STUDY OF FLOW OVER SHALLOW CAVITIES: AN EXPERIMENTAL ANALYSIS

Ana Cristina Avelar, anacristina@iae.cta.br
Instituto de Aeronáutica e Espaço (IAE/ALA), São José dos Campos, SP, Brazil, 1228-904

Nide G. C. R. Fico Jr, nide@ita.br
Instituto Tecnológico de Aeronáutica (ITA), São José dos Campos, SP, Brazil, 12228-900

Olympio Achilles de Faria Mello, oamello@iae.cta.br
Instituto de Aeronáutica e Espaço (IAE) , São José dos Campos, SP, Brazil, 1228-904

Abstract. *An experimental study was conducted to investigate the flow over a shallow cavity. The flow field was quantitatively studied using the two-dimensional Particle Image Velocimetry (PIV) technique, which provides two components of the velocity in a given plane. It was investigate the Reynolds number influence on the oncoming flow for a specified cavity aspect ratio. The experiments were carried out in a subsonic wind tunnel, in the Aeroespace Technical center (CTA) of the Institute of Aeronautics and Space (IAE), Brazil. It was found that at low Reynolds number the flow outside the cavity does not possess enough energy to create a bubble inside it. The present work is a continuation of previous numerical work done by the second author and collaborators, who have already investigated the influence of several parameters on the flow over a shallow cavity.*

Keywords*: Particle Image Velocimetry, Wind Tunnel, Shallow Cavity, Reynolds Number*

1. INTRODUCTION

The flow over cavities is of great interest as it is related to various engineering applications. Solar energy collectors, combustion cambers, and environmental problem are some examples. In the literature many works dealing with this matter are found, but most of them considering cavities with small aspect ratios. Aung (1983) found, experimentally, that for laminar forced convection, the local heat transfer distribution on the cavity floor has a maximum value located between the midpoint of the cavity floor and the downstream wall. Bhatti and Aung (1984) numerically simulated the 2-D, laminar flow over cavities and showed that the heat transfer inside the cavity is influenced strongly by its aspect ratio. Sinha et al. (1982), experimentally, showed that, for laminar flow, the number and shape of the bubbles inside the cavity is influenced by its aspect ratio. Metzger, Bunker and Chyu (1989) simulated numerically the turbulent flow in cavities and concluded that the flow pattern is strongly dependent on the aspect ratio and little influenced by variations in Reynolds number. Gomes (1998) has shown experimentally that the efficiency of heat absorption improves accordingly, if we introduce vertical fences along the perimeter of the collector. The present authors interest in such flows is due to their research related to the flow over flat plate solar energy collectors. Such devices loose energy to the ambient around them mainly by forced convection because of the wind blowing over its upper surface. The heat transfer film coefficient is directly related to the wind velocity impinging over the solar collector (Duffie and Beckman, 1980). The use of a wind barrier around the collector perimeter will create a re-circulation region over its surface, diminishing the mean wind velocity and consequently the heat loss. The resemblance of such flows to the ones over shallow cavities motivated the present work.

The present effort is a continuation of previous numerical work done by the second author and collaborators, who up to this point investigated the following aspects: (*i*) cavity aspect ratio and (*ii*) Reynolds number of the oncoming flow (*iii*) extent of the three-dimensional effects originating at the cavity's sides. These computational results so far are 2-D. However, it should be pointed out that the code is currently being expanded to handle 3-D flows. Thus, at the present stage of the investigation of shallow cavities, it has become important to start an experimental study to corroborate the numerical analysis. The first one (Avelar et al, 2007) investigated cavities of several aspect ratios, AR. All aspect ratios had at least a qualitative agreement with the previous numerical results, except for a particular one, AR=6. For this particular case the experimental results yielded a single bubble inside the cavity, while the experimental results obtained by the present authors showed the presence of two recirculating bubbles. The Particle Image Velocimetry technique, PIV has been used in the experimental investigation. PIV is a non-intrusive experimental technique, which allows the instantaneous 2-D or 3-D measurement of a planar flow field by imaging the reflected light from small particles in the flow that are illuminated by a laser sheet. The acquired images are divided into small interrogations regions (called interrogation windows), which are statistically analyzed to produce a correlation peak that represents the magnitude of the particle displacement within the interrogation region. Similar displacements that are estimated through all of the interrogation regions enables the development of a velocity vector map over the entire region of interest. A good description of this technique is given by Raffel et.al (1988).

In the present work just experimental results of the Reynolds number influence, for a specified aspect ratio value, namely, six, on the flow over the cavity will be presented. The experiments are being conducted in a subsonic wind tunnel, TA-3, in the Aeroespace Technical center (CTA) of the Institute of Aeronautics and Space (IAE), Brazil. The results seem to indicate that the flow outside the cavity must be "strong enough" to be able to create a bubble inside the cavity. For very low Reynolds numbers this is not possible. As the Reynolds number increases a well-formed single bubble was observed. In fact, this is in agreement with the CFD calculations but contrary to the previous experimental results by the authors. It must the remembered, however, that for the earlier experimental work the cavity roughness was much greater than for the present effort. This, in fact, might have an important role in the flow pattern. Certainly, this is one of the aspects to be considered in the next phase of the work.

2. DESCRIPTION OF THE EXPERIMENTS

The experiments were performed in a closed-circuit, open test section subsonic wind tunnel, TA-3. The outlet of its contraction, and the inlet of its diffuser are circular. A schematic of this wind tunnel is represented in Fig. 1. The wind tunnel test section is 0.65m diameters and 0.97m long. A 13 hp motor produces a maximum velocity of 40 m/s through the test section. The experimental apparatus is shown in Figs. 3 through 6.

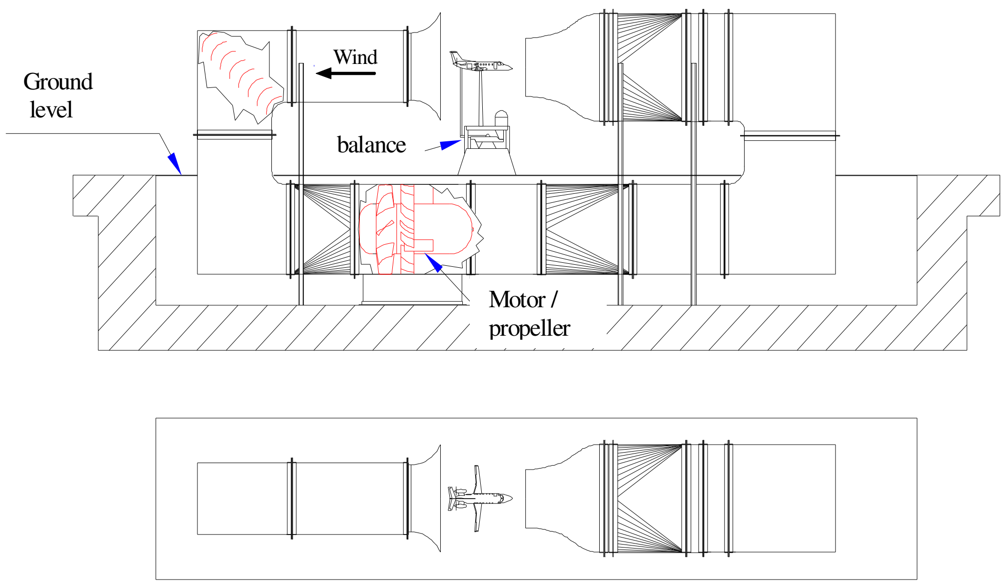

Figure 1. Schematic of the TA-3 Wind tunnel.

The cavity model, Fig. 4, was made of wood and painted in flat black to minimize light reflections. A resin was used to make the cavity model surface very smooth. The model dimensions are indicated in Fig. 6. The model was built in a way that allows the variation of the distance *w*. The model width is equal to the wind tunnel test section diameter, 0.65m. The PIV measurements were carried out for the aspect ratio, l_e/s, value equal to six, and for velocities ranging from 3.5m/s to 32 m/s, corresponding to Reynolds number based on the cavity depth, Re_s, ranging from 0.9×10^4 to 8×10^4.

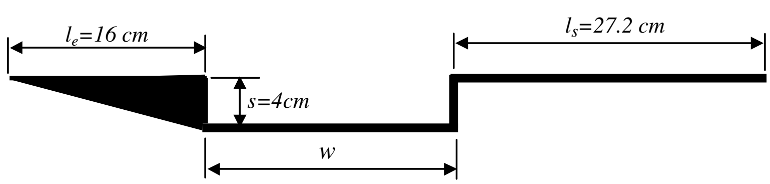

Figure 2. Cavity dimensions.

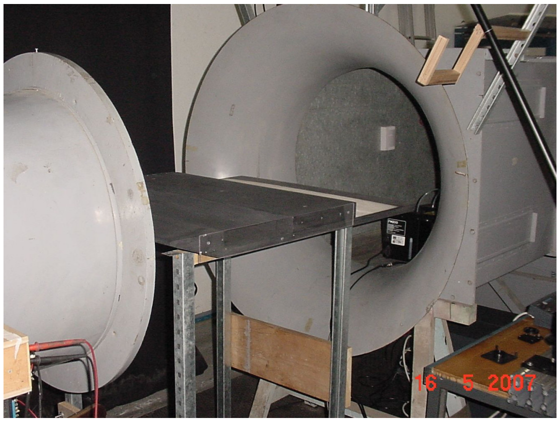

Figure 3. Cavity model in the wind tunnel.

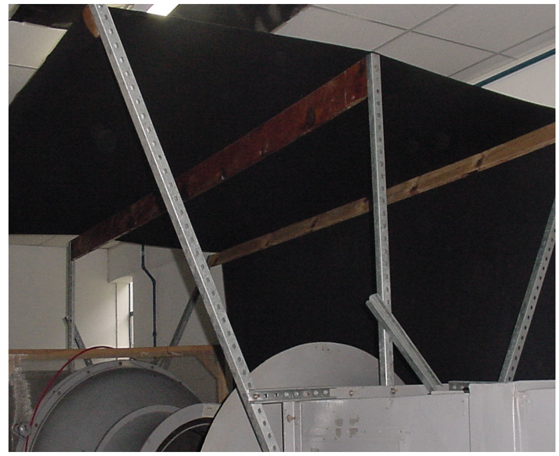

Figure 4. Laser protection.

2.1 THE PIV SYSTEM

A New Wave Nd-YAG 200 mJ dual pulsed Nd:Yag laser, with a repetition rate of 15Hz was used to illuminate the particles introduced into the flow field. A vertical laser light sheet, Fig.6, was created using a flexible mirror arm and a light sheet optics with thickness adjuster. The PIV images pairs were recorded with a digital HiSense 4M camera (built by Hamamatsu Photonics, Inc.) with Nikkor f# 2.8 lenses with 105 mm of focal length. The digital camera was placed on a Dantec Scheimpflug Camera Mounts fixed on an aluminum trail supported by a positioning device, as shown in Fig. 2. The test section flow was seeded with smoke particles, approximately 5 µm in diameter, using a Rosco Fog generator, which was placed inside the wind tunnel diffuser, as can be seen in Fig.5. The measurement process was synchronized and controlled by a Flow Map System Hub and the FlowManager software produced by Dantec Dynamics, Inc. For each PIV measurement, four image pairs were captured in one second, and around 140 image pairs were averaged.

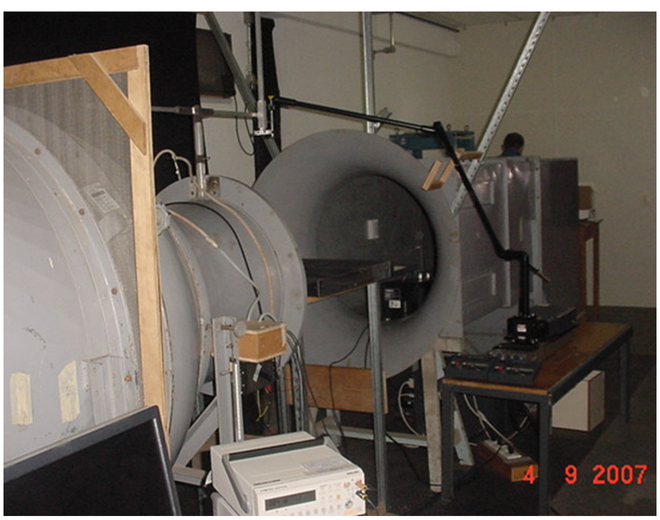

Figure 5. PIV experimental set up.

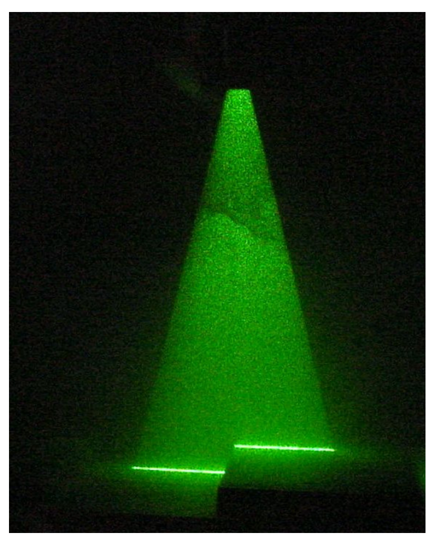

Figure 6. Laser sheet.

A protective black shelter, Fig. 4, was placed around the test section in order to avoid dangerous laser reflections and to avoid environment light interference in the measurements. The measurements were made at the cavity longitudinal central plane.

A Pitot tube probe with differential pressure transducer was used for wind speed monitoring.

3. RESULTS

All results herein are for a cavity with aspect ratio equal to six. The Reynolds number, based on the cavity depth was varied from $0.9x10^4$ to $8.0x10^4$. The central longitudinal plane of the cavity was explored in the present work. The final goal is to have a data bank exploring many longitudinal and transversal planes.

Figure 7 shows the flow for $Re_s = 0.9x10^4$. It seems that the flow outside the cavity does not have enough energy at such low speed to create a bubble inside it. It was also noticed that around x= 250 mm the streamlines tend to bend to the outside of the cavity.

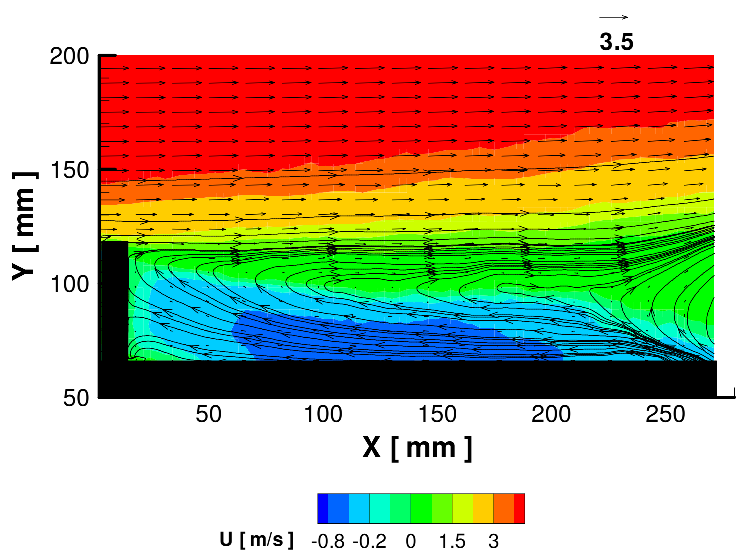

Figure 7. Flow inside the shallow cavity, AR=6, Re = 0.9 $x10^4$

No appreciable change in the flow pattern inside the cavity was observed as the Reynolds number, based upon the cavity depth, increased to $2.0x10^4$. Once again it is not clear the bubble formation inside the cavity, see Fig 8.

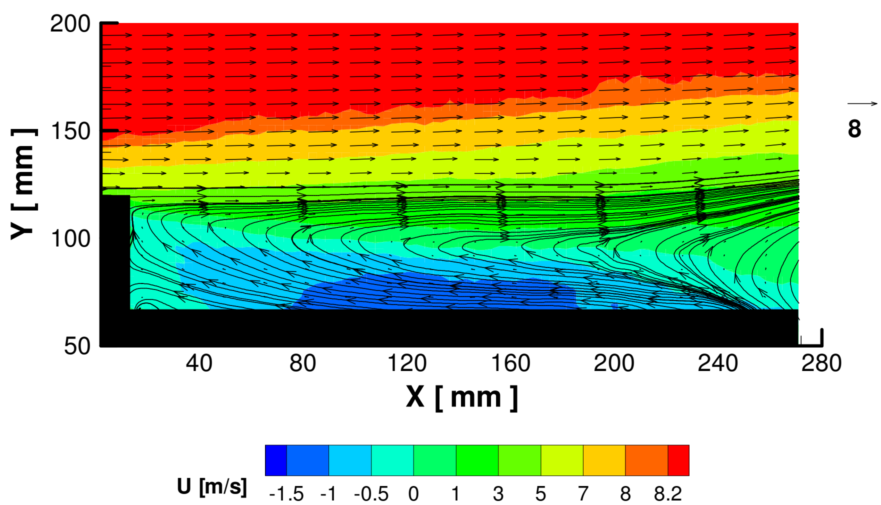

Figure 8. Flow inside the shallow cavity, AR=6, Re = 2 $x10^4$

As the Reynolds number reaches 4.0 x 10^4 it is seem that a single bubble is formed inside the cavity. It seems now that the external flow has enough energy to rotate the flow inside the cavity. This is in agreement with the CFD simulations. On the other hand the previous experimental effort revealed the presence of two bubbles inside the AR=6 cavity. The authors believe that the surface roughness, much more pronounced in that case, might be responsible for the second bubble. As this happens the streamlines at x=250 mm seem to bend inwards instead of outwards as shown in Fig. 7. Figure 10 and Fig.11 show that a secondary bubble is formed at the upstream corner. This particular bubble did not appear in the CFD simulations, suggesting that a more accurate discretization should be used.

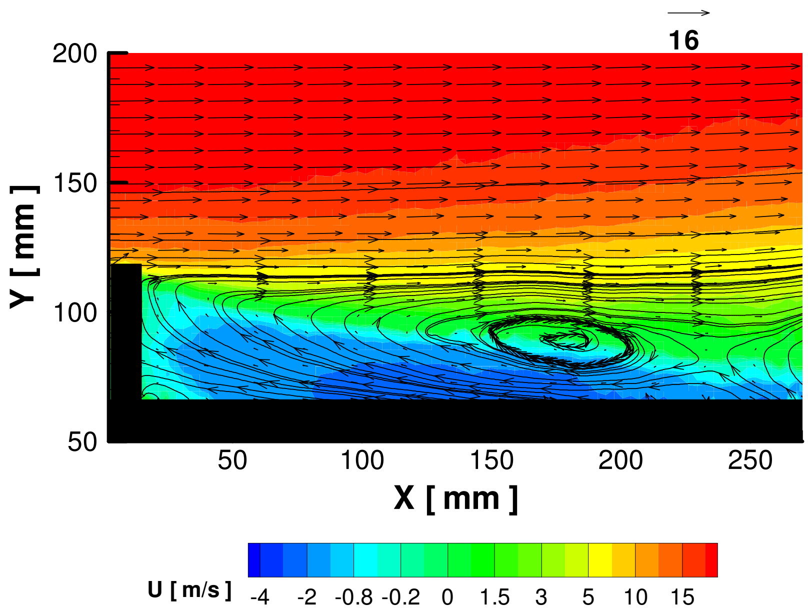

Figure 9. Flow inside the shallow cavity, AR=6, Re = 4.0 x10^4

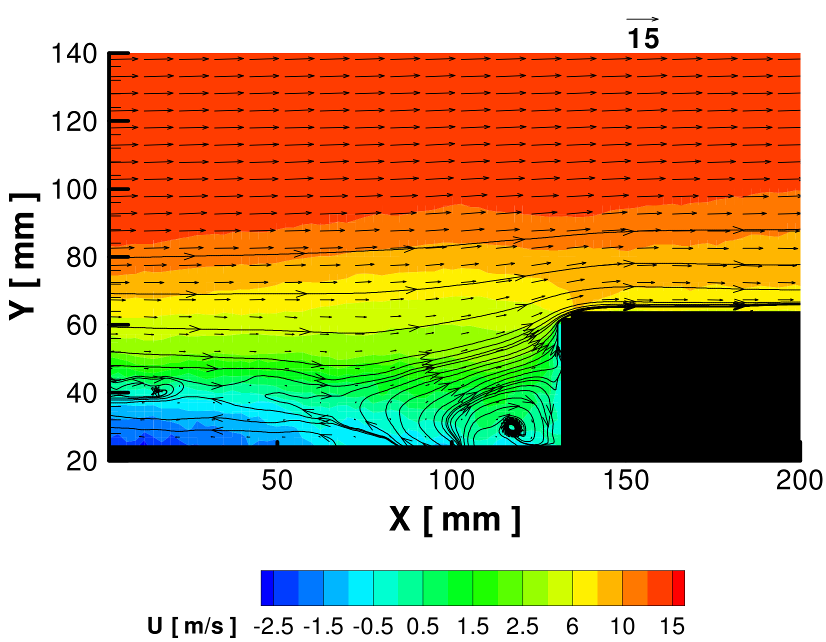

Figure 10. Flow inside the shallow cavity, AR=6, Re = 4.0 x10^4. Detail of the upstream corner.

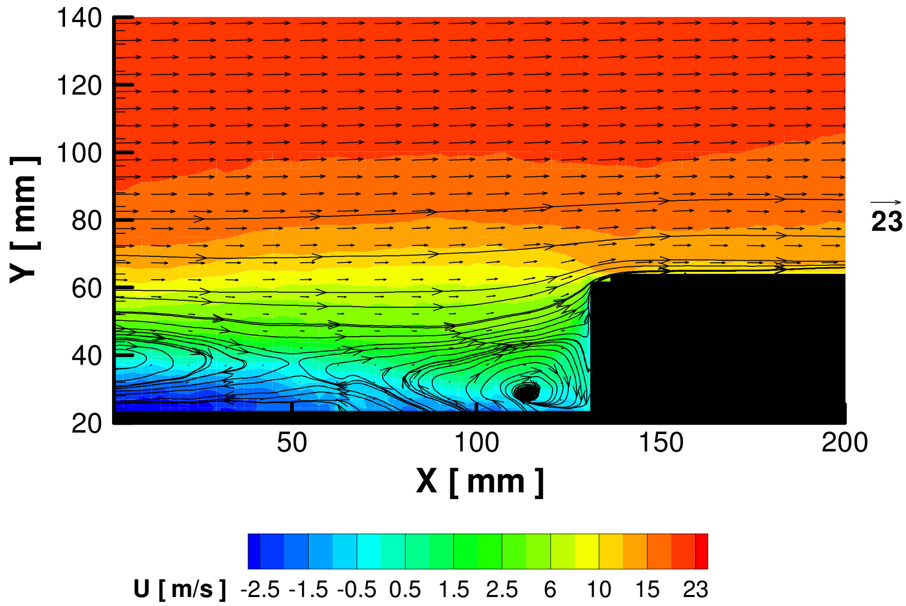

Figure 11. Flow inside the shallow cavity, AR=6, Re = 5.7 x10^4. Detail of the upstream corner.

The higher Reynolds investigated, Re_s = 8.0 x 10^4 and Re_s = 6.2 x 10^4, displays a well-formed recirculating bubble. The qualitative picture is very similar to the one presented in Fig. 9.

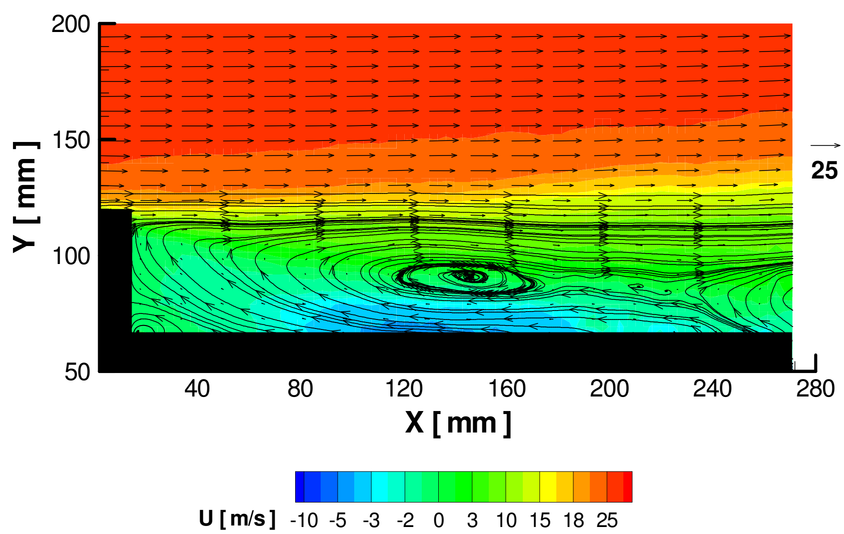

Figure 12. Flow inside the shallow cavity, AR=6, Re = 6.2 x10^4

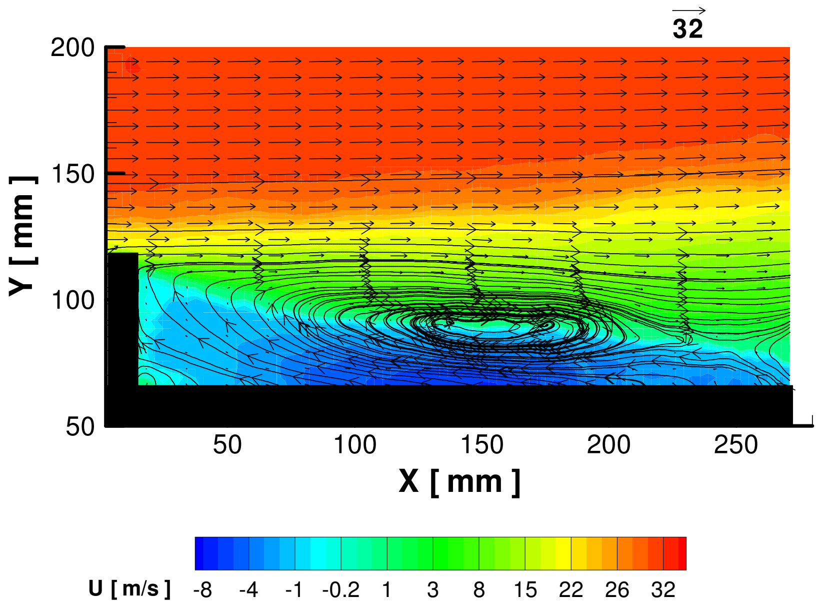

Figure 13. Flow inside the shallow cavity, AR=6, Re = 8 x10^4

4. CONCLUSION

In the present experimental effort, a shallow cavity of aspect ratio equal to six was investigated. The measurements were made at the cavity longitudinal central plane. The results seem to indicate that the flow outside the cavity must be "strong enough" to be able to create a bubble inside the cavity. For very low Reynolds numbers this is not possible. As the Reynolds number increases a well-formed single bubble was observed. In fact, this is in agreement with the CFD calculations but contrary to the previous experimental results by the authors. It must the remembered, however, that for the earlier experimental work the cavity roughness was much greater than for the present effort. This, in fact, might have an important role in the flow pattern. Certainly, this is one of the aspects to be considered in the next phase of the work.

5. ACKNOWLEDGEMENTS

The authors thank FAPESP and Embraer for the financial support through the project 2000/13769-0. They would like to thank the ALA Engineer Matsuo Chisaki, and the technicians José Rogério Banhara and José Ricardo Carvalho de Oliveira.

6. REFERENCES

Aung, W., Bhatti, A., 1984, "Finite Difference Analysis of Laminar Separated Forced Convection in Cavities". *Journal of Heat Transfer*, Vol. 106, No. 1, pp. 49-54.

Aung, W., 1983, "An Interferometric Investigation of Separated Forced Convection in Laminar Flow Past Cavities". *Journal of Heat Transfer*, Vol. 105, No. 3, pp. 505-512.

Duffie, J. A., Beckman, W.A., 1980, "Solar Engineering of Thermal Process". New York: Wiley Interscience.

Gomes D. G., 1998, "Optimization of Flat Plate Solar Collectors", Máster Thesis Dissertation, TIA – Technological Institute of Aeronautics, São José dos Campos, SP, Brazil, (in Portuguese).

Metzger, D. E., Bunker, R. S., Chyu, M. K.,1989 "Cavity Heat Transfer on a Transverse Grooved Wall in a Narrow Flow Channel". Journal of Heat Transfer, Vol. 111, No. 1, pp. 73-79.

Raffel, M., Willert, C., Kompenhas, J.,1998 "Particle Image Velocimetry", Springer-Verlag, New York, Sinha, S. N., Gupta, A. K, Oberai, M. M.,1982 "Laminar Separating Flow Over Backsteps and Cavities Part II: Cavities". AIAA Journal, Vol. 20, No. 3, pp. 370-375.

Avelar, A. C., Fico, N. G. C. R. Jr., Mello, O. A. F.,2007 "Three-dimensional Flow Over Shallow Cavities", 37 th AIAA Fluid Dynamics Conference and Exhibit, 25-28 June 2007, Miami, FL.