

# COMPUTATIONAL SIMULATION OF THE NATURAL CIRCULATION OCCURRING IN AN EXPERIMENTAL TEST SECTION OF A POOL TYPE RESEARCH REACTOR

**Francisco R. T. do Nascimento, Carlos A. S. Lima Junior, André F. S. de Oliveira, Renato R. W. Affonso, José L. H. Faccini, Maria L. Moreira<sup>1</sup>**

Instituto de Engenharia Nuclear – IEN/CNEN  
Rua Hélio de Almeida, 75 - Cidade Universitária, Ilha do Fundão  
21941-906, Rio de Janeiro – RJ

[rogerio.tdn@gmail.com](mailto:rogerio.tdn@gmail.com)  
[souzalima\\_ca@ien.gov.br](mailto:souzalima_ca@ien.gov.br)  
[oliveira.afelipe@gmail.com](mailto:oliveira.afelipe@gmail.com)  
[raoniwa@yahoo.com.br](mailto:raoniwa@yahoo.com.br)  
[faccini@ien.gov.br](mailto:faccini@ien.gov.br)  
[malu@ien.gov.br](mailto:malu@ien.gov.br)

## ABSTRACT

The present work presents a computational simulation of the natural circulation phenomenon developing in an experimental test section of a pool type research reactor. The test section has been designed using a reduced scale in height 1:4.7 in relation to a pool type 30 MW research reactor prototype. It comprises a cylindrical vessel, which is opened to atmosphere, and representing the reactor pool; a natural circulation pipe, a lower plenum, and a heater containing electrical resistors in rectangular plate format, which represents the fuel elements, with a chimney positioned on the top of the resistor assembly. In the computational simulation, it was used a commercial CFD software, without any turbulence model. Besides, in the presence of the natural circulation, a laminar flow has been assumed and the equations of the mass conservation, momentum and energy were solved by the finite element method. In addition, the results of the simulation are presented in terms of velocities and temperatures differences, respectively: at inlet and outlet of the heater and of the natural circulation pipe.

## 1. INTRODUCTION

The issue of safety at nuclear power plants is a primary factor to allow that nuclear energy can be consolidated as a reliable alternative for power generation. Thus, many studies have been conducted in order to improve safety in nuclear reactors. These studies can be done by the construction of test sections, which can correctly reproduce part of the dynamics involved in a nuclear power plant operating in extreme condition. Experimental facilities are necessary to allow estimates about the actual prototypes, safely and at a relatively low cost. On the thermo-hydraulics point of view, many studies are focused on a greater understanding of the natural circulation phenomenon. This phenomenon appears as a feasible alternative to assign a higher level of security to the reactor core when the heat removal system fails, but it can also be used during the normal reactor operation. In reactors such as AP600 / AP1000, natural circulation plays an important role in the passive system of residual heat removal. This system is responsible for the removal of the decay heat from the reactor core when the water supply system fails [1]. Another application of the system described above is found in the Australian OPAL research reactor. In such a reactor, the natural circulation is used for

physics experiments and during refueling [2]. Basically, the main difference between these two reactors is that the AP600 / AP1000 present pressurized cores while OPAL does not, since it is a pool type reactor. In both reactors, the formation of a thermo siphon occurs, when they are submitted to natural circulation conditions, which causes a displacement of the working fluid from the hottest regions to the colder ones, resulting in a heat removal from the hottest regions. When this phenomenon is established, it has a heat exchange cycle that does not depend on any external agent, for example, a pump. From this moment, the velocity fields control depends only on the phenomenon and any variation in the velocities should be attributed to buoyancy forces, which can be related with temperature variations in the system. Thus in order to simplify the study of natural circulation in a test section, it will be analyzed the behavior of different dimensionless numbers, as Grashof (Gr), Reynolds (Re) and Richardson (Ri) and of different parameters, as the temperatures and velocity fields in the flows, by using a computational simulation method with a CFD software, without turbulence model.

## 2. MATHEMATICAL FORMULATION

In order to simulate the thermo-hydraulic behavior of experimental test section, the CFD software used presents several models to solve the Navier-Stokes equations, including turbulence models. However, the computational costs of their application become very high due to the increased the amount of elements assigned to this problem. Therefore, in the present work, two models were used: one to solving the energy equation and other to considering the mass and momentum conservation equations.

### 2.1 Laminar Flux and Heat Flow

To solve the mass and momentum conservation equations, it was used the laminar flow model, where a laminar flow and a Newtonian incompressible fluid is assumed. This model can be defined by the equations:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + \nabla \cdot (\mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) - \frac{2}{3} \mu ((\nabla \cdot \mathbf{u}) \mathbf{I}) + \mathbf{F} \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2)$$

where:

- $\rho$  is the fluid density ( $\text{kg/m}^3$ );
- $\mathbf{u}$  is the velocity vector ( $\text{m/s}$ );
- $P$  is the pressure (Pa);
- $T$  is the transposed matrix of  $\nabla \mathbf{u}$
- $\mathbf{I}$  is the identity matrix
- $\mathbf{F}$  is the volume force vector ( $\text{N/m}^3$ );
- $\mu$  is the fluid dynamic viscosity ( $\text{kg/(m}\cdot\text{s)}$ ).

Meanwhile, to solving the energy conservation equation, it was used the heat flow model, defined by the equation:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (k \nabla T) + Q \quad (3)$$

where:

$C_p$  is the specific heat capacity at constant pressure (J/(kg·K))

$T$  is the absolute temperature (K);

$k$  is the thermal conductivity (W/(m·K));

$Q$  are the heat sources (W/m<sup>3</sup>).

## 2.2 Initial and Boundary Conditions

Since the simulation involves natural circulation, the initial velocity of the fluid was considered zero. For the initial temperature, it was considered the value of 31.7°C for the fluid and for the entire inner part of the tank. According to the optimizing method, the parameters of interest (Cunha *et al* [3] ).are better represented by this temperature. On the other hand, for the tank surface, it is assumed room temperature, while the rest of the outer part of the tank is regarded as being thermally insulated. Finally, the power applied was 8000 W.

## 2.3 Solution of the Conservation Equations

The software (Comsol 4.1[4]) used can solve CFD problems by the finite element method. Thus, the mesh generator uses tetrahedral elements to discretize the three-dimensional domains, triangular elements for the borders and edge elements (sides) to the boundary geometry. On the other hand, the discretization of the equations is done using the Galerkin method (Harari *et al* [5] and Donea *et al* [6]). When this method is used to find the weak form of convection and diffusion equation (Eq. 1), it causes numerical stability problems. Because of this, the use of stabilization techniques is necessary. The software provides three stabilization techniques: diffusion crosswind, isotropic diffusion, and streamline diffusion. The latter process is enabled as default, in order to forcing the equation systems coupling. Moreover, this process becomes necessary when convection is dominant in the considered stream.

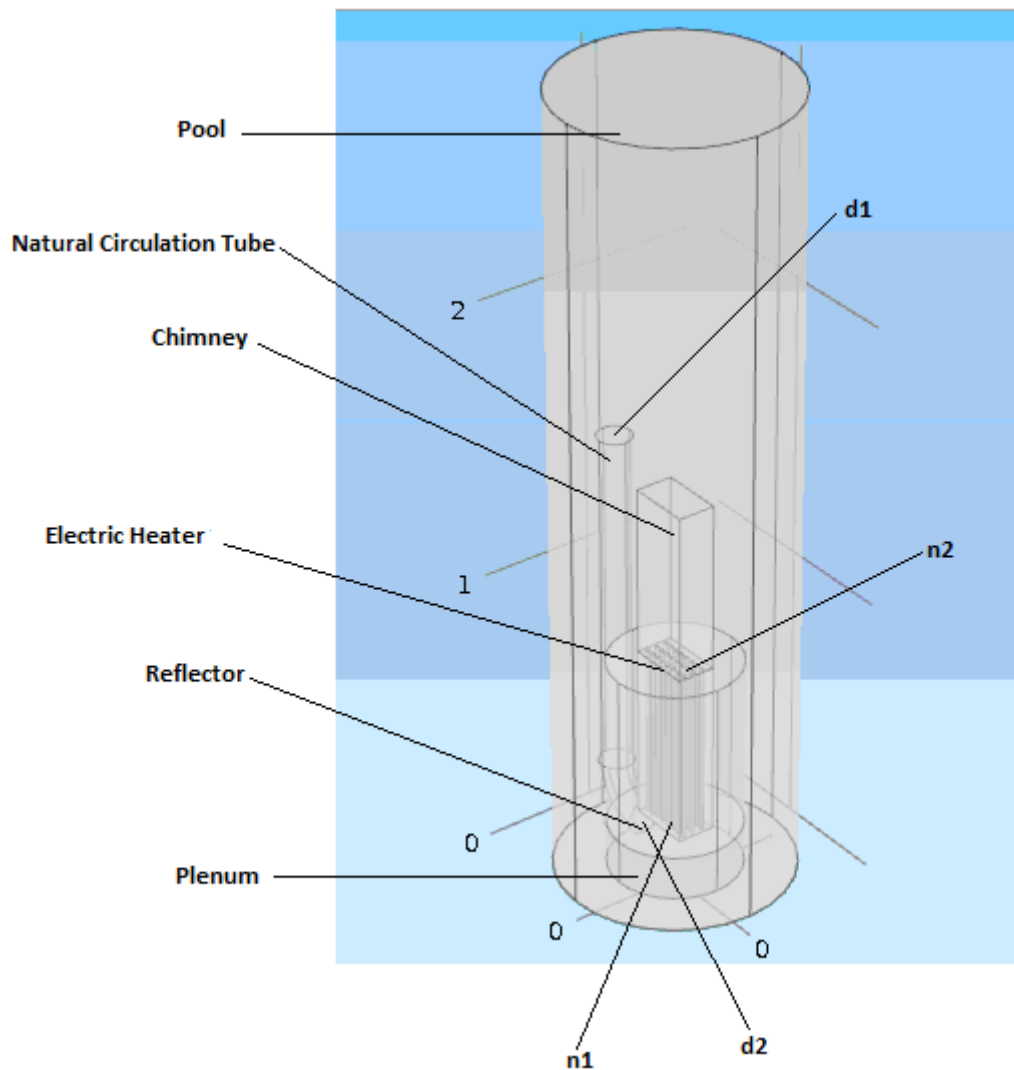
## 2.4 Test Section Description

Considering the objectives of this this work, the definition of some important parameters to study the phenomenon in question can be done suitably by using the OPAL reactor. In order to define the experiment dimensions, it was used the genetic algorithm method (Lima Junior [7]). Based on specific input data, a computer code written in C language verifies the similarity between the prototype and a test section with certain dimensions, returning some

possible dimensions for the desired experiment. Initially, two possible conditions were chosen to be simulated. The criterion for choosing the more adequate dimensions was the condition that demands the lowest power with the highest flow. Table 1 presents the dimensions of the prototype and those of the test section, while Fig. 1 shows a schematic of the test section.

**Table 1: Prototype and Test Section Dimensions**

Component	Radius (m)		Height (m)	
	Test Section	Prototype	Test Section	Prototype
Pool	0.4	2.8	2.8	14.5
Plenum	0.225	2.6	0.157	0.887
Reflector	0.225	2.6	0.615	0.615

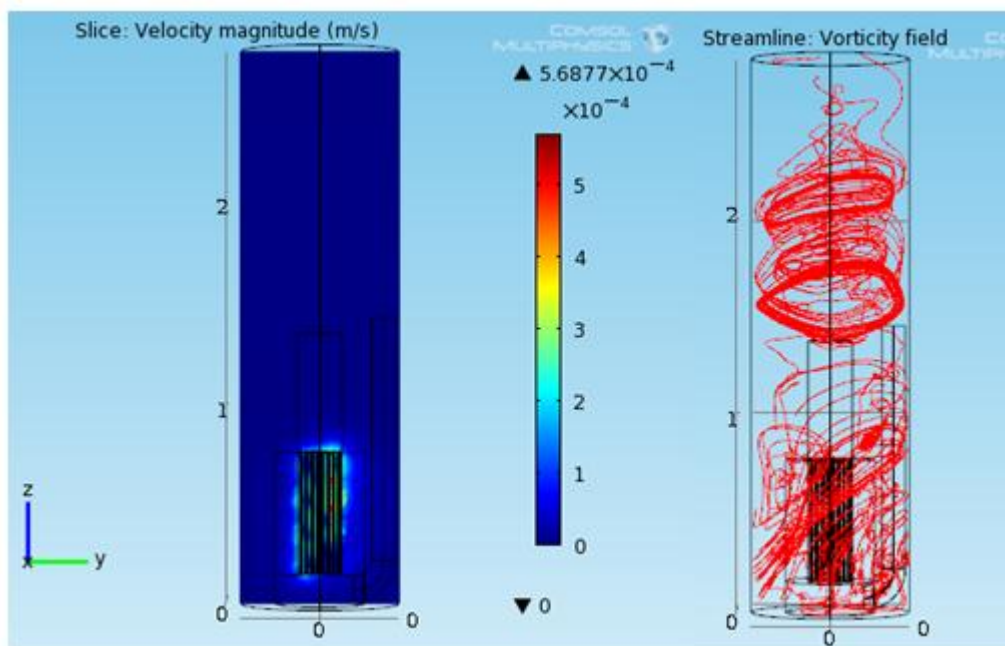


**Figure 1 – Schematic of the Test Section.**

### 3. RESULTS

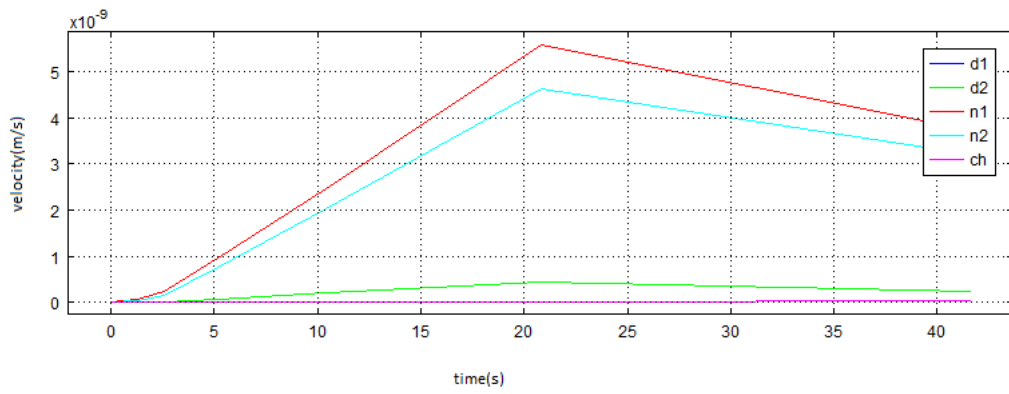
#### 3.1 Velocity Profiles

In Fig. 2, it is shown the average velocity profile on a plane that cuts the circulation tube and the electric heater exactly in their centers, where, it can be observed that the major variations occur in a region opposite to the natural circulation pipe, at the inlet of the electric heater. This shows that the sudden change in flow area that occur when the fluid passes by the outlet of the natural circulation pipe and collides with the plenum wall, probably causes the formation of vortices. Thus, the heat removal is not uniform. Moreover, it can be noted that the greater influence of the velocity on the flow occurs precisely in the Z direction.

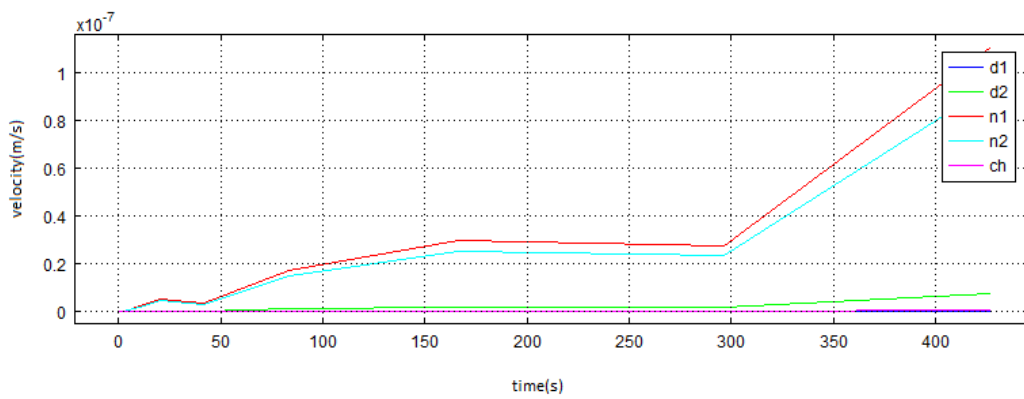


**Figure: 2 Average velocity profiles in the YZ plane passing through the center of the core and vorticity field over the entire section.**

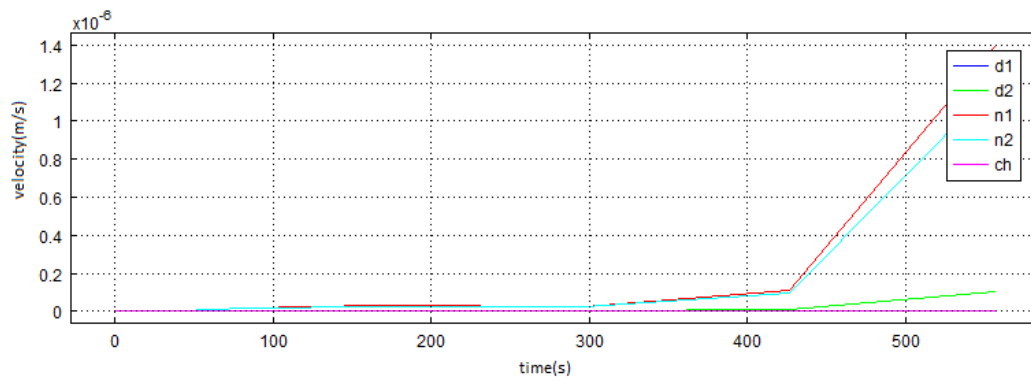
In order to assess the evolution of the velocity profiles in the main circuit pipes, Figs. 3 to 7 show the velocity profiles, after different processing times, for: the inlet of the natural circulation tube “d1” and its outlet “d2”; the inlet and the outlet of the electric heater represented by “n1” and “n2”, respectively, and the chimney outlet “ch”.



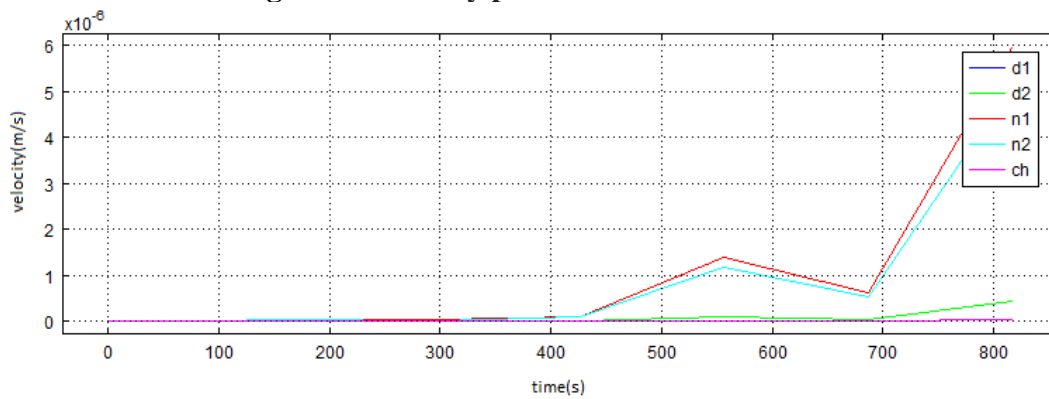
**Figure 3: Velocity profile after 40 seconds.**



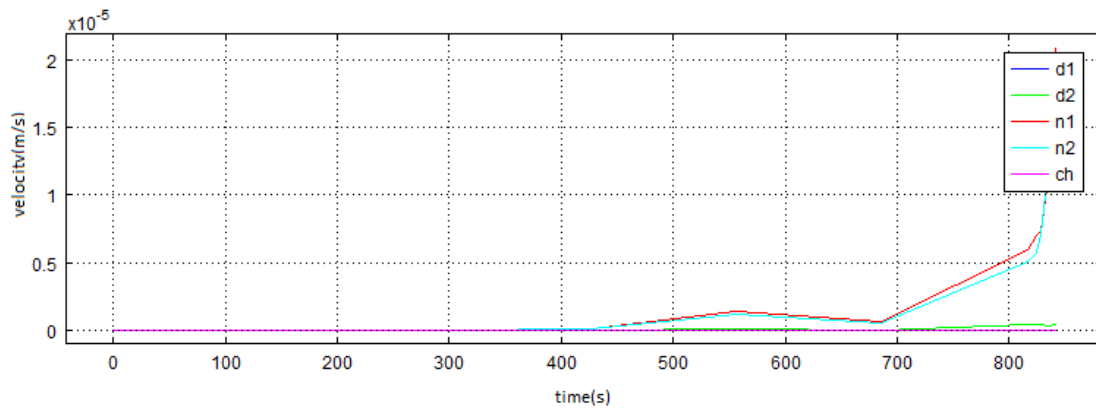
**Figure 4: Velocity profile after 400 seconds.**



**Figure 5: Velocity profile after 500 seconds.**



**Figure 6: Velocity profile after 810 seconds.**



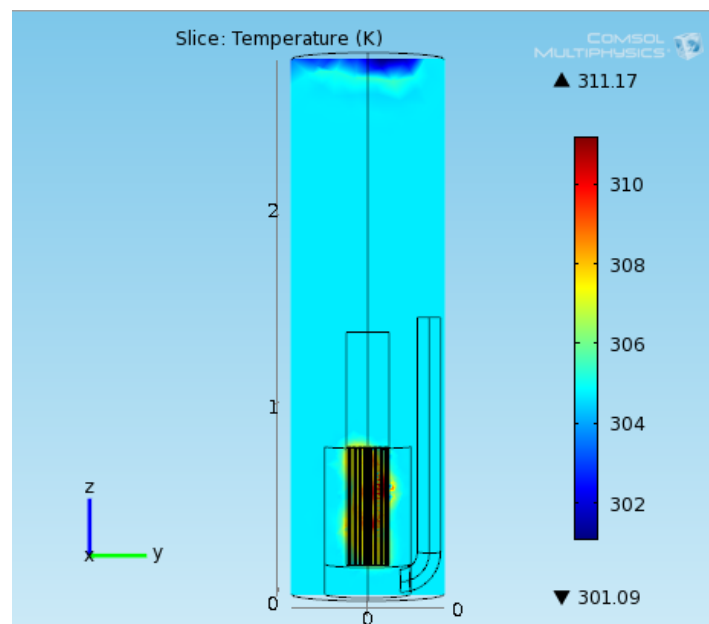
**Figure 7: Velocity profile after 840 seconds.**

In Figs. 3 to 7, it can be observed that the velocity contributions to the natural circulation process are arranged in a descending order, accordingly their importance, as follows:

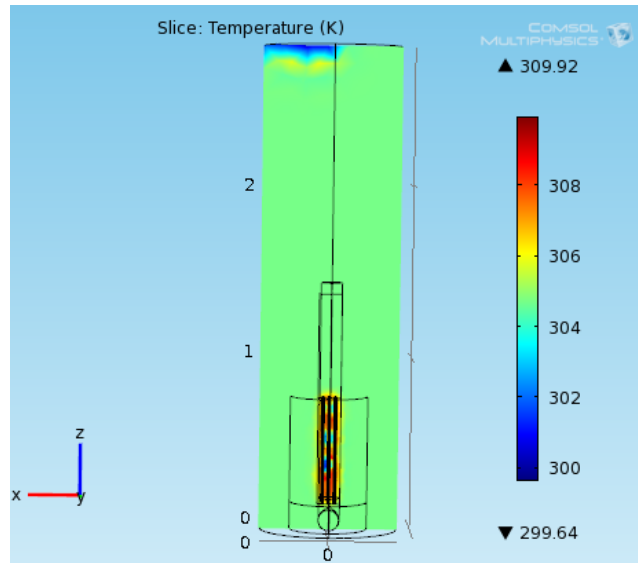
- 1) in the inlet of the core (electric heater) "n1";
- 2) in the outlet of the core "n2";
- 3) in the outlet of the natural circulation tube "d2";
- 4) in the outlet of the chimney "ch";
- 5) in the inlet of the natural circulation tube "d1".

### 3.2 Temperature Profiles

Figures 8 and 9 show the temperature profiles inside the tank for the planes which pass through the centers of the electric heater and of the natural circulation tube in YZ plane and in the XZ plane, respectively.



**Figure: 8 Temperature profiles inside the tank in the YZ plane.**

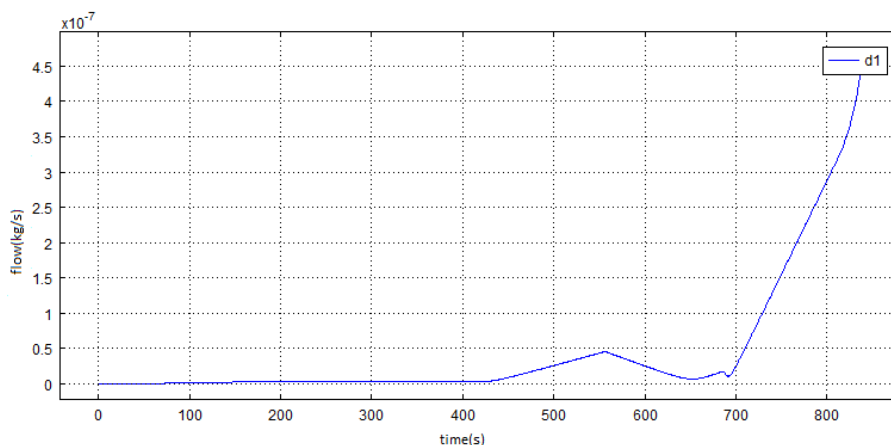


**Figure: 9** Temperature profiles inside the tank in the XZ plane.

According to Figs. 8 and 9, the temperatures do not change significantly along the tank. The higher temperatures occur on the electric heater, but they do not present meaningful variations as well. In Fig. 4 it can be verified the natural circulation occurrence, by comparing the velocity profiles on the chimney "ch" with those on the inlet of the tube "d1" where the curves are almost coincident for various time intervals.

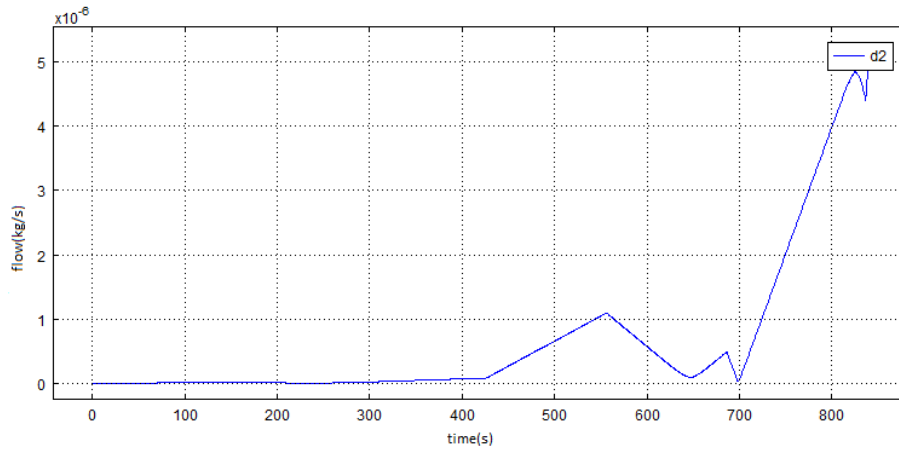
### 3.3 Flow Profiles

The flow rates are presented for the different regions previously defined for the case of velocity profiles: d1, d2, n1, n2 and ch, respectively.

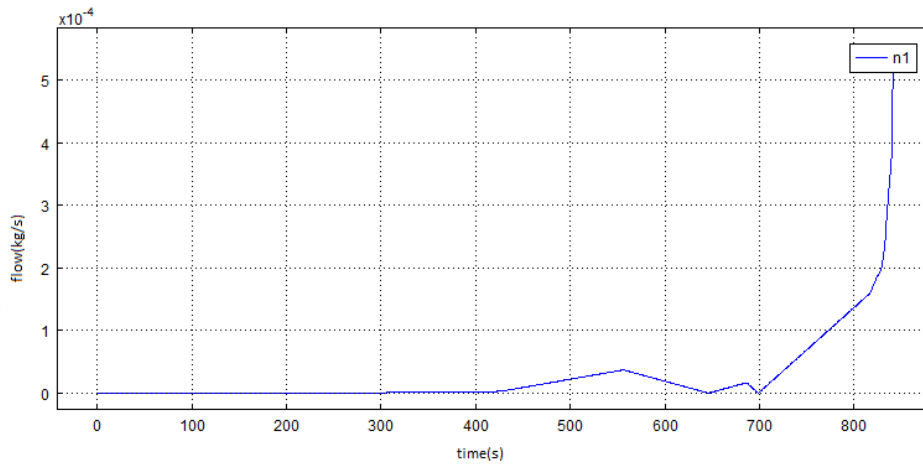


**Figure 10:** Flow profile in the inlet of the natural circulation pipe (d1).

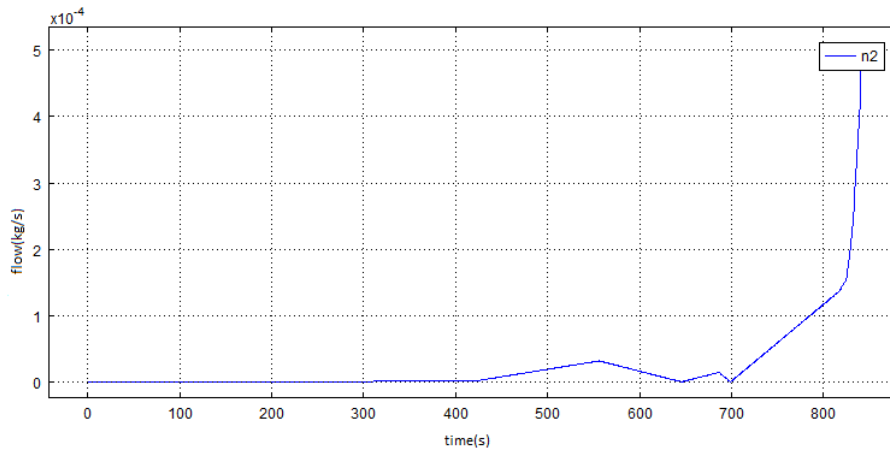




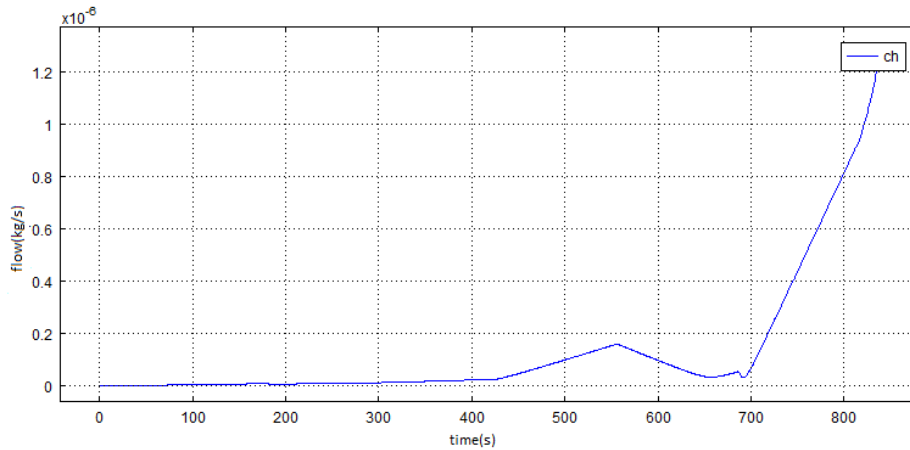
**Figure 11: Flow profile in the outlet of natural circulation pipe.(d2)**



**Figure: 12 Flow profile in the core inlet.(n1)**



**Figure: 13 Flow profile in the core outlet.(n2)**



**Figure: 14 Flow profile in the chimney outlet.(ch)**

Due to variations in the velocity fields and temperature, flow rates also suffer losses and gains according to the considered area. Comparing d1 and d2, it is observed that the flow rate in d2 is much greater than that in d1, due to pressure effects. The reverse effect occurs in the cases of n1 and ch. In these cases, it can be observed a great difference between the flow rates. This difference can be attributed to the temperature difference found in the chimney outlet, to the sudden change of the flow area and to the friction losses suffered when passing through the resistors.

### 3.4 Dimensionless Numbers

In order to evaluate the natural circulation in the proposed circuit with a dimensionless approach, it were calculated the Grashof (Gr), the Reynolds (Re) and the Richardson (Ri) numbers. The results are presented in Tab. 2.

**Table :2 Dimensionless numbers in the core entry.**

Number	Section
Gr	2634.11
Re	0.122
Ri	176975.9

The dimensionless numbers in Tab. 2 were calculated based on the hydraulic diameters in the electric heater inlet of the test section, and on the core inlet of the prototype. The choice of these regions for the calculations was made taking into account the relevance of this region to the study (Soares *et al.* [10]). In the case of the test section, Ri indicated that the flow is governed by natural convection, since  $Ri \gg 1$ . Therefore, according to (Hirata [8]), Nu does not depends on Re and, consequently, on the velocity. These results are in agreement with the relationship given by Kreith [9] for plates and vertical cylinders in natural circulation:

$$h_{cx} = 0,41 \frac{k}{x} (\text{Pr Gr})^{1/4} \quad (4)$$

where  $h_{cx}$  is the heat transfer coefficient,  $k$  is the thermal conductivity,  $x$  is the distance from the edge of interest. Therefore, the Nusselt number, in this case, is a function of Pr and Gr.

**Table 3: Dimensionless numbers in the chimney outlet.**

Number	Section
Gr	7477.49
Re	$2.377 \times 10^{-4}$
Ri	$3.143 \times 10^{11}$

The dimensionless numbers in the Tab. 3 were calculated based on the hydraulic diameter in the electric heater outlet. According to Soares *et al.*[10], the dimensionless numbers at the core inlet of the prototype and at the chimney outlet can be estimated. This is done by considering the flow curves presented in Fig. 14 and by Eq. 5, defined as:

$$M_{ponto} = \rho.V.A \quad (5)$$

where,

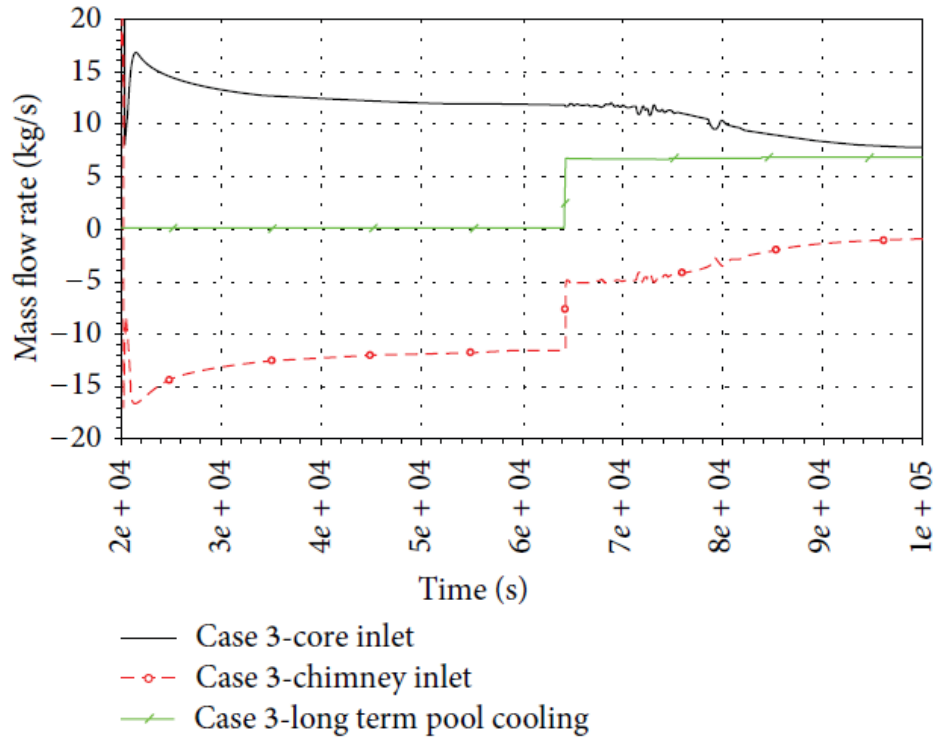
$M_{ponto}$  is in the mass flow rate (kg/s);

$\rho$  is the fluid density ( $\text{kg/m}^3$ );

V is the velocity in the channel outlet (m/s)

A is the drain area ( $\text{m}^2$ ).

In this way, it is possible to estimate the velocities in the core inlet and in the chimney outlet. However in those curves, it is not possible to decouple the inertia effects of the pumps, which can explain the very high values calculated for the velocities. Thus it was found a discrepancy between the values calculated for Re and Ri for the prototype and for the test section, which does not allow an adequate comparison between them.



**Figure 14: Mass flow rate evolutions showing natural circulation (Soares *et al.* [10]).**

#### 4. CONCLUSIONS

Based on the results obtained in the present work, it is clear that, for the simulation time applied, the variations in temperature and in velocity are very small. According to the velocity profiles, it is observed that the natural circulation phenomenon started with greater force at the bottom of the core, i.e. in its inlet. The next location to suffer the natural circulation effects was the region located in the natural circulation pipe outlet. As observed at the velocity curves, this occurs some time later. Thus, a time interval is demanded until the heating effects of the liquid contained in the lower part of the core may be added to the effects of the forces acting on the upper end of the natural circulation pipe and consequently at its bottom. Also with respect to velocity, one can consider that after a certain period of time, there are three distinct velocity profiles: in the chimney outlet, in the natural circulation pipe output, and in the electric heater inlet. Although several aspects must be considered in the analysis of similarity between the test section and the prototype, as the large pressure difference between them, one aspect that could influence on the calculations is the simulation time. The simulated time could be insufficient to allow the velocities to reaching a greater order of magnitude. Perhaps, by applying a higher simulation time,  $Re$  might have values higher than those obtained in this work.

## ACKNOWLEDGMENTS

The authors are grateful to National Council for Scientific and Technological Development (CNPq), the Funding Authority for Studies and Projects (FINEP), the Carlos Chagas Filho Research Support Foundation of the State of Rio de Janeiro (FAPERJ) the Nuclear Engineering Institute IEN / CNEN and the Institute for Innovative Nuclear Reactors (INCT) for the financial support.

## REFERENCES

1. IAEA – International Agency Atomic Energy “*Natural Circulation in Water Cooled Nuclear Power Plant* ” IAEA, Australia, November 2005.
2. ANSTO-Australian Nuclear Science & Technology Organization “*Operational Limits and Conditions for the OPAL Reactor-Summary for Public Relapse*” Document N° RRRP-7230-EDEAN-003-REVO, March 2005.
3. Cunha, J. J., Alvim, A. C. M., Lapa, C. M. F., Pereira, C. M.N.A., Souza Lima Jr., C.A., “*Reduced Scale PWR Passive Safety System Design By Particle Swarm Optimization*” Brazilian Congress of Thermal Engineering and Sciences Belo Horizonte MG November 10-14, 2008
4. COMSOL Multiphysics “*CFD Module User’s Guide*” Version: COMSOL 4.1 Beta April 2010
5. I. Harari and T.J.R. Hughes, “*What are C and h? Inequalities for the analysis and design of finite element methods*” *Comp. Meth. Appl. Mech. Engrg*, vol. 97, pp. 157–192, 1992.
6. Jean Donea and Antonio Huerta “*Finite Element Method for Flow Problems*” John Wiley, Canada 2003.
7. Carlos Alberto de Souza. Lima Junior “*Aplicação da Técnica de Otimização por Enxame de Partículas no Projeto Termo-Hidráulico em Escala Reduzida do Núcleo de um Reator PWR*”, Master thesis-IEN, Rio de Janeiro, 2008.
8. M.H.Hirata “*Análise Dimensional e Leis de Semelhança*”, class notes.  
<http://www.fat.uerj.br/intranet/disciplinas/Mec%E2nica%20dos%20Fluidos>
9. Frank Kreith, “*Princípios de Transmissão de Calor* ”, Edgard Blucher LTD, São Paulo-SP, 1977.
10. Humbeto V.Souares, Ivan D.Aronne, Antonella L.Costa, Claubia Pereira, and Maria Auxiliadora F.Veloso “*Analysis of Loss of Flow Events on Brazilian Multipurpose Reactor Using the Relap5 Code*” Hindawi Publishing Corporation International Journal of Nuclear Energy, Volume 014, Article ID 186189, 12 pages, June 2014.