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**

CFD SIMULATION OF IPR-R1 TRIGA SUBCHANNELS FLUID FLOW

Vitor V. Silva[1,2], A. Santos[1], Amir Z. Mesquita[1], P. S. da Silva[1], C. Pereira[2]

[1]Centro de Desenvolvimento da Tecnologia Nuclear (CDTN/CNEN)
Av. Antônio Carlos 6627
31270-901 Belo Horizonte, MG
{vitors,aacs,amir,psbls}@cdtn.br

[2]Departamento de Engenharia Nuclear
Universidade Federal de Minas Gerais
Av. Antônio Carlos 6627
31270-901 Belo Horizonte, MG
claubia@nuclear.ufmg.br

ABSTRACT

Computational fluid dynamics (CFD) codes have been extensively used in engineering problems, with increasing use in nuclear engineering. One of these computer codes is OpenFOAM. It is freely distributed with source code and offers a great flexibility in simulating particular conditions like those found in many problems in nuclear reactor analysis. The aim of this work is to simulate fluid flow and heat flux in three different configurations of subchannels of IPR-R1 TRIGA reactor using OpenFOAM. The data will be then validated against real experimental data obtained during the operation of the reactor at $100kW$. This validation process is fundamental to allow the use of the software and associated model to simulate reactor's operation at different conditions, namely different power e fluid flow velocities.

1. INTRODUCTION

The aim of this work is to simulate and validate the fluid flow in three different representations of IPR-R1 TRIGA reactor subchannels. The simulation was performed using the open source CFD code *OpenFOAM* [1]. The validation process consisted in comparing the fluid temperatures in the subchannel to the fluid temperature in the reactor. The experimental data used as reference was obtained by [2]. In this context, we consider a broader meaning for *validation*: the objective in this work is not only have accurate results for the numerical simulation and validate it with experimental results, but also identify problems or absence of physical phenomena which can contribute to results as a whole. The process of identification and evaluation of simulation flows is fundamental to improve *OpenFOAM* simulation and the confidence in its numerical results applied to nuclear reactors analysis.

2. METHODOLOGY

As in many CFD simulations, the whole simulation process can be divided in three steps: Pre-processing, Simulation and Post-processing. The next three subsections will present the tasks carried the respective step.

2.1. Pre-processing

The Pre-processing consists mainly of the geometry definition and mesh generation. Although the absence of physics in this phase, about 50% of simulation time is spent in this step [3].

In order to catch fluid flow details in the reactor core without simulating the whole core a set of representative subchannels was modeled. In this set there are three different subchannels: **Triangular subchannel**, **Quadrangular subchannel** and one near the reactor wall, called **Reflector subchannel**. The three representative subchannels (Figure 1) approach was first proposed by [4] in a numerical analysis of IPR-R1 TRIGA using subchannel codes. The use of the same configuration was chosen in this work in order to be able to compare the CFD and subchannels codes in the future and also compare to experimental data collected in these three different subchannels.

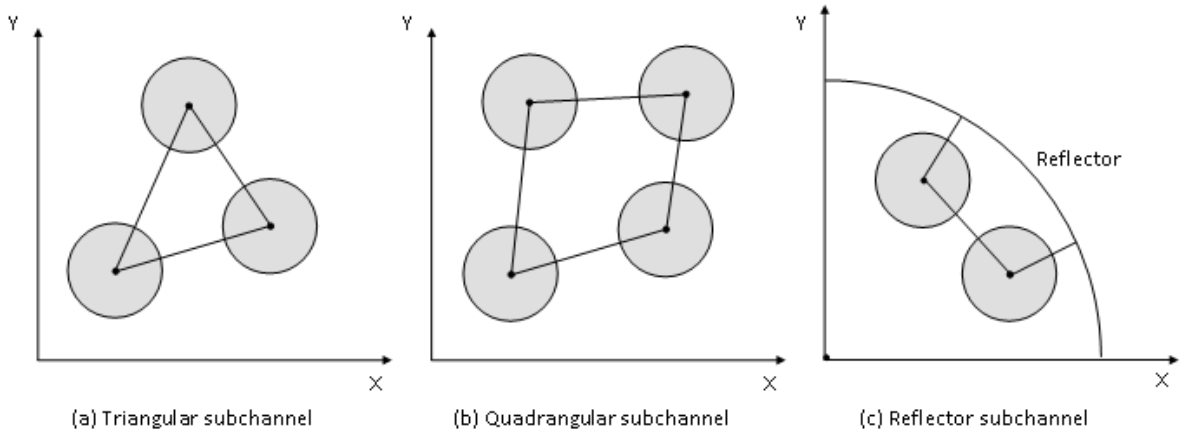

Figure 1: Different types simulated subchannels.

Based on these three types of subchannels, three meshes were created simply by drawing in two dimensions the top of the subchannel and then using extrusion tools to expand it to reactor core's height. The meshes generated were mostly hexahedrical, in order to avoid the non-orthogonal corrections in the simulation process. In table 1 are the main parameters of three meshes used.

Table 1: Three meshes charachteristics

Mesh elements\Subchannel	**Triangular**	**Quadrangular**	**Reflector**
Hexahedra	186,700	357,600	206,000
Prisms	300	400	100
Mesh non-orthogonality	5.09	6.07	5.70

The non-orthogonality parameter gives information of how ortogonal are the mesh cells related to all neighbors. The values presented are an average for all cells in the mesh.

Figures 2, 3 and 4 show, respectively, the triangular subchannel mesh, quadrangular subchannel mesh and the reflector subchannel mesh.

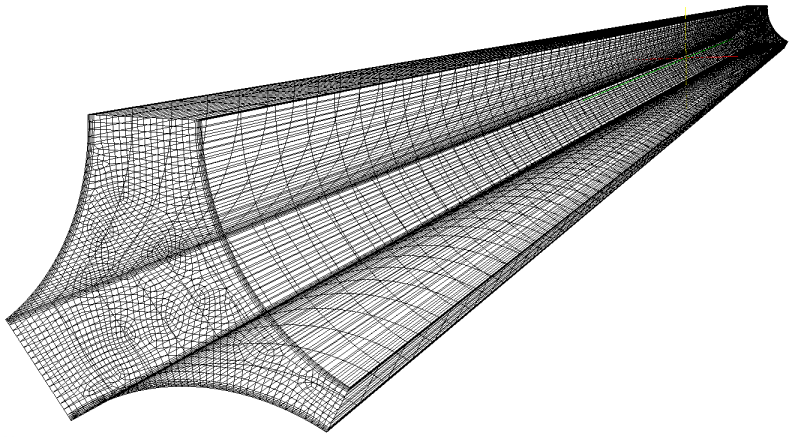

Figure 2: Triangular subchannel mesh.

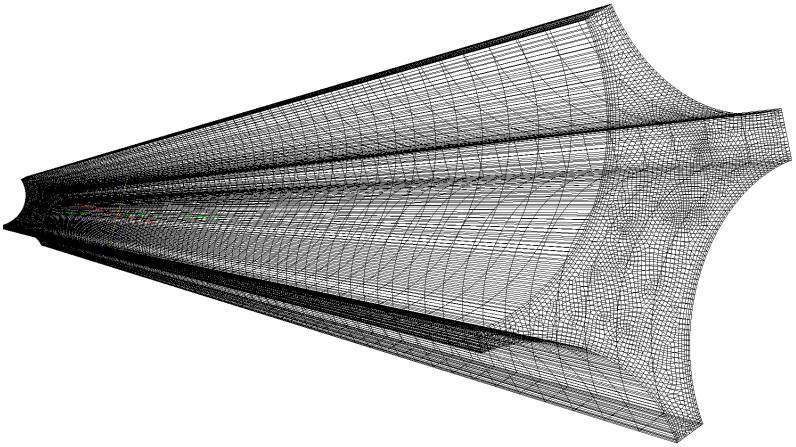

Figure 3: Quadrangular subchannel mesh.

2.2. Simulation

Here are described the simulation conditions adopted. First of all, is the solver used. The solver is, in short, the piece of software which solves the equations representing the physical problem. For this problem the solver chosen is `buoyantBoussiesqPimpleFoam`. This solver is aimed to solve transient problems with turbulent flows, incompressible fluids and heat transfer [5].

The three simulations were defined with the same boundary conditions, turbulence model, relaxation factors and control setup. The turbulence model chosen was the $k - \epsilon$ model. It is a more sophisticated and general turbulence model which includes the effects of transport of turbulence properties, by convection and diffusion. The assumption is that the turbulent viscosity is isotropic. This model has been used frequently for low-speed incompressible flows[6], which is the type of flow being simulated in this work.

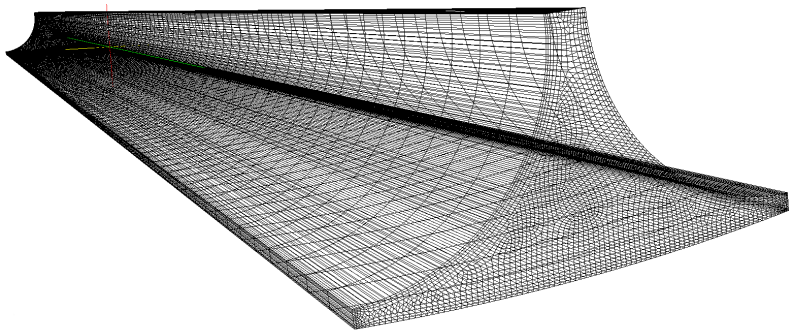

Figure 4: Reflector subchannel mesh.

The reactor's total power considered was $100kW$. The power in each subchannel was obtained supposing the same power in all fuel rods and dividing the total power by the number of subchannels. This average power was then used as the heat flux for fuel walls. The boundary condition used for heat flux is not a default's OpenFOAM boundary condition. Instead, a modified version of turbulent heat flux boundary condition called `TRIGAfuel` which simulates IPR-R1 TRIGA power profile [7] was applied.

The three simulations were carried on the same machine and the time of simulation corresponds to $150s$.

2.3. Post-processing

The post-processing phase includes all steps necessary to consolidate the information from the numerical simulation.

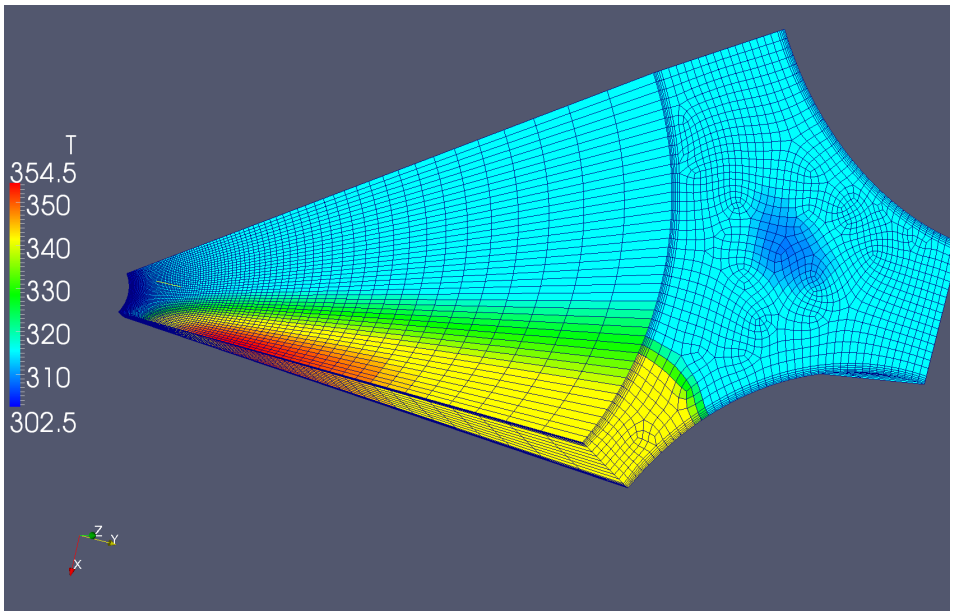

Figure 5: Triangular subchannel fluid temperature.

The results show that there is an important difference in temperature from the fluid flow in subchannels center and the fluid flow in the first row of mesh elements in the domain

not in the heated wall. This is expected due to narrow gap and the uniform velocity distribution in the inlet.

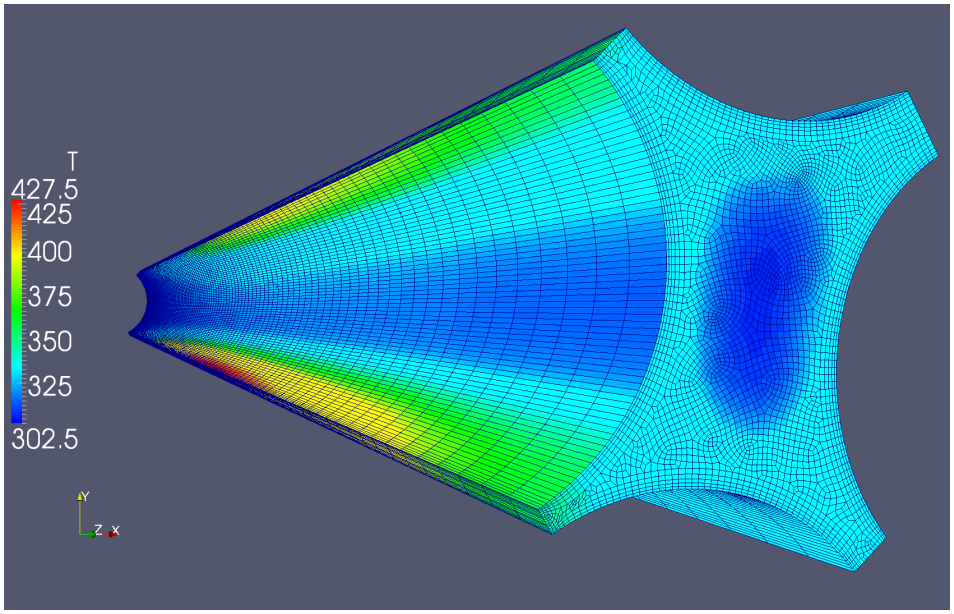

Figure 6: Quadrangular subchannel fluid temperature.

Figures 5, 6 and 7 show, respectively, the triangular subchannel mesh, quadrangular subchannel mesh and the reflector subchannel mesh.

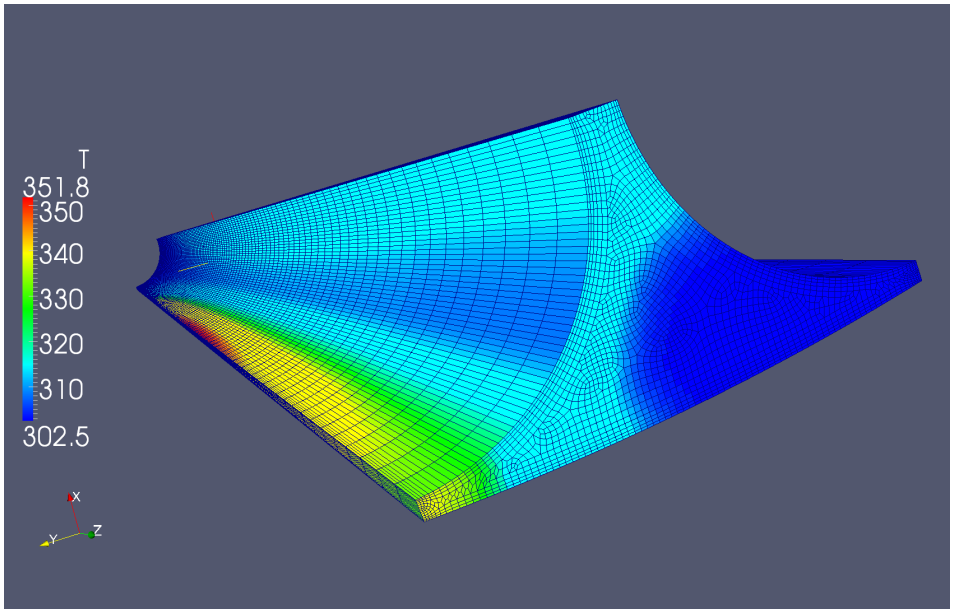

Figure 7: Reflector subchannel fluid temperature.

It is worth remark that in the reflector subchannel presented in Figure 7 there is only on surface which represents a fuel rod. The other surface where the fluid is colder is a graphite element, used to reflect and moderate neutrons.

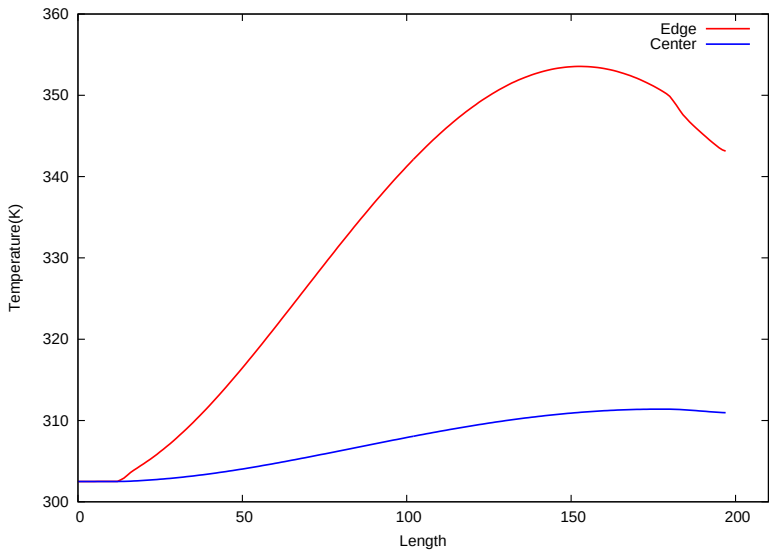

Figure 8: Temperature profile extracted along two lines positioned inside the simulated triangular subchannel.

3. RESULTS

For each subchannel, we took the data along the subchannel length in two positions. One in the fluid near the heated wall center and another in the hottest point near the heated wall. The objective was to check the difference in temperature and the influence of cross-flow absence in this temperature.

In Figure 8 we have the temperature profile in the center and edge of the hottest heated wall. The $\Delta T = 42.61°\text{C}$. In this case, there is no triangular channel in the experimental data to validate this result. However, the difference is above the expected for the simulation conditions and this can be explained by the absence of cross-flow in the hottest edge but also for the direct simulation of the heat flux in the wall. What happens here is that the conjugate heat-flux inside the fuel rod is not simulated and the absence of this physical phenomena causes an overheating of the flow near the wall.

The results shown in Figure 9 and in Figure 10 can be explained the same way as the former figure. The differences in temperature for the quadrangular subchannel are even higher, due to the bigger heated surface for this subchannel. The $\Delta T = 114.61°\text{C}$ is extremely high (and also would lead to ebullition, which is not modeled in the solver used). The $\Delta T = 40.43°\text{C}$ for the reflector subchannel is also high due, mainly, the absence of conjugate heat transfer simulation.

The average temperature in the outlet was of $\Delta T_{quad} = 8.08°\text{C}$, $\Delta T_{triang} = 6.39°\text{C}$ and $\Delta T_{reflect} = 2.03°\text{C}$, agreed in the differences in the experimental work. However, these results are not representative due to the fact of only $150s$ were simulated. In comparison to the experimental data for these subchannels, the values strongly disagree.

Despite the weak results for the temperature comparison, there is an encouraging result.

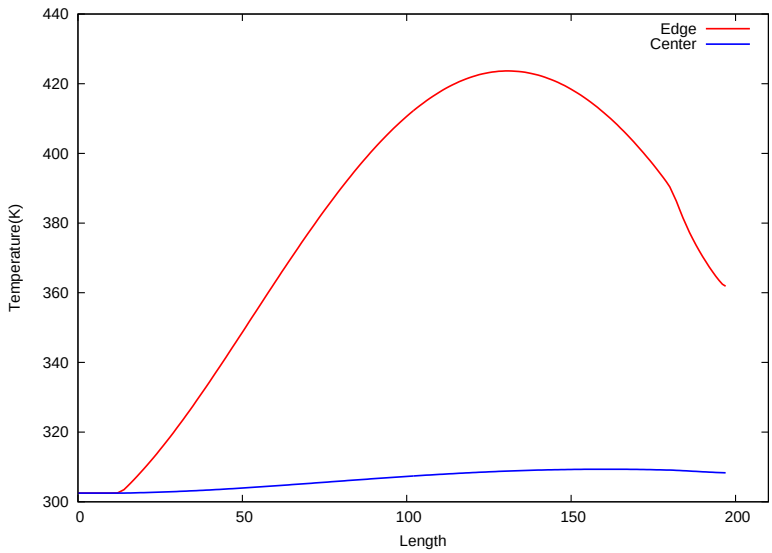

Figure 9: Temperature profile extracted along two lines positioned inside the simulated quadrangular subchannel.

The temperature profiles in all subchannels match in a very good agreement to the temperature variation measured in [2], even better than subchannel codes used in that work. In figure 11 the temperatures simulated in the subchannels near the center of the hottest fuel are plotted in comparison to the temperatures measured in the hottest channel in the reactor, operating in $106kW$ and $265kW$. The discrepancy in the temperature is due to the short time simulation of only $150s$. Despite the differences in value, the curves shape agree with the experimental data.

4. CONCLUSIONS

As a first use of *OpenFOAM* to simulate subchannes fluid flow of IPR-R1 TRIGA the results are encouraging. Although a simplified approach using a fixed value for velocity inflow instead of a real transient approach where the temperature will change according to gravity and fluid density, it was possible to visualize small changes in fluid temperature in the whole subchannels.

The temperatures simulated agreed in shape to the experimental data along the subchannel. Therefore, there were differences in the temperatures mainly due to the absence of cross-flow in the numerical simulation.

The simulation time was of only 150 seconds, not enough to compare the results to a longer time presented in the experimental work (about 28,000 seconds or 8 hours).

However, despite many difficulties faced in this work, *OpenFOAM* presented itself as a flexible - if difficult to use - tool and its open source and free use characteristics are an encouragement to invest time in its use in the nuclear reactors physics field. As many authors in the field attest ([8, 9]), CFD is an indispensable technology for the coming years in the nuclear engineering field.

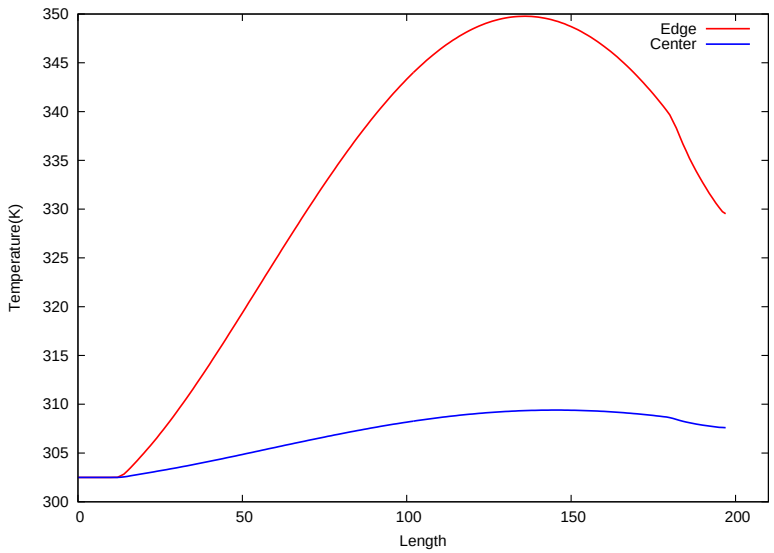

Figure 10: Temperature profile extracted along two lines positioned inside the simulated reflector subchannel.

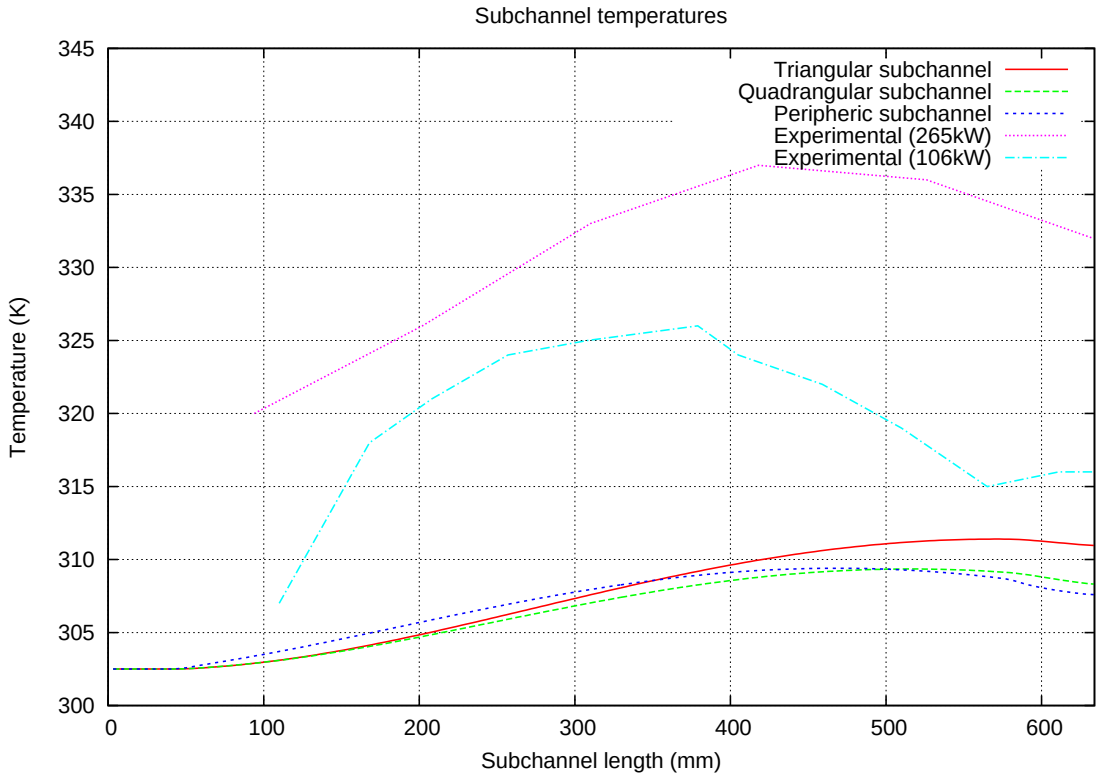

Figure 11: Experimental and simulated data. The simulated data obtained after 150 seconds of simulation.

The simulation performed in this work can be improved in different ways. The first possibility is to add cross-flow to the symmetry planes used in the fluid-fluid intersection in the simulation. Another interesting aspect is to improve the `TRIGAfuel` boundary condition to add fuel materials and conjugate heat transfer. More complex, but also desirable feature, is the neutronic simulation. This could be achieved with a punctual kinetics simulation inside the boundary condition. A careful definition of cross-section must be done in this case, using a lattice code like WIMS [10].

Other future works include the change in this simulation from fixed velocity in the inlet to a velocity dependent to the fluid properties. This kind of transient simulation would reflex the reactor start.

ACKNOWLEDGMENTS

The authors are grateful to the Brazilian research funding agencies, CNEN, CNPq, CAPES and FAPEMIG, for the support.

REFERENCES

1. *OpenFOAM C++ Documentation*, 2013.
2. A. Z. Mesquita, *Experimental Investigation of Temperatures Distributions in the IPR-R1 TRIGA Nuclear Research Reactor*. PhD thesis, Universidade Estadual de Campinas - Faculdade de Engenharia Química, 2005.
3. H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*. Pearson Education Limited, 2nd ed., 2007.
4. M. A. Veloso, "Thermal-hydraulic analysis of the IPR-R1 TRIGA reactor in 250kW," Tech. Rep. NI-EC3-05/05, Centro de Desenvolvimento da Tecnologia Nuclear da Comissão Nacional de Energia Nuclear - CNEN/CDTN, May 2005.
5. *OpenFOAM - the Open Source CDF Toolbox*, February 2013. Version 2.2.0.
6. T. J. Chung, *Computational Fluid Dynamics*. Cambridge University Press, 1st ed., 2002.
7. V. V. A. Silva, A. A. C. Santos, P. S. B. L. D. Silva, A. Z. Mesquita, and C. P. B. Lima, "TRIGA fuel element simulation using OpenFOAM," in *European Research Reactor Conference 2013 - Transactions*, pp. 558–566, European Nuclear Society, 2013.
8. E. Krepper, B. Končar, and Y. Egorov, "CFD modelling of subcooled boiding - Concept, validation and application to fuel assembly design," *Nuclear Engineering and Design*, vol. 237, pp. 716–731, 2007.
9. K. Jareteg, *Development of an Integrated deterministic neutronic/thermal-hydraulic model using a CFD solver*. PhD thesis, Departmente of Applied Physics - Chalmers University of Technology, 2012.
10. T. D. Newton and J. L. Hutton, "The next generation WIMS lattice code: WIMS9," in *Proceedings of PHYSOR*, American Nuclear Society, October 2002.