2015 International Nuclear Atlantic Conference - INAC 2015
São Paulo, SP, Brazil, October 4-9, 2015
ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR - ABEN
ISBN: **978-85-99141-06-9**

A CFD NUMERICAL MODEL FOR THE FLOW DISTRIBUTION IN A MTR FUEL ELEMENT

Delvonei Alves de Andrade[1], Gabriel Angelo[1,3], Edvaldo Angelo[2], Pedro Henrique Di Giovanni Santos[1], Fabio Branco Vaz de Oliveira[1], Walmir Maximo Torres[1], Pedro Ernesto Umbehaun[1], José Antonio Batista de Souza[1], Antonio Belchior Junior[1], Gaianê Sabundjian[1], Adelk de Carvalho Prado[1]

[1] Instituto de Pesquisas Energéticas e Nucleares (IPEN / CNEN - SP)
Centro de Engenharia Nuclear
Av. Professor Lineu Prestes 2242
05508-000 São Paulo, SP
delvonei@ipen.br; dpedro_digiovanni_s@hotmail.com; fabio@ipen.br; wmtorres@ipen.br; umbehaun@ipen.br; jasouza@ipen.br; abelchior@ipen.br; gdjian@ipen.br; acprado@ipen.br

[2] Universidade Presbiteriana Mackenzie
Rua da Consolação 930
01302-907 São Paulo, SP
eangelo@mackenzie.br

[3]Fundação Educacional Inaciana - FEI
Av. Humberto de Alencar Castelo Branco 3972
09850-901 São Bernardo do Campo, SP
gangelo@fei.edu.br

ABSTRACT

Previously, an instrumented dummy fuel element (DMPV-01), with the same geometric characteristics of a MTR fuel element, was designed and constructed for pressure drop and flow distribution measurement experiments at the IEA-R1 reactor core. This dummy element was also used to measure the flow distribution among the rectangular flow channels formed by element fuel plates. A CFD numerical model was developed to complement the studies. This work presents the proposed CFD model as well as a comparison between numerical and experimental results of flow rate distribution among the internal flow channels. Numerical results show that the model reproduces the experiments very well and can be used for the studies as a more convenient and complementary tool.

1. INTRODUCTION

IEA-R1 is a 5 MW pool type research reactor located at IPEN, São Paulo. Its core is composed of MTR (Material Testing Reactors) fuel elements with downward flow. Each fuel element has 18 fuel plates assembled on two lateral support plates, forming 17 independent internal flow channels. The reactor core is assembled in a 5x5 square matrix with 20 fuel elements, 4 control fuel elements and a central Beryllium irradiator. The safe operation of the reactor is guaranteed maintaining suitable safety margins in any operational conditions. These safety margins (DNBR, ONB, CHF and maximum surface temperature) are verified in the thermal-hydraulic analysis (THA) of the core. To perform the THA it is necessary to know some parameters, such as: heat flux distribution, geometric characteristics, material properties and flow rates through the fuel elements. The uncertainties of these parameters are also necessary for the THA. The flow rate through the fuel elements is an important parameter and it is difficult to determine due to the geometric complexity of the core. The International Atomic Energy Agency, IAEA, TECDOC 233 [1] suggests that the flow rate through the fuel elements is the total reactor primary flow rate divided by the number of fuel elements. This value is not completely true because the core has fuel elements and other components as discussed in Torres et al. [2]. A dummy fuel element (DMPV-01) [3] was designed and constructed to measure the core flow rate distribution and also the flow rate distribution among the internal fuel elements channels at the IEA-R1. A numerical model was also developed using ANSYS-CFX®[4] as complementary tool for the flow distribution study in the fuel element as described in the next section. This paper considers this model.

2. EXPERIMENTS

Dummy fuel element DMPV-01, Fig.1, was used to perform an experiment to measure the flow rate distribution among the flow channels. It was assembled in the experimental circuit, Fig. 2. A calibrated orifice plate and a calibrated differential pressure transducer (DPT1) were used to measure the total flow rate through the dummy element and a type K thermocouple measured the fluid temperature during the experiments for water properties corrections. Two pressure probes were constructed with 2.5 mm diameter tube in stainless steel with two pressure taps 475 mm distant. They were assembled inside the flow channels of the DMPV-01 in central region to measure pressure drop together with two calibrated differential pressure transducers (DPT2 and DPT3). Figure 3 shows two adjacent fuel elements and dimensional details of the flow channels. The experiments were performed for three flow rates: 6.1, 5.2 and 4.0 kg/s. The channel flow velocities and flow rates were calculated using pressure drop equation for closed channels.

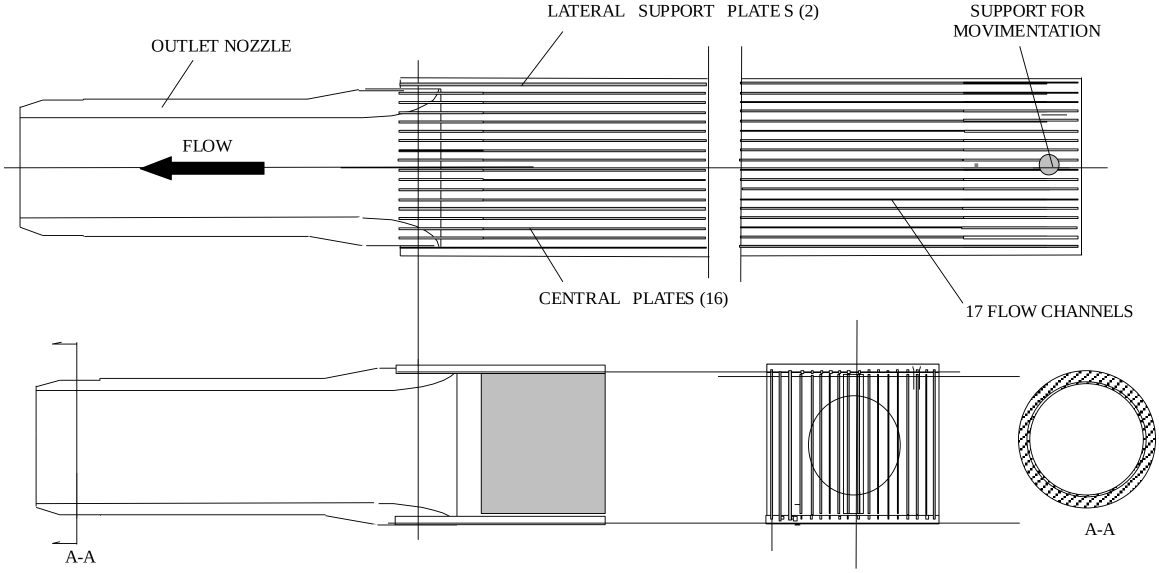

Figure 1. Instrumented dummy fuel element DMPV-01

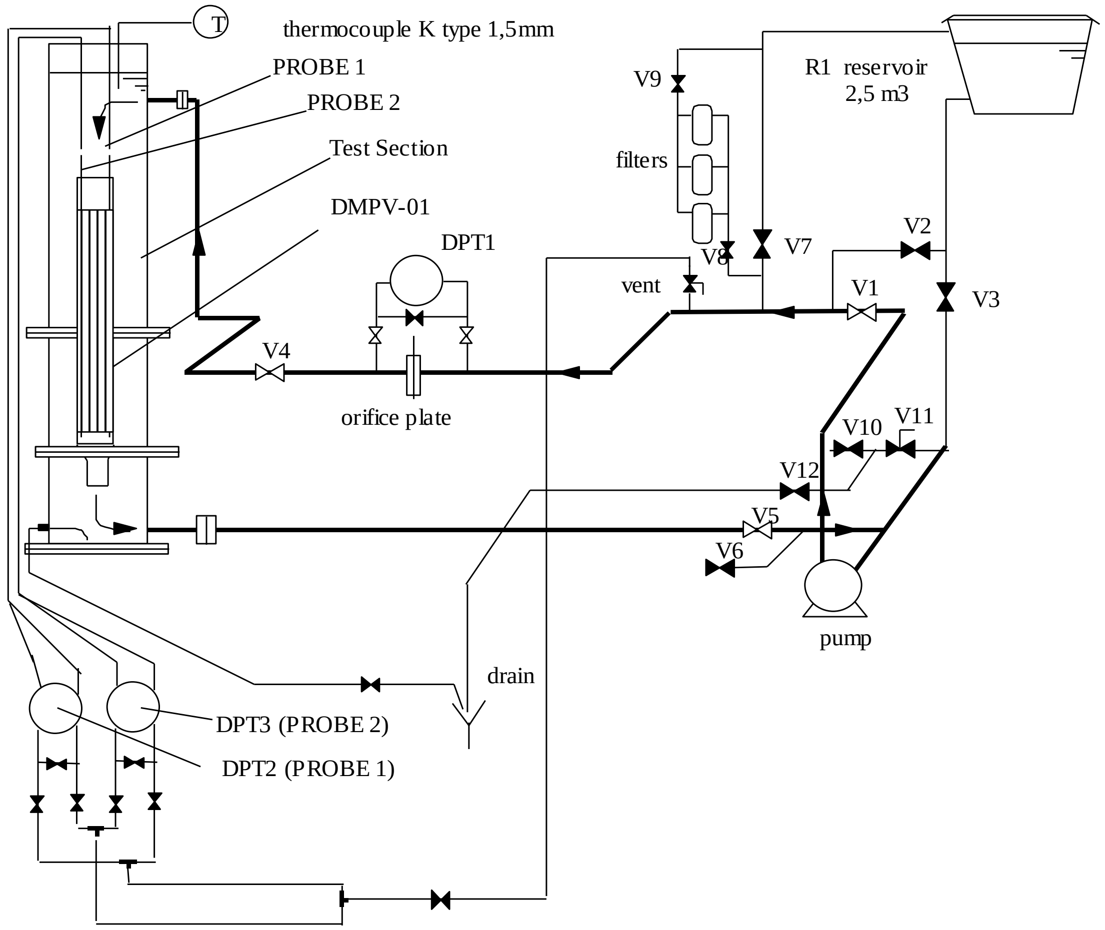

Figure 2. Experimental circuit and DMPV-01

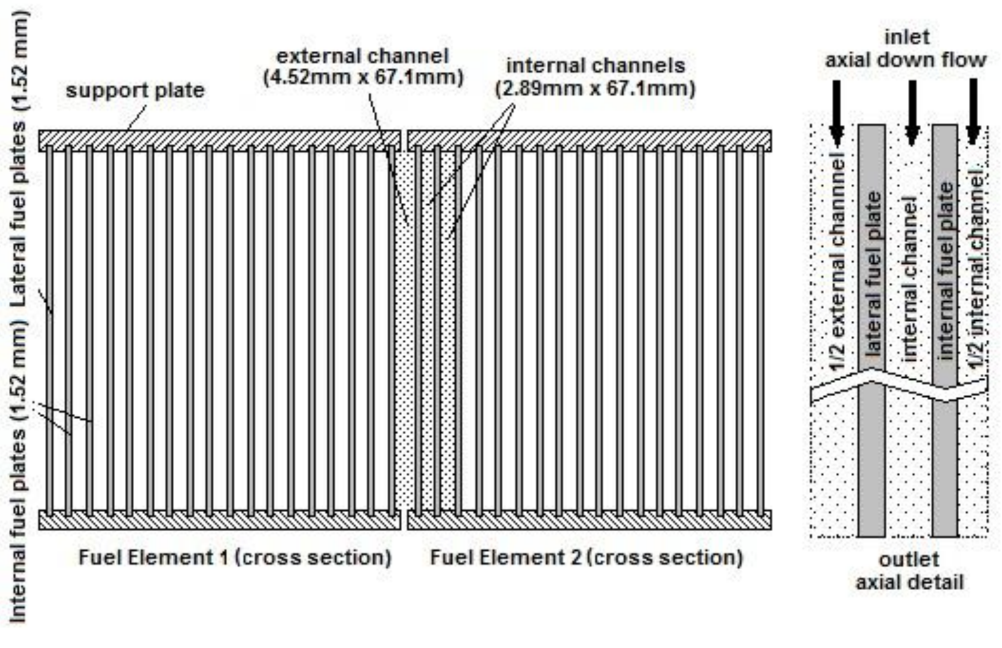

Figure 3. Cross section of two adjacent fuel elements

3. MODEL

The tridimensional fuel element presents symmetry. To save computational time, it is considered only 1/4th of the geometry for the model, as presented in Fig. 4, simulated geometry, Fig. 5, inlet region detail and Fig. 6, outlet region detail.

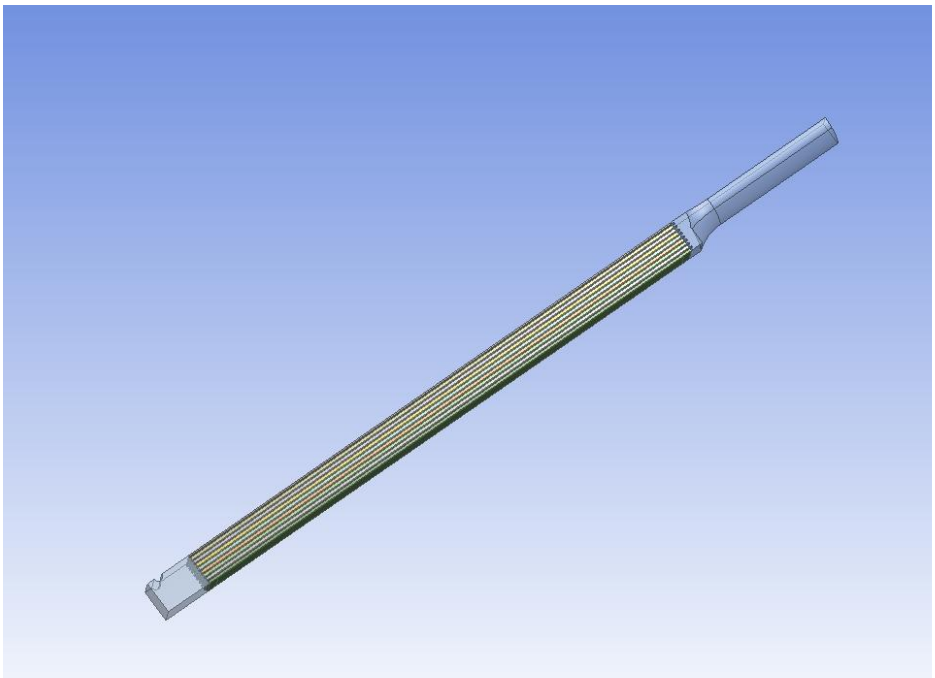

Figure 4 - Geometric model

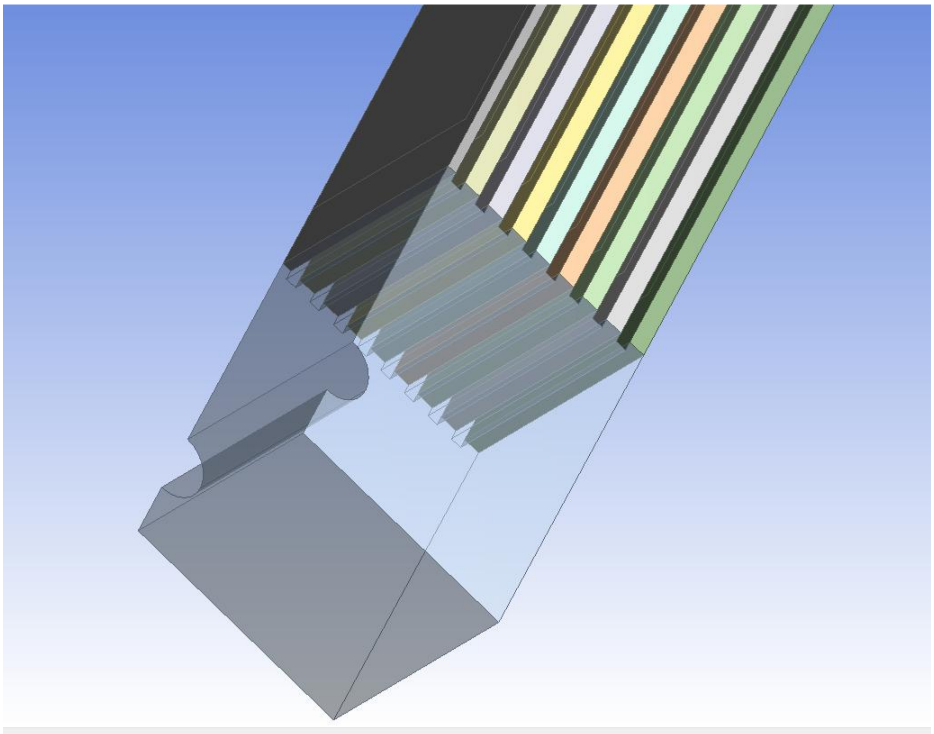

Figure 5 - Inlet region detail

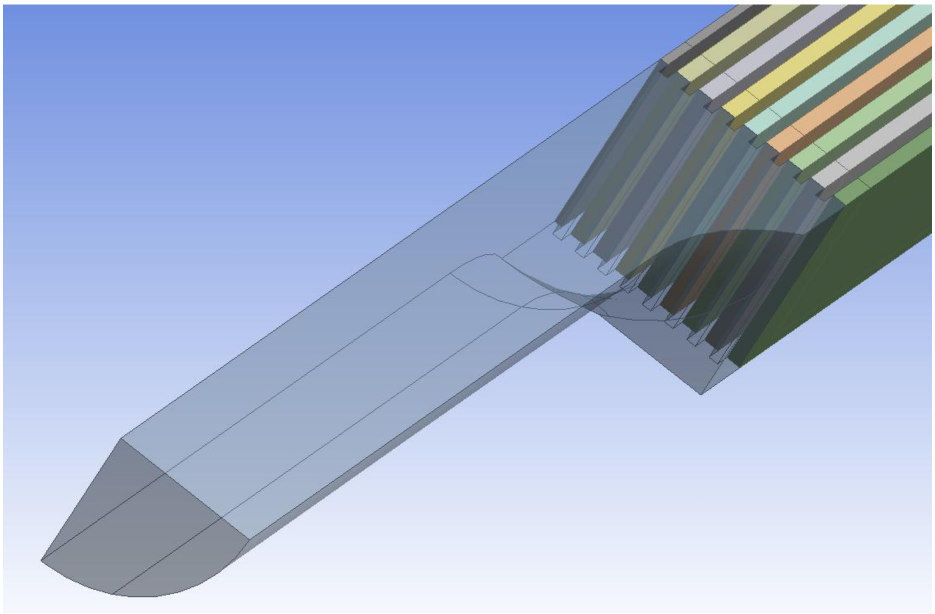

Figure 6 - Outlet nozzle detail

A tridimensional model was developed based on the geometry presented in Fig. 1 using the finite volume method applied to a non structured mesh, Maliska, C. R. [5], Anderson, J. D. [6]. Inflation was considered for all regions next to the walls. Turbulence model considers the standard $\kappa - \varepsilon$ model [7].

Stern, F. at al. [8] and Wilson, R. V. at al. [9] present an approach to define and verify mesh. They discuss about mesh dependency on the results focusing the element size definition in order to validate the CFD models.

The methodology considers, for the same boundary condition, an increase of the mesh density using predefined ratios. This procedure must be performed in such a way that property variation or small variations are not present. When this condition is satisfied the solution is independent of the mesh.

This methodology was used in this paper and the mesh in Figures 7,8 and 9 present the last loop of this interaction. The mesh study was performed using the CFX mesher and resulted in a final mesh, with 3820023 elements.

All simulations were performed in a Xeon dual processor E5520 family, 2.26GHz, with 48 gigabytes of memory.

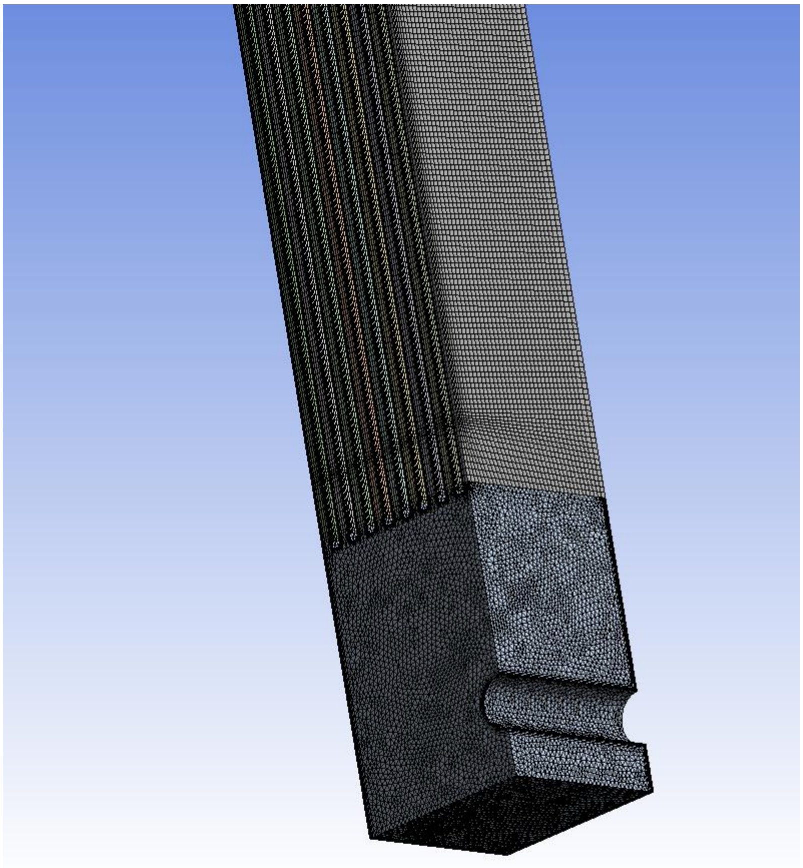

Figure 7 - Mesh - inlet region

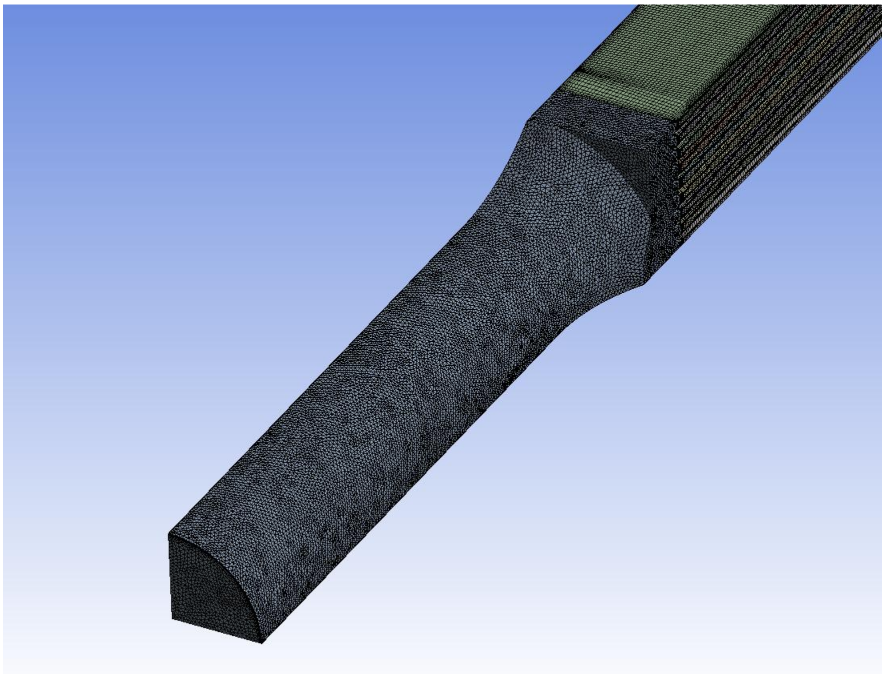

Figure 8 - Mesh - outlet region

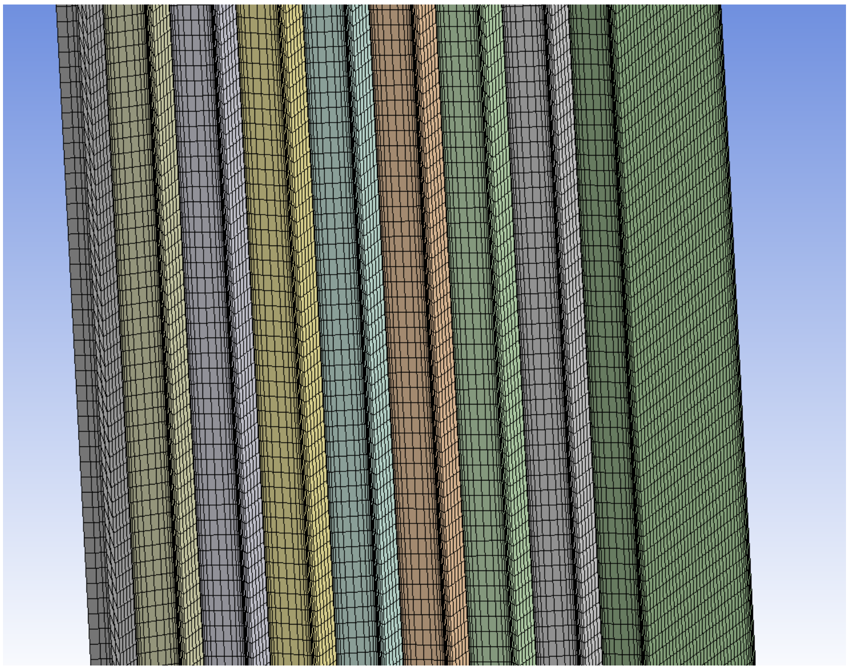

Figure 9 - Mesh - internal channels (fuel plates region)

Water at a reference pressure of 1.5 bar is used in the simulation. As mentioned before, the standard k-epsilon is taken for the turbulence model. At the inlet, three mass flow rates (6.1, 5.2 and 4.0 kg/s) were considered with a static temperature of 27 ºC. At the outlet a zero relative pressure and symmetry at the central region (cuts) were used for all cases simulated in this paper. A high resolution scheme with first order for the turbulence and a residual target of 1.0E-5 was used as convergence criterion.

4. RESULTS

Figure 10 presents the flow distribution among the flow channels for three different flow rates, experimental and numerical results. One can observe that the flow rate in the peripheral channel is lower (10 to 15%) than the average value and depends on flow rate through the fuel element. Taesung, H. and Garland, W. J. [10] present similar results in their studies for McMaster nuclear reactor.

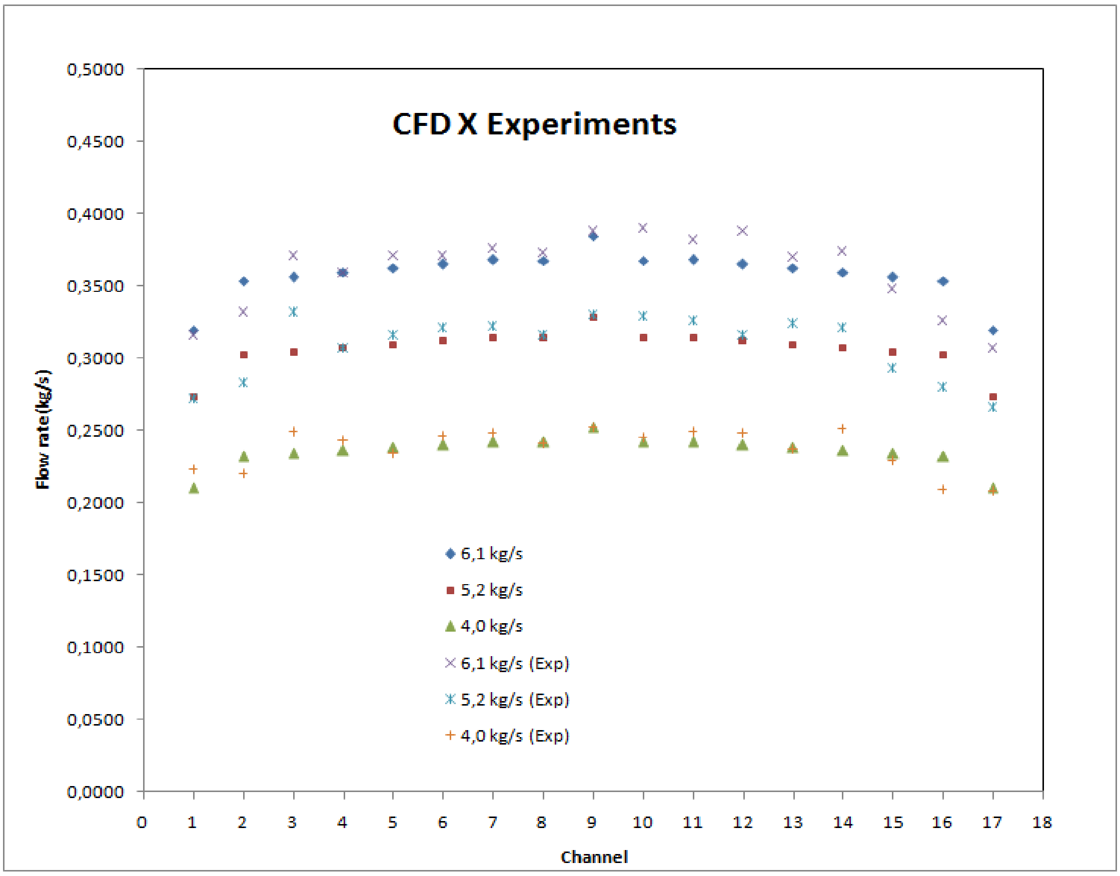

Figure 10 - Fuel element channels flow distribution - experimental x numerical results

4. CONCLUSIONS

This paper presented a comparison between numerical and experimental results of flow distribution among the flow channels. The numerical CFD model proposed in this paper has shown that the model reproduces the experiments very well and can be used as a more convenient or a complementary tool in the determination of the fuel element flow distribution studies. One can observe that the flow rate in the peripheral channel is lower (10 to 15%) than the average value and depends on flow rate through the fuel element. This difference is due to inlet and outlet effects in the element. This behavior is captured very well by the CFX model.

REFERENCES

1. IAEA – TECDOC – 233, “Research Reactor Core Conversion from Use of High Enriched Uranium to Use Low Enriched Uranium Fuel Handbook”, International Atomic Energy Agency, Vienna, Austria, 1980.
2. Torres, W. M.; Umbehaun, P. E., Andrade, D. A., Souza, J. A. B. C. D., “A MTR fuel element flow distribution measurement preliminary results,”*International meeting on reduced enrichment for research reactor* - RERTR, Chicago, USA, Vol. 1, pp.1-6, 2003.

3. Lima, R. M., Oliveira, F. S., “Relatório Descritivo de Fabricação do Elemento DMPV-01”, IPEN Internal Report No.RDF-DMPV-01/01, 2000.
4. ANSYS-CFX – 11.0, *User Manual*, ANSYS Inc., Canonsburg, Pennsylvania – United States, 2007.
5. Maliska, C. R., Transferência de Calor e Mecânica dos Fluidos Computacional, LTC – Livros Técnicos e Científicos Editora S. A., Rio de Janeiro, RJ – Brasil, 2004.
6. Anderson, John D., Computational Fluid Dynamics – The Basics with Applications, McGraw-Hill, New York, NY – United States, 1995.
7. Launder, B.E. and Spalding, D. B., The Numerical Computation of Turbulent Flows, *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, pp. 269-289 (1974).
8. Stern, F. et al., Comprehensive Approach to Verification and Validation of CFD Simulations-Part 1: Methodology and Procedures, Journal of Fluids Engineering, Vol. 123, pp. 793-802, 2001.
9. Wilson, R. V. et al., Comprehensive Approach to Verification and Validation of CFD Simulations - Part 2: Application for Rans Simulation of a Cargo/Container Ship, Journal of Fluids Engineering, Vol. 123, pp. 803-809, 2001.
10. Taesung, H. and William, J. G., Hydraulic study of turbulent flow in MTR-type nuclear fuel assembly, Nuclear Engineering and Design, 236, pp. 975-984, 2005.