10TH INTERNATIONAL SYMPOSIUM ON PARTICLE IMAGE VELOCIMETRY – PIV13
Delft, The Netherlands, July 1-3, 2013

DPIV experimental study of mixed convection in an open cavity

G. Abdelmassih, S. Varela, A. Vernet and J. Pallares

ECOMMFIT, Department of Mechanical Engineering, Universitat Rovira i Virgili, Tarragona, Spain
gorg.abdelmassih@urv.cat

ABSTRACT

Particle Image Velocimetry (PIV) has been used to analyze the flow in an open cavity located on the bottom of a horizontal channel. Two different configurations have been considered. The first configuration corresponds to the isothermal flow and the second has the bottom wall of the cavity heated while the rest of the walls of the cavity and the channel are adiabatic. The local field correlation method (*LFC*) has been implemented on triple image correlation to obtain the mean displacement of the ensemble particles image series hence the triple image correlation algorithm enhances the peak correlation in moving parts that appear in the image. Boundary treatment technique has been applied using the weight function to obtain a good resolution of the velocity fields and achieve an accuracy level equivalent to inner flow location accuracy. A validation algorithm has been implemented for the post-processing in the image to detect and remove the spurious vectors. In order to remove or to reduce undesired light reflections and to homogenize the median illumination clean-up mask approach has been used to tackle the samples of the particle images as a pre-process step to get accurate velocity vectors. The range of Reynolds number was $1000<Re<1500$ for both configurations and that of the Richardson number was $0.01<Ri<10$. The experimentally measured flow structures were analyzed and compared with numerical results.

INTRODUCTION

The flow structure over an open cavity has been a point of interest from plain design research centers because the existence of this type of geometry in many engineering applications [1-6]. Open cavities can be found in many parts in the fuselage of airplanes. For example, one of the most important parts having this kind of shape is the landing gear wells. In solar energy research it has been found that solar cells used for water heating improve their absorption efficiency if wind barriers are installed in the perimeter of the solar cells. As the flat solar cell with wind barriers have the same geometry as an open cavity in medium scales, the research point extended to find the optimum aspect ratio which the flow inside the cavity could be capsulated.

Cooling of electronic components was one of the most vastly applications in small scales for the open cavity. It is simple in design and have a cheap maintenance cost. The electronic component considered as the source of heat and the natural or mixed convection effecting on the flow structure was the focusing point and it was sorted under three main categories depending on the position of the heated wall, e.g. the electronic board, within the enclosure. These categories are assistant flow, opposite flow and heated from below regarding to the wall is exhibited to heat which could be the vertical wall in the stream wise direction, the other the vertical wall or the floor of the cavity respectively. However, there are more and more applications for this type of geometry in different scales can be found in harbor entrance, bomb bay, muffler, etc.

Beside all these applications there is a main reason to make this geometry a point of interest as this geometry is often used to validate the computational model and numerical codes. For this reason, the considerable devoted efforts to study the flow structure and the heat transfer which occurs in flow over open cavities was justified due to their importance in these engineering applications.

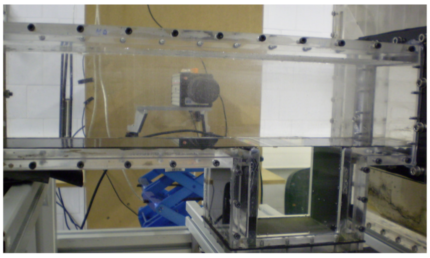
The early studies of flow past cavities were carried out in the 1960s [1-3]. More studies followed to reveal the flow structure and the heat transfer process occurring for both natural and mixed convection for geometries with different aspect ratios [4-7]. The effects of the position of a heated wall on mixed convection in a channel with an open cavity have been studied numerically [10]. The numerical approach considered in [8] is two dimensional with different aspect ratios. These authors found that the maximum temperature values decrease as the Reynolds number and Richardson number increase for all the configurations studied.

Manca et al [9] investigated experimentally the opposing mixed convection in an open cavity with a heated wall bounded by a horizontal unheated plate. The heated wall is on the opposite side of the forced inflow. The results are reported in terms of wall temperature profiles of the heated wall and flow visualization. The ranges of pertinent parameters used in this experiment are Reynolds numbers (Re) from 100 to 2000 and Richardson numbers (Ri) from 4.3 to 6400. Also, the ratio between the length and the height of cavity (L/D) ranges from 0.5–2.0, and the ratio between the channel and cavity height (H/D) is equal to 1.0. The results show that at the lowest investigated Reynolds number, the

surface temperatures are lower than the corresponding surface temperatures for Re = 2000 at the same heat flux. The flow visualization shows that for Re = 1000, there are two nearly distinct fluid motions: a parallel forced flow in the channel and a recirculation flow inside the cavity. For Re = 100, the effect of a stronger buoyancy determines a penetration of thermal plumes from the heated plate wall into the upper channel. Moreover, the flow visualization shows that for lower Reynolds numbers, the forced motion penetrates inside the cavity, and a vortex structure is adjacent to the unheated vertical plate. At higher Reynolds numbers, the vortex structure has a larger extension while L/D is held constant.
Three dimensional numerical studies of the flow and heat transfer characteristics for assisting and opposing incompressible laminar flow past an open cavity have been developed by Stiriba [10] and Stiriba et al. [11]. The results show that the flow exhibits a three-dimensional structure and is steady for Re=100 with Ri ranging from 0.01 to 10 and Re=1000 with Ri ranging from 0.001 to 1.The forced flow dominates the flow transport mechanism and a large recirculating zone from inside the enclosure which results in heat transfer by conduction. Abdelmassih et al [12] performed numerical simulation of incompressible laminar flow in a three-dimensional channel with a cubical open cavity with a bottom wall heated. They noted that the effects over the velocity and temperature distribution of the buoyancy forces acting perpendicular to the mainstream flow are studied for Reynolds numbers (Re) between 100 and 1500; Prandtl number (Pr) is set to 0.7 and Richardson number (Ri) between 10^{-3} to 10^{1}. Most of these studies were numerical for both two and three dimensional configurations with a varying aspect ratio. Few experimental studies can be found in the literature specially using PIV techniques because the maximum velocity inside the cavity is many times smaller than the flow velocity over the cavity [13] which is a measurement challenge.

EXPERIMENTAL SETUP

a)



b)

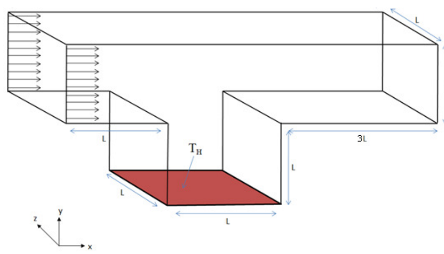


Figure 1 The central channel.

A model of an open cavity located in the bottom of a horizontal channel has been used (Figure 1). The model consists in a cubical cavity with a dimension of 100 mm. The total length of the channel is 500 mm while the length from the inflow tank to the cavity leading edge is 100mm. The material used in fabricate the walls of the central channel is poly-methyl-methacrylate (PMMA). This transparent material admits optical access for the PIV measurements.
The heated wall is made from copper metal. The copper is characterized by high thermal conductivity allowing us to provide a homogenous temperature distribution on the heated wall surface. The cooper plate has been thermally insulated to avoid effects of the surrounding temperature. The bottom wall of the channel and the cavity were painted black in order to reduce the reflections of the laser light sheet and improve the quality of the images at the edges. Six electric resistances (2500 Ohms) are used to heat the copper flat plate. Each resistance has a calibrated thermocouple (type “k”) to monitor the temperature. Thermal paste was used in order to increase the thermal conductivity of the thermal interface between the resistances and the cooper block. As the physical properties of water vary with temperature it has been necessary to implement a temperature control to maintain a constant water temperature during each experiment.
Figure 2, shows the flow cycle. The flow supplied by a main tank, with capacity about $1m^3$, filled with approximately 240 litters of water. A centrifugal pump with flow rates up to $2.5m^3/h$ pumps the water to the inflow tank trough a PVC pipe (25mmID). After the pump the fluid flows through a valve and a flow-meter to control the flow rate. Then the fluid is conducted the inflow tank simultaneously from the top and the bottom in order to avoid creation of eddies. The water in the inflow tank passes through seven mesh plastic screen 1mm×1mm, separated each other with 15 mm distance, spited in two groups separated by a honey camp 10mm×10mm with 30mm thickness. A nozzle is placed at the end of the inflow tank used to direct the water to the channel. Then the flow passes to the central channel. The outflow is collected in a drain tank in a level higher than the central channel and the inflow tank to avoid getting air bubbles into the flow. Finally the water passes from the drain tank to the principle by 3 tubes to close the flow cycle.

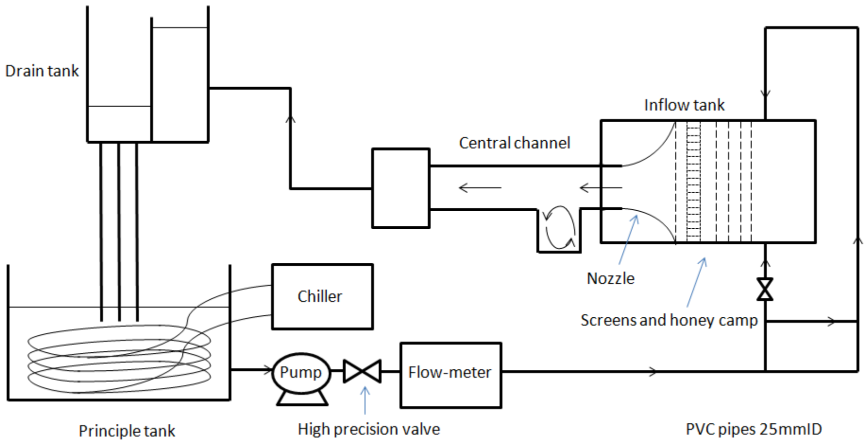


Figure 2 Sketch of the flow cycle

Nd: Yag semi-conductor laser, Monocrom *MP532-3W* were used in this work to illuminate the interested planes. This laser is linearly polarized with 532nm wavelength. It was used in continuous mode at 250 Hz. Optical system composed of 3 lenses is mounted on the beam in order to generate a light sheet of 120 mm in x-axis and 0.7 mm in y-axis.

MotionPro digital cameras, model HS-3, has been used in the PIV recording processes. The sensor of the camera presents a 1280×1024 pixels resolution with square pixels 12µm. A zoom lens Sigma, 28-300mm F3.5-6.3 DG Macro has been used to provide the ability to amplify the specific regions. Polyamide particles with a mean diameter of 50 µm have been used as seeding. Flow at Reynolds 1000 and 1500 have been analyzed without (configuration I) and without (configuration II) heat transfer from the bottom of the cavity whereas the applied Richardson numbers range was from 10^{-2} to 10^{1}.

RESULTS AND DISCUSSION

Different aspects have been considered in order to accomplish the best accuracy possible during the measurements. The maximum zoom of the camera (1024 x 1024 pixels) has been used and the image has focused on the whole cavity and 5% of the flow in the channel. On the other hand, the flow cycle has been running for more than 30 minutes before starting the image capture in order to overcome the fluctuations resultant from pump switching on as well as to avoid the existence of any bubbles in the flow cycle.

The PIV results have been compared with results obtained from numerical simulations. These numerical results have been obtained using the finite volume parallel code 3DINAMICS [12]. The code solves numerically the three-dimensional incompressible Navier-stokes on non-uniform staggered Cartesian meshes. The SMAC-method is used to join continuity and momentum equation, in which, the Poisson equation for the pressure is computed with the biconjugate gradient method (BiCGtab). Convective and the diffusive terms are approximated using SMART scheme and central differences respectively.

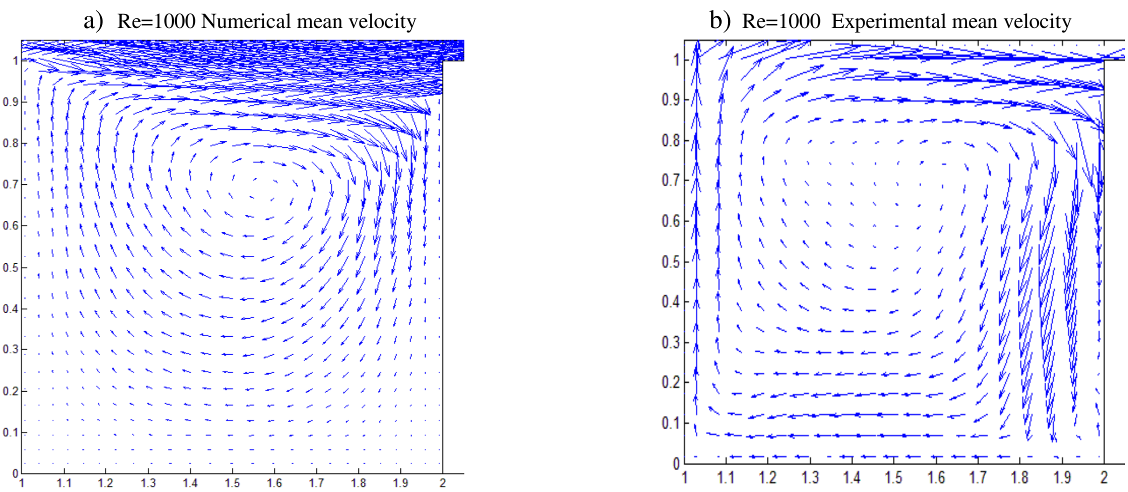


Figure 5 Velocity maps for the mean velocity for Re=1000, a) numerical simulation and b) experimental results

Figure 5, shows the results for Re=1000 for configuration *I*, i.e. isothermal flow. Figure 5.a corresponds to the numerical results while Fig. 5.b shows the measured vector field. It can be seen that there are significant differences mainly because under the experimental conditions the flow was not completely constant. This indicates that even small perturbations in the flow above the cavity produce important differences with respect to the numerical results in the flow inside the cavity.

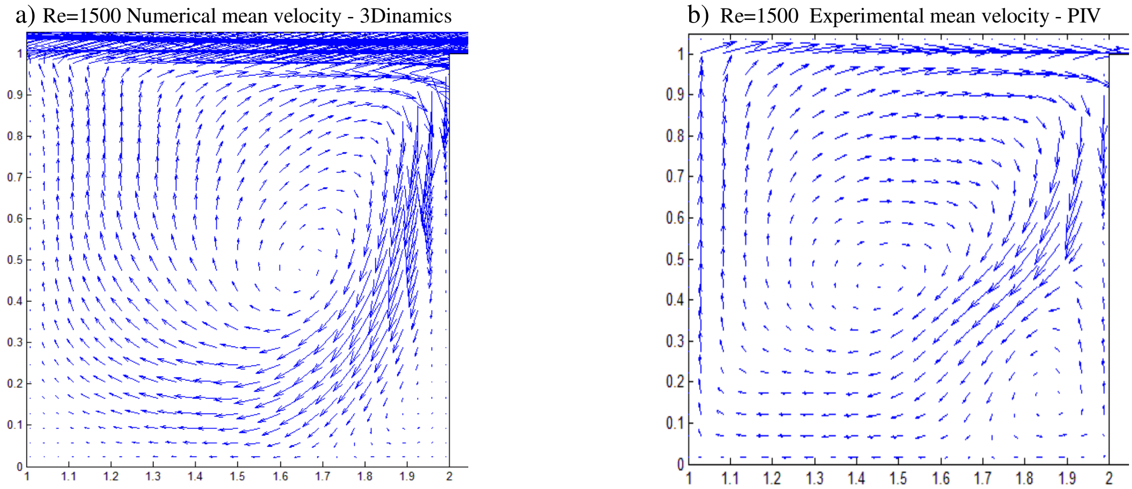


Figure 6 Velocity maps for the mean velocity for Re=1500, a) numerical simulation and b) experimental results

Figure 6 corresponding to Re=1500 (configuration *I*) show that the differences between simulation (Fig. 6.a) and measurements (Fig. 6.b) are reduced in comparison with results at Re=1000 because the reduction of the relative intensities of the perturbation as the flow rate is increased.
For configuration *II*, it have been found experimentally that for small Ri≤1 for each Re the flow is similar to the presented in configuration *I* and this due to the small temperature increment $\Delta T < 2$ºC. For this reason we focused on Ri=10 for each Re.

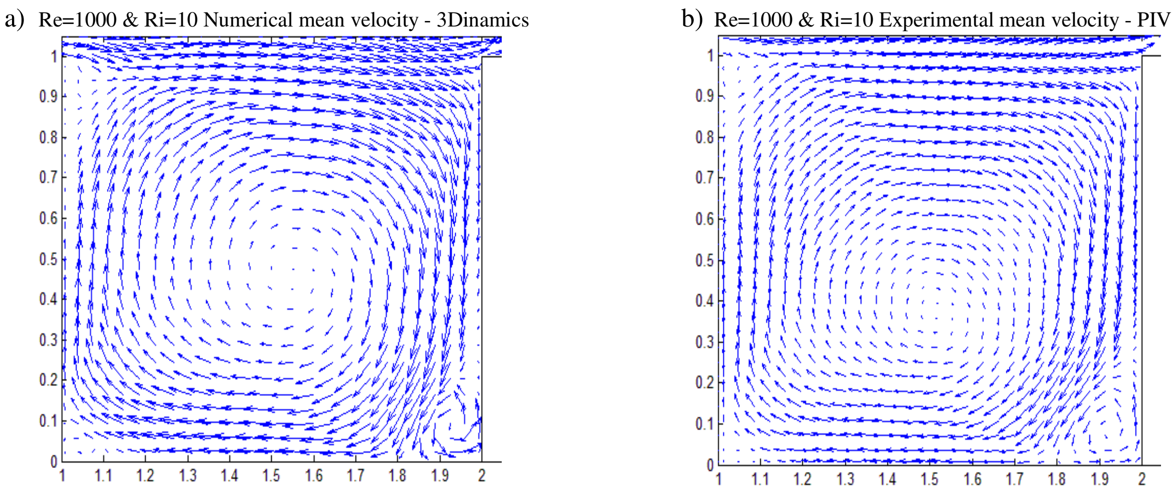


Figure 7 Velocity maps for the mean velocity for Re=1000 and Ri =10, a) numerical simulation and b) experimental results

Figures 7a and 7b, displays the numerical and experimental mean velocity maps of the flow for Re=1000 and Ri=10, respectively. It is obvious that there is a good agreement between the flow structures presenting numerically and experimentally except that the center of the recirculation cell presented by experimental results is 15% lower within the cavity.

Figures 8a and 8b, depicts the numerical and experimental mean velocity maps of the flow for Re=1500 and Ri=10, respectively. A good agreement between the flow structures presenting numerically and experimentally can be found. The main difference can be noted that the center of the recirculation cell presented by experimental results is shifted about 5% laterally.

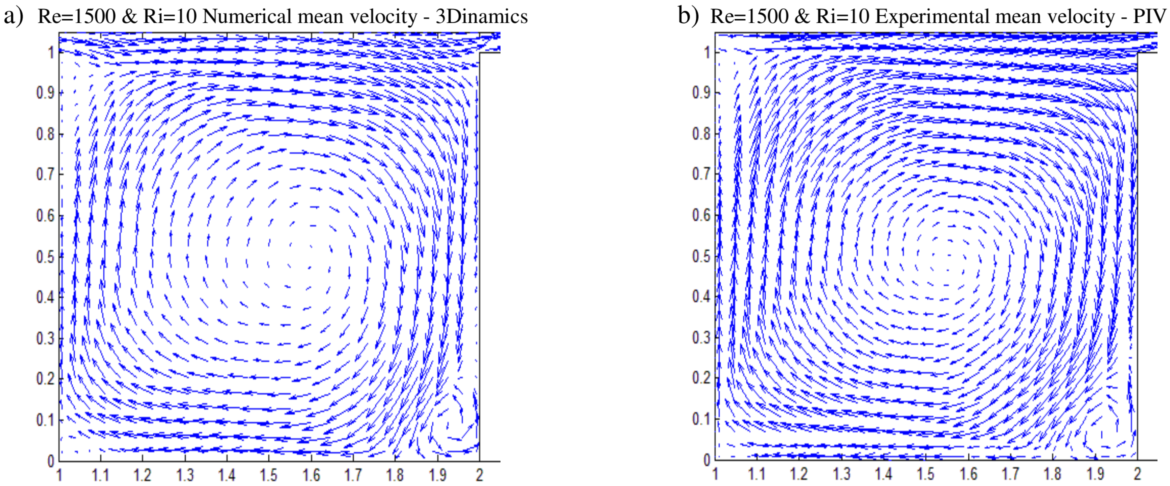


Figure 8 Velocity maps for the mean velocity for Re=1500 and Ri=10, a) numerical simulation and b) experimental results

CONCLUSIONS

Experimental study has been carried out for the flow structure over 3D cubical open cavity heated from below. The results shown that there is a good agreement between the experimental and numerical study except for Re=1000 of the configuration *I*. The effect of the buoyancy is weak for small Ri. Time average flow structure approach is used to study the unstable flow as both studies demonstrated that the flow becomes unstable and complicated for Re=1000 and 1500 with Ri $\geq$1.

ACKNOWLEDGMENTS

This study was financially supported the Spanish Ministry of Science and Technology and FEDER under project DPI2010-17212.

REFERENCES

[1] Burggraf O “Analytical and numerical studies of the structure of steady separated flows” J. Fluid Mech. 24 (1966) pp.113–151

[2] Weiss R and Florsheim B “Flow in a Cavity at Low Reynolds Number” Phys. Fluids 8 (1965) pp.1631–35

[3] Pan F and Acrivos A “Steady flows in rectangular cavities” J. Fluid Mech. 28 (1967) pp. 643–55

[4] Papanicolaou E and Jaluria Y “Mixed convection from an isolated heat source in a rectangular enclosure” Numer Heat Tr. A-Appl. 18 (1991) pp. 427–61

[5] Leong J Brown N and Lai F “Mixed convection from an open cavity in a horizontal channel” Int. Commun Heat Mass 32(2005) pp. 583–592

[6] Manca O Nardini S and Vafai K “Experimental investigation of mixed convection in a Channel With an Open Cavity” Exp. Heat Transfer 19 (2006) pp.53–68

[7] Papanicolaou E and Jaluria Y “Mixed convection from a localized heat source in a cavity with conducting walls: A numerical study” Numer. Heat Tr. A-Appl. 23 (1993) pp. 463–484

[8] Manca O Nardini S Khanafer K and Vafai K “Effect of heated wall position on mixed convection in a channel with open cavity” Numer. Heat Tr. A-Appl. 43 (2003) pp. 259-282

[9] Manca O Nardini S Vafai K, “Experimental investigation of opposing mixed convection in a channel with an open cavity below” Experimental Heat Transfer 21 (2008) 99–114

[10] Stiriba Y “Analysis of the flow and heat transfer characteristics for assisting incompressible laminar flow past an open cavity” Int. Commun. Heat Mass 35 (2008) 901–907

[11] Stiriba Y Grau F Ferré J and Vernet A “A numerical study of three-dimensional laminar mixed convection past an open cavity” Int. J. Heat Mass Tran. 53 (2010) pp.4797–4808.

[12] Abdelmassih G Vernet A and Pallares J “Numerical simulation of incompressible laminar flow in a three-dimensional channel with a cubical open cavity with a bottom wall heated” J. Phys: Conf Ser 395 012099 (2012)

[13] T. J. O’Hern, J. R. Torczynskl and T. Y. Chu, Shear-drive flow in a square cavity: A comparative study using PIV, LDV, and computational simulations, Symposium on laser anemometry: Advances and applications, ASME FED (1994).