2013 International Nuclear Atlantic Conference - INAC 2013
Recife, PE, Brazil, November 24-29, 2013
ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR - ABEN
***ISBN*: 978-85-99141-05-2**

BASIC CFD INVESTIGATION OF DECAY HEAT REMOVAL IN A POOL TYPE RESEARCH REACTOR

André F. Oliveira[1], Maria de L. Moreira[2]

[1]Programa de Engenharia Nuclear, COPPE
Universidade Federal do Rio de Janeiro
Av. Horácio Macedo, s/n, bloco G
Rio de Janeiro, RJ
afoliveira@ien.gov.br

[2]Instituto de Engenharia Nuclear (IEN/CNEN)
R. Hélio de Almeida, s/n, SETER
Rio de Janeiro, RJ
malu@ien.gov.br

ABSTRACT

Safety is one of the most important and desirable characteristic in a nuclear plant. Natural circulation cooling systems are noted for providing passive safety. These systems can be used as mechanism for removing the residual heat from the reactor, or even as the main cooling system for heated sections, such as the core. In this work, a computational fluid-dynamics (CFD) code is used to simulate the process of natural circulation in an open pool research reactor after its shutdown. The physical model studied is similar to the Open Pool Australian Light water reactor (OPAL), and contains the core, cooling pool, reflecting tank, circulation pipes and chimney. For best computing performance, the core region was modeled as a porous media, where the parameters were obtained from a separately detailed CFD analysis.

1. INTRODUCTION

Cooling systems that work by natural circulation are widely used in industries. Especially in the nuclear sector, where facilities need to be intrinsically safe. Nuclear reactors, particularly those more moderns (Generation III +), have used this concept of passive cooling in their projects. They are developed in order to use the gravity force to act removing heat from core under operation, or just to remove residual heat. This is also one of the safety principles of some pool type research reactors where the bulk primary system coolant offers sufficient thermal capacity to absorb the residual heat from core and transport it to external sinks.

This is the case of the australian multipurpose reactor OPAL [2] (Open Pool Australian Light Water Reactor) that, under normal operation, has heat extracted from the core by an upward flow of water supplied by pump suction. This heated fluid is then transferred to decay tanks and heat exchangers. Under the time of shutdown, the pumps suction ceases, and heat is extracted from the core only by the natural circulation of the pool water. This paper is part of a larger study which seeks to get a scaling model for a test section that is similar to a prototype reactor with very similar characteristics to OPAL.

This step seeks to obtain some important information about the process of removing residual heat from reactor, such as the temperature gradient in the pool, the fluid core temperature and mainly the flow rate in the nucleus. For this purpose was used the commercial computational fluid dynamics code Ansys CFX [1].

This code calculates the governing equations of the problem, using fininte volumes discretization, allowing to obtain the fields of temperature, pressure and velocity in domain under study. This software is widely used for simulations where heat removal occurs in laminar or turbulent flows, one-phase or two-phase, including those where there are combustion processes and chemical reactions. This package has been used to evaluate different problem types in nuclear thermohydraulics as boron dilution, erosion, corrosion and deposition; core instability in BWRs; recriticality in BWRs among others.

The computational fluid dynamics technics is currently feasible due to development of technology and use of multicore CPUs in cluster systems. Therefore it becomes possible to solve these complex problems that require large computational power, without resorting to very expensive computing facilities.

2. THE MODEL

The model under study comprises a cylindrical pool of 12 m depth, which is a superestimated operation condition, where the associated storage pools do not contribute to the coolant inventory. That means, the thermal capacity of the system is greatly reduced.

In this model, the core consisting of thin flat plates and irradiation channels, is represented by porous media in order to reduce the computational costs. This core is surrounded by a reflector tank containing heavy water and has a plenum at the bottom.

In shutdown condition, the residual heat existing in the core heats the water from coolant channels, that rises by buoyancy to the pool top.

In this model are contained the natural circulation pipes, that have their valves opened when the reactor is shut down. Then the heated fluid from the top down by them when they lose energy and density increases. This model is shown in figure 1.

During operation of reactor, near to the top of the pool is maintained a layer of hot water, which acts as a radiation shield. As stated previously, this studied model is very similar to the OPAL reactor, differing by them in the operating power (30 MW). Specifically in this study, we consider the residual heat of a nucleus which operated at full power for 6 months.

In order to reduce the computational costs, in this basic analysis, the flow inside the reflector tank was not simulated, as well the heat transfer through solid sections.

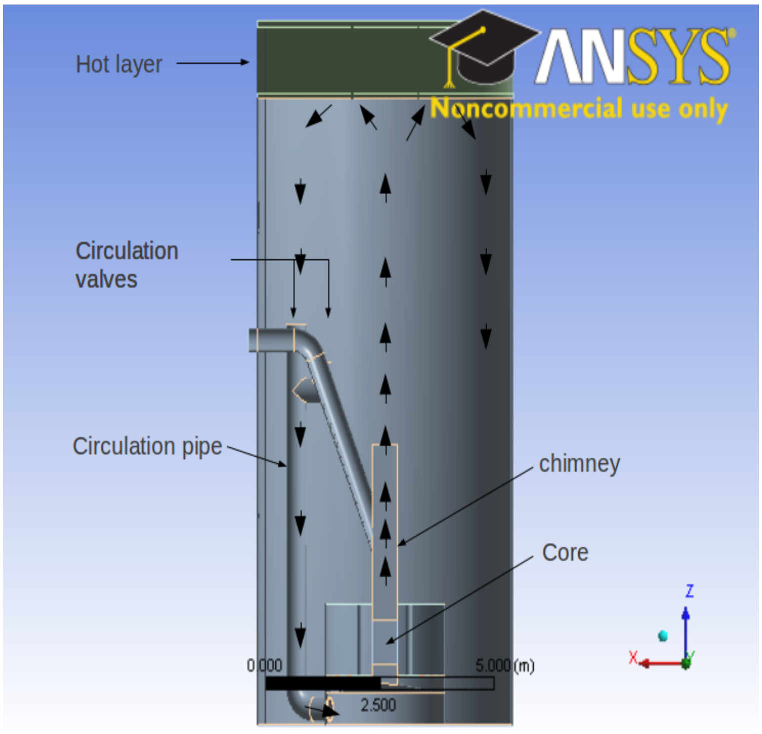

Figure 1: Scheme of the natural circulation in pool reactor showing the reflector tank, chimney and circulation pipes.

2.1. Decay heat

After the shutdown of a nuclear reactor, even if the fission reactions have been ceased, there is the heat generated by decay of fission products. This residual heat must be removed from the reactor core, avoiding that it would be damaged. In the reactor under study this process of heat removal takes place by natural circulation of water in the pool, which is released by the opening of four valves on the top of circulation pipes, as shown in figure 1.

According to chapter 3 of [3], the heat generation for reactor shutdown state is given by the sum of the heat produced by: (1) fission from delayed neutron or photoneutron emissions; and (2) decay of fission products, fertile materials, and others activation products from neutron capture. However, in a few minutes the fissions generated by delayed neutrons tend to be reduced to a negligible amount. Therefore, the major source of heat generation is the fission products decay.

Considering the prototype in study operating at power of 30 MW and subsequently shuting down; the heat supplied to the system is derived only from the fission products existing in the core. And to simulate this simplified heat source, was used an expression for the power, which depends on time and operating power, and considers only the energy released by β and γ emissions. This expression is found in equation (3-70c) of [3].

$$\frac{P}{P_0} = 0.066[(\tau - \tau_S)^{-0.2} - \tau^{-0.2}] \tag{1}$$

In this equation, τ is the time at which the reactor was started, and the τ_S the shutdown time and P_0 the power before the shutdown.

3. SIMULATION

For the simulation purposes, the model was divided into three domains: (a) WATER domain, (b) HOT LAYER domain and (c) CORE domain. For these domains were defined one hour transient simulation, with adaptive time steps ranging between 0.1 and 0.2 seconds.

For these three domains, was used the k-omega turbulent flow model, with the reference pressure of 1 atm, and initial velocity equals zero.

The **a** domain was started at 35°C, the **b** domain at 45°C and the **c** domain at XXX°C as shown in figure 2.

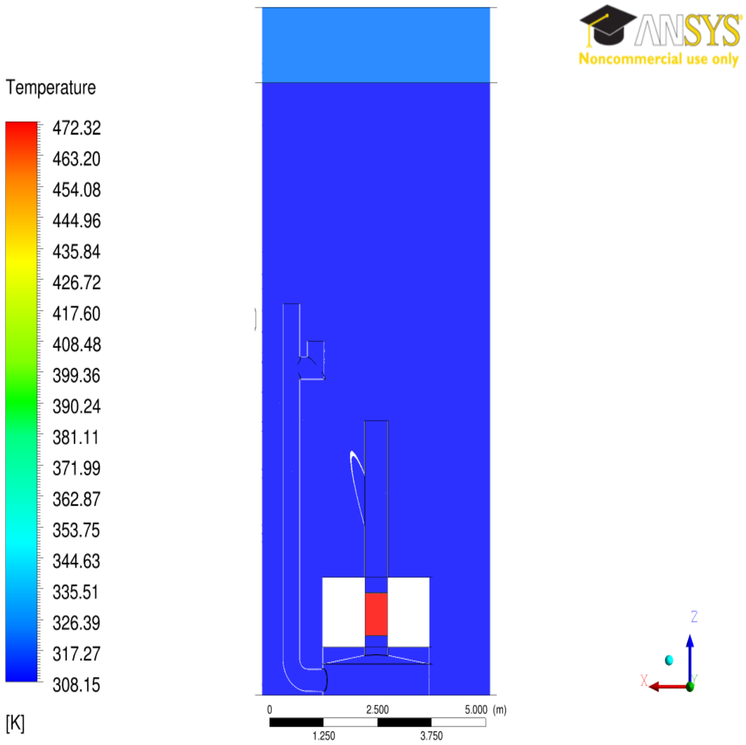

Figure 2: Initial domains temperatures

This simulation is intended to obtain the overall performance of the system. And for that we used the convergence criteria of 1.0e-5 for average RMS residues or a maximum number of 100 iterations.

3.1. Heat source and sink

For a system where the flow is established by natural circulation, the simulation of heat sources and sinks are very important. For this model in study was applied the equation for decay heat proposed by [3] on the core porous domain. For this implementation was necessary to define a subdomain that provides heat to the system. This time, to simplify the problem, all the walls that surround the fluid domain were considered adiabatic except for the pool top, where established a boundary condition of the free sliding with heat loss to outside medium at 25°C.

3.1.1. Core Domain

As mentioned before, the domain that comprises the core of this model is very important. In this domain there is a heat source that will generate the movement of the whole fluid. This domain has a special feature to be modeled as a porous media, because of its 23 fuel assemblies composed by 21 thin fuel plates each, and a complete simulation of channels would be time expensive.

For Ansys CFX code, a porous media can be characterized by its volume fraction, the loss directions and a resistance quadratic coefficient. This coefficient is characteristic of the media in which the flow occurs, and can be obtained from the equation 2.

$$\frac{\partial p}{\partial x_i} = K_Q \mid U \mid U_x \tag{2}$$

Even though there are theoretical and empirical expressions providing the pressure loss for rectangular, circular and even between very thin channels, we decided to infer the value of K_Q via CFD. For that was simulated a detailed core model, varying the average inlet velocity at the core entrance, at the range from 0.01 to 1.1 m/s. For these input velocities were measured the pressure drop, and by mean of these the quadratic coefficient, K_Q with $3509kgm^{-4}$ value. The data from this simulation are shown in table 1 and figure 3.

Table 1: Linear adjust data for quadratic coefficient estimative

Inlet velocity (m/s)	ΔP (Pa)	$\Delta P/\Delta Z$ (m/s^2)	U*Uz $(m/s)^2$
0.01	0.82832	0.59166	0.0001418
0.02	3.19133	2.27952	0.0005739
0.025	4.96228	3.54449	0.0008995
0.030	7.13753	5.09824	0.0012996
0.040	12.6674	9.04814	0.0023165
0.050	19.8106	14.15043	0.0036298
0.080	50.9224	36.37314	0.0093757
0.1	79.7137	56.93836	0.0147182
0.3	722.248	515.8914	0.1358040
0.5	1972.93	1409.235	0.3775020
0.7	3723.21	2659.435	0.729983
0.9	5894.30	4210.214	1.186920
1.0	7126.28	5090.200	1.453300
1.1	8453.54	6038.242	1.744870

On this same domain defining the reactor core, a subdomain has been implemented with a heat source defined by equation 1, whereas the reactor has operated for 6 months at fixed power of 30 MW.

A CFD model for simulating complex systems as a nuclear reactor, is always an optimization task of resources and time. We need to reach the highest possible accuracy in numerical solutions spending a little time and computational resource. This implies a convenient adjust of time and space discretization, in order to obtain solutions close to the true values. For this study was constructed an unstructured finite volume mesh, consisting of tetrahedral elements with edges ranging between 0.0005 and 0.1 m and the result was a 5 million elements mesh.

3.2. Processing

The problem described above was simulated using the small cluster of Instituto de Engenharia Nuclear (IEN/CNEN), which consists of 10 Intel Xeon machines with 12 x5660 cores of 2.8Gz. The communication protocol used was MPI (Message Passing Interface), only distributed over 24 processors (two machines), and the obtained results are shown bellow.

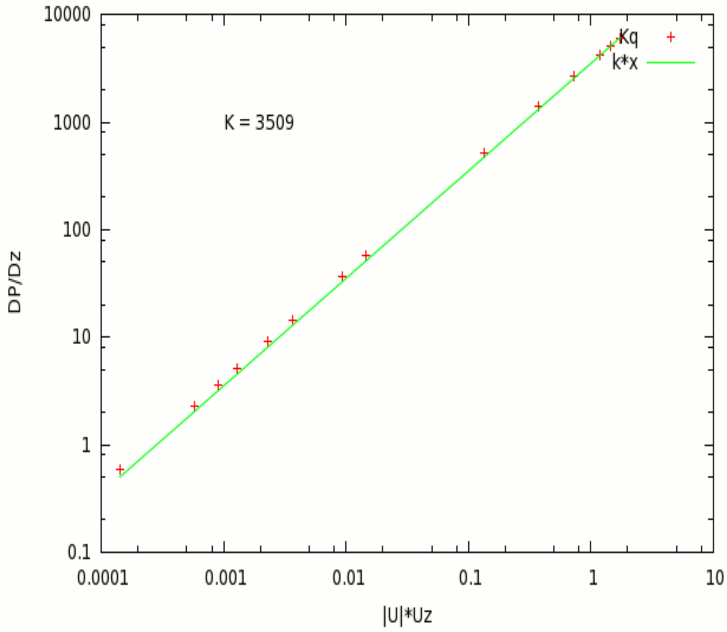

Figure 3: Quadratic coefficient estimative (K_Q)

4. RESULTS

As stated earlier, this computational fluid dynamics study of reactor prototype was performed with the intention to evaluate aspects related to intrinsic safety of the installation, and also to provide data needed for the calculation of important dimensionless groups as Grashof, Reynolds, Stanton, Friccion among others, that characterize a natural circulation flow acording [4]. To obtain these values is important to evaluate the core temperature and velocity, as well as in the circulation pipes and pool. The values of this simulation are shown below.

4.1. Temperature Distribution

Under normal operation, as explained earlier, the average temperature of the pool water varies around 35°C, and the temperature of the hot layer is maintained at around 45°C. These are initial temperature of the problem shown in figure 2. After this shutdown the temperature distribution changes rapidly, as can be seen in the figures 5 and 6. Considering safety aspects of the installation is possible to check in figure 4, that the maximum temperature reached in the simulated domains is not large enough to damage the structure of the fuel. After one minute of transient the fluid temperature within the core is reduced to a level close to the pool temperature.

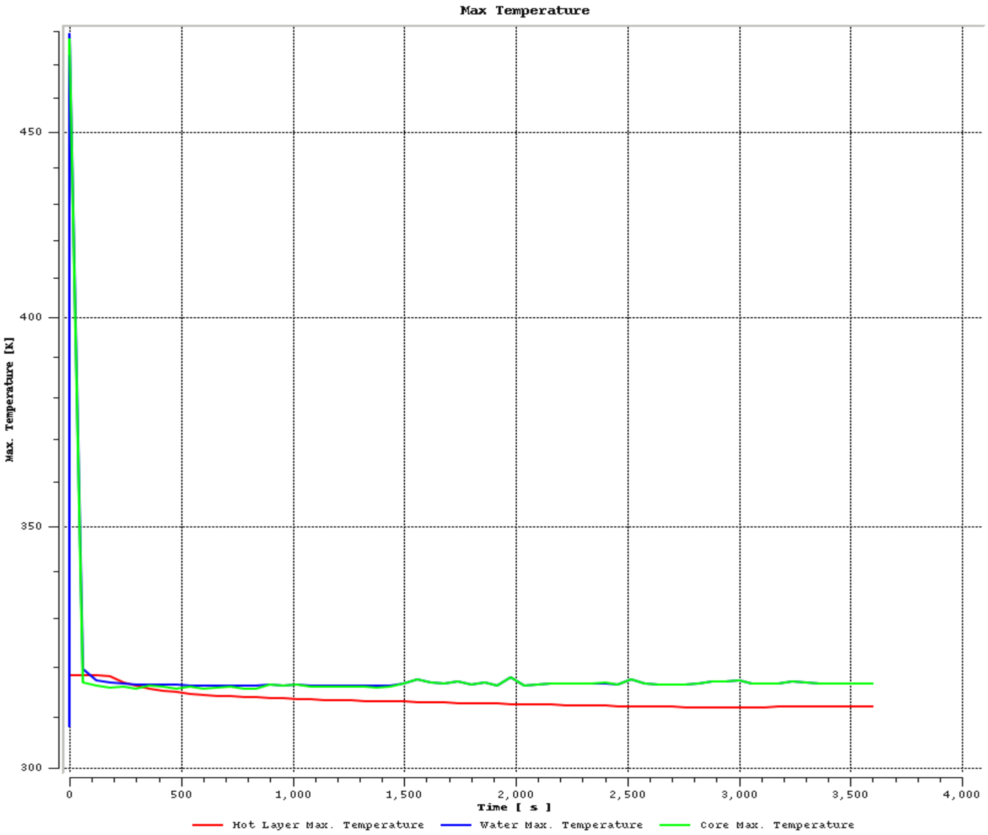

Figure 4: Maximum temperature in corresponding domains of the pool, hot layer and core

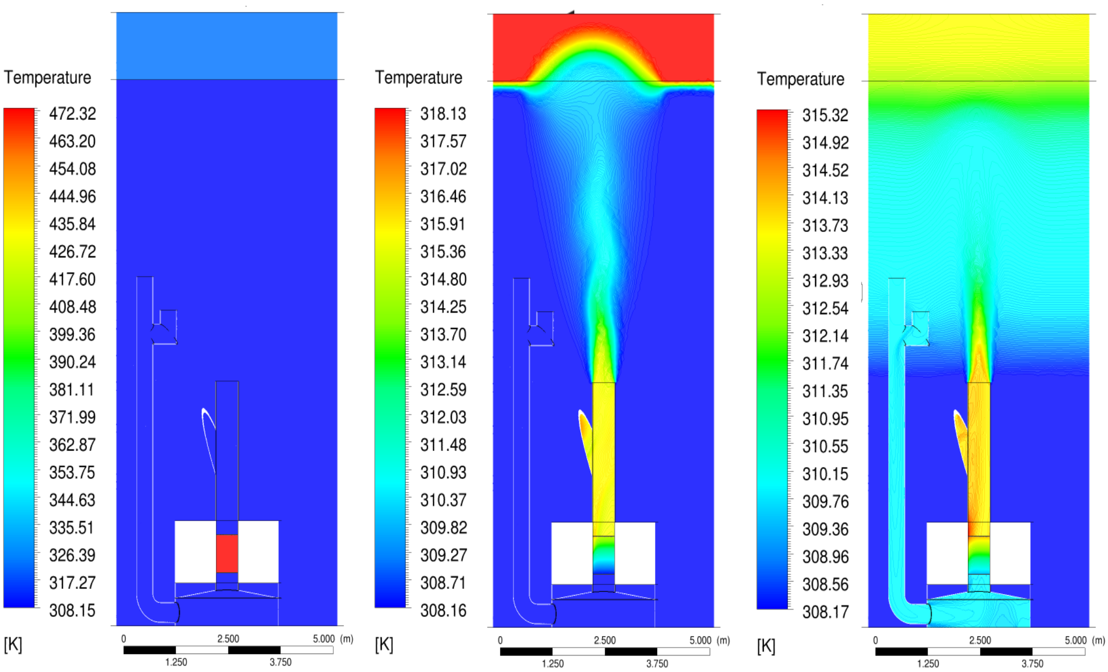

Figure 5: Temperature distribution in the pool at the instants t = 0 t = 60 t = 900 s.

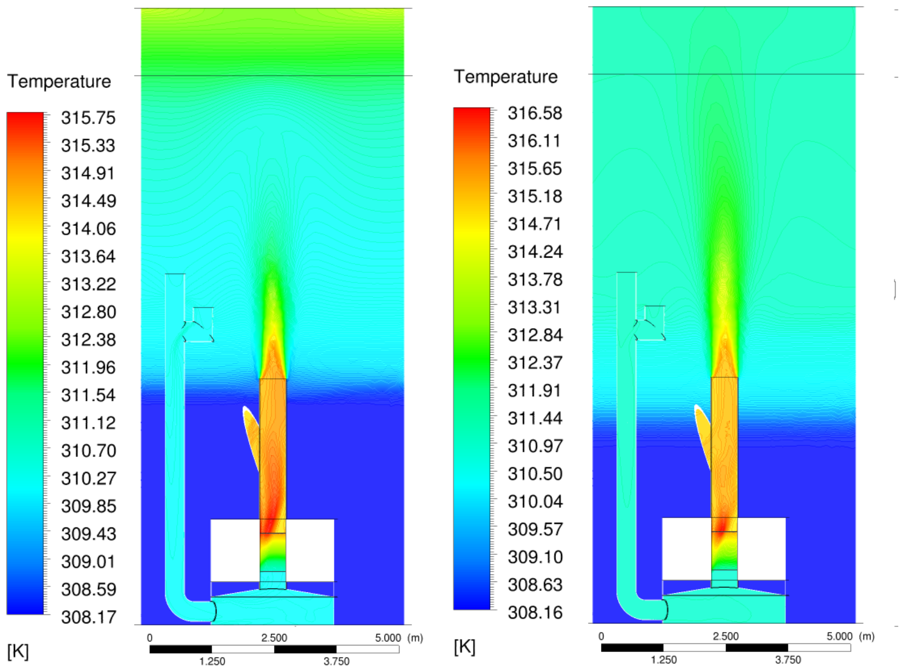

Figure 6: Temperature distribution at the times t = 1800 and t = 3600 s.

4.2. Velocity

After one hour transient, there is a marked reduction in velocity profile at the core entrance, and hence the velocity profile of the pool, as can be seen in figures 7 and 8, which respectively show the Reynolds number in the core output and the velocity in a plane that cuts the core, the pool and the natural circulation pipes.

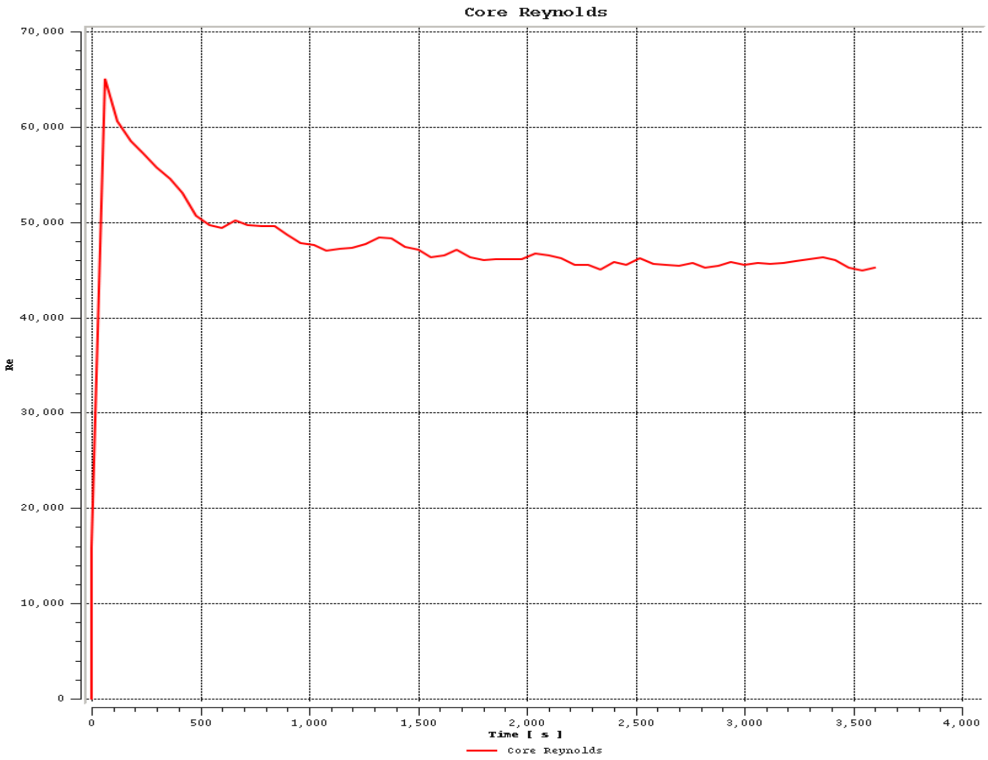

Figure 7: Reynolds number at core outlet

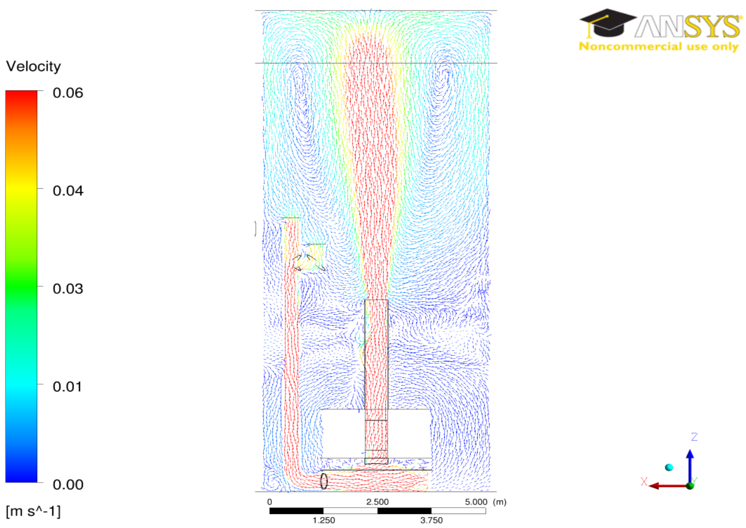

Figure 8: Velocity profile in a vertical plane that cuts the core, pool and the circulation pipes.

5. CONCLUSIONS

The CFD simulation developed in this work shows the possibility of using this numerical technique for study of flows in which some parameters are unknown. Specifically in this study, was possible to obtain information concerning the safety of the installation, as the fluid temperature in the early moments after the reactor shutdown, checking the formation of a temperature gradient of about 8 degrees between the hot and cold sources of the system . This information is very important for obtaining dimensionless groups which characterize this type of flow.

Associated with this temperature gradient, was obtained the fluid velocity, which in the period of 1 hour reaches 0.16 m/s at the core outlet. This is a very important parameter when is intended to obtain scaling systems. For according to the KATAOKA and ISHII formulation, the flow rate is a function of the number of friction, which is function of Reynolds number. Thus requiring a reference value for velocity.

This study verified the need for more realistic initial conditions for this problem, for example the initial velocity fields of domains, especially within the core, to reach a more detailed evaluation of safety aspects. These more detailed studies are in developing and will be presented in a future work.

ACKNOWLEDGMENTS

Thank CNPq, COPPE/UFRJ and IEN/CNEN for supporting this research.

REFERENCES

1. "Ansys CFX", `http://www.ansys.com/` (2013).
2. "Open Pool Australian Light water reactor", `http://www.ansto.gov.au/AboutANSTO/OPAL/index.htm` (2013).
3. N. E. Todreas and M. S. Kazimi. *Nuclear Systems I - Thermal Hydraulic Fundamentals*, Taylor & Francis, United States (1993).
4. M. Ishii, I. Kataoka "Scaling laws for thermal-hydraulic system under single phase and two-phase natural circulation", *Nuclear Engineering and Design*, **81**, pp. 411–425, (1983).