

## **WATER AND LEAD-BISMUTH EXPERIMENTS: FLUENT AND STAR-CD SIMULATION**

**Alberto Peña, Gustavo A. Esteban and Alberto Abánades**  
University of the Basque Country, Bilbao, Spain.

### **Abstract**

In the framework of the ASCHLIM project, a group of European experts in thermal hydraulic calculation has been clustered to evaluate the state of the art of CFD (Computational Fluid Dynamics) codes applied to ADS designs. The work of this group, supported by the Vth Framework Program of the European Union, has been focused on CFD analysis of the experiments that includes thermal hydraulic phenomenology associated to current transmutation design options. Some experimental data related to the MEGAPIE project (MEGAWatt Pilot Experiment) were analyzed. This paper includes the CFD calculations performed in our institution in that sense.

## Introduction

The University of the Basque Country has been involved in different CFD code calculations, validating them by comparing computational and experimental results. Sometimes, the simulations are post-test calculations, in which experimental data are available, and some other times, pre-test calculations have been performed in search of the best design configuration for each desired purpose.

Within the MEGAPIE project, a group of CFD code oriented experiments have been designed, wherein different cooling conditions of the spallation target window for a neutron source are tried. An important objective of the analysis focuses on describing the heat transfer viability of the particular window design prepared for MEGAPIE. At the same time, new measuring tools are being developed, and CFD code feasibility when the fluid is a heavy liquid metal (HLM) is being checked.

The experiments dealing with HLM are being planned to be carried out in the THEADES loop in the KALLA facility of the IKET (Institute of Nuclear and Energy Technologies) in the FZK in Germany. This activity called KILOPIE (KILOWatt Pilot Experiment) involves two experiments with different geometries and thermal conditions of the HLM.

In parallel to these HLM experiments, other experiments have been carried out with water using almost the same 1:1 geometry constructed in plexiglass. This experiment, called HYTAS, allows drawing relevant conclusions for the HLM case because the same Reynolds number is used apart from the same geometry scale. In addition, visualization techniques may be used, and corrosion or high operational temperatures are not present.

## HYTAS experiment

### *Experiment lay-out*

In Figure 1, the facility for the HYTAS experiment is depicted. A plexiglass module representing approximately a 1:1 scale of the MEGAPIE geometry was built for its use with water.

Figure 1. Facility for the HYTAS experiment (courtesy of Knebel et al, 2002)



Two different geometries were used for the experiment considering the presence and absence of a by-pass. This by-pass is a rectangular tube whose dimensions are  $20 \times 10 \text{ mm}^2$ , and it is used to avoid stagnation points in the fluid close to the window that may entail hot spots when using HLM. The mean flow rate is  $140 \text{ m}^3/\text{h}$  coming down an annular gap, and going up through an inner cylinder. When the by-pass is considered, the rate between mean flux and the flux through the by-pass is 1:15. More details may be checked in KNEBEL et al., 2002.

**Simulation set-up**

FLUENT is the CFD code used for these calculations. Only half of the geometry has been used due to the symmetrical lay-out of the display. However, the turbulence motion is a three dimensional phenomenon, and therefore there is no symmetrical flow in real problems. Figures 2 and 3 show the meshing for the HYTAS geometry without and with by-pass, respectively.

The meshing for the HYTAS experiment without the by-pass consists in 223723 mixed cells, generated by the GAMBIT code, and for the geometry with by-pass the number of cells is 165300. For both calculations, the turbulence model is the RNG  $k-\epsilon$  with the standard wall functions, and the discretisation scheme for the convective terms is the second order UPWIND.

The walls of the display are considered adiabatic. A water flux of  $140 \text{ m}^3/\text{h}$  through the annular gap has been considered in both cases. However, in case of the by-pass an additional flow of hot water equaling to  $9.3 \text{ m}^3/\text{h}$  through the rectangular tube has been taken into consideration, the temperature of the secondary flux being 50 K hotter than the main flux through the annular gap.

Figure 2. **Meshing for the HYTAS experiment without by-pass**

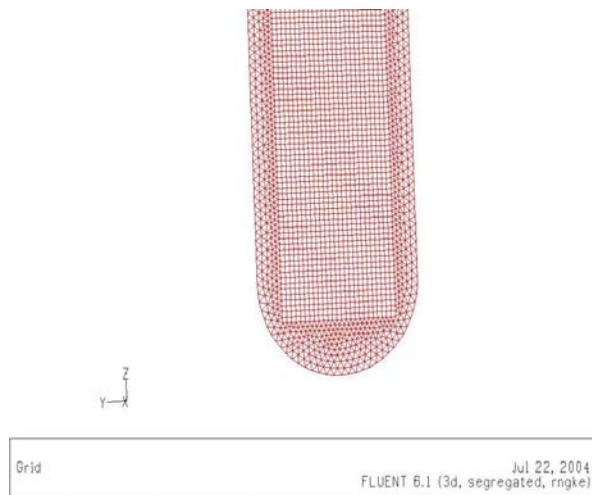
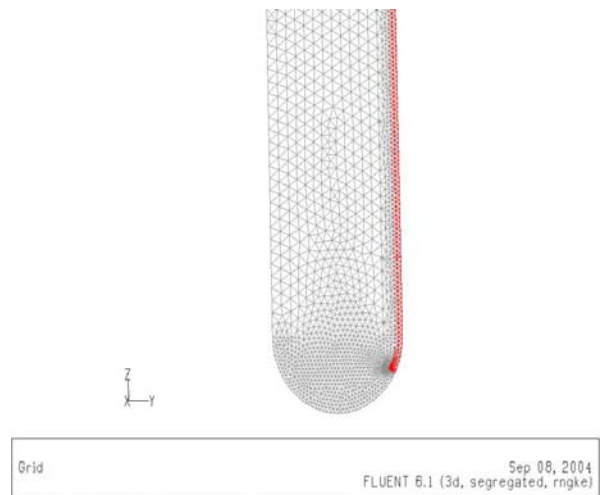


Figure 3. **Meshing for the HYTAS experiment with a by-pass**



## Results and discusion

Figures 4 and 5 show the velocity vectors in the window zone for both geometries. Calculations have been done with the stationary and transitory equations leading to the same results.

Figure 4. Velocity vectors in the symmetry plane. Geometry without by-pass.

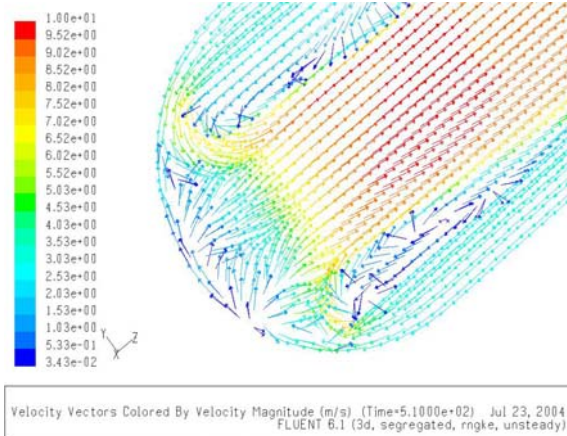
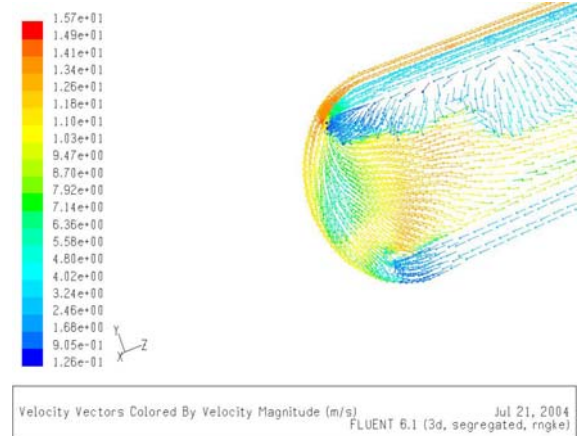


Figure 5. Velocity vectors in the symmetry plane with by-pass.

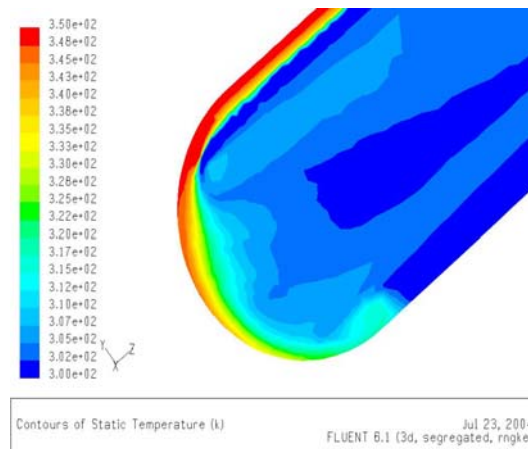


In Figure 4, there are some stagnant points near the window that do not appear in Figure 5. Stagnant points will be a source of hot points in the fluid that may involve problems of cooling when located close to the window. The window is the component that must be well cooled, because it is the zone where the proton beam is going to impact. As a consequence, the dangerous development of hot spots has been avoided with the bypass and the cooling efficiency of the window improved. In the experiment the stagnant points are not in a fixed position, though in the FLUENT calculations this is not achieved.

Although stagnant points disappear when there is a by-pass, this new element may provoke recirculation from the bypass upward into the main flux annular gap (Figure 5). The flow velocity in the by-pass should be checked to avoid these phenomena.

Figure 6 shows the temperature distribution in the HYTAS experiment CFD calculations, when there is a by-pass, and it can be seen that there are no hot-points foreseen.

Figure 6. Contours of temperature in the symmetry plane. Geometry with by-pass.



The two coming together streams mixture is very fast, as the temperature at the inlet of the inner cylinder is very homogenous.

### Heated jet experiment

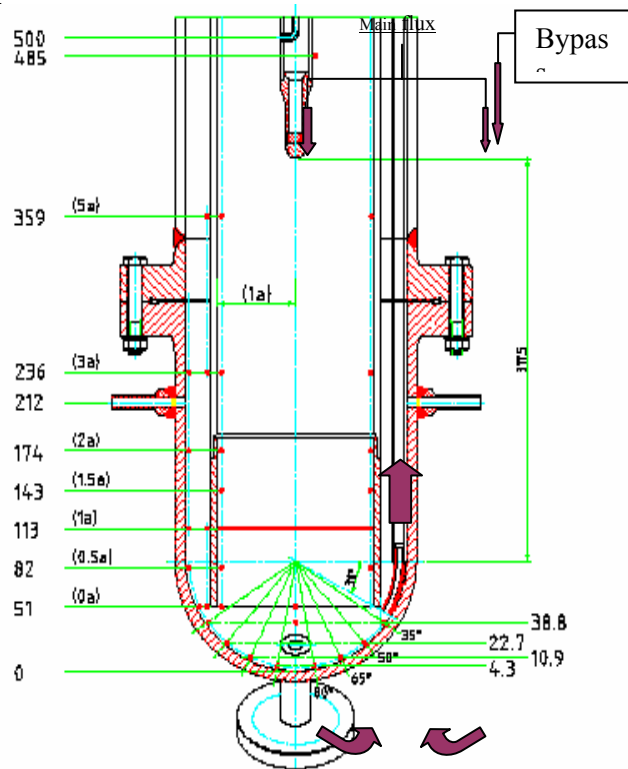
Using almost the 1:1 geometry as in HYTAS, other experiments are going to be performed with Pb-Bi in the THEADES loop in the KALLA laboratory (Figure 7).

Two experiments will be done in the mentioned loop:

- A turbulent mixture of two jets at different temperatures and flow rates.
- The cooling of a spallation target window with a similar geometry as the MEGAPIE, and with similar heat fluxes.

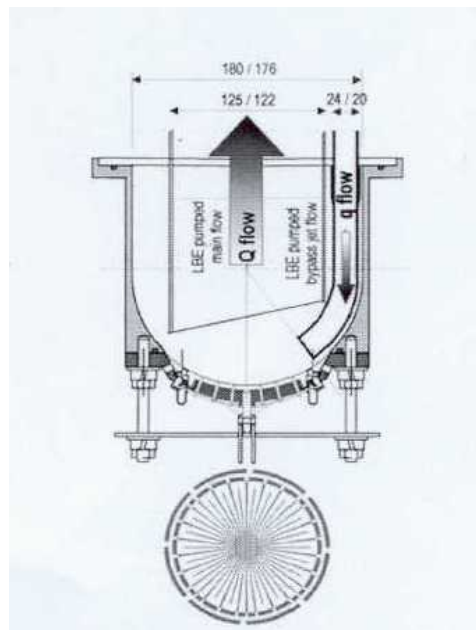
The first of the experiments is the so-called heated jet experiment, and the scheme of the facility is shown in Figure 7 and Figure 8. The difference between the schemes is the inner cylinder inlet configuration, a little bit slanted in Figure 8.

Figure 7. Facility for measurements. Numbers show the z coordinates, in order to fix the thermocouples position.



The maximum flow rate injected through the by-pass is  $4 \text{ m}^3/\text{h}$ , with a temperature of  $350 \text{ }^\circ\text{C}$ . The mean maximum flow rate is  $40 \text{ m}^3/\text{h}$  at  $300 \text{ }^\circ\text{C}$ , coming trough the annular gap (for more details see Knebel et al, 2002).

Figure 8. Second designed geometry, with the inner cylinder inlet slightly slanted. (courtesy of Knebel et al, 2002)



### ***Simulation set-up***

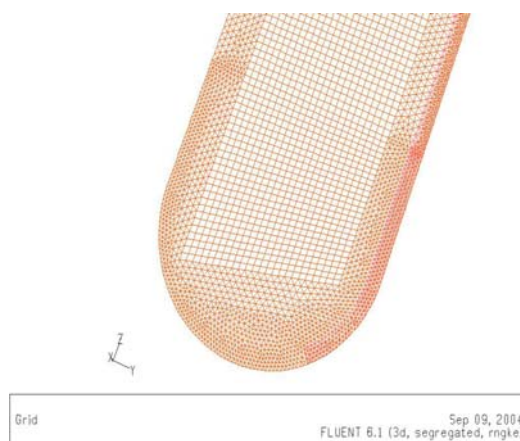
The calculations made for the configuration in Figure 7. were made with two CDF codes: STAR-CD and FLUENT. The domain used for the calculations with both codes is the same as the one used for the HYTAS experiment (Figure 3.).The boundary conditions for the simulation have been the following:

Main Flux:  $T^a$ : 300 °C and  $Q$ : 18 m<sup>3</sup>/h

Flux through the by-pass:  $T^a$ : 350°C and  $Q$ : 2 m<sup>3</sup>/h

Then, for the configuration in Figure 8 (inner cylinder inlet slightly slanted), the grid is the one in Figure 9, and calculations have been made with FLUENT. The number of cells is 386783, and the boundary conditions are the ones of the other simulation

Figure 9. **Heated jet experiment modelling grid. Slanted geometry.**



In both cases, the 3D geometry is only half of the total facility, the turbulence model is RNG k- $\epsilon$ , and the discretization scheme for the convective terms is first order UPWIND.

### ***Results and discussion***

In Figures 10 to 13, it can be seen the difference between the calculations of both codes. Calculations have been done for the configuration of Figure 3. Recirculation is foreseen, but in a different way if one code or the other is considered. The main reason for these discrepancies is the different treatment of the thermal boundary layer. The other models, the grid and the boundary conditions are the same.



Figure 10. Contours of temperature. FLUENT calculation. Heated jet experiment.

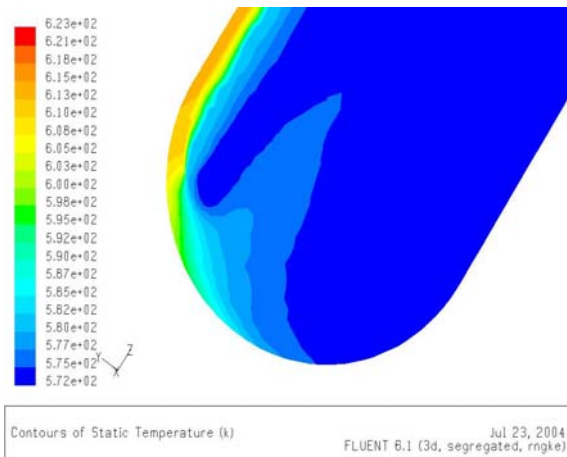


Figure 11. Contours of temperature. STAR-CD calculation. Heated jet experiment.

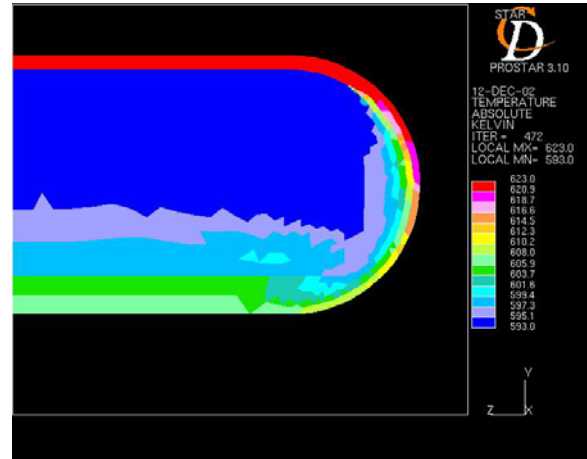


Figure 12. Velocity vectors. FLUENT calculation. Heated jet experiment.

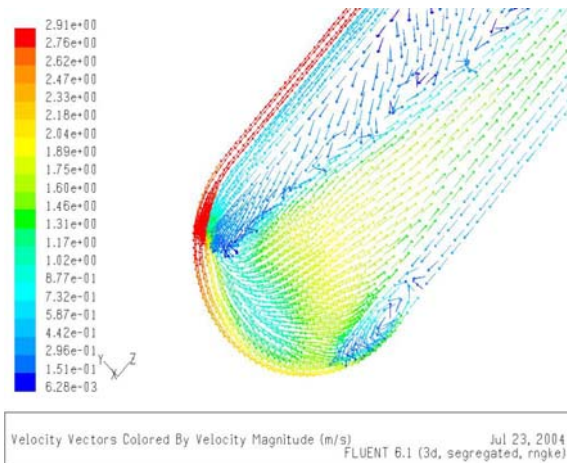
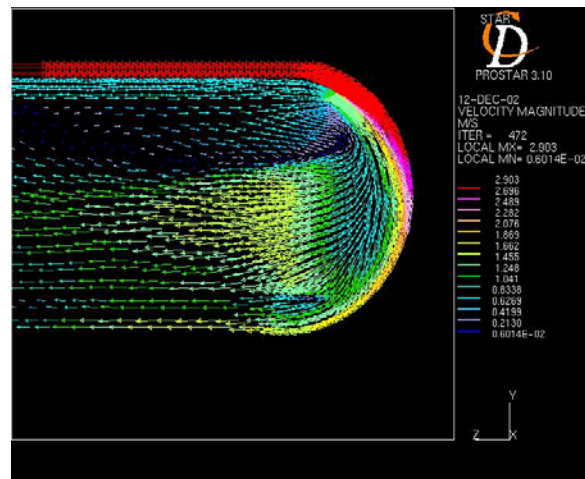


Figure 13. Velocity vectors. STAR-CD calculation. Heated jet experiment.

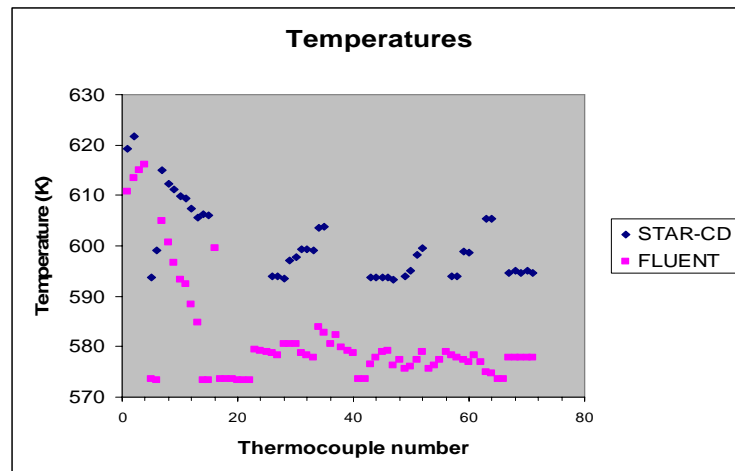


Comparing Figures 5 and 6 with 10 and 12, it can be highlighted that using water for experiments is a good idea for preliminary studies, but results with liquid metals are different, so, experiments with liquid metals are necessary. Recirculation zones are foreseen in different places for water and for liquid metals.

The comparison between FLUENT and STAR-CD temperature values can be seen in Figure 14, the STAR-CD ones being very high compared with the FLUENT ones. Heat transfer calculated by STAR-CD is smaller than the one calculated by FLUENT, and that is because of the way the codes consider the heat transfer coefficient to the wall. Previous experiences (ASCHLIM, 2003) show that the correlations for this coefficient in commercial codes must be changed from the default ones.



Figure 14. FLUENT and STAR-CD temperature values at the heated jet thermocouple positions



Also the change in the geometry configuration leads to different results (see Figures 15 and 16), although the recirculation problem is still without solution.

Figure 15. Contours of temperature. FLUENT calculations. Heated jet experiment. Slanted geometry.

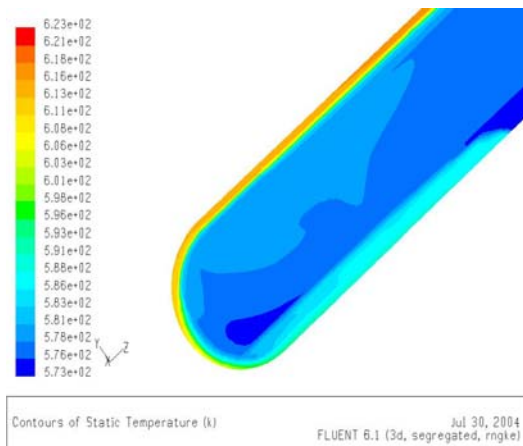
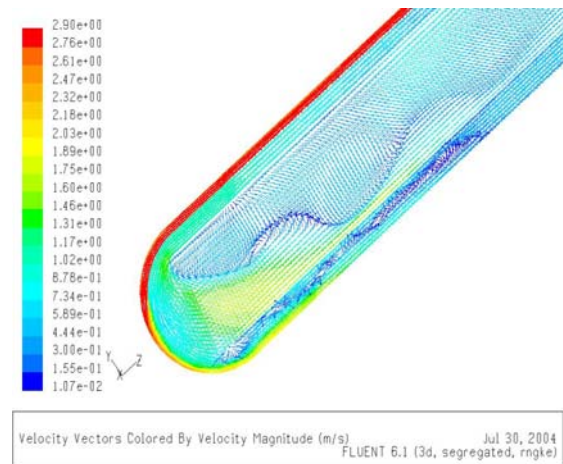


Figure 16. Velocity vectors. FLUENT calculations. Heated jet experiment. Slanted geometry.



## KILOPIE experiment

### Experiment description

This is the second experiment to be done in the THEADES loop. The main objective is to analyse how the target window can be cooled, with similar heat fluxes as MEGAPIE.

The difference between the turbulent mixed experiment and this one has been mentioned in another section. The bottom hemisphere is dismantled and it is substituted by a surface heating system that will provide a heat flux of  $140 \text{ W/cm}^2$ . The scheme for the experiment is the one shown in Figure 8. The flow rates through the annular gap and through the by-pass are the same, but the inlet temperature is  $180 \text{ }^\circ\text{C}$  for both fluxes.

### Simulation set-up

The calculations have been done with FLUENT, there is no benchmark exercise available though. Therefore, the calculations could be used as pre-test calculations in order to understand what is going to happen in the experiment, and could help to explain the results, once they are available.

The geometry depicted in Figure 9 was the one used for the heated jet experiment, and also for this KILOPIE experiment. The boundary conditions, as well as the physical models, are the same as in the other experiment, except for the heat flux input. The inlet temperatures for the fluxes are 180 °C, and there is a heat flux boundary condition in the wall of the hemisphere, with a value of 140 W/cm<sup>2</sup>.

### Results and discussion

Figures 17 to 19 show the temperature contours and the velocity vectors for this KILOPIE experiment.

The velocity field is very similar to the one in the heated jet experiment. Recirculations cannot be avoided with these configuration and boundary conditions, unless another by-pass is considered or the flow rate is changed.

Figure 17. **Temperature contours at the symmetry plane. KILOPI experiment**

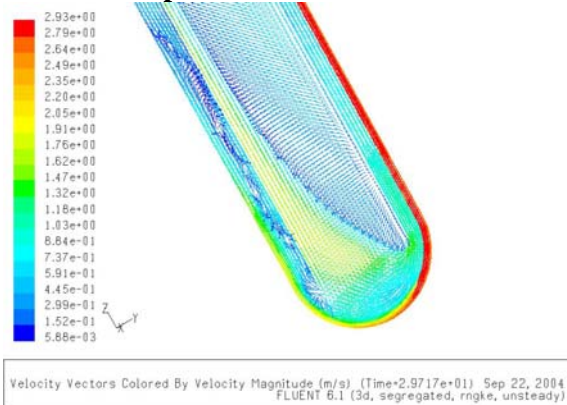


Figure 18. **Temperature contours at the hemiespheric plane.**

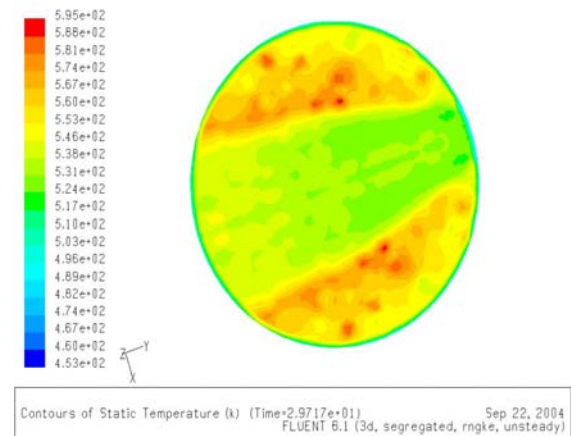
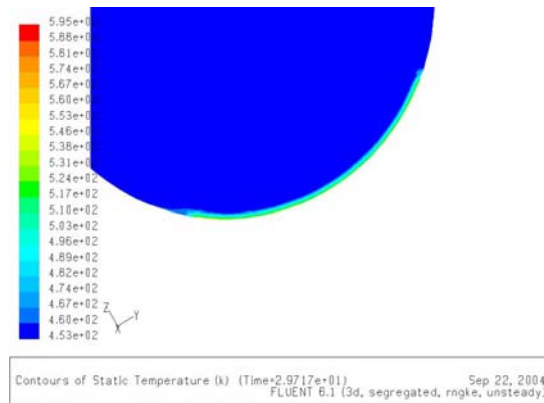


Figure 19. **Temperature contours at the window. Symmetry plane.**



There are no stagnant points and the maximum temperature is given at the window (595 K). It can be observed that there is almost an instantaneous cooling of the device, because not very far from the window the temperature is again the inlet one (453 K).

A very effective mechanism to cool the window is the by-pass flow. In Figure 18, the coldest zone of the window is due to the by-pass flux flowing over it. Having a by-pass section that expands the flux could improve the thermal conditions of the window.

## Conclusions

Some recirculation is foreseen in the experiments with liquid metals and water. STAR-CD temperatures are higher than the FLUENT. The discrepancy is thought to be due to the different near wall treatment employed by both codes for the thermal boundary layer behaviour.

The codes do not see major differences using different inlet region configurations, and although the cooling of the window seems to be guaranteed (no hot points are foreseen), the flow pattern is not properly driven into the inner tube.

Mass flow rates are higher in the heated jet experiment compared with the HYTAS experiment with water. This can contribute to the better flow patterns obtained. Fewer critical points appear in the heated jet experiment calculations, so it is a good result in order to continue with the experimental set-up.

## REFERENCES

- [1] J.U. Knebel, M. Daubner, S. Gnieser, R. Stieglitz (2002), *R. Water and Lead-Bismuth Experiments for Megapie*, FZK, 2002.