ISTITUTO NAZIONALE DI FISICA NUCLEARE

Sezione di Genova

INFN/TC-95/09
15 Febbraio 1995

M. Olcese:

WATER DYNAMICS INSIDE THE COUNTING TEST FACILITY (CTF) TANK AT THE GRAN SASSO UNDERGROUND LABORATORIES

SIS-Pubblicazioni
dei Laboratori Nazionali di Frascati

INFN - Istituto Nazionale di Fisica Nucleare
Sezione di Genova

INFN/TC-95/09
15 Febbraio 1995

WATER DYNAMICS INSIDE THE COUNTING TEST FACILITY (CTF) TANK AT THE GRAN SASSO UNDERGROUND LABORATORIES

M. Olcese
INFN–Sezione di Genova, Università di Genova, via Dodecaneso 33, I–16146 Genova

Abstract

The results of a convective phenomena analysis of a liquid inside a large vessel with few flow obstacles show that a significant natural circulation can occure even with very small temperature gradients.

I have been in charge of analysing the convective fluid motion inside a cilindrical tank (10m in diameter, 10 m in height) and I think that should be useful to report in details the calculation approach and the results.

1. - INTRODUCTION

A fiesibility test (called CTF) of the proposed BOREXINO (1) detector to be built in the future, is now nearly ready to start at the Gran Sasso Underground Laboratories in Italy.

The BOREXINO experiment is expected to be able to observe neutrinos coming from the sun using a ν-e as a detection reaction. The detection medium will be a liquid scintillator contained in a spheric vessel several meters in diameter positioned at the center of a large cilindric tank full of water.

The CTF test facility (see fig. 1) is a small scale of the proposed BOREXINO detector. The scope of CTF test is to check the possibility to reach both in the scintillator and in the water (which works as a γ shield) ultrahigh radioactive purity in order to allow the detection of approximately 80 events (ν-e scattering reactions) per day predicted for BOREXINO.

A clean-up system is used to remove the radioisotopes from the water while it is also very important to minimize the effects due to the radon generated on the tank walls and transported by the water towards the inside vessel containing the scintillator. For this reason it is important to study the water flow patterns inside the tank and the possibility to control them avoiding as much as possible the above transport phenomena of radon.

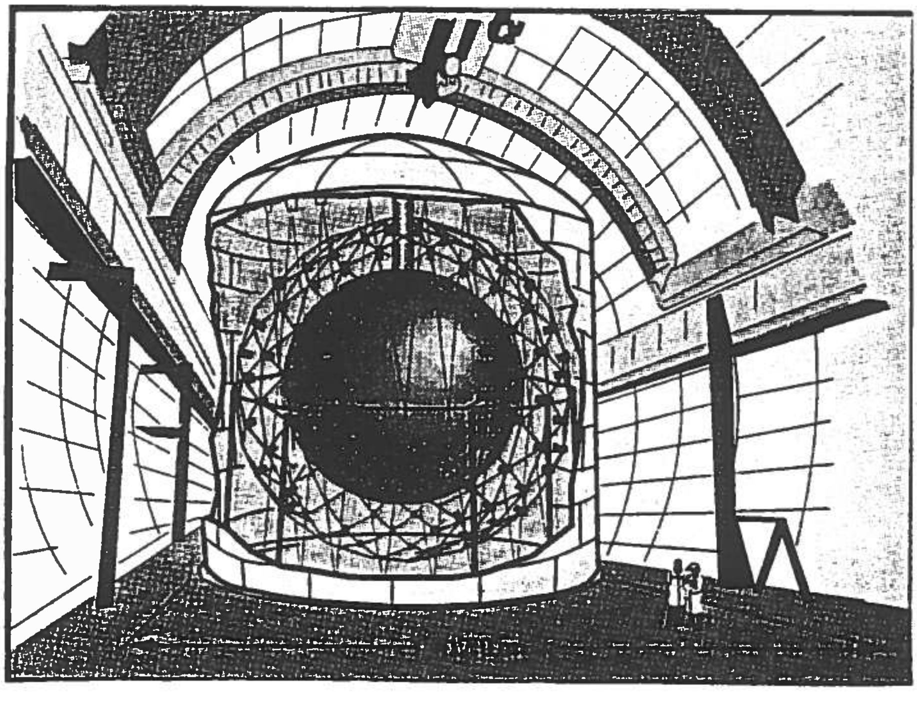

FIG. 1 - Conceptual design of CTF.

2. - CTF FLUIDODYNAMIC SYSTEM

The geometry of CTF tank is shown in figure 2. The tank is a big cilinder made by many steel plates welded together and clothed on the inside surface by a thin layer of plastic compound (PERMATEX).

There are two feedwater ring at the bottom and one outlet ring at the top. The water inside tank has an upper free surface with nitrogen small overpressure. The water is ricirculated in a closed loop consisting, outside the tank, of a pumping, deionizing and demineralizing system. The outside surface of the tank is in touch with room air. The tank foundation consist of different layers of steel and concrete with at the very bottom a draining pabble with a constant temperature water flow.

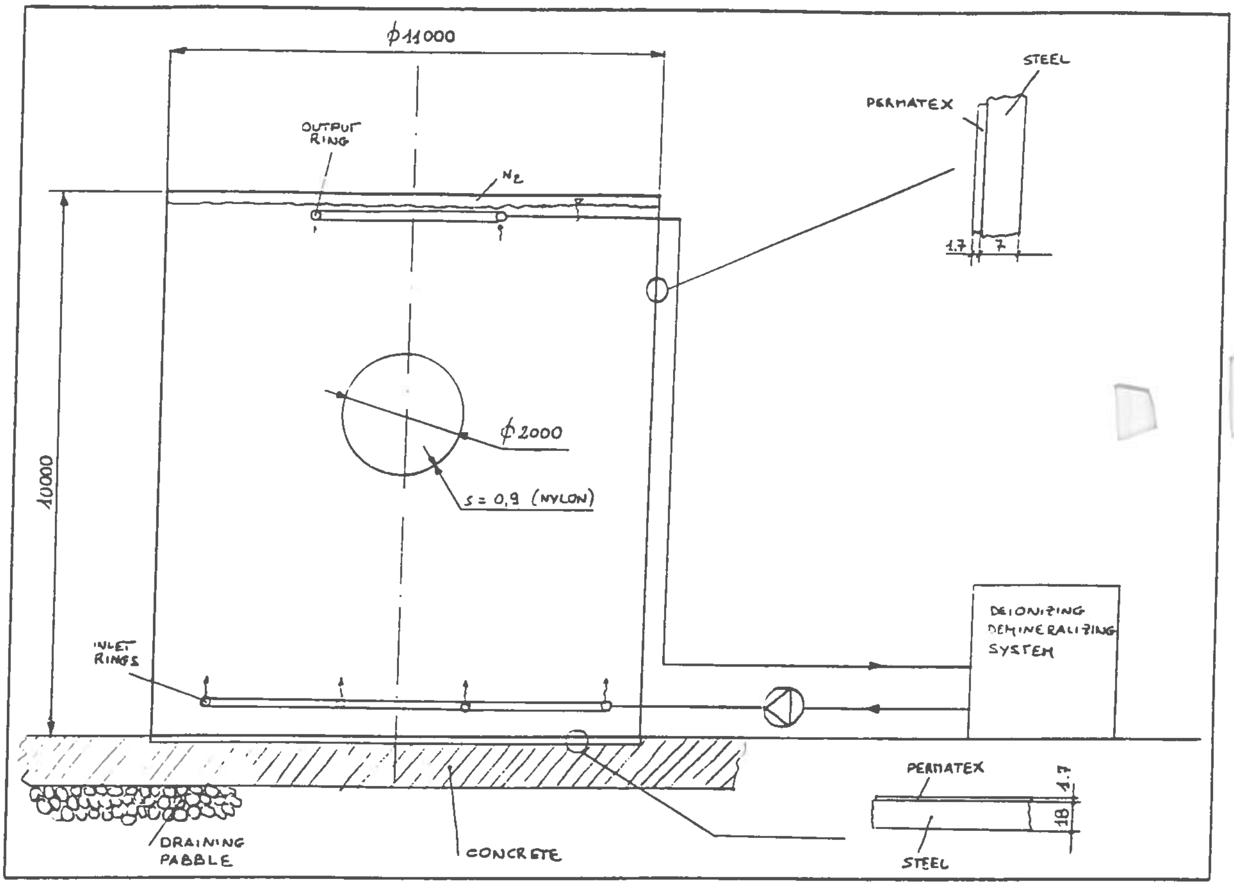

FIG. 2 - Geometry of CTF tank.

3. - THE BASIC FLUIDODYNAMICS

Neglecting the heat exchange through the tank walls and supposing that the water motion is only governed by the forced circulation imposed by the feed water flow rate (about 3 m^3/h) a Reynolds number of about 50 can be calculated inside the tank: the flow regime should be fully laminar. In these conditions the water streamlines should be quasi-vertical with the exception of the region adjacent to the inside parts (central vessel, photomultiplier assemblies.

Heat transfer and feed water energy input causes buoyancy phenomena to appear which overlap a natural circulation on the forced convection: the result is that the mass and heat transport mechanism become much more complex.

It is well known that the forced convection is governed by the Reynolds (Re) number (viscous forces) while the natural convection by the Grashof (Gr) number (buoyancy forces). The ratio Gr/Re^2 is the parameter that allows a good prediction of the flow regime (2):

. $Gr/Re^2 < 1$ forced convection

. $Gr/Re^2 > 1$ natural convection

With a so small (50) Reynolds number it can be calculated that, even very small thermal gradients (less than 1°C) results in $Gr/Re^2 > 1$ that is in natural convection flow regime.

It is well known (2) that the transition between laminar and turbolent flow occurs when Re=2200 in isothermal flows, while such limit decreases when natural convection phenomena appears. In a geometry like CTF tank it is difficult to evaluate if the flow is fully laminar or if turbolence plays a significant role.

In order to have a realistic prediction of the flow patterns inside the tank a computer simulation has to be performed due to the complexity of the phisical problem. It is also difficult to understand the distribution of the turbolence energy inside the tank. For this reason a computer code with a k-ε (2 equations) model of turbolence has been used to perform the simulation.

4. - "FLOTRAN" COMPUTER CODE

The FLOTRAN (3) computer code solves the basic conservation equations (mass, momentum, energy) and two additional transport equations for the kinetic energy of turbolence (k) and its dissipation rate (ε) which will be used to evaluate the eddy viscosities and diffusivities in a standard k-ε model.

The k-ε model of turbolence has gained considerable popularity and is one of the most widely used turbolence model in commercial software. It has shown successfull performances in simulating a huge class of turbolent flows involving both near-wall and free-shear flow phenomena. Its basic assumption is that turbolence behaves much like molecular diffusion, and momentum/heat exchange arising from turbolence can be modeled through local addition of "extra" viscosity/diffusivity termed the turbolent eddy viscosity and diffusivity.

5. - BASIC CALCULATION ASSUMPTIONS AND BOUNDARY CONDITIONS

In order to avoid the major complexity of a fully three dimensional simulation, the problem has been considered to be axisimetric.

There have been neglected (due to their small volume compared with the total water volume) the influence of the inside support structure and of the photomultipliers assemblies themselves (this is a conservative assumption since leads to maximize the flow velocities).

The input (output) flow rate has been assumed to came out from two rings (one at output) with only one axisymetric orifice equivalent (same cross section area) to the serie of small holes.

The mesh used to perform the analysis is shown in fig 3. A particular care has been applied when modelling the near-wall region within the boundary layer. The finite element model has been also extended to simulate the different layers of the bottom and side walls together with the nylon central thin vessel with scintillator fluid inside.

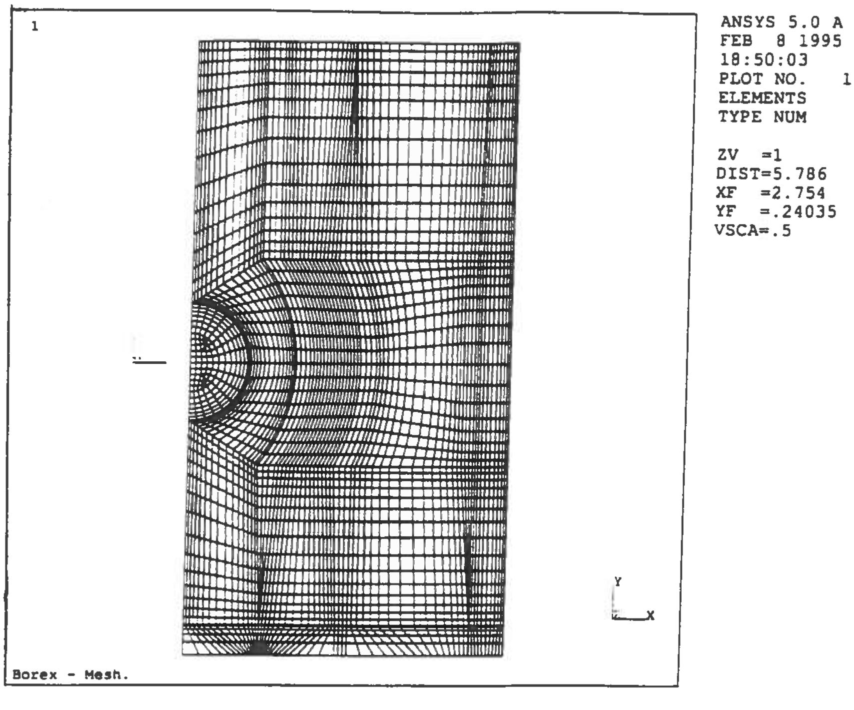

FIG. 3 - FLOTRAN discrete model: mesh.

The major problem consists in applying the appropriate boundary conditions. A constant uniform heat exchange coefficient has been applied on the outside surface of the side wall, calculated assuming free convection in air through the following formula (2):

$$h = 0,131 \frac{(\rho 2 g \beta \Delta T/\mu^2)^{1/3}}{Pr^{1/3}} \qquad (a)$$

where:

ρ	is the fluid density
g	is the gravity acceleration
β	is the fluid thermal expansion coefficient
ΔT	is the wall-bulk temperature difference
μ	is the fluid dynamic viscosity
Pr	is the Prandtl number of the fluid

valid for wide vertical surfaces in free convection. Assuming a ΔT of 5°C, which has been proved by the calculation, the equation (a) gives h=2 $W/(m^2 {}^\circ C)$.

The top surface of the fluid has been considered to be adiabatic, such an assumption is valid since the heat exchange between the inside water and the upper thin layer of nitrogen can be neglected if compared with the heat exchange through the walls. On the very bottom surface of the foundation concrete a constant uniform temperature of 5°C has been imposed, which is the mean temperature of the water flowing through the draining pabble.

The feed water inlet temperature depends on the amount of heat exchange in the closed loop of the demineralizing and deionizing systems. Due to the complexity of such a system and to the small amount of the loop flow rate if compared to the total water volume in the loop part outside the tank, totally installed in the same room of the tank, it can be properly assumed that the water reaches the equilibrium with the room air temperature (assumed to be 16,5°C, that is the mean value detected in the experimental hall) before entering the tank.

6. - RESULTS OF CALCULATIONS

A steady state run of FLOTRAN code based on the finite element model and with the boundary conditions above described has been performed. Figure 4 plots the fluid velocity vectors within each cell of the calculation frame. Figure 5 plots the fluid stream lines.

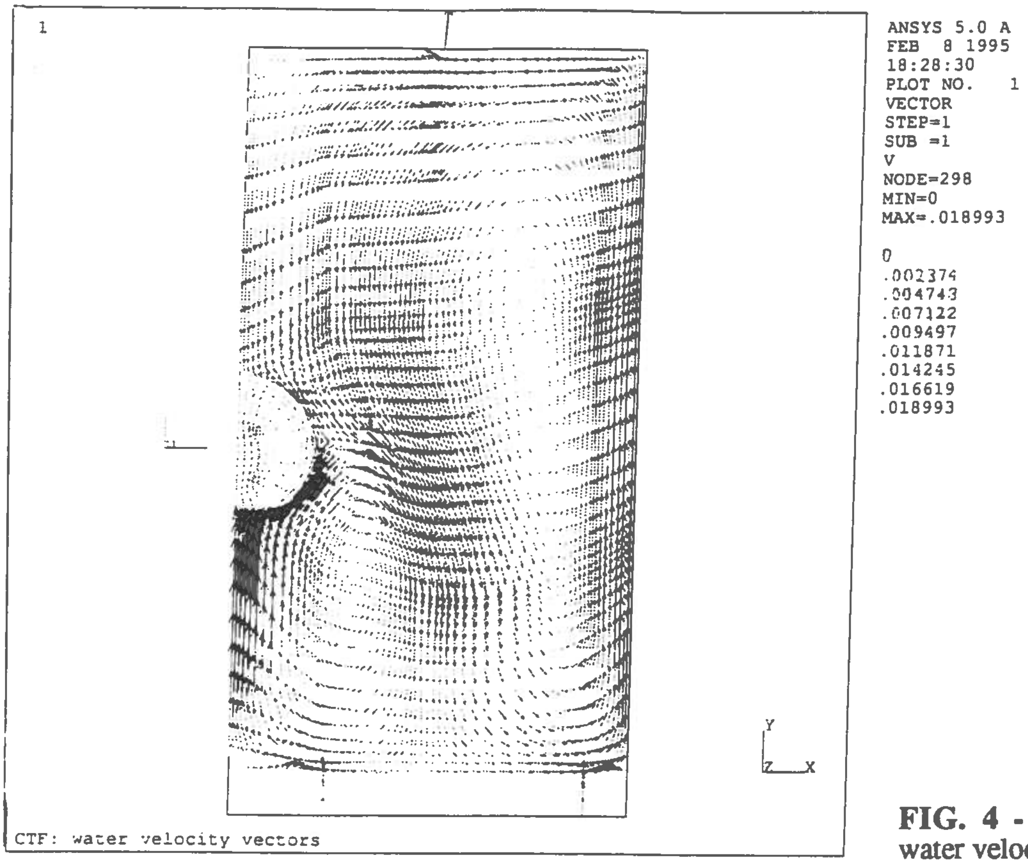

FIG. 4 - CTF tank water velocity vectors.

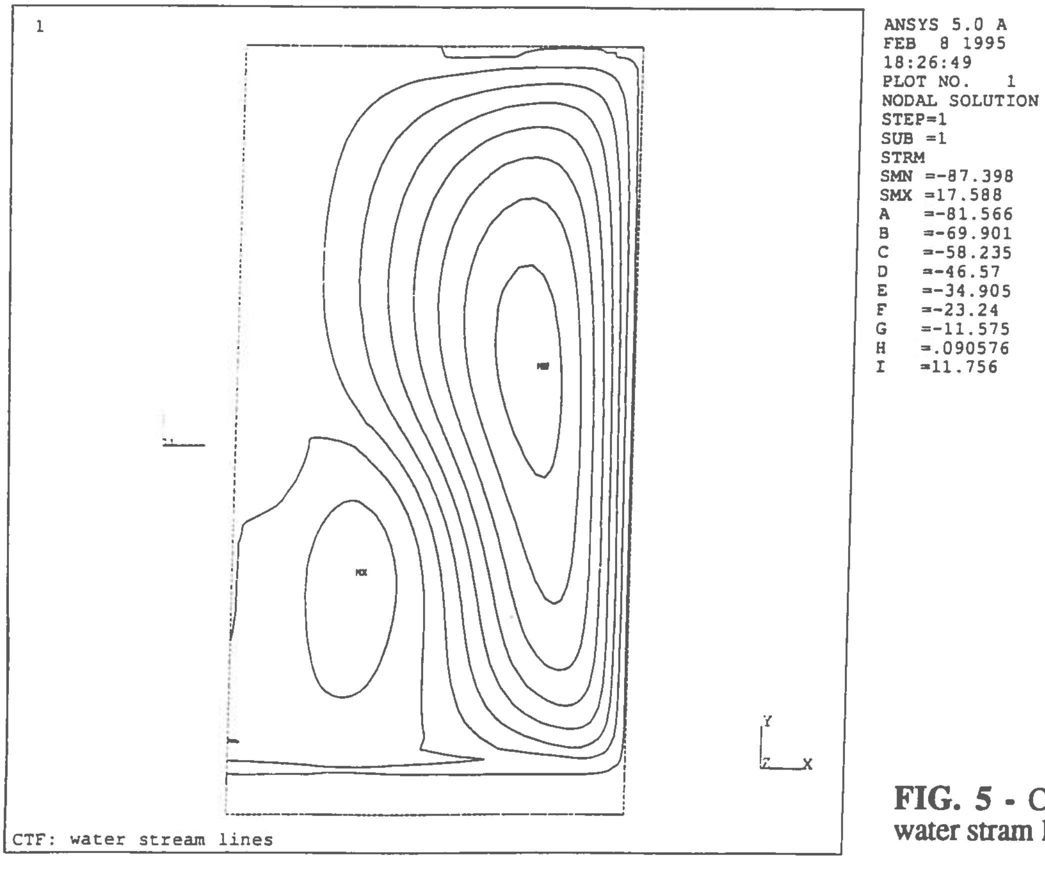

FIG. 5 - CTF tank: water stram lines.

It can be observed that two steady state vortexes dominate the fluid pattern, feeded by both the buoyancy forces due to the water heating in the near wall region and the inlet feed water temperature higher than the mean inside. Figure 6 plots the fluid velocities inside the central vessel; it can be pointed out the induced toroidal circulation due to the thermal gradients on the outside surface of the vessel.

The main topics of the results are the following:

- the mean inside temperature is equal to 10 °C and it is almost uniform due to the mixing effect of the turbolence

- the total side wall temperature gradient (ca. 6°C) is concentrated in the air (the film resistance is dominant)

- the total bottom temperature gradient (ca. 5°C) is concentrated in the concrete (which thermal resistance is dominant)

- the maximum velocity (1,8 cm/s) occurs in the region below the central vessel, as a conseguence of the density gradient between the input water and the water inside.

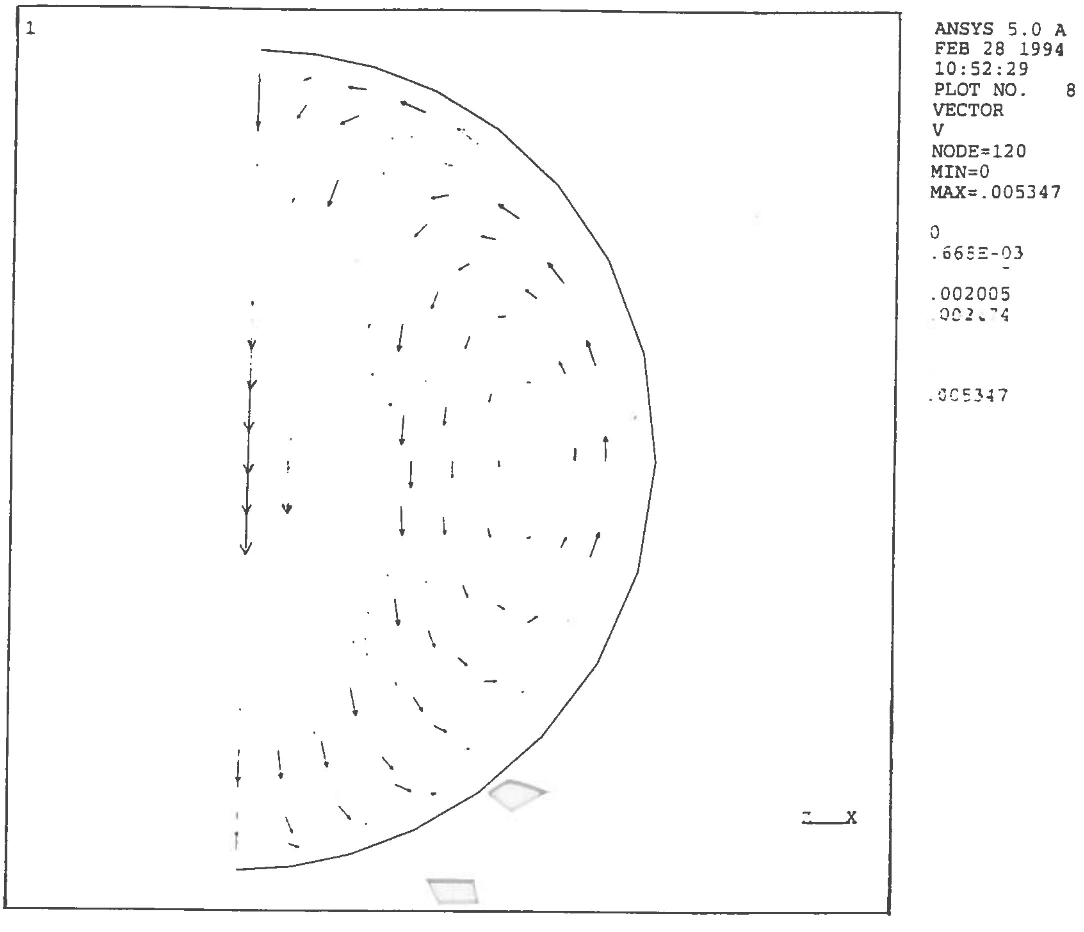

FIG. 6 - CTF central vessel: scintillator velocity vectors.

7. - CONCLUSIONS

Even if this analysis has been performed through a computational fluid dynamic computer code which simulates the turbolence effects with a well proved k-ε model several calculation assumptions and simplyfying hypothesis has been made.

For these reasons the present analysys has to be considered to be very preliminar. A more accurate simulation of the water fluidodynamic inside the CTF tank will be possible as far as some input data will be fixed as a result of the experimental measurements. However as preliminary conclusion we can state that even with small thermal gradients the natural circulation inside the tank is significant (fluid velocities with order of magnitude of cm/s appears) and dominant on the forced convection.

The flow patterns are quite complex and as a function of the boundary conditions several steady state vortexes appears.

REFERENCES

(1) G.Bellini, M. Campanella, D. Giugni, R. Raghavan, "Borexino at Gran Sasso: Proposal for a real time detector for low energy solar neutrinos" (1991).

(2) Frank Kreith, "Principles of Heat Transfer" (1973).

(3) FLOTRAN Computationally Efficient Finite Element Software for Fluid Flow and Heat Transfer Problems - COMPUFLO, Inc. (1988).