EXPERIMENTAL VERIFICATION OF CFD & THERMAL HYDRAULICS CODES BY QUANTITATIVE FLOW VISUALISATION

R.K. Singh, A.K. Kansal, N.K. Maheshwari, P.K. Vijayan and V.M. Joshi
Bhabha Atomic Research Centre, Mumbai, India

Abstract

Complex flow fields are encountered in many reactor components and processes. Measurement and analysis of various flow parameters are very important for optimal design, experimental determination of safety margins and verification of CFD and thermal hydraulics codes. Development of image capture hardware and digital image processing technique in Particle Image Velocimetry (PIV) has made possible to map complex flow fields instantaneously at thousands of points with very high temporal and spatial resolution. PIV is a non intrusive and very flexible technique. In this technique using synchronized operation of laser and CCD camera, seeded flow is illuminated by pulsing laser sheet and images of seeded particles are recorded on CCD camera. The displacement of the particles is measured in the plane of the image and used to determine the velocity of the flow. Image plane is divided into small interrogation regions. Velocity vectors are calculated with the help of cross correlated images obtained from two time exposures. This paper describes 2D PIV System used, flow mapping and verification of CFD codes for pipe flow, submerged jet, thermal stratification in water pool and Fluidic Flow Control Device (FFCD) proposed to be used in advanced accumulator of Emergency Core Cooling System (ECCS).

1. Introduction

Most innovative nuclear reactor systems under development are single-phase systems. Present day CFD codes are considered to be robust for single-phase systems and hence increasing application of these codes is expected for such reactors. However, verification and validation of CFD codes for reactor systems and components is not adequate. For verification and validation of CFD codes, one must be able to measure the multi-dimensional velocity, temperature and concentration fields. Often velocity, temperature and concentration measurements are possible with simple experimental techniques at a few selected points only. Even such techniques are intrusive in nature and affect the flow, temperature and concentration fields being measured. In the past decade or so, techniques for practically non-intrusive measurement of the field variables have become possible, thanks to successful development of Particle Image Velocimetry (PIV), Planar Laser-Induced Fluorescence (PLIF), etc.

Westerweel et al. [1] carried out PIV measurement for fully developed turbulent pipe flow for Reynolds number of 5300. Doorne and Westerweel [2] applied Stereoscopic PIV to measure the instantaneous three component velocity field of pipe flow over the full circular cross-section of the pipe.

Mi et al. [3] performed an experimental investigation for turbulent jet issuing from a round sharp-edged orifice plate into effectively unbounded surroundings. Planar measurements of velocity were conducted using PIV in the near and transition regions. The Reynolds number, based on the jet initial diameter and velocity, was approximately 72,000. The instantaneous and mean velocities, Reynolds normal and shear stresses were obtained. The centerline velocity decay and the half velocity radius were derived from the mean velocity. Westerweel et al. [4] investigated experimentally, the flow at the outer boundary of a submerged self-similar turbulent jet by means of combined PIV and laser-induced fluorescence (LIF) measurements. A fluorescent dye was used so that the LIF data could be used to discriminate between the jet fluid and the ambient fluid. The axial velocity, Reynolds stress, and vorticity are determined relative to the jet boundary.

Buoyancy driven natural convection flows are widely encountered in many of the engineering applications. In particular, in the presence of sufficiently strong heat source, natural convection currents can be set up even in large water pools. The eventuality under such conditions would be a thermally stratified pool with a steep temperature gradient along a vertical plane in the pool [5, 6]. A lot of analytical and experimental work, in particular reference to heated enclosures, has been done by Batchelor [7], Gill [8], Cormack et al. [9], Imberger [10], Patterson & Imberger [11] apart from others. Also, with regard to natural convection phenomenon in cylindrical geometries, Martini & Churchill [12], Sabzerari & Ostrach [13], Brook & Ostrach [14], Deaver & Eckert [15] among others have reported their findings. The studies have indicated the importance of the parameters governing the flow phenomenon like Rayleigh number, Prandtl number and the aspect ratio of the enclosure (the ratio of height to the width of the enclosure). Corvaro and Paroncinia [16] performed numerical and experimental analysis to study the natural convection heat transfer in a square cavity heated from below and cooled by the sidewalls. A 2D PIV was utilized to measure the velocity fields at different Rayleigh numbers. Measured quantities were compared with the numerical results. Even though such studies are of immense help in understanding the dynamics of the phenomenon, the findings are very much influenced by the geometry and imposed boundary conditions. Hence, the conclusions drawn from such works may not be directly applicable to specific reactor systems.

Regarding confined swirling flow, Lewellen [17] cited a number of papers dealing with confined vortex flow phenomena. Vatistas et al. [18] presented analytical and experimental studies to investigate selected fluid parameters in vortex chambers. From the analytical model, the dimensionless quantities of the core size, static pressure drop across the chamber and radial static pressure distribution are determined as a function of the chamber geometry. Zhang and Hugo [19] employed stereo PIV to study a vortex generated via tangential injection of water in a 57 mm diameter pipe for Reynolds numbers ranging from 1,118 to 63,367. Method of decreasing optical distortion and PIV calibration technique were addressed by them. Boysan and Swithenbank [20] numerically predicted turbulent vortex flow in cyclone chamber with the aid of a two-equation model of turbulence by employing a two-viscosity concept.

A particle image velocimetric facility has been set-up for flow mapping. Such facility plays important role in basic research, design optimisation, Computational Fluid Dynamics (CFD) code development and its validation. Flow in circular tube, submerged jet, natural convection in a water pool and fluidic flow control device (FFCD) used in advanced accumulator of Emergency Core Cooling System (ECCS) have been studied using PIV system. Theoretical analysis has also been performed. The paper deals with the experimental setups used, results obtained by using PIV. The theoretical analysis performed has been compared with results obtained with the PIV system.

2. Particle Image Velocimetry System

Particle Image Velocimetry (PIV) is a non-intrusive optical technique for the measurement of flow velocity vectors at many (e.g. thousands) points in a flow field simultaneously and provides instantaneous vector fields and displays velocity vectors in real-time [21]. The system contains pulsed laser source, optical components for laser sheet generation, camera, synchronizer and analysis software as shown in Figure 1. Laser is used as it has very short pulse duration (~5-7 ns) which can freeze any motion. The other reason is that only laser light can be focused into a thin enough light sheet so that only particles in that plane are imaged. A special camera is utilized for taking images so that it can store the first image (frame) fast enough to be ready for the second exposure. To synchronize the laser source and camera, a synchronizer is provided. For all the experiments, glass hollow spheres having density of 1.1 g/cm^3 (close to water) and size of 10 μm are used as tracer particles.

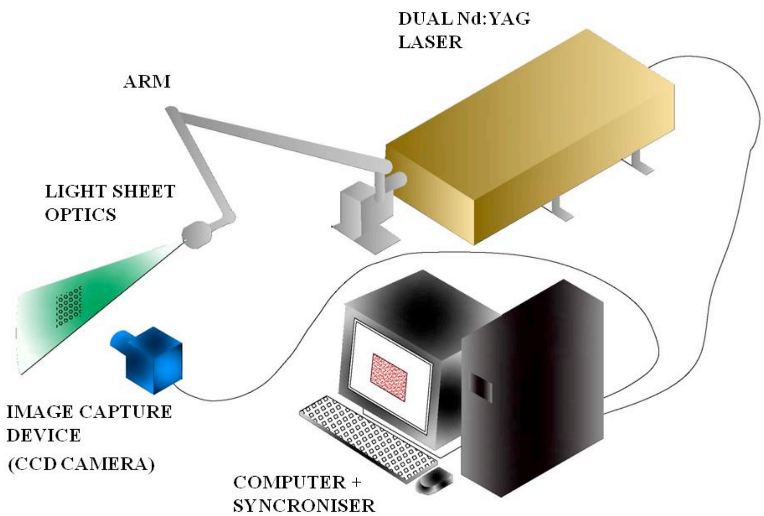


Fig. 1: Typical 2D PIV Setup

2D Velocity vectors are obtained by analysis of displacement of tracer particles on images in a known time. In order to determine the path taken by the particles, a statistical cross-correlation is performed. The peak cross-correlation yields a velocity vector which describes the average direction and speed of particle displacement. Thus, the result of a PIV analysis is a complete velocity vector field for the selected area [22].

3. Experiments preformed for various geometries

Flow in circular tube, submerged jet, natural convection in a water pool and fluidic flow control device (FFCD) used in advanced accumulator of Emergency Core Cooling System (ECCS) have been studied using PIV system and are discussed below.

3.1 Flow in circular tube

The test facility consists of a test section, pump, rotameter and sump. The test section is comprised of a glass tube having inner diameter of 22 mm and a square glass enclosure around the glass tube filled with water. The purpose of water filled square glass enclosure is to minimise the image distortion from the curved surface of pipe. Test section used is shown in Fig. 2a. Experiments were carried out for different flow rates. Figure 2b (i and ii) shows the velocity profile for flow rate of 0.4 lpm and 30 lpm respectively at different axial locations. The Reynolds numbers calculated for these flow rates are 480 and 36000 respectively. It can be seen from the Fig. 2b that measurement taken by PIV system gives parabolic velocity profile for laminar region and flat velocity profile for turbulent region. Figures 2c (i) and (ii) depict comparison of measured non-dimensionised axial velocities as a function of non-dimensionised radial distance with the computed by CFD. The axial locations at which these profiles are shown are at a distance of 13D and 6.25D from inlet for laminar and turbulent flow respectively, where D is inner diameter of the circular tube.

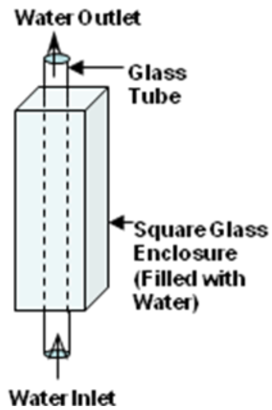


Fig. 2a: Test section for flow through circular tube

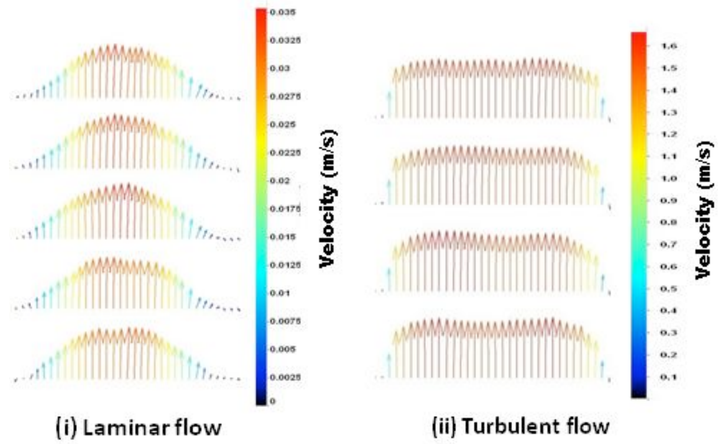
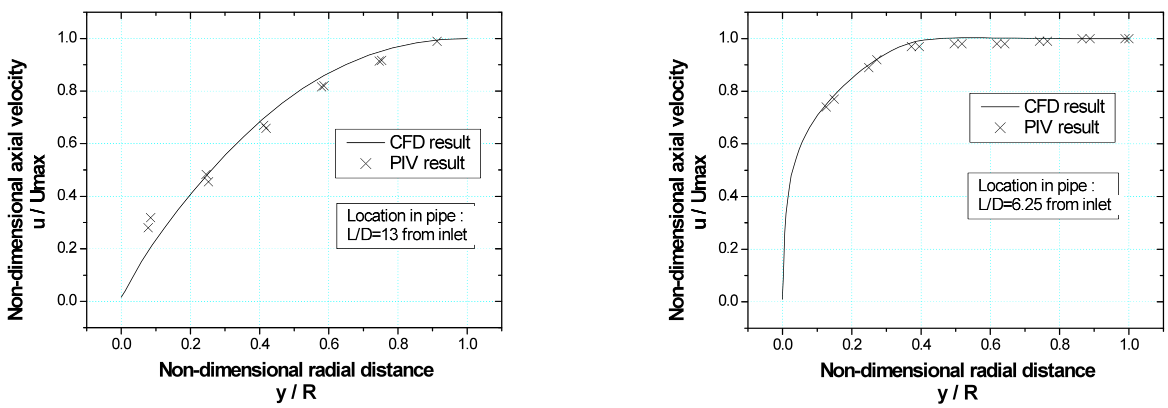


Fig. 2b: Velocity profile in a pipe by digital PIV at different axial locations



(i) Laminar Flow (ii) Turbulent Flow

Fig. 2c: Comparison of velocity at pipe cross section by PIV and CFD code

3.2 Submerged water jet

In the secondary shutdown system of Pressurised Heavy Water Reactor (PHWR) and advanced water cooled reactor, poison is injected into calandria through perforated tubes to shutdown the reactor. Poison coming out of the perforated tubes is in the form of submerged jets. Distribution of poison by submerged jet in presence of calandria tubes is very complicated problem in reactor.

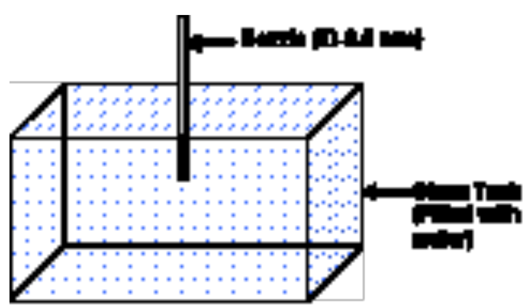


Fig. 3a: Schematic of Nozzle in water tank

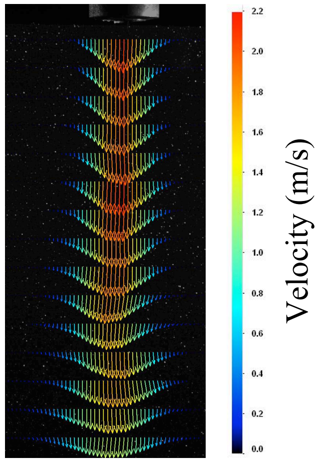


Fig. 3b: Velocity profile at different axial distances of submerged water jet by PIV system

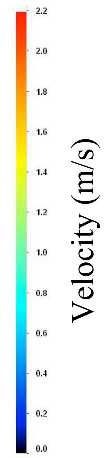


Fig. 3b: Velocity profile at different axial distances of submerged water jet by PIV system

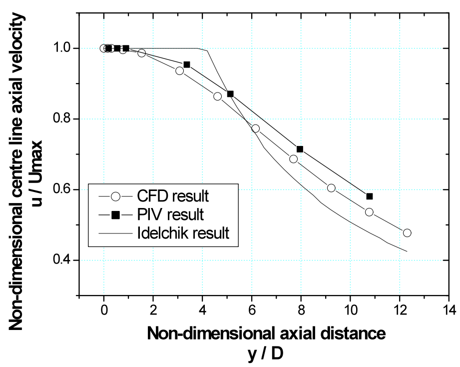


Fig. 3d: Comparison of PIV results, CFD results and Idelchik results for jet axial velocity

As a first step, single submerged jet without any obstacle is studied using PIV. In submerged jet, as the jet progresses the velocity of the jet decreases in axial direction due to expansion of jet in transverse direction. In this process the total momentum remains constant at all cross sections in the submerged jet [23]. Experimental setup used is consisted of water tank, tube of inner diameter of 6.5 mm, pump and rotameter. A flow of 4 lpm is provided through tube. Fig. 3a shows the water tank with circular tube of inner diameter of 6.5 mm for issuing water jet. Fig. 3b shows the velocity profile of jet and its broadening as it progresses by using PIV. Fig. 3c shows the velocity contours of jet using PIV. CFD analysis is also carried out for experimental conditions. Figure 3d shows the comparison of PIV results with CFD results and analytical results [24] for jet axial velocity.

3.3 Natural convection in water pool due to vertical plate heater

In advanced nuclear reactors, large water pools have been employed for removal of core decay heat passively during reactor shut down. In this system, heat exchangers connected to primary circuit are immersed in large water pool. These heat exchangers transfer heat from primary circuit to water pool by buoyancy driven natural convection phenomena. During heat transfer process, thermal stratification with a steep temperature gradient along the vertical plane occurs in the water pool.

Typical applications affected by this phenomenon include the passive core decay heat removal system of Advanced Heavy Water Reactor (AHWR). This system comprises of the Isolation Condensers (IC) submerged in Gravity Driven Water Pool (GDWP) which acts as the heat sink. For such systems, apart from the reduced heat transfer, the vertical temperature gradient may also pose a serious problem for the concrete structure housing the pool. Also, if there is rapid evaporation, undesirable pressurization of the concrete structure may also result [25]. Under reactor shutdown conditions, the steam generated in the core as decay heat gets transported to the IC tubes by natural circulation. The steam condenses in the IC tubes and the heat is transferred to GDWP. This can be a long transient lasting several hours and may extend to a few days. In the process, the GDWP water tends to get stratified with time thereby gradually reducing the heat transfer from the IC tubes to the GDWP.

To ensure satisfactory performance of the system under consideration, it becomes mandatory to study and analyse the fundamental mechanism underlying this phenomenon. Also, towards the optimization of the design of such systems in which thermal stratification is expected to occur, techniques to maximize or minimize the effect are also required to be evolved.

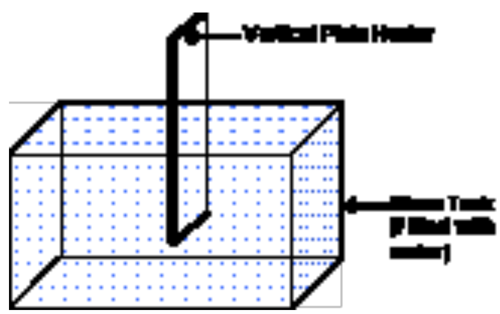


Fig. 4a: Schematic of vertical heater in glass tank

Natural convection flow pattern in water pool due to vertical plate heater is studied using PIV system. The experimental setup consists of vertical heater plate immersed in a water tank as shown in Fig. 4a. The heater plate was given electrical power of 313W. As the plate is heated, boundary layer flow develops adjacent to the plate. PIV measurements taken after 15 s of switching on heater are shown in Fig. 4b. CFD analysis has been also carried out to study the natural convection flow pattern. Figure 4c shows the CFD results for velocity distribution inside the water tank at 15 s.

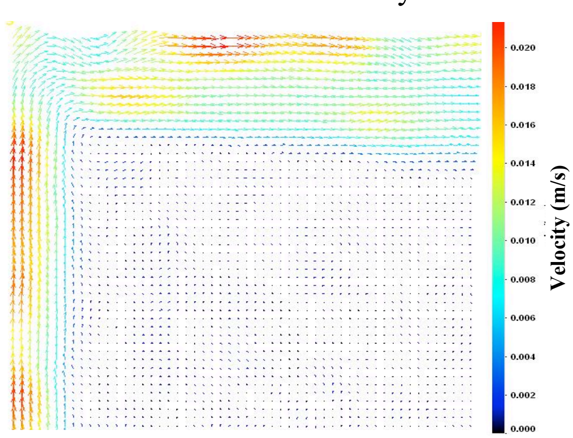


Fig. 4b: Velocity vector by PIV system

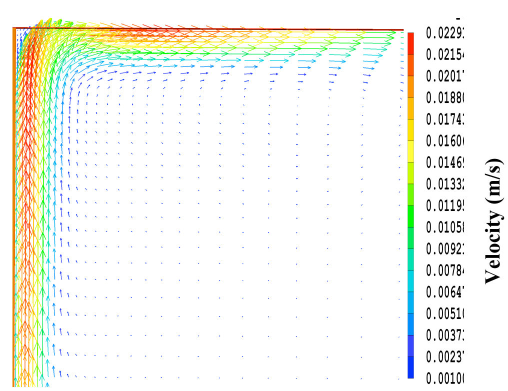


Fig. 4c: Velocity vector by CFD code

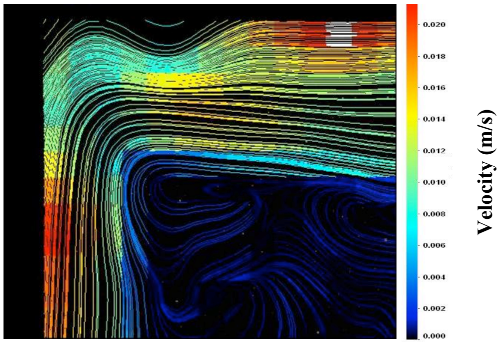


Fig. 4d: Velocity stream lines and contours by PIV

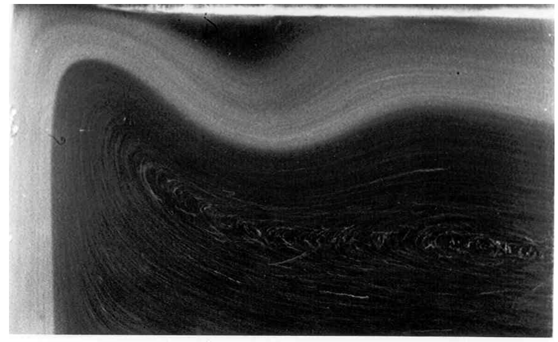


Fig. 4e: Flow visualization with aluminium particles

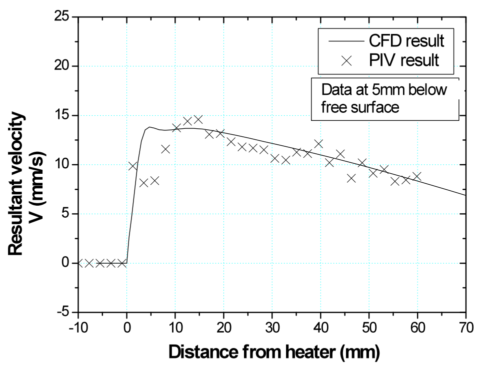


Fig. 4f: Comparison of PIV and CFD results at 5 mm below free surface of water

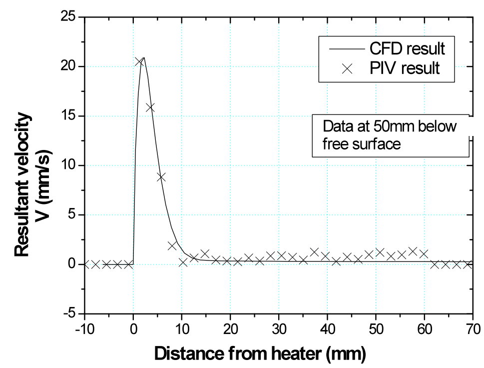


Fig. 4g: Comparison of PIV and CFD results at 50 mm below free surface of water

The flow visualization is done with aluminium particles (as shown in Fig. 4e matches closely with stream line pattern obtained from PIV system as shown in Fig. 4d. It can be observed from Fig. 4d and Fig. 4e that fluid moves up along the vertical heater to the free surface. At the free surface, the detaching boundary layer is reflected downwards. However, due to its higher temperature (lower density), it is found to rise back to the free surface and flows along the free surface horizontally towards the wall of the container. Comparison of resultant velocities from PIV and CFD results as a function of distance from vertical heater is given in Figs. 4f and 4g for 5 mm and 50 mm below free surface respectively.

3.4 Fluidic Flow Control Device

Emergency Core Cooling System (ECCS) is provided to limit the fuel temperature rise within acceptable limits in the event of Loss Of Coolant Accident (LOCA). The advanced accumulators in advanced nuclear reactor injects large amount of cold water by passive means, at high pressure, directly into the core for short period of time and then a relatively small amount of cold water for large period of time to quench the core. This objective is achieved by incorporating

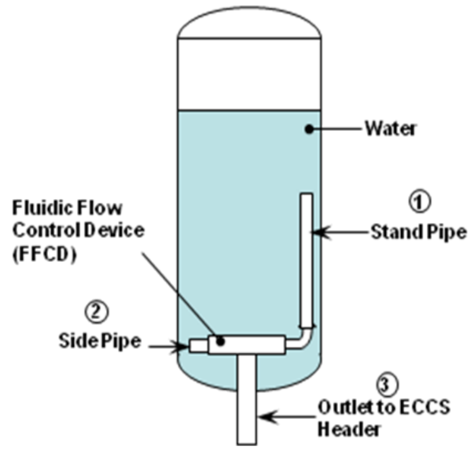
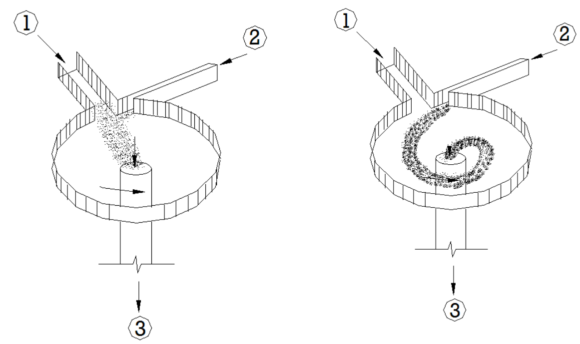


Fig. 5a: Accumulator tank with Fluidic Flow Control Device (FFCD)



(i) Large flow rate from Stand pipe

(ii) Reduced flow rate from Side pipe

Fig. 5b: Details of FFCD

a Fluidic Flow Control Device (FFCD) at the bottom of accumulator tank as shown in Fig. 5a, which reduces the flow after some time, by passive means. This feature enables to extend the accumulator discharge. During initial period, the water level in the advanced accumulators is above the stand pipe, the water enters the vortex chamber of the FFCD through both stand pipe (radial) and side connection (tangential) as shown in Fig. 5b (i), and after some time when the water level in the accumulators falls below the top of the stand pipe, the water enters the chamber through the side pipe only, which is tangential to the vortex chamber as shown in Fig. 5b (ii). This causes the formation of vortex, which increases the flow resistance, and hence reduces the flow rate passively.

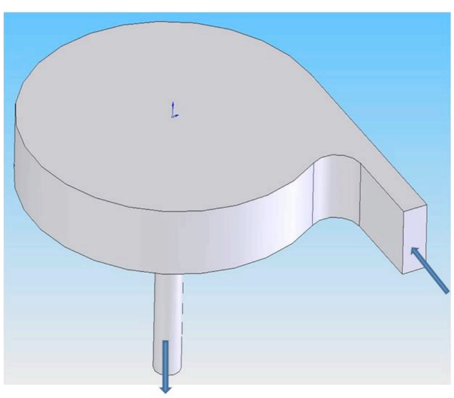


Fig. 5c: Schematic of Glass Fluidic flow device used in PIV experiment

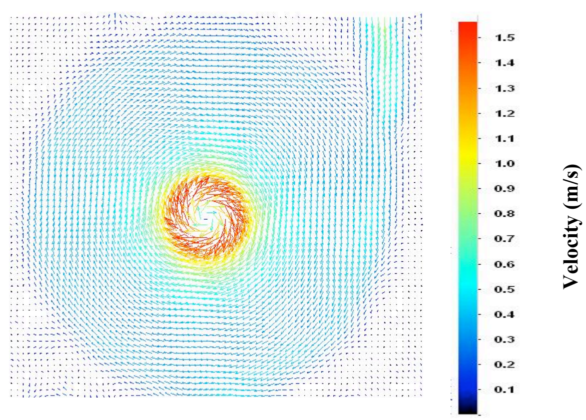


Fig. 5d: Velocity vector pattern using PIV system

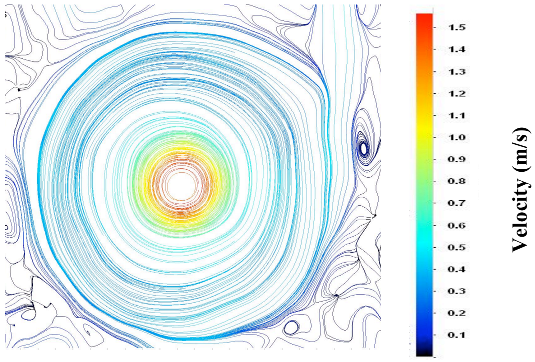


Fig. 5e: Velocity stream line and contour using PIV system

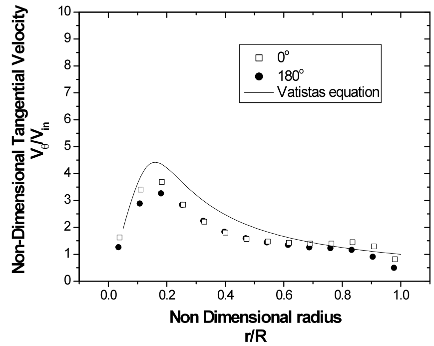


Fig. 5f: Tangential velocity distribution using PIV system

FFCD for PIV experiment was fabricated from glass as shown in Fig. 5c. Only the side pipe and outlet are considered for study the vortex in FFCD as vortex study is of our interest. It has main shell of 95mm diameter and 20 mm height. The experiments were conducted for various flow rates. Circular mid plane is studied using the PIV system for velocity pattern and the result is shown in Fig. 5d. Fig. 5e shows the stream line plot generated by PIV software. Fig. 5f shows the variation of non dimensional tangential velocity with non dimensional radius using PIV system and pattern matches with empirical correlation proposed by Vatistas et al. [26]. Results shows increase in tangential velocity to a peak value with radius than decrease.

4. Conclusions

Flow pattern studies have been carried out for various geometries pertaining to nuclear reactor systems. To have confidence in PIV system measurements first flow pattern in circular tube has been obtained for laminar and turbulent flows. A good agreement has been found between CFD and PIV results. PIV measurements have also been taken for a submerged jet. A comparison of axial centre line velocity has been performed between CFD studies and PIV measurement.

The flow pattern studies subsequently have been carried out to study the flow pattern generated by heated plate in the pool of water. A comparison of velocities, obtained in CFD analysis and by PIV measurement at various locations has been carried out. About the flow pattern obtained in fluidic flow control device flow measurement has been carried out for a swirling flow using PIV system. The flow pattern obtained is similar to empirical equation given by Vatistas et al. [26].

Nomenclature

D	nozzle diameter (m)
r	radius (m)
R	inner radius (m)
u	centre line axial velocity (m/s)
U_{max}	centre line maximum axial velocity (m/s)
V	resultant velocity (mm/s)
V_{in}	Inlet velocity (m/s)
V_θ	Tangential velocity (m/s)
y	distance (m)

5. References

[1] J. Westerweel, A.A. Draad, J.G. Th. Van der Hoveven and J. Van Oord, “Measurement of fully-developed turbulent pipe flow with digital particle image velocimetry”, Experiments in Fluids, vol 20, 1996, pp 165-177

[2] C. W. H. van Doorne and J. Westerweel, “Measurement of laminar, transitional and turbulent pipe flow using Stereoscopic-PIV”, Experiments in Fluids, vol 42, 2007, pp 259-279

[3] J. Mi, P. Kalt, G. J. Nathan and C. Y. Wong, “PIV measurements of a turbulent jet issuing from round sharp-edged plate”, Experiments in Fluids, vol 42, 2007, pp 625-637

[4] J. Westerweel, T. Hofmann, C. Fukushima and J. C. R. Hunt, “The turbulent/non-turbulent interface at the outer boundary of a self-similar turbulent jet”, Experiments in Fluids, vol 33, 2002, pp 873-878

[5] P.F. Peterson, Scaling and analysis of mixing in large stratified volumes, International Journal of Heat & Mass Transfer, 37, 1994, 97-106

[6] J. Christensen and P.F. Peterson, A one-dimensional Lagrangian model for large-volume mixing, Nuclear Engineering and Design, 204, 2001, 299-320.

[7] G.K. Batchelor, Heat Transfer by free convection across a closed cavity between vertical boundaries at different temperatures, Journal of Applied Mathematics, 12, 1954, 209

[8] A.E. Gill, The boundary-layer regime for convection in a rectangular cavity,Journal of Fluid Mechanics, 26, 1966, 515

[9] D.E.Cormack, L.G.Leal and J.Imberger, Natural convection in a shallow cavity with differentially heated end walls, part-I-Asymptotic Theory, Journal of Fluid Mechanics, 65, 1974, 209

[10] J.Imberger, Natural convection in a shallow cavity with differentially heated end walls, part-III-Experimental results, Journal of Fluid Mechanics, 65, 1974, 247

[11] J.Patterson and J.Imberger, Unsteady natural convection in a rectangular cavity, Journal of Fluid Mechanics, 100, 1980, 65

[12] W.R. Martini and S.W. Churchill, Natural convection inside a horizontal cylinder, AIChE J, 6, 1960, 251

[13] A. Sabzevari and S. Ostrach , Experimental studies of natural convection in a horizontal cylinder, Journal of Heat Transfer, 3, 1974, 100

[14] Brook, I.H. & Ostrach, S., An experimental investigation of natural convection in a horizontal cylinder, Journal of Fluid Mechanics, 44, 1970, 545.

[15] F.K. Deaver and E.R.C. Eckert , An interferometric investigation of convective heat transfer in a horizontal fluid cylinder with wall temperature increasing at a uniform rate, Journal of Heat Transfer, 4, 1970, Paper NC 1.1.

[16] F. Corvaro and M. Paroncini , Experimental analysis of natural convection in square cavities heated from below with 2D-PIV and holographic interferometry techniques, Experimental Thermal and Fluid Science 31 (2007) 721–739

[17] W.S. Lewellen, "A review of confined vortex flows", NASA CR-1772, July 1971

[18] G.H. Vatistas, S. Lin and C.K. Kwok, "Theoretical and experimental studies on vortex chamber flows", AIAA journel, Vol. 24, No. 4, April 1986, 635-642

[19] Zherui Zhang and Ronald J. Hugo, "Stereo particle image velocimetry applied to a vortex pipe flow", Experiments in Fluids (2006) 40, 333–346

[20] F.W.Rysan and J. Swithenbank, "Numerical prediction of confined turbulent vortex flows", Report No. ADA102849,Sheffield Univ (United Kingdom) Dept of Chemical Engineering and Fuel Technology, Sep 1980

[21] M. Raffel, C. Willert and J. Kompenhans, "Particle Image Velocimetry", A Practical Guide, Springer Verlag, Berlin, 1998.

[22] R. J. Adrian "Twenty years of particle image velocimetry", 12th International symposium on applications of laser techniques to fluid mechanics, Lisbon, 2004, July 12-15.

[23] G. N. Abramovich, "The Theory of Turbulent Jets", The Massachusetts Institute of Technology Press, 1963.

[24] I.E. Idelchik, "Handbook of Hydraulic Resistance", Begell House Publishers, Second Edition

[25] N.V.Satish Kumar, N.K.Maheshwari, P.K.Vijayan, D.Saha and R.K.Sinha, Studies on the phenomenon of thermal stratification in side heated enclosures, 6th ISHMT-ASME Heat & Mass Transfer Conference and 17th National Heat & Mass Transfer Conference, 2004, Kalpakkam, India.

[26] Georgios H. Vatistas, V. Kozel and W.C. Mih, "A Simpler Model for Concentrated Vortices", J. Experiments in Fluids, Vol 11,1991 ,73-76