International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:7, No:7, 2013

Sidecooler Flow Field Investigation

L. Manoch, J. Matěcha, and J. Novotný

Abstract**—**One of the aims of the paper is to make a comparison of experimental results with numerical simulation for a side cooler. Specifically, it was the amount of air to be delivered by the side cooler with fans running at 100%. This integral value was measured and evaluated within the plane parallel to the front side of the side cooler at a distance of 20mm from the front side. The flow field extending from the side cooler to the space was also evaluated. Another objective was to address the contribution of evaluated values to the increase of data center energy consumption.

Keywords**—**CFD, Sidecooler, Stereo PIV.

I. Introduction

NOWADAYS, the energy consumption in data centers has been a highly debated topic. One data rack can be equipped with device with performance up to several tens of kilowatts. This performance, when under load, has to be cooled efficiently, with the lowest costs. Index PUE (Power Usage Effectiveness) is one of the indicators of cooling effectiveness. It is the ratio of total energy consumption to the IT equipment consumption. This index, however, tells nothing about the manner and type of cooling. It is easy to influence the index, e.g. if we place UPS to servers, this part of consumed energy moves to the denominator, thus lowering the coefficient of PUE.

$$PUE = \frac{\text{Total facility power}}{\text{IT equipment power}} \quad (1)$$

Nowadays, hot and cold aisles are one of the standards used for cooling data centers. In this compilation, the areas of suction and blowing from servers / data racks are completely separated. Cold air, without mixing with hot air, is supplied to the data rack inlet. This assumption can be used only in case the basic conditions are met (e.g. blanking off the space between servers, tightness of cold aisle, etc.). This compilation allows for different solutions of cooling, one of them being cooling with the help of a side cooler.

Side coolers are generally placed between individual data racks and their number is derived from the power spectrum of individual fitted data racks.

L. Manoch is with the CTU in Prague, Faculty of Mechanical Engineering,Czech Republic (corresponding author to provide phone: 00420224352800; e-mail: lukas.manoch@ fs.cvut.cz).

J. Matěcha and J. Novotnýis are with the CTU in Prague, Faculty of Mechanical Engineering, Czech Republic (e-mail: jan.matecha@fs.cvut.cz, jan.novotny@fs.cvut.cz).

II. Sidecooler

As a starting model for both experimental measurement and numerical simulation, a real side cooler produced by Conteg Company was used. Specifically, the model AC-SO-CWx42-30x120 was used.

This model has a maximum cooling performance of 15 kW and is fitted with 5 radial fans. The use of radial fans is justified by large pressure drop of the heat exchanger through which the air is sucked from the hot aisle and cooled to the desired temperature afterwards in the area of the heat exchanger.

Using radial fans is one of the reasons of flow field investigation at the side cooler outlet. For this type of cooler, it would be logically preferable to use axial fans in order to minimize energy losses connected with degradation of the flow field inside the side cooler.

In order to achieve desired flow rate, velocity increases at the side cooler outlet. If the aisle cooling is used, a relatively big swirl of cold air can be assumed inside the cold aisle and in such circumstances the servers suction can be negatively affected.

If the non-isle cooling is used, another negative aspect is brought to the already unsuitable environment. High speed at the side cooler outlet can lead to separation and isolation of the data racks.

III. Experiment

The goal of the experimental measurement was to measure and evaluate the flow field at the side cooler outlet, i.e. at the inlet to the cold aisle area.

The measurement itself was carried out outside the aisle and the heat transfer and the exchange within the whole side cooler were not considered.

The measurement plane was at a distance of 30mm from the front part of the side cooler and was parallel to the front part. The reason for such distance was the ability to record uniform flow field without possible defects from the elements which form a part of the side cooler front parts.

The flow field was mapped with the help of the Stereo PIV method [1], [2].

The resulting velocity field was measured in nine positions for four flow rates (the fans performance on the control panel was set to 40, 60, 80 and 100%) and created after evaluation. The flow field was measured and evaluated in each position in the area of approximately 200mm x 200mm. Individual velocity fields were partially overlapped. For illumination of the measurement plane, a laser (BSL by QUANTEL Company) with performance of 220mJ in pulse and a frequency of 15 Hz was used for Stereo PIV method. Fog generator produced by SAFEX Company was used for generating the seeding particles. Two cameras with 2048x2048, 12-bit resolution were used for recording the image. The cameras were placed at an angle of 45° to the measurement plane and were located on the supporting structure allowing for their vertical and horizontal shift. Each camera recorded 100 double images, which enabled calculation of the instant 2D velocity fields with the help of adaptive correlation. These fields created the 3D velocity fields using the calibration method. The mean value was evaluated for each position. The process of creation to one velocity field for each mode was carried out in MATLAB.

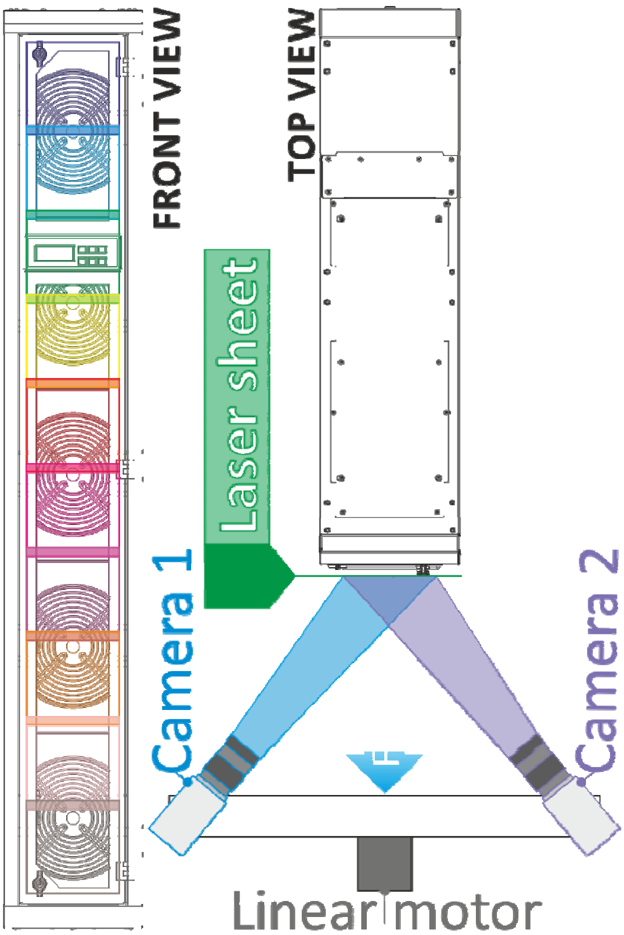

Fig. 1 Experimental measurement setup

IV. Numerical Simulation

A. Geometry

A simplified geometry of the side cooler with main dimensions based on the actual cooler was created to provide for numerical simulation. In order to simplify, all fastening material and additional inner area was removed with little or no influence on the flow field. Only the outlet part from the side cooler was considered. The beginning of the area was chosen in a place where the air flow exits the heat exchanger that was not simulated and enters the suction part of the radial fans.

A real model geometry, appropriately simplified, was again used for the radial fans geometry. The fans motor area was removed and replaced by a cylindrical area. Similarly, the fan's safety cap was removed and only the main structure was left and fitted in the cross-sectional direction in terms of vector exiting from the fan's blades.

Within the whole side cooler and fans geometry, the wall was not considered with zero thickness. All the considered elements had a minimum thickness of 2mm.

Another geometric element – plane, which enabled mapping the velocity field during the experimental measurement - was added to the model. This plane was created because of easier data exporting and clearly defined areas for evaluation. This plane represented the only geometric element with zero thickness, but given the nature of boundary conditions applicable to this element, the numerical error was not considered in the solution.

6 separate volumes were created in the resultant geometry. This number was based on MRF (Multiple Reference Frame) hypothesis where 5 volumes were defined by each fan and represented rotating areas, the remaining volume defined the stationary calculation area.

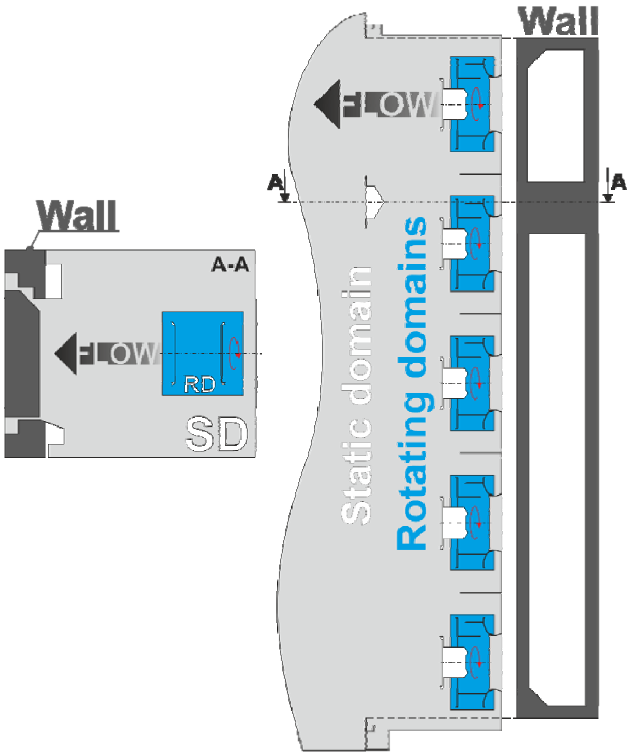

Fig. 2 Numerical simulation domains (gray – stationary domain, blue – rotating domains, dark gray – walls)

B. Mesh

The mesh was created in the preprocessor ANSA v14.1.1. Within the mesh creation, the area of boundary layer was not considered on any geometry. The mesh was smoothed at proper places. The reason for neglecting this important element of the mesh was the assessment of the resultant integral value outside the geometric area of the side cooler and its parts. Evaluation of any values was not considered within the geometric elements of the side cooler and given this assumption a whole mesh was created. This attitude concerns also the element size outside the inner area of the side cooler. The plane at a distance of 30mm from the front part of the side cooler is the most important area for this case. As mentioned above, the nature of the flow field exiting from the side cooler is of great importance. The mesh was proportionally adjusted due to the nature of the investigation.

The resulting two-dimensional mesh was created solely by three-wall elements. The bulk mesh was then created by four-wall elements in the total amount of approximately 6.2 million of elements.

C. Solver

Solver ANSYS CFX v14 was used for numerical simulation.

Boundary conditions for the entire volume were defined where the GGI (General Grid Interface) boundary condition with defined shift given by rotation of particular areas at the interface of each successive volumes was defined.

For regions representing radial fans, a rotating mesh was defined with constant speed of 2450 rev/min. All rotating areas were solved with the help of MRF method.

An under pressure related to the reference value of pressure was defined at the inlet to the computational area. The reference value of pressure was set at 1atm. The under pressure corresponded with pressure drop of the heat exchanger. The value was determined experimentally and gained a constant value of 180 Pa. This method was chosen because the heat exchanger was not contained in the numerical simulation. The heat exchanger however affected the area of suction in the radial fans.

A boundary condition with zero velocity at the walls was chosen for geometry representing the walls of side cooler and fans.

For remaining areas representing general boundaries to the open space, a boundary condition of a pressure inlet/outlet with zero pressure related to the reference value of pressure was chosen [3].

The whole numerical solution was divided into two separate phases. In the first phase, the whole problem was solved with the help of stationary solver for given boundary and initial conditions. In the second phase, the stationary solution was used as an initial condition for non-stationary solution [4].

The whole solution was considered to be isothermal with constant temperature of 25°C.

For advection a transient discretization of second order accuracy was used.

RANS turbulent model RNG k-epsilon was used for the solution [5].

TABLE I
COMPARISON OF FLOW RATE FOR EXPERIMENTAL AND CFD DATA AT SPECIFIED PLANE

Method	Flow rate $\dot{V}$ [m^3/s]	Difference [%]
Experiment	0.7054	Reference value
CFD	1.004	29.74

V. EVALUATION

Flow rate comparison at specified plane was used for results evaluation. For results from numerical simulation area weighted average calculation of plane normal velocity (2) was used because the grid elements had not same area.

$$\dot{V}_{CFD} = \int_A v_v \cdot d_A . \tag{2}$$

Flow rate calculation from experimental measurement was performed on equally spaced grid.

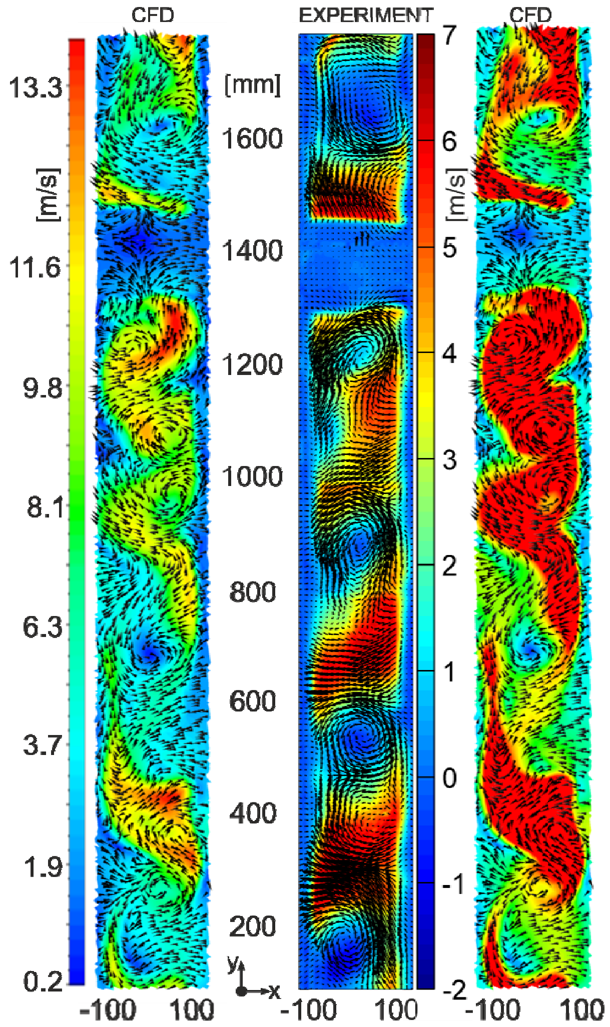

Fig. 3 Velocity filed comparison (first from the left – CFD with CFD scale, second and third – experimental and CFD with experimental measurement scale)

Velocity fields shown above (Fig. 3) represent 100% of sidecooler fan power. For experimental measurement it was not possible control fan revolutions directly. In case of numerical simulation the fan revolutions were directly specified.

VI. Conclusion

For chosen conditions the difference between experimental measurement and numerical simulation was about 30%. Results indicate that turbulent model and hypothesis for rotating domain were not selected proper for this case.

Acknowledgment

Project (TA01010184 / Research and development solutions data racks, cooling and transport systems for data centers) is / was solved with the financial support of TA ČR.

References

[1] J. Novotný, “New algorithms for processing data obtained by PIV measurements”, Thesis, CTU in Prague, 2009.
[2] J. Novotny, L. Manoch, “The criterion of choosing the proper seeding particle”, Journal of Mechanics Engineering and Automation, Volume 2, 2012.
[3] L. Manoch, J. Nožička, P. Pohan, “Investigation of flow in data rack”, EPJ Web of Conferences, Volume 25, 2012.
[4] L. Manoch, J. Matěcha, P. Pohan, “Flow field mapping in data rack model”, EPJ Web of Conferences, Volume 45, 2013.
[5] L. Manoch, J. Novotny, L. Nováková, “Investigation of flow field in data rack”, 22nd International Symposium on Transport Phenomena, 2011.