VELOCITIES MEASUREMENTS FOR PRESSURIZED WATER REACTOR RESEARCH USING PIV-CFD

J. Stefanini*, G. Mignot**, V. Saldo**, G. Baroi***, T. Conte***

(*)TSI France Inc., Marseille, France
(**) CEA Cadarache, France
(***) Framatome, Paris, France

ABSTRACT

Since several years, a R&D action, based on numerical simulation and experiments of flow into PWR vessels was initiated at Framatome and CEA. The aim is to reach a better understanding of the thermal hydraulic phenomena and their consequence into the whole nuclear vessel. This paper presents a comparison between numerical simulation and experimental data obtained using PIV technique. Measurements have been performed into the hot legs of a 1300 MW PWR mock-up.

The experiment was carried out on a scaled down PWR model. The facility represents the upper plenum of such reactor with its internal components (control rods guides, support columns, …) and its hot legs. The 450 l/s nominal flow rate is symmetrically distributed on 4 hot legs through regulation gates.

One TSI PIV system including 200 mJ laser and high-resolution cross correlation camera (1K x 1K pixels) was used to describe the flow behavior. For practical reasons and to avoid image distortion due to circular optical access, the camera is located at the end of the hot leg, which is equipped with a flat glass window.

Measurements were carried out in one hot leg and we will present in this paper data obtained in three different cross section planes (perpendicular to the main flow direction).

The numerical simulation has been done using a standard CFD code that allows computing thermohydraulics phenomena into complex geometrical situation. It is solving 3D Navier Stokes equations using finite volume method and allows computing steady and unsteady conditions.

In order to make a realistic comparison between experimental results and numerical simulation, we averaged the PIV data over 100 instantaneous velocities fields.

The results show a good agreement between experimental data and CFD regarding the velocity spatial distribution and general flow behavior. However it is necessary to investigate with more detail in order to have a good agreement not only when comparing the general flow behavior but also comparing numerical data (velocity magnitude).

1. INTRODUCTION

Knowledge of the thermal hydraulic behavior in a Pressurized Water Reactor (PWR) is an important element of the reactor operation (safety and performance). To study hydraulic phenomena, numerical simulation becomes a very useful tool to compute thermohydraulics flows through complex geometry. However, CFD codes need to be supported by experimental data in order to validate the different models.

In this paper, we will present in a first part PIV measurements obtained into a region (hot legs) of a PWR upper plenum mock up. The upper plenum and hot legs are key zones, which govern the flow located on the downstream side of the upper core plate. These flows are complex from a geometrical and physical

point of view. The velocity measurements are included in a large experimental project intended to contribute to a better understanding of the general flow structure in this part of the reactor and also to support numerical computation.

We will also present a comparison between experimental data (PIV) and CFD results.
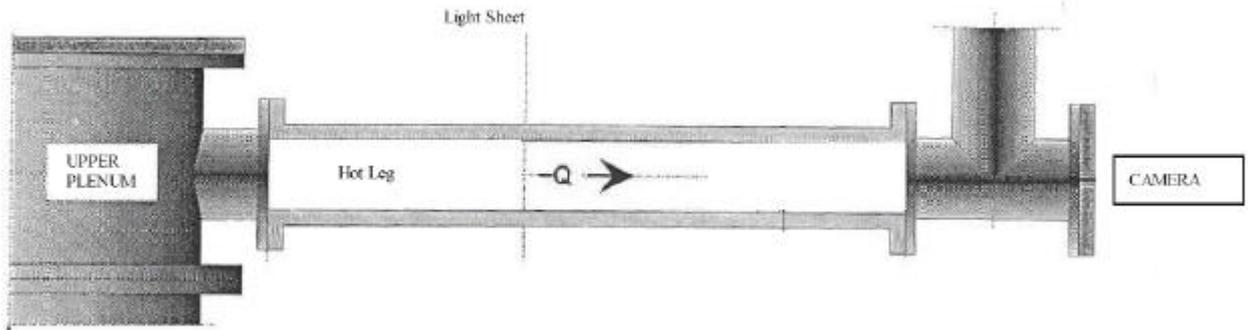
2. EXPERIMENTAL SETUP

The experiment was carried out on a scaled down ($1/5^{th}$) PWR model, BANQUISE. The facility represents the upper plenum of such reactor with its internal components (control rods guides, support columns, …)

and its hot legs. The 450 l/s nominal flow rate is symmetrically distributed on 4 hot legs through regulation gates.

One PIV system was used to describe the flow behavior. For practical reasons and to avoid image distortion due to circular optical access, the camera is located at the end of the hot leg, which is equipped with a flat glass window.

Measurements were carried out in one hot leg (figure 1) and we will present in this paper data obtained at four different cross section planes (perpendicular to the main flow direction) located between the beginning and the end of the hot leg.


Figure 1 Experimental apparatus

3. PARTICLE IMAGE VELOCIMETRY

3.1 Experimental Setup

The Particle Image Velocimetry (Figure 2) system is based upon a TSI PowerView system, including 200 mJ dual Yag laser (Spectra Physics), Light arm, PIVCAM 10-30 high resolution cross correlation camera (1K x 1K resolution), Synchronizer and "Insight Ultra" windows NT based software for acquisition, processing and post processing.
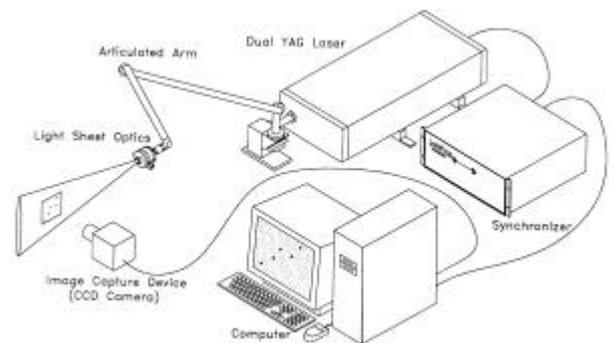
Figure 2 PIV system

3.2 Seeding

Because of the large water quantity (several hundreds m^3), it was not possible, mainly du to cost, to use particles as hollow glass spheres or metallic coated particles. We decided to use 4 μm nylon particles with a density of 1,05 g/cm^3. They are well adapted for water flows studies because of their small size and density close to water but the main drawback is because of the mean diameter it becomes necessary to use high energy laser.

4. NUMERICAL SIMULATION

4.1 Methodology

As the geometry of an upper plenum is very complex, we have done a modelization of an entire upper plenum with its four hot legs. The model stretches from the upper core plate to the end of the hot legs.
The study is divided into two parts: the first to determine the distribution of flow rate into the upper core plate and the second to simulate flows into the upper plenum and hot legs.

4.2 Presentation of the Star-CD code

The CFD code Star-CD is developed by Computational Dynamics in GB and the interface by Adapco in USA. This code developed for many CFD problems uses the Finite Volume method to solve hydraulics' equations. This code allows the modeling of complex geometry's with unstructured meshes and is applied to a wide variety of incompressible or compressible, laminar or turbulent flows, with or without heat transfer between fluid and solid. Framatome undertook a validation of the code with analytical and experimental results using Quality Assurance procedures, which completes for its specific applications the validation of the code done by CD-Adapco.
With Star-CD we can use two algorithms for solving Navier-Stokes equations:

- SIMPLE algorithm for steady-state calculations which allows a very quickly convergence of the calculations to a steady state solution.

- PISO algorithm (for Pressure Implicit with Splitting of Operators) for transient calculations.

The modeling of Banquise mocks-up use the RNG k-ε (Renormalisation Group k-ε) model for turbulence modeling.
The model that we have created is presented on Figure3. It represents a whole plenum and the four hot legs. We have chosen to modelize a whole plenum in order to take account of the internal structures in the four quarters of the upper plenum, which induce dissymetries between the hot legs. All these internal structures are modelized in the upper plenum as control rods guide tubes and support columns but their geometries have been simplified especially concerning the internal structures of control rods guide tubes.
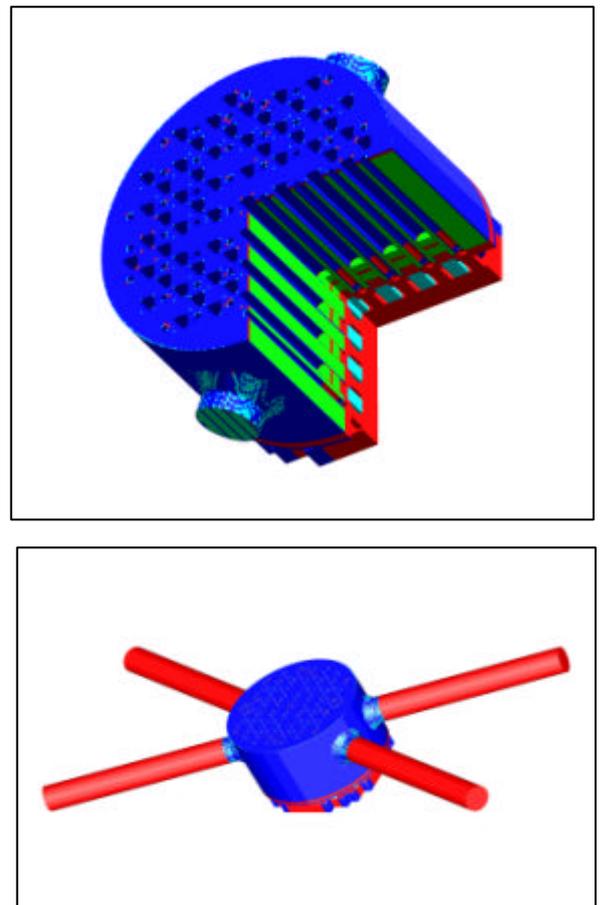

Figure 3 Mesh and geometry presentation.

5.   RESULTS

5.1   General Flow Behavior (numerical data)

Figure 4 shows some streamlines coming from assemblies H1, G3 and C4 and how the fluid is organizing in hot legs. We see that streamlines coming from central holes of the upper core plate tend to go to the top of the hot leg whereas the streamlines coming from peripheral holes tend to go to the bottom of the hot leg. This figure shows also a swirl effect.
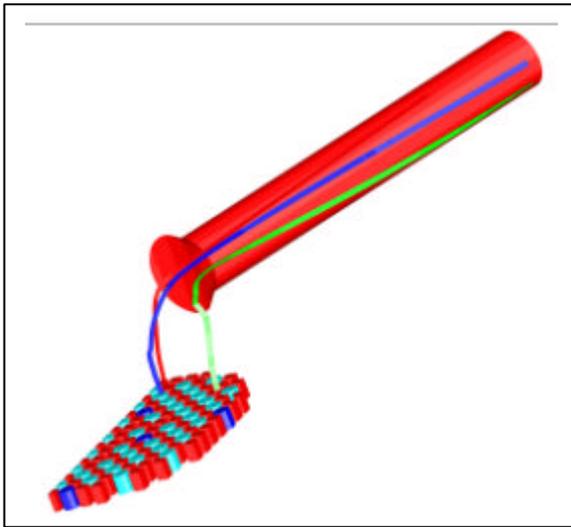
Figure 4 View of streamlines into upper plenum and hot leg 3

5.2   Results Comparison (PIV-CFD)

Experimental measurements have been carried out on 190 mm x 190 mm field with 32 x 32 pixels interrogation cell (6 mm^2 spatial resolution).

The comparisons between numerical and experimental results are concerned with velocity and vorticity profiles in different cross sections of hot leg number 3. These four different planes are equally spaced along the hot legs from the beginning to the end. Figures 5 and 6 show respectively the mean velocity and vorticity maps for each of these sections. We have to point out that the velocity fields correspond to the velocity projection on the cross section plane without taking into account the axial velocity which is the main component (80% of the velocity magnitude). PIV data have been averaged over 100 instantaneous velocities field.

We can see that the main flow structures are well described and especially the number of vorticies and their locations in the hot leg for different cross-section planes. The modeling seems to be able to describe the swirl effects in the hot leg.

However, vortices locations are not exactly identical and also looking at mean velocities fields, numerical simulation seems to over estimate the velocity magnitude compared to experimental data obtained by PIV. This may be due to the inlet boundary conditions used for flow rate on the upper core plate.

We can see from the velocities fields (figure 5) that if close to the upper plenum (hot leg entry), the flow shows very strong vortex, the magnitude of the vorticities fields, as shown figure 6, is decreasing when moving away from the upper plenum. This phenomenon corresponds to energy diffusion containing into the flow.

6.   CONCLUSION

This study contributes to a better understanding of the flow behavior into the upper plenum and the hot legs of a 1300 MW PWR reactor. A PIV technique has been use to caracterize the flow into one hot leg of a 1/5th PWR mock-up. We demonstrate the capability of such technique to investigate industrial flow.

The second step of this work was to use experimental data in order to validate numerical simulation. The first results have shown a good agreement qualitatively between CFD and PIV. However looking at quantitative results, CFD over estimates velocity magnitude obtained by PIV measurements.
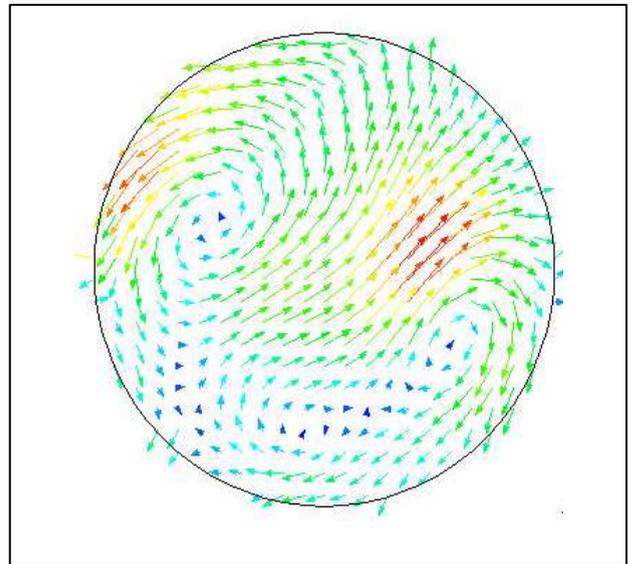
The next goal of this study will be to investigate the flow into different hot legs and also onto the upper plenum using PIV. These new experimental data will be use to validate numerical simulation.
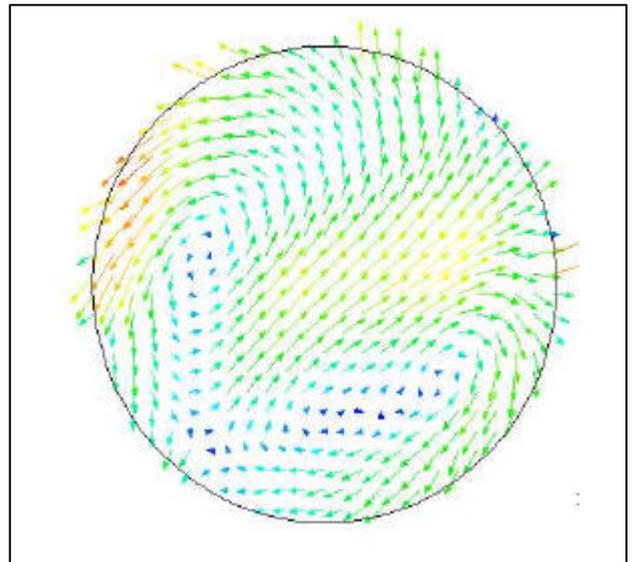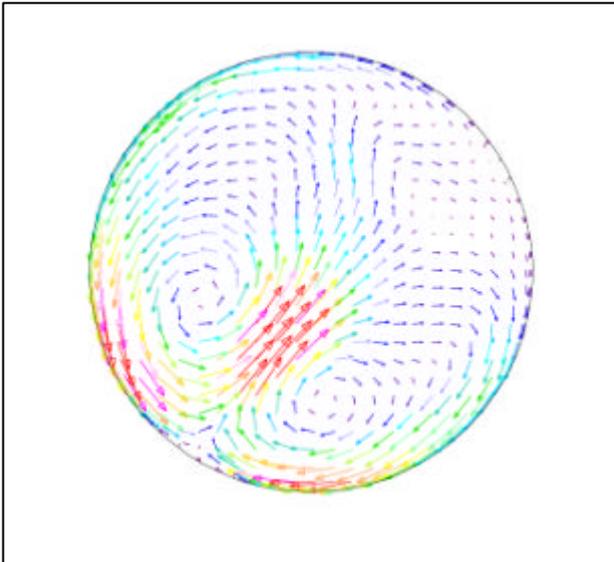
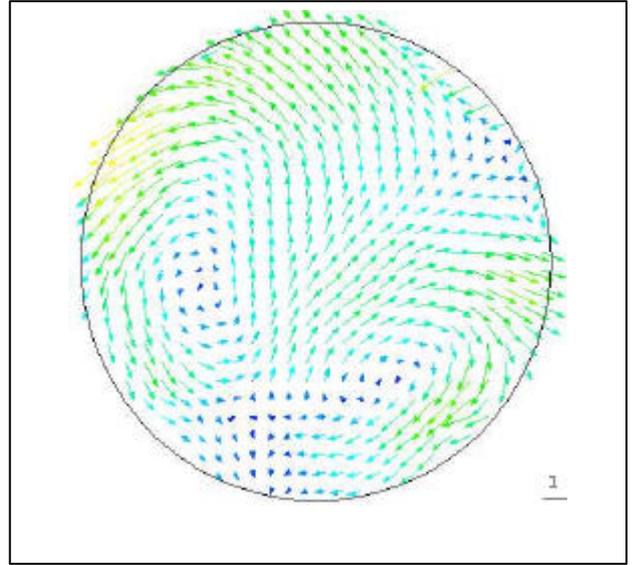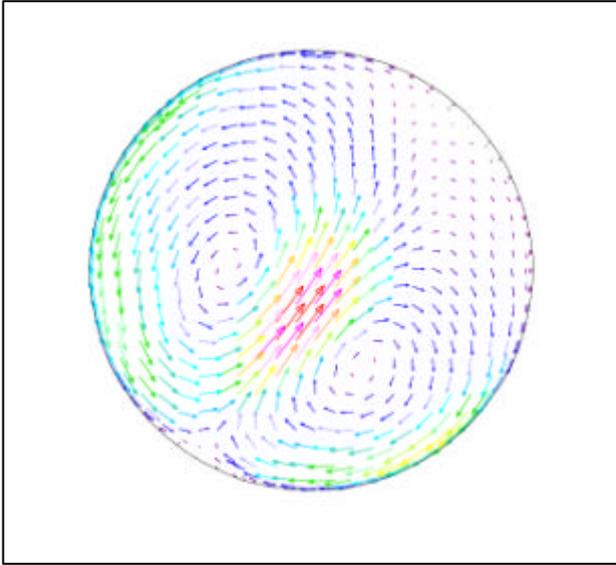# Figure 5 Velocities fields in hot leg 3
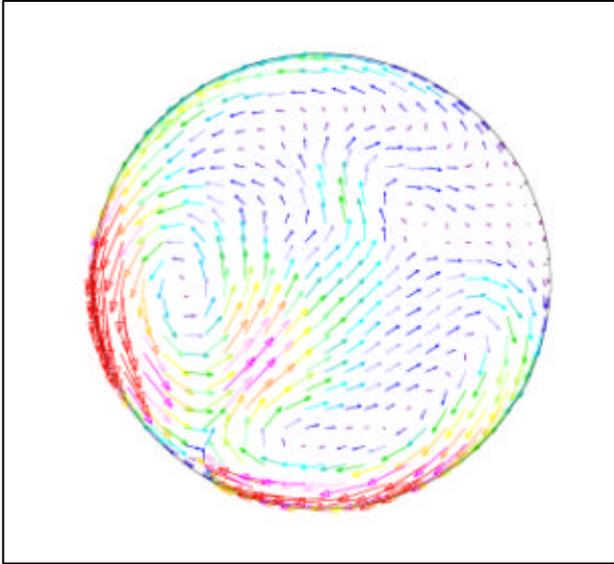
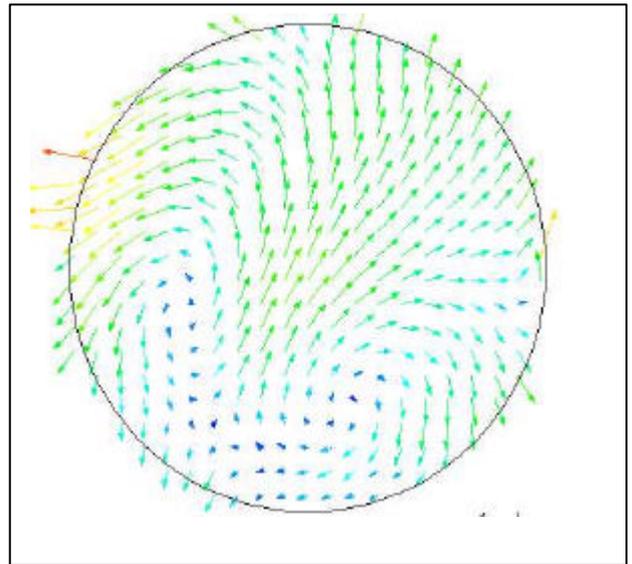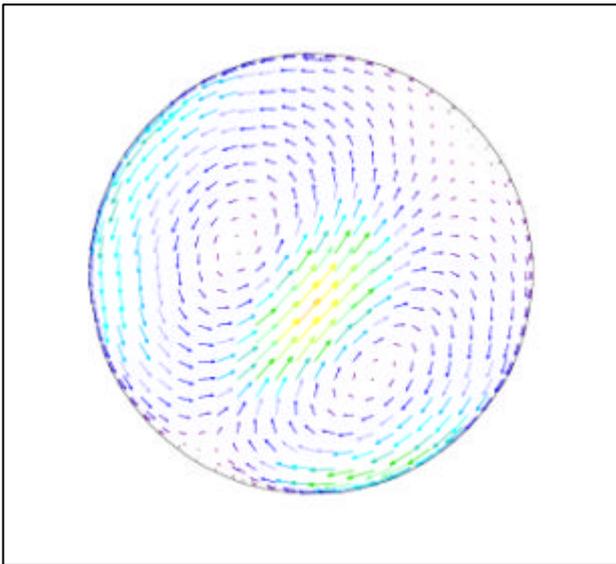**CFD RESULTS**

EXPERIMENTAL RESULTS

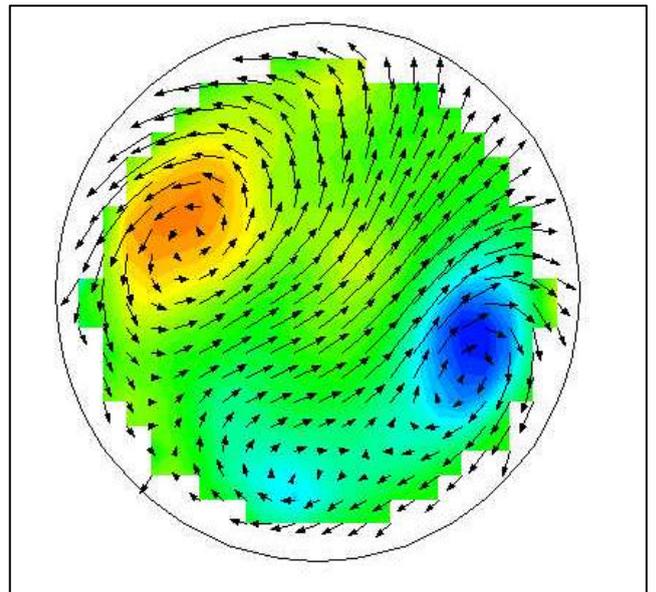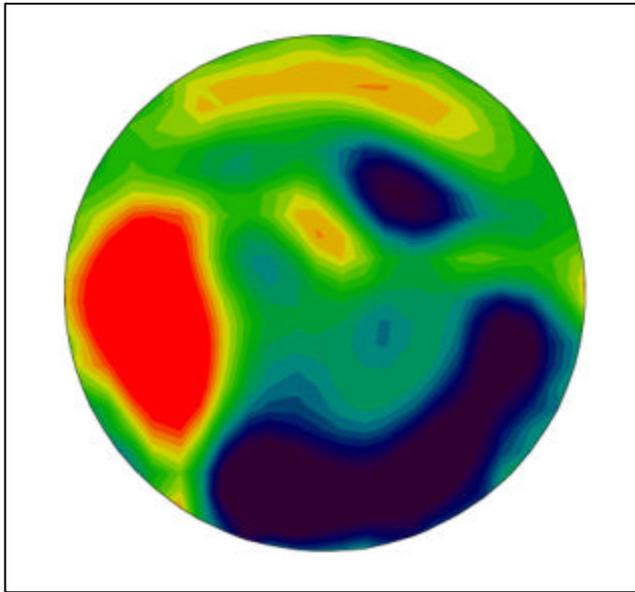**Hot leg entry**

1/3 of the hot leg

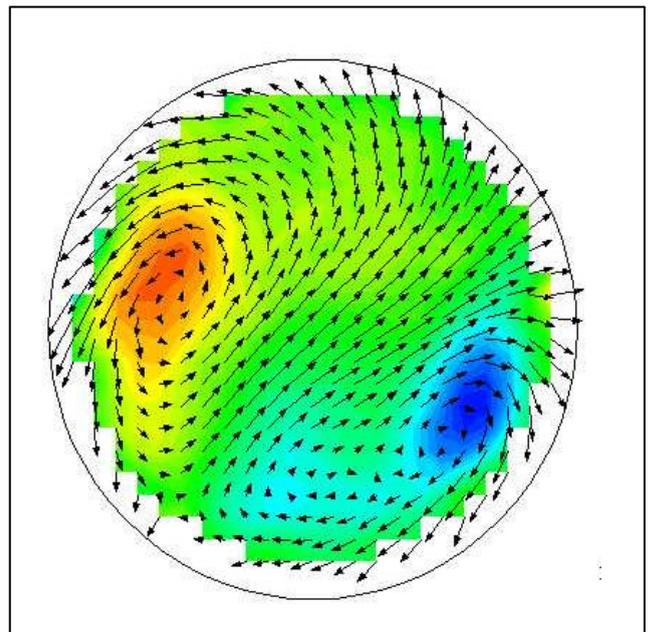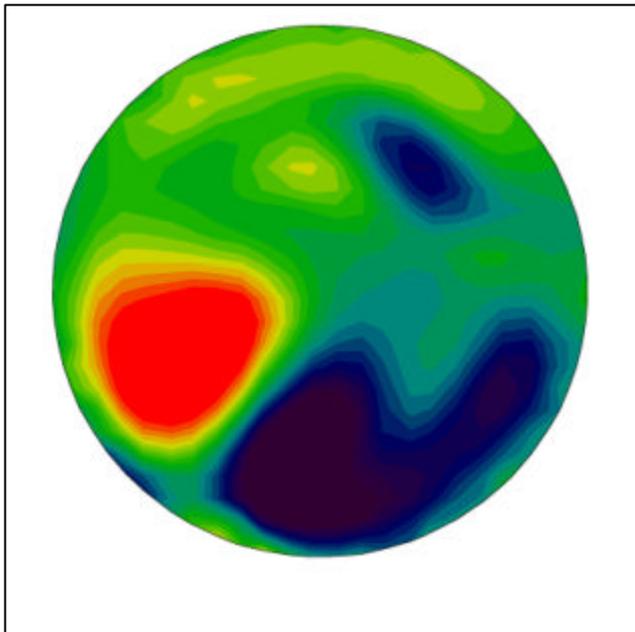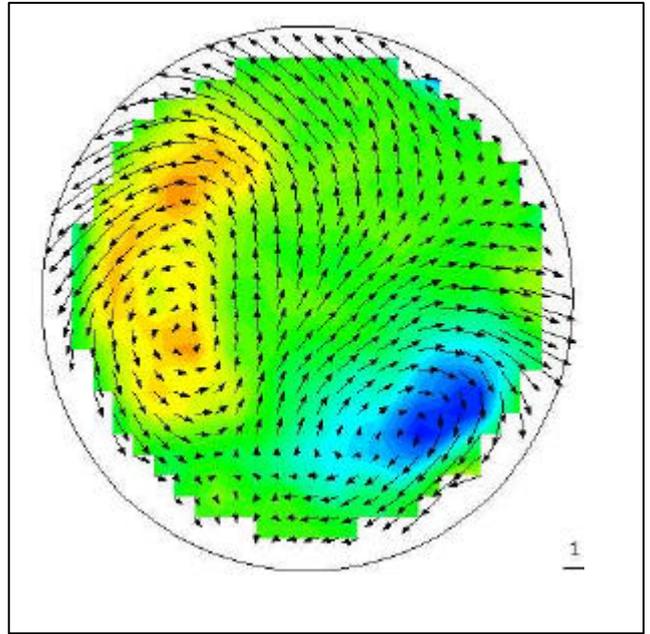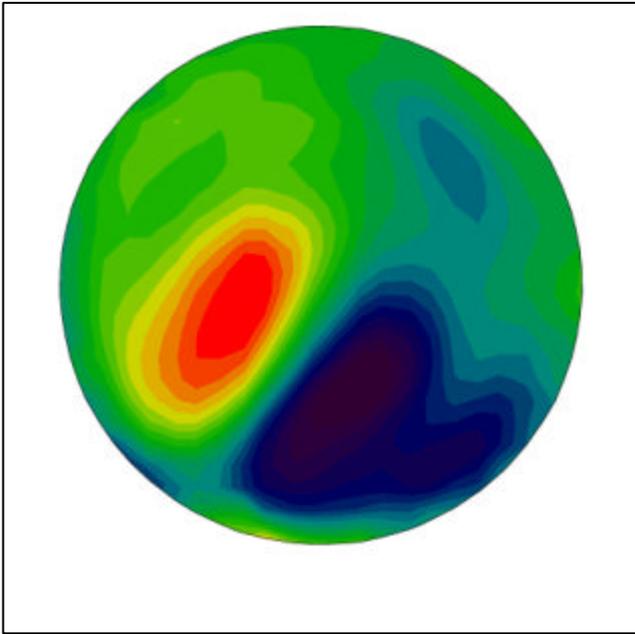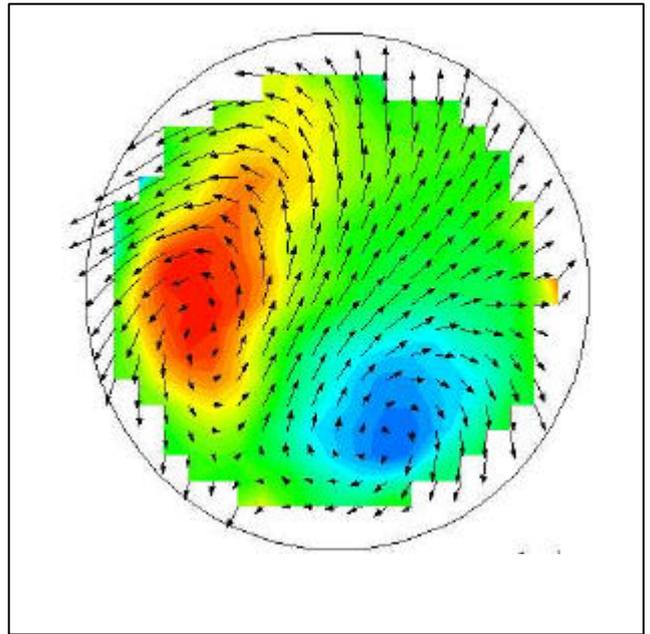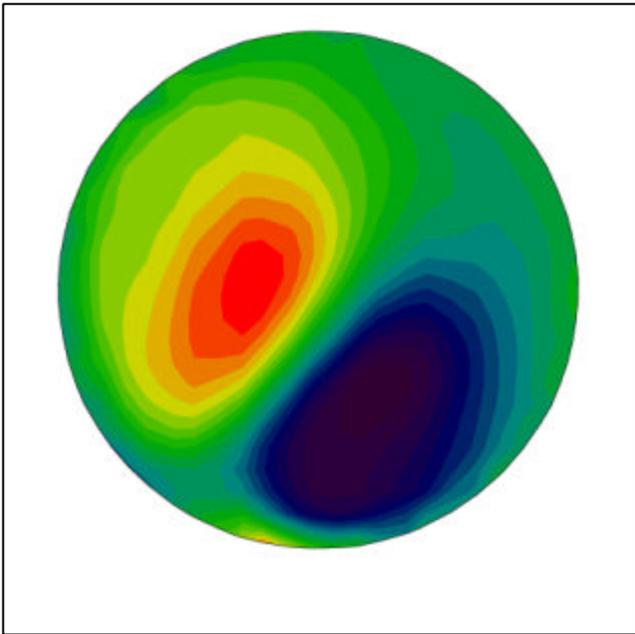**2/3 of hot leg**

End of the hot leg

Figure 6 Vorticity fields in hot leg 3

CFD RESULTS   **EXPERIMENTAL RESULTS**

Hot leg entry

1/3 of the hot leg

2/3 of hot leg

End of the hot leg