



---

## Moderator Temperature Reactivity Feedback in the Calandria Tank of Embalse Nuclear Power Plant based on a CFX Analysis

José D. Jiménez Díaz, Alejandro I. Lazarte

Nuclear Regulatory Authority (ARN), Av. Libertador 8250, CABA, BNP 1429C, Argentina  
e-mail: [jjimenez@arn.gob.ar](mailto:jjimenez@arn.gob.ar), [alazarte@arn.gob.ar](mailto:alazarte@arn.gob.ar)

---

**Abstract** In this paper is presented the temperature, fluid velocity distributions in moderator tank (calandria tank) by means of ANSYS CFX 15.0 software, and the moderator reactivity feedback of a CANDU-6 reactor in steady state conditions. The domain is shaped by the calandria vessel, the 380 horizontal calandria tubes, the 8 inlet distributors and the 2 outlet pipes. Though, some of reactor internals are not considered. The geometrical data is based on the designer blueprints, and the inlet flow, outlet pressure and radial and axial thermal power profiles were imposed as realistic boundary conditions. Different operation modes were evaluated, and the flow pattern within the calandria vessel was obtained. The results was compared with the open literature such as other 3 dimensional software and experimental test. The comparison showed the flow pattern in the calandria vessel could be dominated by momentum, buoyant or both of them forces. Finally It was determined that for the operative conditions of the CANDU-6 the flow pattern in the calandria is dominated by momentum and buoyant forces, and the moderator reactivity feedback map was obtained by the moderator three dimensional temperature distributions.

**Keywords** Computational Fluid Dynamics, Moderator, Calandra, CANDU-6

---

### 1. Introduction

The use of CFD software for preliminary designs and optimization of nuclear components has increased in the recent years. The CFD tools shall be considered as a complement to existing engineering tools, usually one dimensional, which under certain assumptions have been validated by experimental data and verified with other systems codes assumed validated. CFD calculations allow users to analyze in depth, with the corresponding models and approaches within their range of validity, complex flow patterns which in many cases are available only through detailed experiments. Certainly, CFD simulations do not replace experiences or current knowledge of some special physical configurations. In particular, this tool allows users to analyze complex behaviors in an approximate way and based on engineering judgment could be accepted as physically accepted.

A CANDU-6 reactor, which description is being performed in the followings sections, has the so called calandria tank that contained the neutron moderator fluid. The moderator fluid circulation, mass flow and temperature distribution play a key role during reactor operation. The use of CDF code may become an interesting tool for analyzing the fluid moderator distributions during reactor normal operation, and obtaining the reactivity distribution in different calandria regions

There are few references in the literature that deal with the moderator mass flow rate and temperature distribution within the calandria tank. For instance, in [1], it is stated the importance in knowing the flow patterns of the moderator fluid in the calandria tank at different operating conditions because it might be useful for predicting thermal stratification and neutron feedback oscillations.



In [2] it is described an experiment called “SPEL” which is a simile calandria tank, but it was not exactly scaled compared to a real one. In spite of this, however, it had very similar geometrical and heated characteristics allowing the experiment a adequate representation of the calandria for capturing the expected physical phenomena. By changing mass flow rate and power in the SPEL experiment, it could be easily recognized three flow patterns: a) dominated by momentum, b) a mixed flow pattern and, c) dominated by buoyant force. In abovementioned work they also simulated the calandria tank with real dimensions using Fluent® and imposed specific boundary condition such as power and inlet fluid velocity. The author could reproduce by simulations the three flow patterns obtained in the experiments.

On the other hand, in [3], the temperature distribution was determined by a simulation using 14.0 CFX, and the results were validated against an experiment of a scale calandria tank. In this experience an asymmetrical distribution was obtained and showed that the flow pattern was governed by moment and buoyant forces.

Moreover in [4] a transient analysis was made by performing simulations with FLUENT-12. Similar results were obtained by [2]. He concluded that the temperature distribution is three-dimensional and not symmetrical. He also mentioned the existence of the three flow patterns.

In the work of by [5] where the flow pattern was analyzed by means of the code CUPID, with the porous medium approximation, and imposing a radial power distribution. In [6], they analyzed the subcooled boiling nearby the calandria tubes by CFD simulations and determined that the subcooled boiling is flow pattern dependent. These particular phenomena may become relevant in the reactor operation by the control in the neutron flux shape distributions and to avoid possible pressure tube boiling.

Most of the works abovementioned are relatively recent because of the computing power needed for getting adequate results due to the complex geometry.

In this paper the temperature distributions and the flow pattern, in the calandria tank of a CANDU reactor, are obtained by simulations in CFX academic version 15.0 [7] and the reactivity distribution by analytical calculus.

The geometry was built using the blueprints of the calandria tank and its components. The meshing was limited by the computational resources in the stage of the simulation in CFX, so only one mesh was used in this analysis. The results will be validated by the conceptual comparison with the literature flow pattern. And the temperature distribution will be used to calculate the reactivity distribution by means of an analytical relation.

### Description CANDU reactor

The CANDU6 reactor is a reactor that uses natural uranium as fuel and heavy water as coolant and moderator. These reactors generate approximately a thermal power of about 2000 MWth [8]. The CANDU-6 core has a horizontal array of 380 coolant channels, denominated pressure tubes, of approximately 6 m in length. The nuclear fuel is located inside the pressure tubes arranged in twelve rod-bundle of 37 fuel rods elements. Each fuel rod consisted of a zircaloy tube that contained the uranium pellets.

The CANDU reactor has two cooling circuits arranged in “Figure 8” at 100 bar, approximately. The cooling fluid enters to the reactor at 260 °C and goes out at about 310 °C, in saturated conditions with a small amount of flow quality. Each of these circuits is composed of two main pumps, two steam generators and four headers: two inlet and outlet headers [9]. A view of the mentioned components is presented in **Error! Reference source not found.**

The moderator system is a closed heavy water circuit at low pressure, ranged from 7 bar (pump sump pipe) to 2 bar and low temperature, compared to the primary system, ranging from 44 °C to 71 °C [9]. The moderator fluid thermalizes fast neutrons coming from nuclear fission through collisions with atoms of the moderator fluid contained inside the calandria tank.

Within the calandria tank (Figure 1) are, in addition, the reactor internals, such as vertical guides tubes for 4 absorber rods, 21 adjuster cobalt rods, 28 safety rods for the shut-off system, 6 liquid vessel that control reactivity, 6 horizontal nozzles where it is injected a neutron poison belonging to second shutdown system guides tubes for reactor instrumentation (26 vertical and horizontal 5 arrayed) as it is mentioned in [9]. All these tubes have an extremely complex geometrical configuration for the calandria tank.

The moderator receives approximately 103 MW of heating power. About 100 MW are due to neutron thermalization and the remaining amount due heat transfer coming from the coolant. Both, moderator and



coolant fluids are separated by the pressure and calandria tubes, and in between the two concentric tubes there is CO<sub>2</sub> which acts as a thermal insulator.

The moderator circuit consists of two centrifugal pumps; each one was designed for a nominal rate of 100%. During normal operation, the heavy water is taken from the bottom of the calandria tank, goes through heat exchangers to extract heat delivered to moderator and the cooled water returned to the calandria. The water is injected by 8 nozzles located upon the calandria tanks lateral sides (at the level of the median plane). More geometrical detailed on calandria internals will be give later on this work.

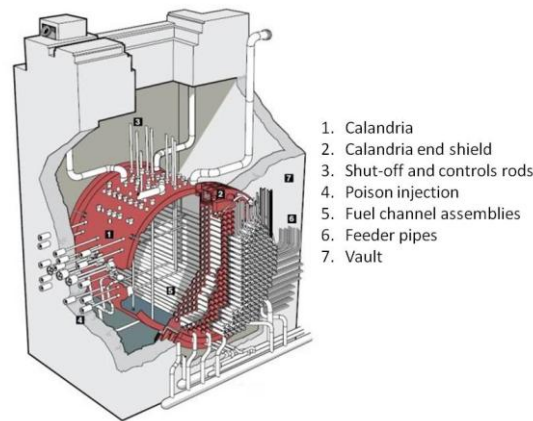


Figure 1: View of the calandria tank and its internal components (image taken from <https://canteach.candu.org/Page>).

### Flow regime map in calandria tank

When the moderator fluid enters the calandria tank; depending on the flow velocity, power and other geometric characteristics; various single phase flow regimes may come up. Typically, a characterization of flow regime map is done based on geometric and physical variables is relevant of the problem.

In [7], by the experimental results, three types of flow patterns were identified: a) dominated by momentum, b) mixed c) dominated by buoyant force. These patterns are shown in **Error! Reference source not found..**

The first pattern, **Error! Reference source not found..a**, is characterized by the formation of a stagnation point due to the interaction of the two jets entering on both sides of the calandria tank. As could be checked this stagnation point is located 90° respect to the horizontal axis at higher position of the tank. In the **Error! Reference source not found..b** is shown the mixed flow pattern that holds due to a competition between the inertial and buoyant forces, and considering that a heat source may cause an asymmetrical flow. In this regime, the stagnation point is located over one side of the calandria tank, typically at the angular position to the outlet nozzle. Finally, the last pattern dominated by buoyant forces shows that cold water (at slow velocity) may go to downwards whereas the hot water in center of the calandria goes towards the top the tank.

It is accepted in the literature that the flow regimes could be classified into three types according to the dimensionless Archimedes number (Ar). This dimensionless number reads

$$Ar = \frac{g\beta QD}{c_p \rho AV^3} \quad (1)$$



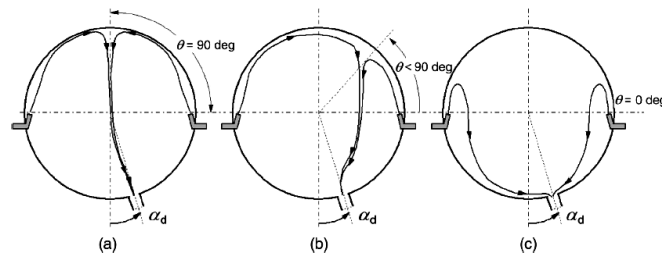


Figure 1: Flow patterns identified in the moderator tank; a) Flow dominated by momentum, b) Mixed flow, c) Flow dominated by Buoyant forces [1].

where  $\rho$  is the fluid density,  $V$  is the inlet velocity,  $g$  gravity acceleration,  $\beta$  is the thermal expansion coefficient,  $Q$  is the total power,  $D$  the diameter of the calandria,  $c_p$  is the specific heat, and  $A$  is the inlet area of the fluid to the calandria. Table 1 shows the pattern flow as function and its corresponding  $Ar$  number range.

**Table 1:** Archimedes number ranges for the three flow regimes within the calandria tank.

Regime	Archimedes number
Flow dominated by momentum	$Ar < 0.07$
A mixed flow	$0.07 < Ar < 0.45$
Flow dominated by buoyant force	$Ar > 0.45$

### CFD model of the calandria tank

In this section it is presented a brief description of the geometrical CFD model. The later includes the calandria tank and some of the internal (guide tubes, pressure tubes, among other) considering the blueprints of the Embalse Nuclear Plant used for construction, according to the designer [9]. A simplified two dimensional sketch is shown in Figure 2.

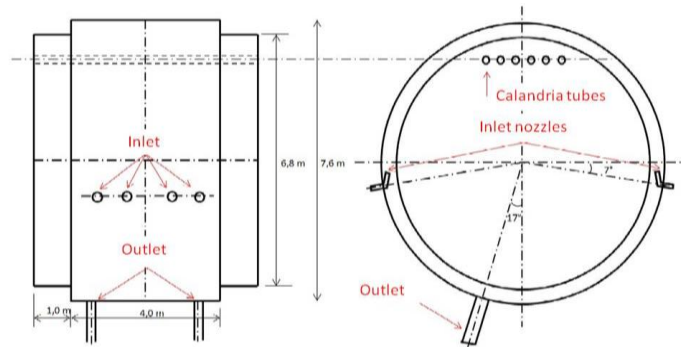
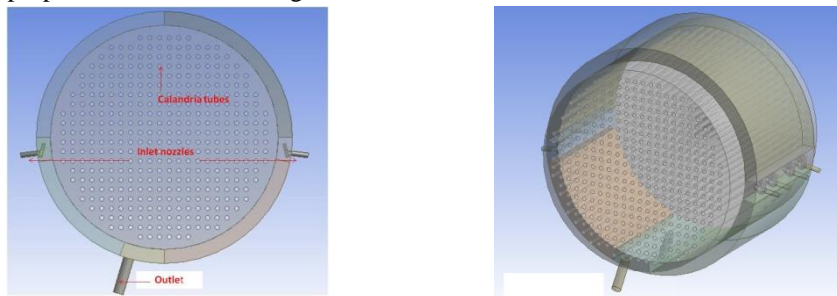


Figure 2: Lateral and frontal view of the calandria tank

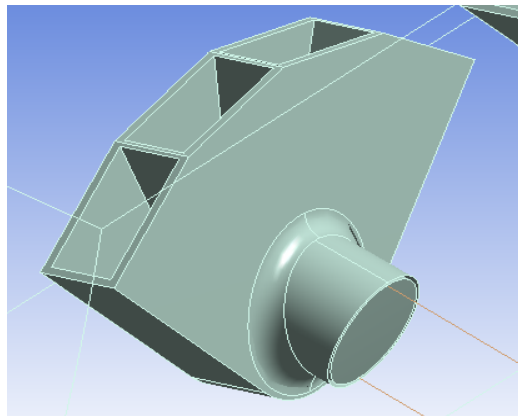
Inside the calandria tank, vertically distributed there are several pipes that each one contains the control mechanisms, controls rods and so forth. These guide tubes has a pipe diameters ranging from 6 to 16 mm. Between these vertical pipes and the horizontal calandria tubes come up very small free spaces with a few millimeters length. Therefore, for generating an acceptable mesh taken into consideration those narrows gap, a huge number of additional elements are needed imposing a large computation time. Based on references cited here and the purposes of this work, the guide tubes for control rods or instrumentation were not considered.



*Figure 3: Front view of the calandria tank.*

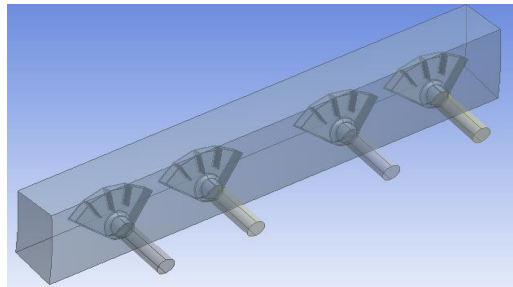
Figure 3 represents the frontal view of the model. The moderator come into the calandria tube through the inlet nozzle upon the lateral sides of the tank. Each injection pipe has a 90 degree elbow close to the tank as it is show in the left side of Figure 3. This elbow was not modeled and straight pipe was considered, instead. The inlet and outlet pipes were partially represented in order to achieve a velocity field of turbulent flow fully develops.

The calandria tank has 8 inlet nozzles; half on them on one lateral side and the other half nozzles in the opposite lateral side. Each axial axis of each inlet nozzle has an angle of  $14^\circ$  with respect to vertical axis, and the pipe diameters is about 15 cm. Also, each outlet nozzles (located at the bottom) has a  $7^\circ$  angle with respects to the horizontal axis and shifted  $17^\circ$  respect to the vertical axis. The geometry of the nozzles is based on blueprints and it is shown in Figure 4. As can be seen, the fluid enters to the moderator by tank by four equal rectangular sections.



*Figure 4: Screenshot of one of the modeled nozzle*

On Figure 5, it is sketch the right side nozzles array arrangement. As could be seen, the nozzles are arranged in pairs and not equally separated.



*Figure 5: Generally view of the inlet nozzles to calandria*

Meshing of this complex geometric was quite difficult trying to achieve our goal of getting the flow pattern and running in single laptop. For accomplishing this, the domain was divided into different parts for meshing with diverse methods in each one. By doing this, it was also possible to structured fine meshes.

The mesh was done using WORKBENCH of ANSYS 15.0. The domain was divided in: “upper”, “left”, “right” “Inner”, “inlet” and “outlet”. In each region was defined an “extruded” meshing type that was “inflated” from the calandria tubes. The parts “right header” and “left header” were defined a method of automatic meshing, allowing a tetrahedral unstructured mesh elements in those regions.

In the Figure 6 it could be seen the mesh in the calandria tank and some calandria tubes close to the wall. For the calandria tubes, we selected hexahedral elements with a size ranged between 19 mm and 25 mm; hexahedral elements of 65 mm for the reflector in the calandria tank were defined. In addition, in **Error! Reference source not found.** it is sketched the inlet nozzle meshes. For these regions it was defined tetrahedral, unstructured and 20 mm sized elements.





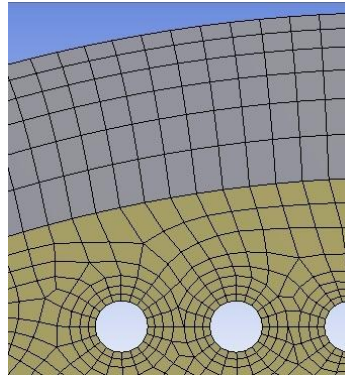


Figure 6: Vertical cut of the calandria tank mesh. It could be visualized calandria tubes and tank meshes. Finally, we generated one mesh of 6.5 million of hexahedral and tetrahedral elements approximately.

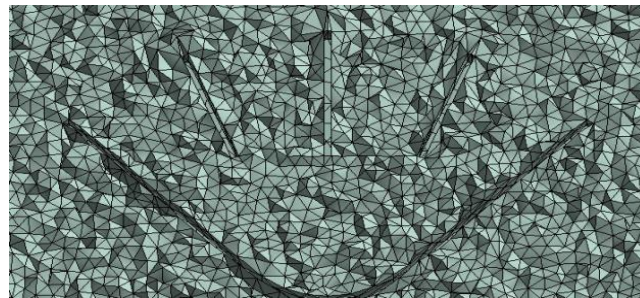


Figure 7: Side view of the network in the calandria tank

### Boundary conditions and simulation processing

The problem consists of a three dimensional, no isothermal, single phase flow. The turbulence model selected was the k- $\epsilon$  with a scalable function. For the sake of brevity, the conservative and constitutive equations included in the software solver are not being repeated here. The boundary conditions for the selected three cases are detailed in Table 2:

**Table 2:** Boundary conditions for the simulated cases.

Boundary condition	CASE 1	CASE 2	CASE 3
Inlet mass flow rate (kg/s)	2060	1030	515
Inlet temperature (K)	317.15	317.15	317.15
Outlet pressure (bar)	2	2	2
Heat delivered by conduction (MW)	3	3	3
Heat delivered by moderation (MW)	100	100	100
No. Archimedes	0.02	0.12	0.96
Expected flow pattern	Flow dominated by momentum	Mixed flow	Flow dominated by buoyant force

The cases are identical except for the inlet mass flow rate. The first case could be operative understand as the startup of the backup pump. The second case is the normal operating conditions of a CANDU6 whitest the case 3 is a hypothetical flow reduction up to the 50 % of the nominal for determining whether the model can adequately capture the particular phenomenology previously seen in other CFD models, software and experiments.

The heavy water properties were added as expressions in function of the temperature taken from the IAPWS.

### Boundary conditions and simulation processing



The moderator fluid receives power from different sources. One of them is by heat conduction through calandria tubes to moderator and the other by neutron thermalization. These sources generate a power distribution with a radial and axial specific shapes as it is illustrated in **Error! Reference source not found.** . All walls with the exception of the calandria tubes were considered smooth and adiabatic, and hence, no heat is lost to environments or other components.

The radial power profile, denoted by  $g(r, \theta)$ , was taken from the [5], whereas the axial power distribution, denoted by  $f(z)$ , corresponds to an average power profiles of 380 fuel channels at normal full power operation considering nominal position of reactivity controls mechanism. The normalized power profiles must fulfill that:

$$\frac{P_T}{V} \iiint k f(z) g(r, \theta) r dr d\theta dz = 1 \quad (2)$$

where  $P_T$  total power,  $V$  the calandria tank volume of 223 m<sup>3</sup>, and the  $k$  the normalization value.



Figure 8: Normalized radial (left) and axial (right) power profile

The implemented boundary condition in ANSYS was the total power density per unit of volume denoted as  $q'''$ , accordingly to **Error! Reference source not found.**. This quantify was obtained as the ratio of the 100 MW of delivered power by moderation and fluid volume in calandria ( $V_{cal}$ ) multiplied by the radial distribution  $g(x, y)$  and axial  $f_z(z)$  as in Figure 8.

$$q''' = \frac{100 \text{ MW}}{V_{cal}} g(x, y) f_z(z) k \cong 448 \frac{kW}{m^3} g(x, y) f_z(z) k \quad (3)$$

Besides, the power delivered by conduction wall, the heat flux distribution ( $q''$ ) was determined by (4) as the ratio of 3 MW by the total lateral area of the 380 calandria tubes ( $A_{Tub}$ ) multiplied by the radial  $g(x, y)$  and the axial distribution  $f_z(z)$ .

$$q'' = \frac{3 \text{ MW}}{A_{Tub}} g(x, y) f_z(z) k \cong 4050 \frac{W}{m^2} g(x, y) f_z(z) k \quad (4)$$

In Figure 9 it is shown the calandria tank domain with the power sources already implemented. The scale shows the heat flow in each calandria tube.

For getting a steady state simulations it was defined the outlet fluid temperature as convergence parameter, verifying that the target parameter met the expected values. It was defined a time step of 10 seconds and convergence was reached with errors about of  $1 \times 10^{-3}$  for the residuals.

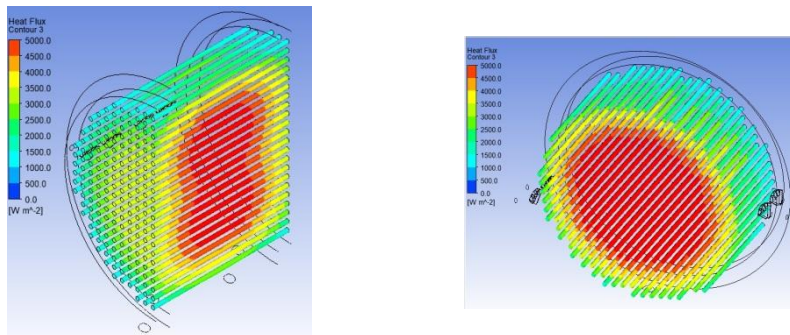


Figure 9: Three dimensional power distribution



## Results

Three simulations were done. The streamlines were constructed using velocity field ranging from 0 m/s up to 2 m/s. In Figure 10 is shown the three flow pattern obtained changing the inlet velocity. These results are similar to those presented in the work of [1] and the expected flow regimes shown in [7].

The left figure shows the flow pattern is dominated by momentum where two inlet jets after leaving the nozzles go up towards the top of the tank and, then after interacting they fall down symmetrically (case 1). When setting case 2, a mixed flow pattern was obtained as shown in the middle of the Figure 10. It could be easily check the two inlet jets go up to the top of the tank and then interact they fall down asymmetrically. When considering Case 3, the flow pattern obtained was dominated by the buoyant force; the two jets have a low velocity and after coming into the tank and go towards the bottom of the tank rather symmetrically allowing the hottest fluid in the middle going up. This shows that the developed model adequately represents experimental results in the literature and by other researchers.

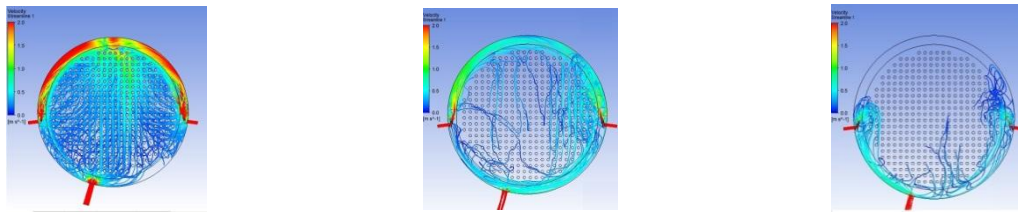


Figure 10: From left to right, streamlines for Case 1, 2 and 3

In the Figure 11, Figure 12 and Figure 13, the temperature distribution on plane x-y, at  $z = 0$ ,  $z = 2.5$  m and  $-2.5$  m, respectively, for the three cases simulated are shown. For case 1 in  $z = 0$ , the two jets interact among them producing a low temperature region on upper part of the calandria; but this effect disappear for plans  $z = 2.5$  m and  $-2.5$  m and a more symmetric distribution could be found and almost no stratification is devised. It could be seen that regions close to both inlet nozzles do not have equal temperature and asymmetry appears in case 2.

In one of the nozzle a homogeneous fluid temperature is seen whilst, in the other region, a fluid thermal stratification show up. Form safety point of view this is not a relevant issue but, form operational point of view, this difference may have impact on nuclear fuel burnup and reactivity feedback as well as on liquid control level for controlling neutronic flux profiles.

Furthermore, it is interesting to check that for the z-planes studied the fluid temperature distribution for case 3 is highly stratified. The highest temperature is on the upper part of the calandria in opposition of the case 1.

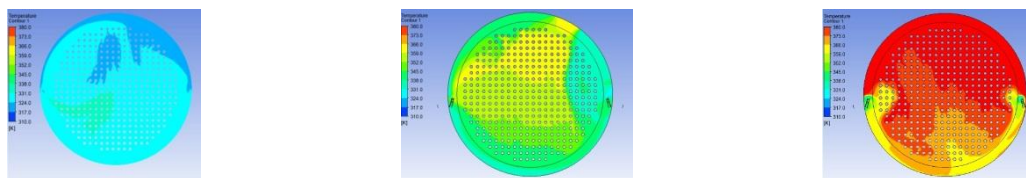


Figure 11: Temperature distribution from left to right for Cases 1, 2 and 3 at  $z = 0$ .

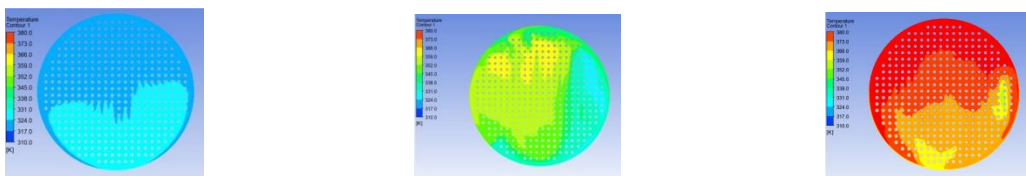


Figure 12: Temperature distribution from left to right for Cases 1, 2 and 3 at  $z = 2.5$  m.





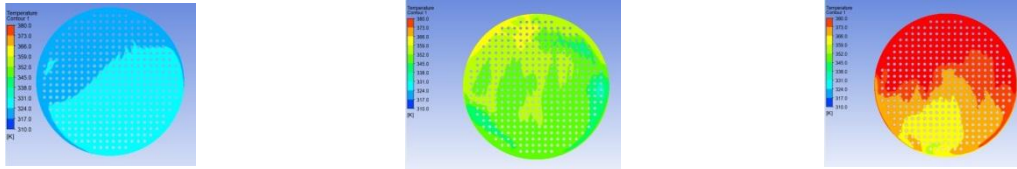


Figure 13: Temperature distribution from left to right for Cases 1, 2 and 3 at  $z = -2.5$  m.

A comparison, of the fluid temperature obtained by means of this model for Case 2 and those published by [7], is shown in the Figure 14. The temperature profile is quite similar in trend and shape. The maximum and minimum values are also equals, indicating the consistent results regarding power distribution.

The most remarkable discrepancies are the increase and decrease slopes. Near to the extremes the temperature is lower than the literature results, and opposite to this in the middle the temperature is higher than the literature results. The differences could be caused by the radial distribution considered.

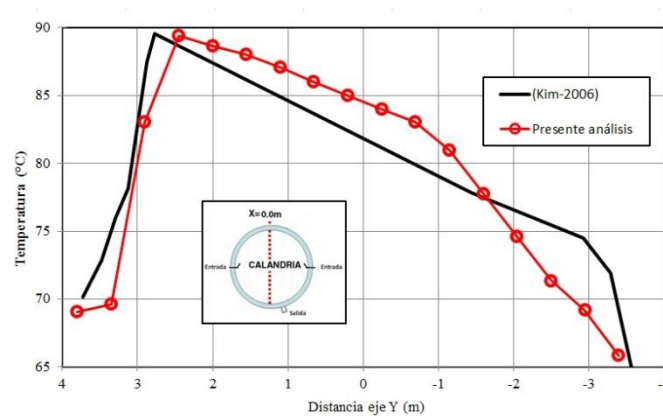


Figure 14: Temperature profil,  $x = 0$  and  $z = 0$  for Case 2 mixed flow

On the other hand, other possible cause of the discrepancy is on the little differences in the scale experiment simulated by [3]. It is important to mention that the Figure 14 is comparing results between one model with real dimensions of the calandria tank with other model scaled from a calandria tank.

In conclusion although there are differences in the temperature across the vertical center line of the calandria tank, the results maintain the tendency of the literature results.

### Reactivity feedback

It was mentioned that the neutron reactivity feedback could be affected, for instance, by the moderator temperature distribution, among other changes in coolant and moderator densities, or fuel temperature, or control rods positions, etc.

We will focus on moderator temperatures changes, therefore the reactivity feedback could be calculated by:

$$\rho_{mod\_T} = \frac{\delta \rho_T}{\delta T} (T_{mod} - T_0) = \sum_{i=1}^{N_{cell}} \frac{\frac{\delta \rho_T}{\delta T} (T_{mod_i} - T_0) V_i}{V_{cal}} \quad (5)$$

Where  $\rho_{mod\_T}$  is the total moderator reactivity,  $\frac{\delta \rho_T}{\delta T}$  is the moderator reactivity feedback (estimated in a PHWR of CANDU type 0.082547 pcm/K),  $T_0$  is a reference temperature (71 °C),  $T_{mod_i}$  is the moderator temperature in cell  $i$ ,  $V_i$  and  $V_{cal}$  are the cell and total calandria volumes, respectively. In order to calculate the reactivity due to moderator temperature some approximations should be done due to the huge numbers of cells and volumes within the model. The simplification is as follow: the calandria was divided in 4 vertical planes (named A, B, C, D), 3 radial rings and 8 azimuthal sectors (@t 45°) as it shown in Figure 15, numbered from 1 to 24.



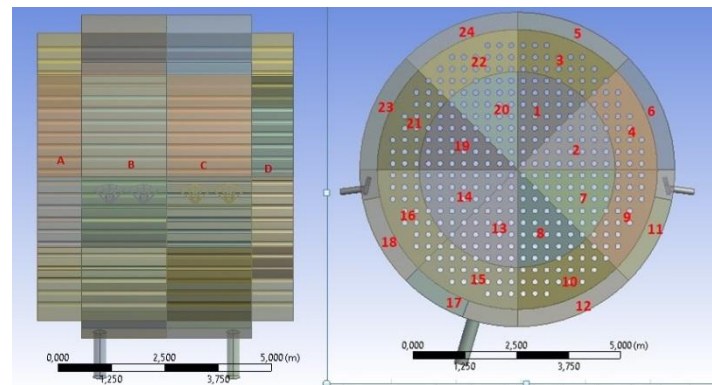


Figure 15: Mapping used for calculating reactivity feedback

Using the above mapping, we determined an average temperature within each sector. These mean temperatures were obtained by taking the average along a line close to the geometrical median of each sector. The volume of each sector was also calculated aided by a CAD software due to the complex geometry. Then the total reactivity feedback due to moderator temperature was estimated using (5); results for the three mass flow rates studied are shown in Table 3.

**Table 3:** Reactivity feedback due moderator temperature for three inlet mass flow rates

Case	Moderator mass flow rate (kg/s)	$\rho_{mod,T}$ (mk/K)	$\rho_{mod,T}$ (\$/K)
1	2060	-158	-0.3
2	1030 (normal operation)	40	0.075
3	515	229	0.43

It could be realized that despite of asymmetric flow and the temperature distribution shown in Figure 11, Figure 12 and Figure 13, the total reactivity feedback is just 40 pcm or 0.075 \$ for the normal operation. These values are relative small when comparing with the reactivity insertion due to, for instances, the liquid control zone. It has to be remarked that case 1 and 2 are not symmetric as it would be expected.

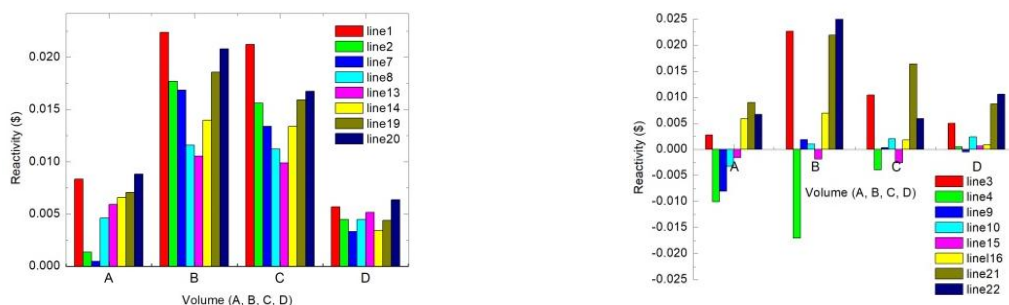


Figure 16: Reactivity feedback by the different azimuthal and radial rings (central ring at the left, and second ring in the right).

Figure 16 shows reactivity feedback by each line and rings for during normal operation (using mass flow rate of the case 2). On the left it could be easily seen that reactivity for the four vertical planes and the first radial ring, is for all lines positive and the contribution by each volume (A, B, C, D) rather symmetric.

However, in the second ring, the reactivity contribution is positive and negative depending on the azimuthal sector. For those cells closed to the inlet nozzles the feedback is negative and it changes sign when fluid starts to heat up.

## Conclusions

In this paper the temperature distributions and the flow pattern in a calandria tank in a CANDU6 were determined by steady state simulations with ANSYS CFX 15.0, academic version. The model considered all calandria tubes, flow nozzles and outlet pipes, but do not take into account other internals components. The total



power used was 103 MW following an axial and radial distributions as expected (obtained) in this reactor.

Three possible inlet mass flow rate boundary were simulated cases and we obtained similar flow pattern consistent with other models made in CFD and experimental results. It was verified that with the lowest mass flow rate and high fluid thermal stratification came up. This is because the flow pattern is dominated by buoyant forces instead of momentum. When mass flow rate increases, close to normal operation, a mixed flow pattern appear. Despite the fluid temperature stratification is lower the other case, an asymmetric fluid distribution show up.

From temperature distributions, we obtained the moderator fluid temperature reactivity feedback performing some approximation in the calandria and taking mean values. In spite of the temperature asymmetries, the reactivity feedback is a positive (40 pcm), but small quantity compared with other mechanism for controlling core reactivity.

### Acknowledges

The authors would like to acknowledge the discussion and contribution to this work by Eng. Juan Carlos Ferreri, president of the National Academy of Sciences of Buenos Aires.

### References

- [1]. Zadeh, F., Etienne, S., & Teyssedou, A. (2017). Computational Fluid Dynamic Simulation of the Flow Moderator IN CANDU-6 Nuclear Reactors. *Journal of Nuclear Engineering and Radiation Science*, 3(10115) 1 - 13.
- [2]. Sarchami, N. Ashgriz, & M. Kwee. (2012). Three Dimensional Numerical Simulation of a Full Scale CANDU Reactor Moderator to Study Temperature Fluctuations. *International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering*, 6(3): 221-227.
- [3]. Kim, H., & Chang, S. (2015). Computational Fluid Dynamics Analysis of the Canadian Deuterium Uranium moderator at the stern test Laboratories INC. *Nuclear Engineering Technology*, 47(15): 284-292.
- [4]. ANSYS. (2013). Help System - Guide CFX, ANSYS® Academic Research. Release 15.0.
- [5]. Jeong, J., Choi, S., & Yoon, H. (2015) Thermal hydraulic analysis of the CANDU moderator tank cupid using the code. *Progress in Nuclear Energy*, 85(15): 541-547.
- [6]. Yoon, C., Rhee, W., & Min, B. (2004). 3-D CFD Analysis of the CANDU-6 Moderator Circulation Under Normal Operating Conditions. *Journal of the Korean Nuclear Society*, 36(6): 559-570.
- [7]. Kim, M. (2006). Analysis on fluid flow and heat transfer inside Calandria vessel of CANDU-6 using CFD. *Nuclear Engineering and Design*, 236(06): 1155-1164.
- [8]. Lazarte, A., Corso, S., & Schiliuk, N. (2016). Model with central nuclear Embalse RELAP5: broken steam line inside the containment. XLIII meeting of Argentina Association of Nuclear Technology. Buenos Aires, November 21-25.
- [9]. AECL. (1978). Design Manual Assembly Calandria. DM-18-31200-01-01.

