Numerical and experimental analysis on flow behavior and energy separation in a commercial Ranque-Hilsch vortex tube

Prof. Dr.-Ing. habil. Karl Bühler
Fakultät Maschinenbau
und Verfahrenstechnik (M+V)

Badstraße 24
77652 Offenburg
Tel.: 0781 205-268
E-Mail: k.buehler@fh-offenburg.de

1948: Geboren in Achern
Lehre als Mechaniker
1971: Ing. (grad.) Ingenieurschule Offenburg
1975: Diplom Maschinenbau Universität Karlsruhe
1979: Promotion über Thermokonvektion
1985: Habilitation über Rotierende Strömungen,
Privatdozent Universität Karlsruhe
1986: Professor Universität Karlsruhe
Seit 1991: Professor an der Hochschule Offenburg
Seit 1994: Mitglied des Instituts für Angewandte Forschung (IAF)
der Hochschule Offenburg
1999: Forschungsaufenthalt an der University of Colorado
at Boulder, USA
Zahlreiche Veröffentlichungen auf dem Gebiet der Thermo- und Fluiddynamik, darunter Strömungsmechanik (Springer 1991), Grundzüge der Strömungslehre (Teubner 2008), Hütte-Grundlagen der Ingenieurwissenschaften (Kap. Strömungsmechanik)

Forschungsgebiete: Strukturbildung in der Thermo- und Fluiddynamik,
Anwendungen der Computeralgebra, Nummerische Strömungssimulation CFD

3.3 Numerical and experimental analysis on flow behavior and energy separation in a commercial Ranque-Hilsch vortex tube

Prof. Dr.-Ing. habil Karl Bühler
Prof. Dr.-Ing. Ulrich Hochberg
Dipl.-Ing. (FH) Boris Kubrak

Abstract

This study focuses on the experimental and numerical investigations on a commercial Ranque-Hilsch vortex tube. Ranque-Hilsch vortex tubes have many applications in industry and production as they can generate a very cold flow just from pressurized air .e.g. machine tool cooling. Main objective of this study is the energy separation in the flow field which results in a temperature drop on the cold exit of the tube. This was investigated experimentally by measuring the outlet temperature on the cold exit and the pressure drop on the flow restrictor valve on the hot exit. At a pressure drop of 0.5 bar the vortex tube showed the best performance by reaching a cold exit temperature of –16.7 °C. The Inlet flow was pressurised air at 20 °C and 6 bar.

The numerical analysis was carried out by full 3D steady state CFD-simulation using the commercial software ANSYS CFX 11.0. The three dimensional model represented a 120° sector of the tube using periodic boundary conditions. A comparison between different turbulence models ($k - \varepsilon$, RNG $k - \varepsilon$, $k - \omega$, SST) was carried out. The classic $k - \varepsilon$ two layer turbulence model showed the best results compared to the experiment. The energy separation and the drop in cold exit temperature are highest when the viscous work term is included into the energy equation. These effects of including the viscous work term into the energy separation have also been investigated.

Introduction

The flow rotating around any axis is called a vortex. Ranque-Hilsch vortex tubes separate a compressed fluid into two streams, one hot and the other one cold. The vortex motion is created by tangential injection of the compressed fluid into a scroll chamber, creating a strong circular flow field. In a counterflow vortex tube like the one investigated here a fraction of the feed gas exits as cold through the central zone at one end of the tube. The balance fraction of flow exits as hot peripheral stream at the other end of the tube. This effect was first observed by Ranque [1] in 1933 and later investigated by Hilsch [2] in 1947.

In the meantime many theories postulated to explain the temperature separation [3, 4].

Several numerical investigations by using commercial CFD Codes have been performed more recently in order to understand the flow behaviour and the energy separation mechanism [5-7].

Experimental Analysis

The experimental setup is shown in Figure 3.3-1. On the left side is the hot exit with the throttle valve, on the right side the cold exit. By adjusting the throttle valve the massflow fraction can be controlled. The cold exit temperature is highly sensitive to the throttle valve setting on the hot exit. In order to set a comprehensive boundary condition for the CFD simulation the pressure before the throttle valve was measured. On the right is the cold exit where the exit temperature was measured by a Pt100 resistance thermometer.

Fig. 3.3-1: *The vortex tube installed in the laboratory*

The inlet was connected to pressurized air supply at a constant relative pressure of 6 bar.

At a relative back pressure of 0.5 bar before the throttle valve the temperature separation was highest seeing the cold exit temperature dropping down to -16.7 °C.

CFD-Simulation

A three-dimensional numerical model of the Ranque-Hilsch Vortex tube was delveloped using the commercial CFD Software ANSYS CFX 11.0. The solving was performed in parallel computing on two CPUs. The fluid model represented compressible air as ideal gas. Specific heat, thermal conductivity and dynamic viscosity were considered to be constant. In order to reduce computational effort the fluid domain was designed as a 120° sector of the tube as it has three tangential injectors into the scroll chamber as can be seen in Figure 3.3-3. The boundary conditions where set to a constant inlet massflow and pressure boundary conditions on both the cold and hot exit. The throttle valve on the hot exit was not modelled but represented by a pressure boundary condition set to 0.5 bar static relative as measured experimentally. The Cold exit pressure boundary condition was set to 0 bar static relative pressure. The reference pressure was 1 bar. To estimate the fluid temperature the equation of total energy was solved for every run once with viscous work included and once without considering the viscous work. A tetrahedral mesh with prism boundary layer mesh and refinement near the walls consisting of 5 million elements was used for the analysis. In total 4 different turbulence models models ($k - \varepsilon$, RNG $k - \varepsilon$, $k - \omega$, SST) were tested.

Results

Figure 3.3-2 shows the results from the CFD Simulation compared to the experimental measurement. The cold exit temperature is significantly lower when the viscous work term is included into the energy equation. For a better understanding of the energy separation mechanism a streamline that exits on the cold exit was created (Figure 3.3-3). The effect of radial energy transfer from the inner region to the outer region can be seen from Figure 3.3-4 which shows the temperature along the streamline. After the air enters the scroll chamber it is traveling in the peripheral area towards the hot exit on the right. The streamline reaches a stagnation point where the flow approaches the inner zone and changes direction to exit through the centre towards the cold exit on the left. When the gas is traveling in the peripheral zone towards the stagnation point it heats up. The viscous work due to the strong shear stress in the near wall region contributes to this heat up. When the fluid travels back to the cold exit in the centre zone of the vortex it is hotter than the outer regions. This counterflow results in an radial energy transfer from the inner region to the outer region. Figure 3.3-4 also shows an slight increase of the fluid temperature when the air expands in the diffuser on the cold exit.

A mesh sensitivity study showed better results with the finer mesh in place. Inital simulations with a much coarser mesh didn't see such low temperature on the cold exit. Remarkable is also the fact, that the simulation became highly unstable and aborted unless an initial condition in form of a constant rotation of the fluid along the main axis was introduced. The simulation has seen a significant effect of the viscous work on the cold exit temperature. The classic $k - \varepsilon$ model showed the best result in good agreement with the experimental analysis.

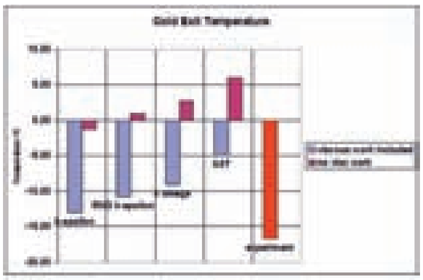

Fig. 3.3-2: *Results for cold exit temperature. The cold exit temperature is significantly lower when the viscous work term is included into the energy equation*

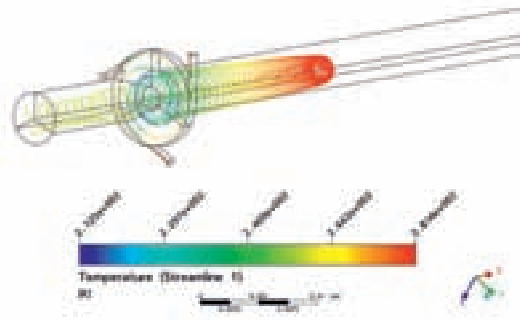

Fig. 3.3-3: *Streamline from inlet to cold exit*

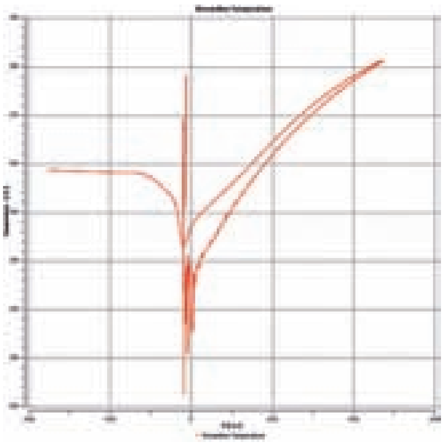

Fig. 3.3-4: *Temperature along Streamline. The fluid heats up when reaching the stagnation point on the right. On the way back through the fluid in the centre has a higher temperature than in the outer region causing a radial energy transfer*

References

[1] G.J. Ranque, Experiences sur la detente giratoire avec productions simultanes d'unechappement d'air chaud et d'un echappement d'air froid, J.Phys. Radium 4 (7), (1933) 112-144

[2] R. Hilsch, The use of the expansion of gases in a centrifugal field as cooling process, Rev. Sci. Instrum. 18 (2) (1947) 108-113

[3] A.F. Gustol, The ranque Effect, Phys.-Usp. 40 (6) (1997) 638-658.

[4] B. Ahlorn, S. Groove, Secondary flow in vortex tube, Fluid Dyn. Res. 21 (1997) 73-86

[5] W. Froehlingsdorf, H. Unger, Numerical investigations of compressible flow and energy separation in Ranque-Hilsch vortex tube, Int. J. Mass Transfer 50 (2007) 821-832

[6] H.M. Skye, G.F. Nellis, S.A. Klein, Comparison of CFD analysis to empirical data in commercial vortex tube, Int. J. Refrig. 29 (2006) 71-80

[7] U. Behera, P.J. Paul, Numerical investigation on flow behaviour and energy separation in Ranque-Hilsch vortex tube, Int. J. of Heat and Mass Transfer (2008)