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**

PRESSURE DROP CALCULATION IN A FUEL ELEMENT OF A POOL TYPE REACTOR

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

[1]Instituto de Engenharia Nuclear (IEN/CNEN – RJ)
Av. Hélio de Almeida, 75, Cidade Universitária
21941-906 Rio de Janeiro, RJ
vlassance@ien.gov.br
malu@ien.gov.br

[2]Programa de Engenharia Nuclear, COPPE
Universidade Federal do Rio de Janeiro
Av. Horácio Macedo, s/n, bloco G
21941-598 Rio de Janeiro, RJ
afoliveira@ien.gov.br

ABSTRACT

Even with the advances of hardware in computer sciences, sometimes it is necessary to simplify the simulation in order to optimize the results given the same calculation runtime. The object of this study is a thermodynamic analysis of the core of a pool type research reactor, focusing on natural circulation. Due to the high geometrical complexity of the core, the scale transfer process becomes an essential step to the thermodynamic study of the reactor. This process takes place by determining the effective equivalent properties obtained from a detailed simulation of the core and transferring them to a porous medium having a coarse mesh while preserving the overall characteristics. In this way, it will be able to obtain the quadratic resistance coefficient K_Q by calculating the pressure drop inside the fuel element. To observe in detail the behavior of this flow, longitudinal and transversal cross sections will be made in different points, thereby observing the velocity and pressure distributions. The analysis will provide detailed data on the fluid flow between the fuel plates enabling the observation of possible critical points or undesired behavior. The whole analysis was made by using the commercial code ANSYS CFX ver. 12.1. This is study will provide data, as a first step to enable future simulations which will consider the entire reactor.

1. INTRODUCTION

Pool type reactors that use natural circulation and passive cooling are highly dependent on the geometry of its components, since the water should flow easily throughout the nucleus in order to cool it down. In this context, parameters as: power, neutron flux, water flow and geometry that should be studied, so such design features can be known and conformed to the project targets. Being familiar with the behavior of the flow of water within the fuel element

enables to predict the temperature distribution, estimate possible critical points and undesirable behavior so as to analyze the scenario as whole.

The pressure drop is defined as the difference of pressure between two points in fluid transport system. The pressure drop increases proportional to the shear forces within the network. The larger the numbers of physical barriers which hinder the fluid flow, the larger the pressure drop. This is exactly what happens in this case, once the fuel element is composed by several parallel plates sustained by lateral supports, which leads to the formation of innumerous narrow channels where the coolant passes. Therefore, the friction between the water and the wall plates will have great importance.

The modern computer modeling techniques allow the description of the complex geometry of a pool type reactor at a detail level sometimes incompatible with the computing capacity of available computers. Due to the high geometrical complexity of the core, the scale transfer process becomes an essential step to the thermodynamic study of the reactor. This process takes place by determining the effective equivalent properties obtained from a detailed simulation and transferring them a porous medium having a coarse mesh while preserving the overall characteristics.

2. OBJECTIVES

A nuclear research reactor is composed of sophisticated geometric structure. If it was made a single model containing all the details, a simulation of high complexity would be created, which implies great computational cost. Given this situation, it was decided to subdivide the simulation into two parts. The first, whose product is the target of this study, is to generate a global coefficient of pressure drop in the nucleus from a detailed model of the fuel element. The second part – left for a further work – will be to model the entire reactor and use this coefficient to represent the pressure in the nucleus. This feature reduces the complexity of the simulation reactor without loss of information.

To observe in detail the behavior of this flow, longitudinal and transversal cross sections will be made in different points, thereby observing the velocity and pressure distributions. On the basis of this information, one can predict possible critical points inside the element. That is, the points where the flow velocity is low, thus heat removal and where vortexes are generated.

3. METODOLOGY

3.1. Symmetry

In 3D modeling, the symmetry planes allow the reduction of domain size, optimizing and simplifying the simulation analysis. The simple definition of this plan will increase the grid

resolution for the same CPU runtime. In the present case, we adopted the ¼ symmetry as shown in Figure 1.

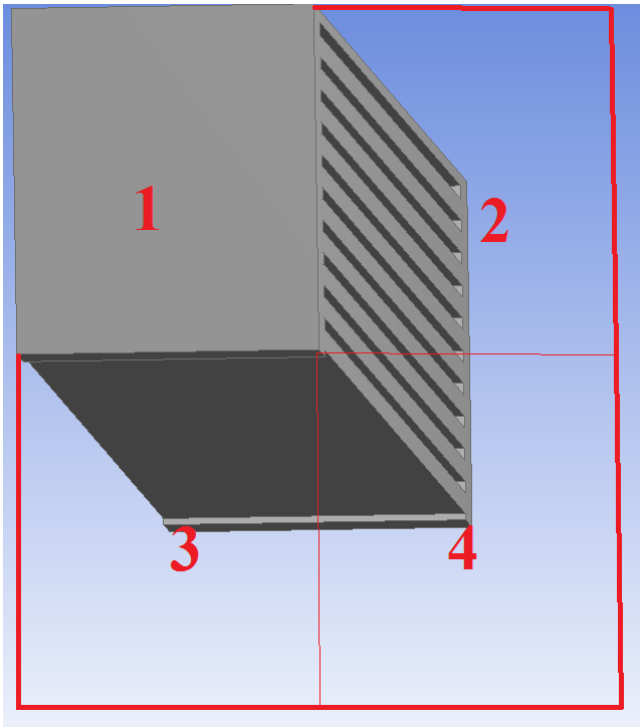

Figure 1: ¼ symmetry of the fuel element.

3.2. Porous medium

In the present case, the target is to calculate the quadratic resistance coefficient K_Q by the formulation of a porous medium [1] to replace the friction forces due to the core geometry. Its definition is a solid with voids distributed more or less uniformly throughout the bulk of the body. So, the reactor core will be considered a porous medium to simplify and optimize further calculations, since there's no need to consider its complex structure in order to simulate the natural convection in the reactor pool.

$$\frac{\partial p}{\partial x_i} = -K_Q |\boldsymbol{U}| U_i \tag{1}$$

So

$$K_Q = -(|\boldsymbol{U}|\, U_i)/(\partial p/\partial x_i) \tag{2}$$

Where $^{\partial p}/_{\partial x}$ stands for the pressure drop and **U** is the velocity.

3.3. Turbulence Models

The baseline two-transport-equation model solving for kinetic energy k and turbulent dissipation ε [2]. Turbulent dissipation is the rate at which velocity fluctuations dissipate. This is the default k–ε model. Coefficients are empirically derived; valid for fully turbulent flows only. In the standard k- ε model, the eddy viscosity is determined from a single turbulence length scale, so the calculated turbulent diffusion is that which occurs only at the specified scale, whereas in reality all scales of motion will contribute to the turbulent diffusion. The k-ε model uses the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. Performs poorly for complex flows involving severe pressure gradient, separation, and strong streamline curvature [3]. The author's self-investigation for flow through a pipe is consistent with the statements that this model is valid for flows without separation and for fully turbulent flow, which doesn't fit our case.

The k–ω model considers two-transport-equation model solving for kinetic energy k and turbulent frequency ω. This model allows for a more accurate near wall treatment with an automatic switch from a wall function to a low-Reynolds number formulation based on grid spacing. Demonstrates superior performance for wall-bounded flows. Shows potential for predicting transition. As the previous model, the k–ω model also uses the gradient diffusion hypothesis. Solves one equation for turbulent kinetic energy k and a second equation for the specific turbulent dissipation rate (or turbulent frequency) ω.

Finally, the adopted model was the Shear Stress Transport (SST). Because it is a variant of the standard k–ω model which combines the original k-ω model for use near walls and the standard k–ε model away from walls by the means a blending function, and the eddy viscosity formulation is modified to account for the transport effects of the principle turbulent shear stress [4]. The transition and shearing options are borrowed from standard k–ω.

3.4. Simulation data and boundary conditions

The stationary analysis was made based considering the parameters in table 1.

Table 1. Simulation Data

Domain material:	Water
Temperature	25°C
Morphology	Continuous
Reference pressure	1,000 atm
Heat transfer model	Isothermal
Turbulence model	SST
Mass flow rate	0,227 kg/s

This mass flow rate was adopted from a previous simulation where the entire pool of the reactor was considered in the moment when the reactor is shut down, thus the pumps are turned off and the natural convection takes place in the cooling of the core. The wall surfaces are assumed to be smooth and no-slip conditions.

3.5. Mesh generation

The grid of the fuel element was generated by using the CFX-MESH 12.1 software. The three dimensional mesh was created using advancing front and inflation 3D according to the following parameters shown in Table 2.

Table 2: Inflation parameters

N° of inflated layers	10
Expansion factor	1,2
Minimum internal angle	2,5°
Minimum external angle	10,0°

Considering the reduced width of the channels (2,45mm) compared to the length (600mm) the cell size could not be reduced because sliver faces would be generated. A sliver face may result in an undesirable mesh. In Figure 2 we are able to see the created mesh.

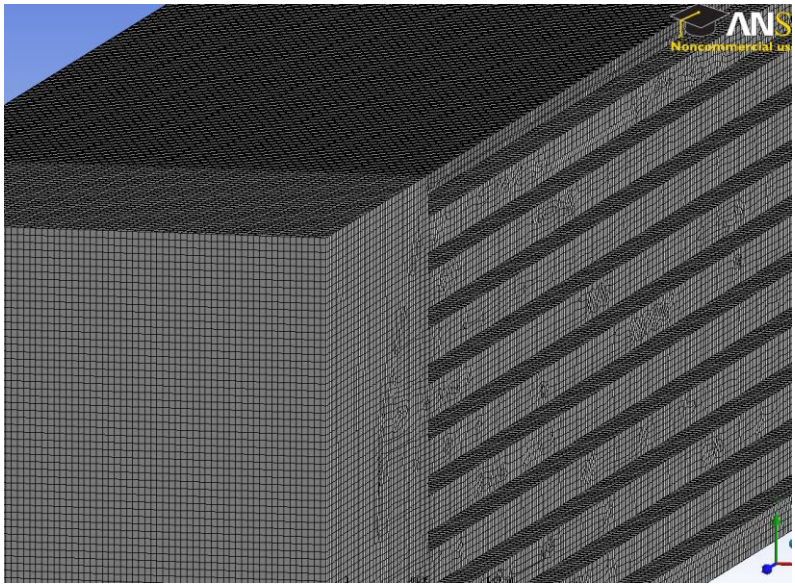

Figure 2: Detail of the mesh.

There were generated 8449680 nodes and 7093548 elements. For the convergence, there were set a maximum of 200 iterations with the criteria of 1E-7 RMS. All the calculations were produced by the code Ansys CFX 12.1, using a cluster formed by 12 CPUs containing 12 cores each at 3.0 MHz

4. RESULTS

The results were made focusing the analysis of the pressure and velocity distributions inside the fuel element. Figure 3 shows a complete image of the fuel element highlighting the pressure along the water channels.

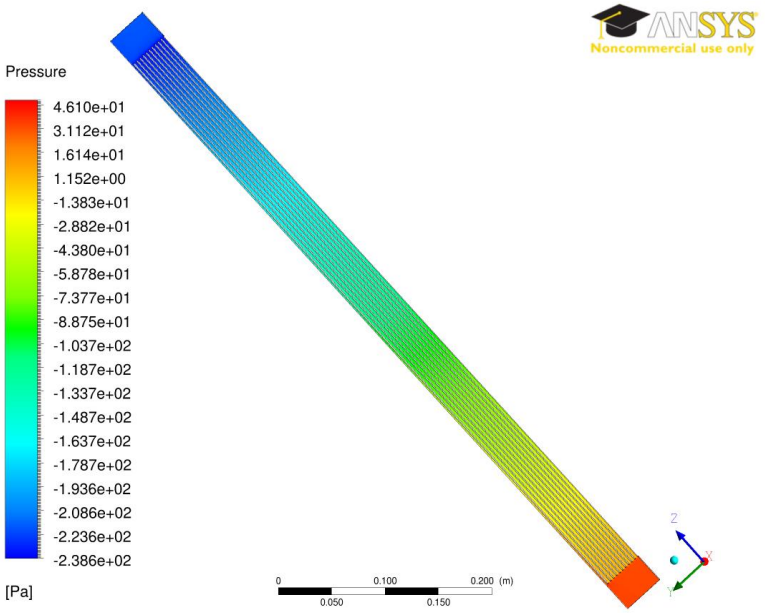

Figure 3: Pressure along the fuel element.

The next images, Figure 4 and Figure 5, show a zoom of the pressure and velocity distribution respectively, at the input of the fuel element.

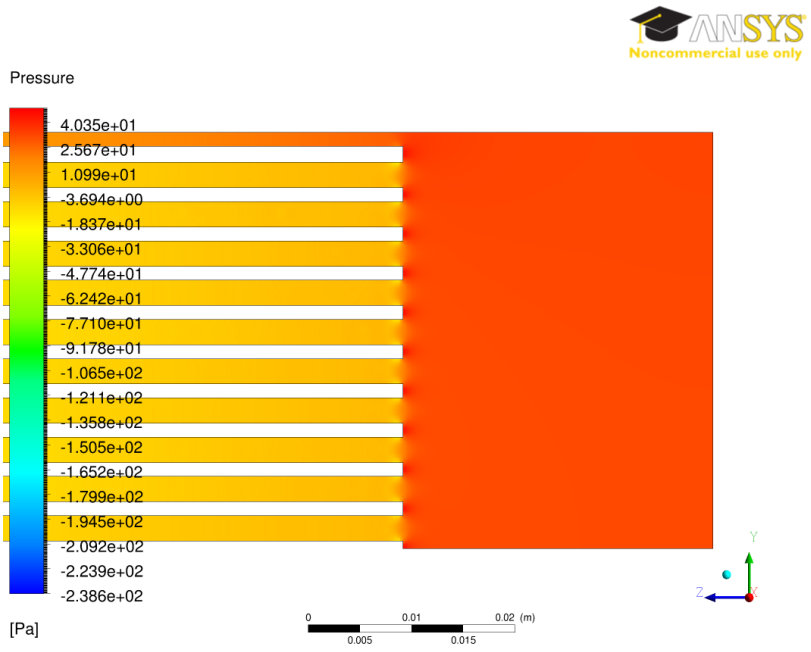

Figure 4: Pressure at the input of the fuel element.

In Figure 4, the first coolant channel, considering the upside down direction, doesn't show a significant pressure drop, meaning that the velocity inside of it has not been increased as in the other channels. In Figure 5, the velocity distribution is plotted.

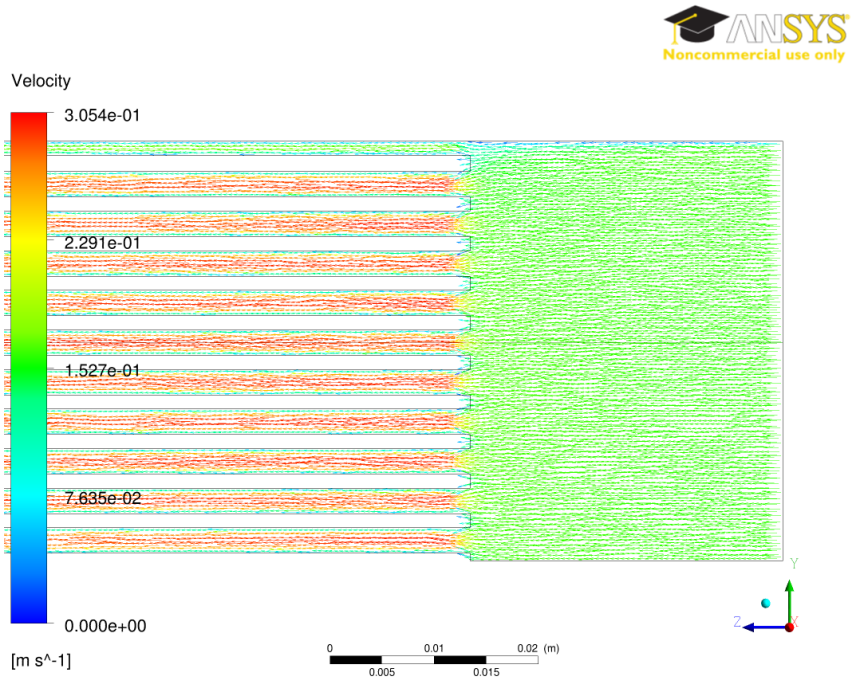

Figure 5: Velocity distribution at the input of the fuel element.

Figure 6 shows the velocity distribution at the output.

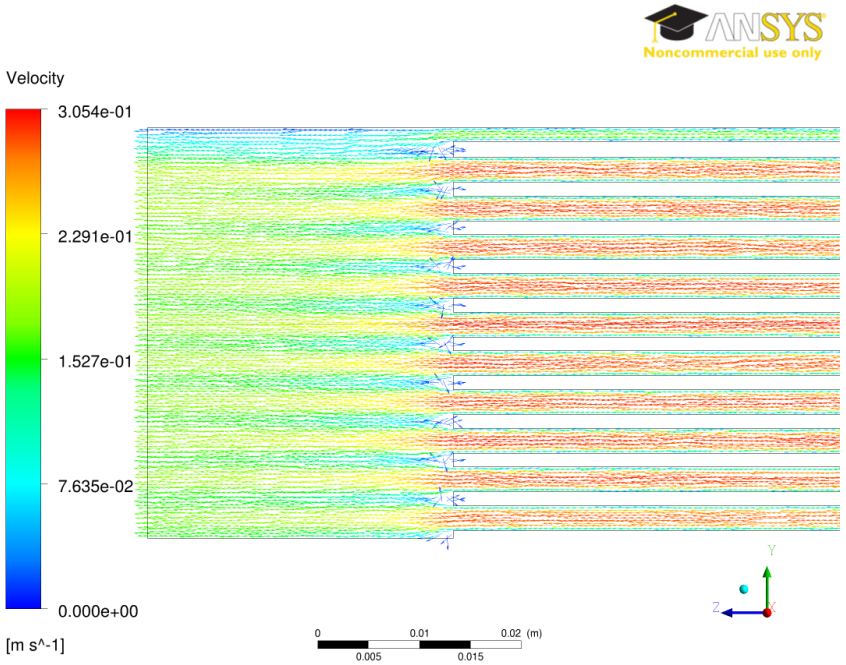

Figure 6: Velocity distribution at the output of the fuel element.

It can be noted in Figure 6 that vortexes are formed in the end of the fuel plates where the flows are reattached.

The total pressure drop calculated between the input and the output was 241,7 Pa.

5. CONCLUSIONS

The results show to be in good agreement with the expected behavior of the fluid inside the fuel elements. The effects of the pressure drop could be clearly noticed in the model. The behavior of the fluid inside the fuel shows that the first channel won't have a good cooling efficiency due to the low flow velocity. This effect is due to the shear currents in the near wall, given that this channel is the one that is closer the fuel wall. This could generate an overheating of the surrounding fuel plates.

Another fact that should be noticed is the formation of the vortexes in the end of the fuel plates. On those points the velocity is close to zero, thus the heat removal. This can become a problem given that this is the region where the coolant should reach one of the highest temperatures in the fuel, since it has been heated all way along the fuel.

Besides these two side remarks, the main goal was reached: the pressure drop has been calculated and now it will be possible to generate a precise quadratic resistance coefficient to be used in the porous medium. Accurate data will be provided in order to enable the simulation of the entire reactor.

REFERENCES

1. J. Tu, G. H. Yeoh, C. Liu, *Computational fluid dynamics: a practical approach*, Elsevier, Oxford, UK (2008).
2. Ansys, Inc, *Ansys CFX Tutorials*, Ansys Inc, Canonsburg, USA (2010)
3. D. C. Wilcox, "Reassessment of the scale-determining equation for advanced turbulence models", *AIAA Journal*, Vol. 26, pp 1299-1310 (1988)
4. D. C. Wilcox, *Turbulence Modeling for CFD,* DCW Intustries, La Cañada, California (2006)