中國機械工程學刊第四十卷第三期第 299~306 頁(民國一百零八年)
Journal of the Chinese Society of Mechanical Engineers, Vol.40, No.3, pp 299~306 (2019)

Heat Transfer Enhancement in a Calandria Based Reactor With a New Inlet Design - A CFD Analysis

V. V. Joshi* and Suneel Patil**

Keywords： Calandria, Nuclear reactor efficiency, Heat exchangers.

ABSTRACT

Controlled cooling of the fuel channels in a calandria based reactor is important to avoid any hazardous aftermath. On the other hand, efficient cooling of the fuel channels can also ensure better utilization of the fuel in electricity generation. In this regard, simple modifications which can cause significant increase in the efficiency are appreciable; in particular, in the case of existing nuclear reactors. In this paper, we have discussed (a) the thermal and flow characteristics of the heavy water inside the traditional calandria based reactor and, (b) the effect of the introduction of a splitter near the heavy water inlet on the cooling efficiency. It is observed that the mass flow rate and inlet orientation have a significant effect on the convective heat transfer from the fuel channels to the heavy water. The underutilization of the heavy water in cooling the fuel channels in the traditional calandria design was speculated from the CFD analysis. The effect of introduction of a simple splitter (aerofoil shaped body) near the inlet was hence studied by carrying out CFD simulations. The latter seems to cause considerable decrease in the temperatures inside the calandria. The comparative analysis has shown that the splitting of the flow near the inlet can keep the calandria operating temperature under the safe limits for the same mass flow rate but for higher thermal power output.

INTRODUCTION

The nuclear reactor primarily consists of Calandria, Steam Generator, Emergency Cooling System, and, primary and secondary moderator (heavy water) circulation system. Calandria is a high pressure reactor vessel made of stainless steel. Inside the calandria, hundreds of horizontal fuel channels are arranged in a hexagonal pattern. The fuel channels contain the fuel (uranium) in the form of fuel bundles. Due to fission reaction, a large amount of energy is released in the form of heat. Primary fluid carries part of this energy (heat) to the boiler where it converts ordinary water into steam whereas the secondary fluid which is circulating around the fuel channel is mainly acting as a moderator. The fuel channels must be always under the safe operating conditions and hence, heat loss by convection to the surrounding heavy water is important.

To understand the flow phenomena inside the calandria, both experimental and numerical studies are being carried out. The important and preliminary observations of the operation of CANada Deuterium Uranium (CANDU) reactors were reported in [1]. The authors conducted a series of experiments at Sheridan Park Engineering Laboratory. These experiments could provide the basic idea of thermal fluid flow characteristics inside the calandria. Similar experiments were reported in [2, 3] to identify the effect of inlet mass flow rate and internal heating on flow and temperature distribution inside the calandria. Experiments under isothermal conditions were reported in [4] to understand the flow characteristics. However, it is difficult to carry out the experimental studies on the full scale calandria model and hence, the numerical analysis can be economic and informative. In this paper, we discuss the details of the CFD analysis of the calandria.

Some of the past CFD studies were carried out using a simplified geometry model and by approximating porous media for the matrix of the calandria tubes inside the calandria [5, 6 and 7]. In [6] the authors have mentioned that the flow inside the calandria can be either momentum dominated flow, or buoyancy dominated flow, or mixed type of flow. The flow pattern depends on the heat load and inlet mass flow rate. A series of numerical simulations for the analysis of CANDU reactors were carried out using Canadian

Paper Received September, 2016. Revised May, 2017, Accepted May, 2017, Author for Correspondence: V. V. Joshi

* *Associate Professor, School of Mechanical Engineering, Vellore Institute of Technology, Vellore, Tamil Nadu, 632014, India.*

** *Modeling Engineer, Lam Research India Private Limited, Bangalore, Karnataka, 560071, India*

Algorithm for Thermo-hydraulics Network Analysis (CATHENA) [6]. This algorithm is capable to simulating transient two phase flow with accounting solid conduction. The numerical study was three dimensional and was to understand moderator circulation for CANDU-6 reactor accounting radial and axial power distribution. The prediction of maximum temperature of the moderator for a given moderator flow rate was possible. The detailed thermal and fluid flow characteristics were reported in [8] based on the CFD analysis of the calandria. The information can be used effectively during the initial stages of design. The capacity of a moderator to remove heat from the calandria is depends on its geometric parameters, two different inlet and outlet locations have been analyzed by Arsene et al [9] for CANDU reactor and shown that there is an increased heat removal rate for modified configuration. However, not much work has been done in this direction. Hence, the effect of calandria inlet and outlet orientations, moderator inflow rate, and flow distribution on the cooling of fuel channels need to be studied. The analysis can help us make the reactors more safe and efficient.

In the present study, the numerical experiments were planned to understand (a) the effect of inlet orientation (θ) and moderator flow rate on the fuel channel temperature, and numerical simulations for the cases of 1200MW and 1800MW thermal loads have been carried out. And, (b) the effect of splitting of the flow near the inlets of the calandria for the two different thermal load cases. Rationale behind these numerical simulation was to make a remark on the cooling enhancement due to the splitting of the flow near the inlets. The present work is a continuation of previous study on the effect of moderator inflow angle on the performance of calandria [10]. The study has suggested the limits for inlet angle for a given inlet flow rate. From our previous study it was found that the inlet angle between 30°–60° (with respect to the horizontal) gives optimum heat removal for all considered heat loads.

DETAILS OF THE CFD MODEL

The schematic view of the considered calandria model is as shown in figure 1. The figure also shows the placing of a splitter plate near the inlet. Due to the splitter plate, the total moderator mass flux at the inlet ($\dot{m}$) splits into two parts, i.e., the mass flow from below the splitter ($\dot{m}_L$) and from above the splitter ($\dot{m}_U$) as shown in figure 1. There are total 553 fuel channels (diameter = 0.132 m), equally spaced (square pitch = 0.288 m) and are always surrounded by the moderator. The calandria is 8m in diameter and 8m in length. Inlets are located on the two sides of the calandria symmetrically opposite to each other and outlet is located at the bottom. Splitter plate has a NACA0012 airfoil shape to reduce any pressure loss and is placed near the center of the inlet opening. Its distance from the calandria center is 4m (refer to figure 1). The CFD simulations have been carried out for various splitter plate orientations (β) which are given in the table 2.

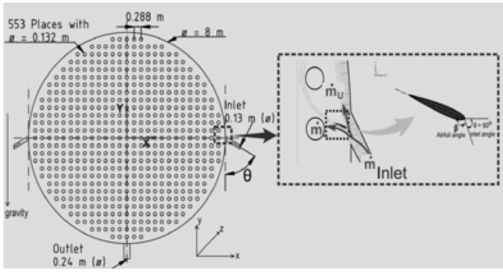


Fig. 1. Detailed view of calandria geometry considered for present study and a side inset indicates the placing of splitter with respect to the inlet angle (θ).

The simulations have been carried out for steady state, incompressible conditions considering both convection and radiation inside the calandria. Reynolds-averaged Navier–Stokes equations (RANS) are solved for the water domain (refer to equations from 1, 2, and 4).

$$\frac{\partial(\bar{u}_i)}{\partial x_i} = 0 \tag{1}$$

$$\bar{u}_j \frac{\partial(\bar{u}_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_j} - \frac{1}{\rho}\frac{\partial \overline{u_i' u_j'}}{\partial x_i} + f_i \tag{2}$$

Where ρ is the density of the fluid, u_i represents velocity vector, x_i represents the axis in Cartesian coordinate system, f_i is the body force term and is non-zero only in the gravity direction in the present case. The outlet was considered to be at atmospheric pressure. No-slip wall conditions were specified for all the solid walls. τ represents the shear stress due to the velocity gradients, and p is pressure.

Boussinesq model has been used to take buoyancy into consideration. The Boussinesq model is used when the density variations are driven by the temperature variation. The buoyancy source term is approximated as,

$$\rho - \rho_{ref} = -\rho_{ref}\,\beta_t\,(T - T_{ref}) \tag{3}$$

and,

$$\frac{\partial(e+p)\bar{u}_i}{\partial x_i} = \frac{1}{\rho}\frac{\partial(\bar{\tau}_{ij}\bar{u}_j)}{\partial x_i} + f_i\bar{u}_i + \frac{1}{\rho}\frac{\partial(\dot{q}_i)}{\partial x_i} \quad (4)$$

A standard k-ε turbulence model (equations 5-10) has been used as it has been validated for the past few calandria case studies [8, 11, and 12]. The convergence criteria was set as 10^{-4}.

Table 1. Details of applied boundary conditions.

Boundary type	Assigned condition	Value
Inlet	Mass flow	Varying 'Re'
Outlet	Static pressure	'0' atm (Relative pressure)
Fuel channels	No slip, Constant heat flux	654×10^3 and 981×10^3 [W/m^2]
Calandria wall	No slip, Adiabatic	q = 0

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\varepsilon - Y_M + S_k \quad (5)$$

Where, Y_m represents the contribution of fluctuating dilatation in compressible turbulence to the overall dissipation rate. This is zero in the present work as the fluid is water (incompressible).

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\left(P_k + C_{3\varepsilon}P_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (6)$$

Turbulent viscosity,

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (7)$$

Production of k

$$P_k = -\rho\overline{u_i u_j}\frac{\partial u_j}{\partial x_i} \quad (8)$$

Effect of buoyancy, P_b

$$P_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t}\frac{\partial T}{\partial x_i} \quad (9)$$

Model constants are,

$$C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad C_{3\varepsilon} = -0.33, \quad C_\mu = 0.09, \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \quad (10)$$

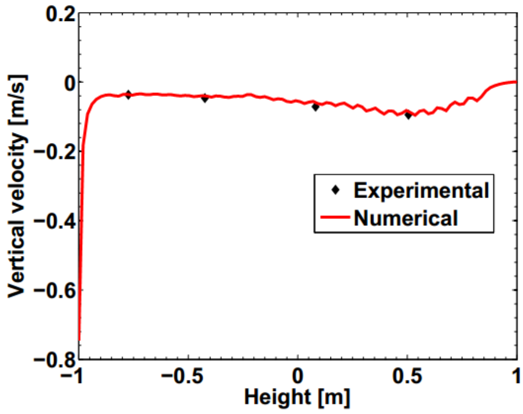
Heat flux boundary condition is specified for fuel channel surface, corresponding to 1200 MW and 1800 MW thermal power. The simulations were carried out for different mass flow rates, and, different inlet and splitter orientations. Inlet temperature is considered as 44.2 °C based on a previous case study [13]. The details of boundary conditions are tabulated in table 1.

In this study, body fitted unstructured grid is generated using ANSYS Meshing as the domain was complex to go for a structured grid. However, the validation and grid dependence studies for the unstructured grid has been reported in [10]. Higher grid density regions were created near the walls with the help of prism layers. The grid dependence studies were repeated for five different grid sizes (control volumes) (refer to table 3). The maximum fuel channel temperature and temperature at three selected sections (refer figure 3) were observed as criteria for grid independence. The variation in theses temperatures due to increase in the number of elements from 3.5 to 4.6 million (refer table 3) was about 1 °C for the applied heat flux being in terms of megawatts. Compromising with the time and memory space requirements for simulations, mesh size of 3.5 million (mesh 2) was considered (refer table 3) for further analysis. The solution process was an iterative one. Numerical simulations were performed with an Intel machine (core i7-3770, 3.40 GHz). For one complete calculation with 3.5 million mesh size required 7-8 h.

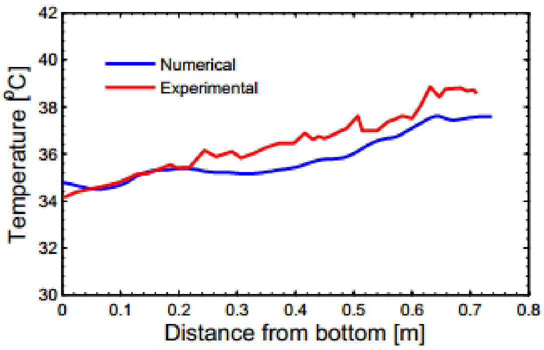
Because of the geometrical symmetry in the construction of calandria, only half portion was modeled. In the present work, phase change process has been neglected for simplicity. The heating of the fuel channels is assumed to be uniform throughout the considered fluid domain.

Validation

The numerical model was validated with the available isothermal and non-isothermal literature experimental observations. Velocity and temperature distribution along the centerline were compared. Figure 2a shows the comparison of vertical velocity variations along the vertical center line of the calandria between present simulated results and the isothermal experimental



(a)



(b)

Fig 2. Validation of numerical model with the literature experimental work: (a) Vertical velocity comparison along a vertical centre line [3], and (b) Temperature comparison along a vertical centre line [1].

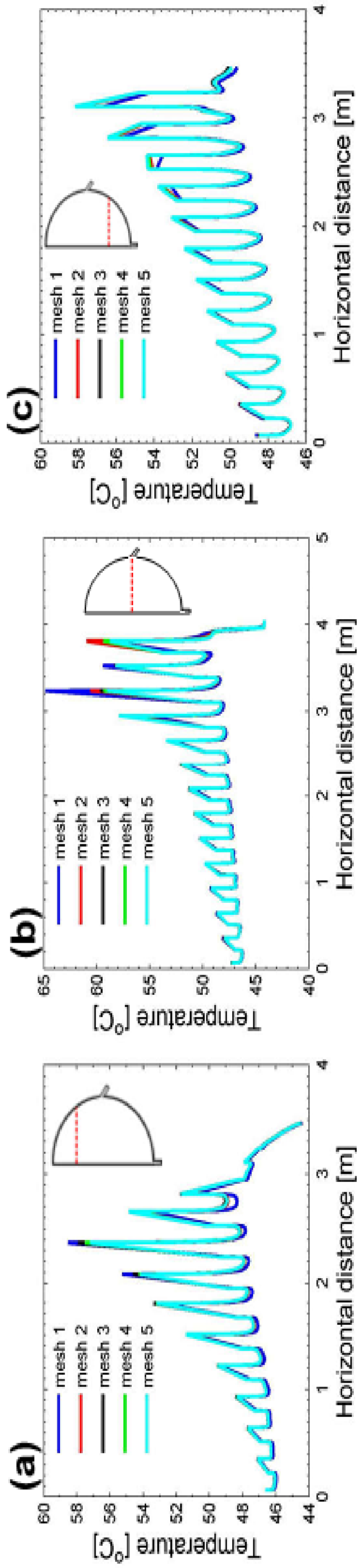


Fig. 3. Temperature at different regions for different mesh. a. Horizontal line 2 m above the calandria center, b. Horizontal line passing through the calandria center and c. Horizontal line 2 m below the calandria center.

Table 2. Inclination of splitter (β) with respect to inlet ($\theta = 60°$) for 1800 MW thermal power

Inlet angle (θ)	Mass flow rate ($\dot{m}$) [kg/s]	splitter angle (β)
60°	80	60
60°	80	80
60°	80	100
60°	80	120
60°	80	140
60°	80	160
60°	80	180

Table 3. Mesh parameters and maximum fuel channel temperature for grid sensitivity study

Case	Inlet angle (θ)	Inlet Re (10^6)	Total number of elements (million)	Maximum T_{CT} (°C)
Mesh 1	75	2.1	2.0	69.8
Mesh 2	75	2.1	3.5	70.4
Mesh 3	75	2.1	4.6	71.6
Mesh 4	75	2.1	5.8	71.1
Mesh 5	75	2.1	7.1	71.1

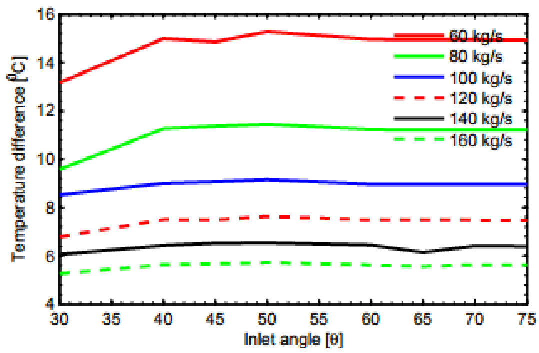
work [3] for the mass flow rate of 2.4 kg/s. The negative velocity values indicate that the fluid moving in a downward direction. For isothermal case, the present simulated values match well with the experimental work. The present numerical model was also validated with a non-isothermal experimental case study by koroyannakis et al [1], where the heat load was 100 kW and mass flow rate was 2.4 kg/s (refer to figure 2b).

RESULTS AND DISCUSSION

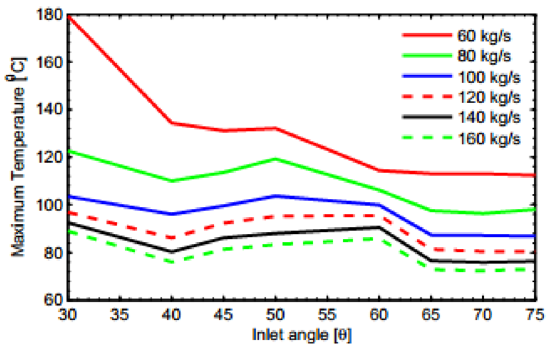
The analysis of the basic geometry (without splitter near the inlet) with the fuel channels subjected to a heat flux value of 654 KW/m^2 which corresponds to the thermal power output of 1200MW is discussed first. This analysis was carried out to arrive at the values of the mass flow rate and the inlet angle where the efficiency can be improved. Further analysis was carried out at these values and in the latter case, the heat flux at the fuel channel surfaces was increased to 1.5 times (981 KW/m^2) which corresponds to 1800MW). The simulations were carried out for different mass flow rates. This analysis essentially answers which mass flow rate for 1800MW case can cause similar temperature values as that of the 1200MW without splitter plate. Further, the introduction of splitter plate was analyzed to determine the possibility of lowering the requirement of higher mass flow rate for 1800MW to operate under the similar conditions as that of the 1200MW case.

Basic calandria geometry (without splitter near the inlet):

This analysis has been carried out for the case of 1200MW thermal power. Figure 4a shows the variation of ΔT (temperature difference between inlet and outlet) for different inlet orientations and different mass flow rates. It is observed that any increase in the inlet angle above 40° with respect to the horizontal (clockwise) has negligible effects on the resultant temperature difference. Apart from this, it is also observed that lower mass flow rate cases have higher temperature differences. The corresponding maximum temperatures of the fuel channel surfaces are as shown



(a)



(b)

Fig. 4. Temperature difference and maximum temperature variation for different inlet angles (θ), 1200MW thermal power: (a)Temperature difference (ΔT), and (b) Maximum temperature (T_{max}).

in the figure 4b. Some of the fuel channels either in the top region or below the inlet close to the calandria wall attained maximum temperature values greater than 100 °C for the lower mass flow rates (60kg/sec and 80kg/sec) and is not under the safe limits. But the lower mass flow rates ensure low pumping power requirements. Hence, there was an opportunity to optimize the operating temperature. In this regard, further analysis has been carried out for 80kg/sec mass flow rate with inlet angle 60°.

Case 2: Similar analysis for thermal power = 1800MW and comparison

As a first case study, a different set of simulations were carried out to see the extent to which the ΔT and maximum temperature values change if the thermal power would be increased from 1200MW to 1800MW. A comparison of the simulated results for 1200MW

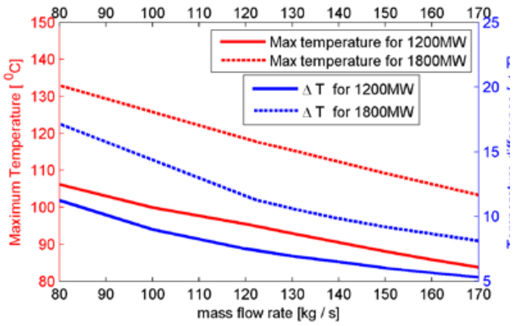


Fig. 5. Variation of maximum temperature and temperature difference between inlet and outlet for different mass flow rates.

and 1800MW is as shown in the figure 5. The maximum temperature values in the case of 1800MW thermal power were greater than 100 °C even when the mass flow rate is doubled. The streamlines and the local temperature variations along the streamlines for the two cases are as shown in the figure 6. It is interesting to note that, the incoming heavy water goes towards the top of the calandria along the calandria surface and then comes down due to gravity. This has a major drawback that the incoming cooler heavy water does not flow close to most of the fuel channels in the calandria. It was also observed that the volume of high temperature region of the fluid in the lower half of the calandria increased in size, due to the increase in the thermal power.

Introduction of splitter plate near the inlet

The idea of using the splitter plate was to split the mass flow to make the cooler incoming heavy water to flow close to as many fuel channels as possible. This can ensure enhancement in the heat transfer. For splitter angles β = 60° and β = 80°, the mass flux from the bottom of the splitter ($\dot{m}_L$) was not effective and the flow was along the calandria wall; whereas for higher β values (i.e. β > 80°), part of the incoming cooler heavy water ($\dot{m}_L$) is coming in contact with some of the fuel channels as shown in figure 7. For β > 100°, the mass flux from the upper and lower sides of the splitter plate ($\dot{m}_U$) join together and flow towards the upper part of the calandria and the region below the inlet close to the calandria wall is negligibly affected by the flow (refer figure 7). For all considered splitter angles the flow diverting towards the bottom part of the calandria can be seen only in splitter angles β = 60°, 80°, and 100°. Hence, for further analysis of volume

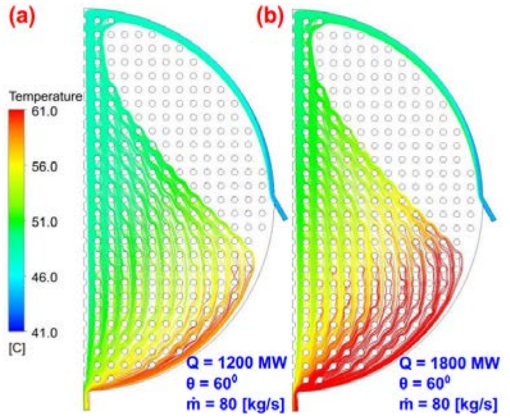


Fig. 6. Stream lines with temperature values (a) 1200MW, (b) 1800MW.

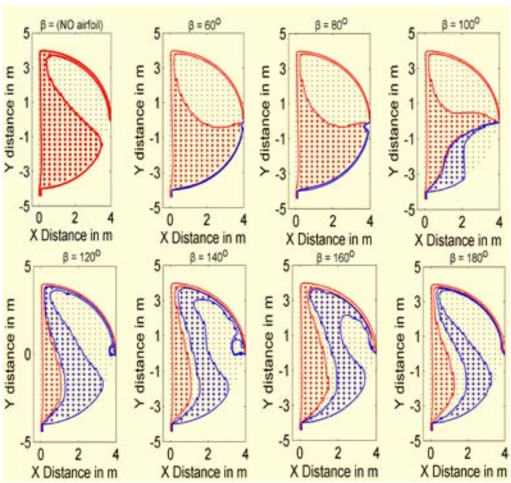
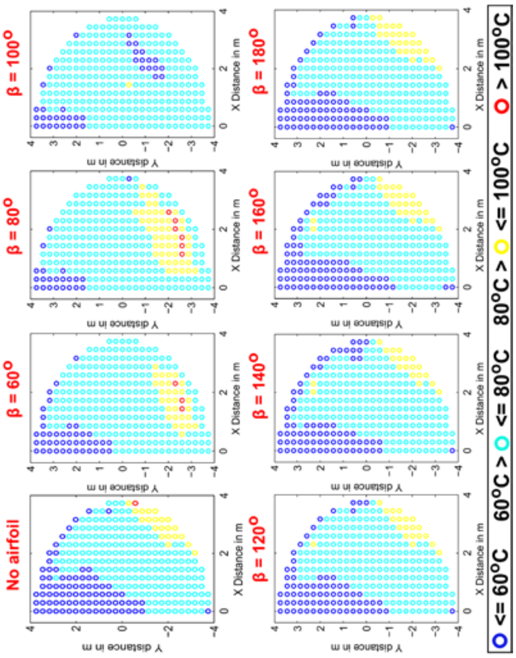


Fig. 7. Amount of volume covered by the stream line for different splitter angles (β) where, blue region indicates the volume covered by the mass flux coming from below the splitter ($\dot{m}_L$) and red region indicates the volume covered by the mass flux coming from above the splitter ($\dot{m}_U$) for 1800MW, θ=80°, ṁ=80kg/sec.

covered by the mass flux ($\dot{m}_L$) only these three cases have been considered.



(a) (b)

Fig. 8. Average fuel channel temperature for different splitter angles (β).

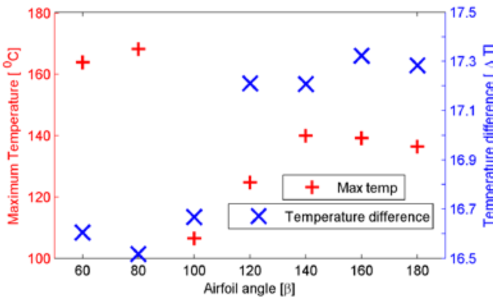


Fig. 9. Variation of maximum temperature and temperature difference with respect to splitter angle (β).

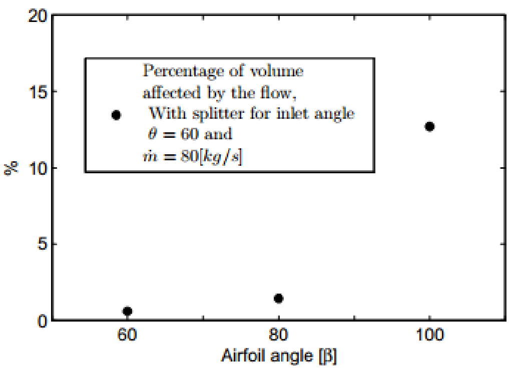


Fig. 10. Percentage of volume affected by the mass flux coming from below the splitter ($\dot{m}_L$).

The average temperature of the fuel channels is as shown in the figure 8. The fuel channels crossing the temperature of 100 °C are highlighted with red color and it can be seen that there is only one fuel channel falling between the temperature range of 80 °C to 100 °C and remaining are below 80 °C for splitter angle of β = 100°. For all the considered splitter orientations, the temperature of the fuel channels in the upper part of the calandria and close to the surface of the symmetry are less than 60 °C. This indicates that the inlet mass flux effectively cools the upper part of the calandria, close to the surface of the symmetry irrespective of the splitter angle. Figure 9 shows the effect of splitter angle on the maximum temperature and temperature difference between inlet and outlet. The maximum temperature is higher in β = 80 °C and it is found to be minimum for β = 100 °C and from the figure 8 we can see that for β = 100° only one fuel channel fall in the average temperature range of 80 °C to 100 °C. If we increase splitter angle further the maximum temperature also increases. The temperature difference was relatively lower in case of β = 60° and β = 80° as compared to that in case of β = 100°; this indicates relatively lesser heat transfer in case of the former two β values. However, for safe operation of calandria based reactor, it is recommended to have the temperature difference below 18 °C [13]. All the considered cases do satisfy this condition as shown in figure 9. But the one with relatively higher temperature difference, lower temperatures in the whole calandria volume, and lower water mass flow rate is preferable. Unlike β = 60° and β = 80° cases, the temperatures were much lower in case of β = 100° (Refer figure 9). This ensures better safety and lower temperatures of operation with β = 100°.

The percentage of volume covered by the flow for different splitter orientations is shown in figure 10. The percentage of volume covered by the mass flux $\dot{m}_L$ in the case of β = 60°, and 80° is not effective as none of the fuel channels come under this volume (refer to figure 7). The case of β = 100° being the optimum as compared to other splitter plate orientations. Further, the percentage of volume covered by cooler heavy water is more in the case of β = 100° as compared to that in the case of without splitter plate (refer to figure 7). From the average fuel channel temperature plot (refer figure 8) it was observed that for β = 140° the temperature distribution was not uniform and the maximum temperature was found to be 140 °C for the same (refer figure 9). The percentage split of inlet mass flux [% = $\{\dot{m}_L/\dot{m}\}\times100$] where, $\dot{m}$ is the inlet mass flow rate (80 [kg/s])} for splitter angle β = 100° is around 50%. The percentage of fuel channels fall within the streamline is proportional to the volume covered by it.

CONCLUSION

In the considered existing calandria design, the incoming heavy water seems to flow along the surface of the calandria towards the top and then flow down due to gravity. The inlet angle has least effect on the heat transfer for angles greater than 40°. If the thermal power output has to be increased then it demands corresponding increase in the heavy water flow rate.
The present work has shown that the use of splitter plate near the inlet of the calandria can considerably increase the cooling efficiency for the same heavy water flow rate. Splitter plate orientation has strong influence on the streamline pattern and the temperature distribution within the calandria. Splitter plate orientation close to 100° with respect to the inlet is found to be optimum. The inlet mass flux is effectively moving upwards for splitter angle greater than > 100° which is not favorable in this case.

Present studies show that the maximum temperatures inside the calandria can be reduced to about 25 °C without increasing the mass flow rate and hence, no accidental risks. Increase in the percentage of volume covered by the cooler water flow is possible with the introduction of splitter plate. Thus the heat transfer enhancement between fuel channels and the heavy water could be achieved. It is found that the mass flux ratio required to keep the fuel channels' operating surface temperature within the safe limit is around 50%.

REFERENCES

1. Koroyannakis, D., R. D. Hepworth, and G. Hendrie. "An experimental study of combined natural and forced convection flow in a cylindrical tank." Report-AECL 100, pp TDVI-382, (1983).
2. Huget, R.G., Szymanski, J.K., & Midvidy, W.I. "Status of physical and numerical modelling of CANDU moderator circulation." Proceedings of the 29th annual conference of the Canadian Nuclear Association and 10th annual conference of the Canadian Nuclear Society, Vol 1-3, (p. 3v1200), Canada, (1989)
3. Huget, R. G., J. K. Szymanski, and W. I. Midvidy. "Experimental and numerical modelling of combined forced and free convection in a complex geometry with internal heat generation." Proceedings of 9th International Heat Transfer Conference. Vol. 3, pp 327-332, (1990).
4. Ravi, S. D., N. K. S. Rajan, and P. S. Kulkarni. "Computational and experimental studies of fluid flow and heat transfer in a Calandria based reactor." Computational Fluid Dynamics 2008, pp 233-238 (2009)
5. Kulkarni, P. S., N. K. S. Rajan, and R. S. Tupake. "A study and analysis of fuid dynamic and heat transfer behaviour in a calandria based reactor." Computational Fluid Dynamics Journal, vol-13, pp 79-84 (2004).
6. Yoon, Churl, Bo Wook Rhee, and Byung-Joo Min. "3-D CFD Analysis of the CANDU-6 Moderator Circulation under Normal Operating Conditions"." Journal of the Korean Nuclear Society, 36.6, pp 559-570 (2004).
7. Yoon, Churl, Rhee, Bo Wook, & Min, Byung Joo. "A study on hydraulic resistance of porous media approach for CANDU-6 moderator analysis." Proceedings of the KNS-KARP Joint spring meeting, (pp. CD-ROM). Korea, Republic of: KNS (2002).
8. Kim, Manwoong, Seon-Oh Yu, and Hho-Jung Kim. "Analyses on fluid flow and heat transfer inside Calandria vessel of CANDU-6 using CFD." Nuclear Engineering and Design, Vol 236.11, 1155-1164, (2006).
9. Arsene, Radu, Ilie Prisecaru, and Ştefan Nicolici. "Improvement of the Thermal hydraulic characteristics in the Calandria Vessel of a CANDU 6 Nuclear Reactor." UPB Science Bulletin, Vol 75.4, pp 251-262, (2013).
10. Rajan N.K.S, Kulkarni P. S., and Suneel M. P., 2014. "Numerical investigation of the performance of a calandria based reactor under varied orientation of injection", Eighth international conference on computational fluid dynamics, China, pp. CD-ROM, (2014).
11. Yoon, C., B. W. Rhee, and B. J. Min. "3-D Moderator Circulation Simulation of CANDU6 Nuclear Reactor Using CFX-4.3." Proceedings of the Korean Nuclear Society Conference. Korean Nuclear Society, pp. CD-ROM, (2001).
12. Yoon, Churl, Bo Wook Rhee, and Byung Joo Min. Validation of a CFD model for predicting CANDU-6 moderator circulation against SPEL experiments. No. KAERI/TR--1999/2001. Korea Atomic Energy Research Institute, (2001).
13. Bajaj, S. S., and A. R. Gore. "The Indian PHWR." Nuclear Engineering and Design, Vol 236.7, pp 701-722, (2006).