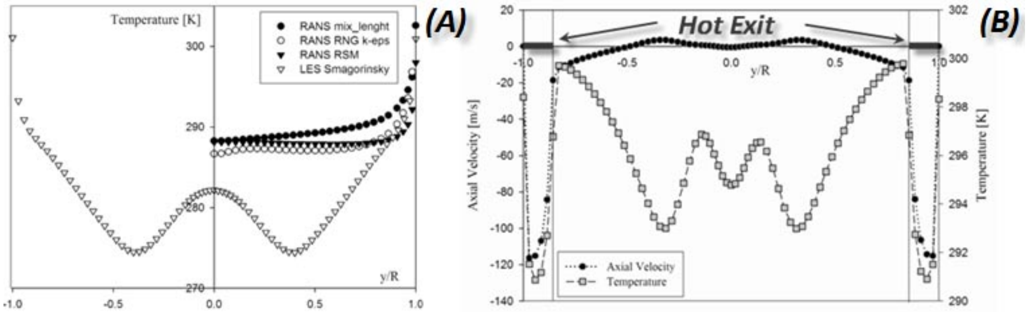

Fig. 3. – Temperature radial profile in a central section of the tube (A) and at the hot exit (obtained by LES) (B).

simulations, RSM results showed a better agreement with LES ones. In particular, using RNG k-ϵ an overestimation of axial velocity and consequently an underestimation of the swirl velocity occurs, as we can expect. Anyway RNG and RSM thermal separation predictions show little differences and both predictions are quite far from LES ones. LES simulations results have been compared with the literature ones showing a good agreement. Numerical diffusion has been considered and its contribution is probably negligible due to high convection velocity. An increase in LES time extent is at the moment in progress.

REFERENCES

[1] Skye H. M., Nellis G. F. and Klein S. A., *Int. J. Refrig.*, **29** (2006) 71.
[2] Eiamsa-ard S. and Promvonge P., *Renew. Sustain. Energy Rev.*, **12** (2008) 1822.
[3] Farouk T. and Farouk B., *Int. J. Heat Mass Transf.*, **50** (2007) 4724.
[4] Ricci R., Mazzieri M., Secchiaroli A. and Savelli E., *Raffreddamento di componenti elettronici di potenza mediante dispositivi ad effetto Ranque-Hilsch:caratterizzazione sperimentale e realizzazione di un prototipo*, in *Proceedings of the 26th UIT Conference, Palermo, 2008* (Edizioni ETS) 2008.
[5] Wilcox D. W., *Turbulence Modeling for CFD*, second edition (DCW Industries California, US) 1998.
[6] Fluent Inc., *FLUENT 6.3.26 User's Guide* (2006).
[7] Ricci R., D'Alessandro V., Secchiaroli A., Montelpare S. and Mazzieri M., *Raffreddamento di componenti elettronici di potenza mediante dispositivi ad effetto Ranque-Hilsch:simulazione numerica del flusso interno ed esterno*, in *Proceedings of the 26th UIT Conference, Palermo, 2008* (Edizioni ETS) 2008.