Tenth International Conference on Computational Fluid Dynamics (ICCFD10), Barcelona,Spain, July 9-13, 2018 **ICCFD10-270**

Fluid Flow and Heat Transfer Analysis in a Calandria Based Reactor for Different Fuel Channel Configurations

P. S. Kulkarni*, N. K. S. Rajan*, Suneel M. P** and Vijesh Joshi***
Corresponding author: psk@aero.iisc.ernet.in; pskdhar@gmail.com

*Indian Institute of Science, Bangalore, INDIA
**HCL Technologies Ltd, Bangalore, INDIA.
***School of Mechanical Engineering, Vellore Institute of Technology, Vellore, INDIA.

Abstract: Continuous heat removal from the fuel channels in the calendria based reactor is important to avoid any hazardous aftermath. The primary objective of present study is to analyze heat removal capacity of calendria for four different fuel channel configuration. As a design optimization point of view there will be many design parameters need to optimize, in the former analysis, it was observed that the mass flow rate and inlet orientation have a significant effect on the convective heat transfer from the fuel channels to the moderator. The combination of mass flow rate and the inlet angle leading to lowest maximum temperatures inside the calendria was found to cause the underutilization of the calendria. Hence the latter part of the analysis has been carried out in the direction of improving the efficiency of the calendria by splitting the mass of flow near the inlet. In the present paper, the discussion has been done on heat removal capacity of calendria based reactor for inline and staggered fuel channel configurations. The numerical study shows that square pitch 90^0 inline arry configuration is better as compared to other considered configurations.

Keywords: Calandria, Computational Fluid Dynamics, Heat Transfer.

1 Introduction

Calandria is essentially a nuclear reactor core and it consists mainly of fuel channels, pressure tubes control rods. Heavy water (moderator) flows through the calandria for mainly (i) to moderate the neutrons and (ii) to keep the pressure tubes at lower temperatures. Hence, it is necessary to minimize the operating temperature of the pressure tubes during loss of primary fluid and failure of emergency core cooling. The moderator inside the calandria must be capable of removing all the heat generated inside to keep calandria tube temperature within the safe limit in situations like loss of coolant by accident. There are many parameters which govern the heat transfer and internal fuel channel temperature distribution inside the calandria during an event such as loss of coolant by accident. In order to understand the fluid flow and heat transfer phenomena inside the calandria there have been many experimental studies on scaled test facilities which are capable of reproducing important thermohydrulic phenomena and it is practically difficult to carry out experiments on operating nuclear reactors. Koroyannakis et al. [1] first conducted a series of experiments at Sheridan Park Laboratory (SPEL). The dimensions of a scaled model are not exactly mimicking the real calandria geometry, but the scaled model was capable of reproducing important thermohydrulic phenomena. The earlier experiments [2] [3] were conducted for different configurations like, with and without pressure tube banks, with and without heat load, enforced symmetry by blocking flow at the vertical mid-section and for different mass flow rates.

Earlier CFD studies [4], [5] have used simplified geometry representation by approximating porous media for matrix of the calandria tubes inside the calandria. The experimental studies have given a detailed flow structure inside the calandria, the flow inside the calandira can be classified into three groups namely a. Momentum dominated flow, b. Buoyancy dominated flow [5] and c. mixed type of flow which occurs during transition from buoyancy dominated flow to momentum dominated flow or vice versa. These flow patterns depends on rate of heat load and inlet mass flow rate. Isothermal experiments [6] were carried out on a scaled calendria model for fixed inlet angle and mass flow rates. Numerical results validated against the isothermal experimental results and non- isothermal studies have done to estimate tolerance band for safe operation during events like loss of primary coolant by accident. Earlier numerical studies have shown that the CFD analysis of fluid flow and heat transfer are quite complex due to both the geometry and the nature of mixing in the calandria.

A series of CANada Deuterium Uranium (CANDU) reactors are analyzed by using Canadian Algorithm for Thermohydraulics Network Analysis (CATHENA). This algorithm is capable to simulating transient two phase flow by accounting solid conduction. Yoon et al. [5] computationally studied 3D moderator circulation for CANDU-6 reactor accounting radial and axial power distribution. The predicted maximum temperature of moderator is around 82.9 ^{o}C for the inlet moderator velocity of 2m/s. Kim et al. [7] given a flow regime map for buoyancy dominated, momentum dominated and mixed type of flow by considering Reynolds (Re) and Archimedes (Ar) dimensionless numbers for different inlet velocity and heat load. This information can be used effectively during the initial stages of design. Similar studies on CANDU reactor have shown that for low inlet velocity the flow inside the calandria is dominated by buoyancy and a small fluctuation in fuel channel temperature can create circulation zones [8] and which may lead to asymmetric flow.

In previous numerical studies [6], [9] it was observed that the mass flow rate and the inlet orientation have a significant effect on the convective heat transfer from the fuel channels to the moderator. The best combination of inlet angle and mass flow rate was found by brute force approach [10]. An inlet angle between 30^{0} to 60^{0} (with respect to horizontal) was found to be optimum for different heat load conditions and for mass flow rate of 172 kg/s. Even do the maximum calandria tube temperatures below the boiling point temperature the most of the flow was cooling effectively the upper half of the calandria tubes. Geometric modification near inlet (introduction of a splitter) has a greater effect on the flow behavior [11]. The simple introduction of a splitter (airfoil shaped body) was found to have considerable effect on the temperature inside the calandria by splitting the inlet flow into two directions and effectively cooling most of the calandria tubes. A simple geometric modification has resulted to use the same calandria geometry for higher heat loads i.e., 1200MW to 1800MW. In case of calandria with the splitter, total mass flux at the inlet ($\dot{m}$) splits into two parts, i.e. the mass flux which is coming from below the splitter and from above the splitter. There were total 553 fuel channels (diameter = 0.132 m), equally spaced (square pitch = 0.288 m) and are submerged in the moderator. NACA0012 airfoil geometry was considered as a splitter and was placed at the center of the inlet and 4m from the calandria center.

In the present paper, the effect of four different fuel channel configurations on the fuel channel temperature and pressure drop has been investigated numerically using commercial CFD package ANSYS CFX for 600MW thermal load and with inlet Re of $8.43X10^{5}$. Buoyancy effect is included with Boussinesq approximation. k-ϵ turbulence model is used to predict turbulent flow [7], [12]. The present work is a continuation of previous study on effect of moderator inlet injection angle, inlet mass flow rate and geometry modification near inlet on performance of calandria based reactor, where the brute-force approach has been used to arrive at optimum range of inlet angle for constant inlet flow rate.

2 CFD model and its validation

In the present study numerical investigation is carried out for different fuel channel configurations (Inline array and staggered array) as shown in Figure1 for a specified mass flow rate and heat load condition. Heat removal capacity has been evaluated for different fuel channel configuration. Changing the fuel channel configuration will affect the pressure drop as well as heat transfer and fluid flow behavior. The objective of the present work is to primarily look into the temperature distribution inside the calandria and to evaluate pressure drop. This study is important to comment on the effect of fuel channel configuration on the minimization of the operating temperature of the calandria during events like loss of coolant by accident and emergency core cooling.

Table 1: Details of considered geometry and fuel channel configuration and mesh size

Type	Rotation β	X-Pitch m	Y-Pitch m	Total Tubes	Mesh Size millions
Square pitch 90^o In-line array	90^o	0.264	0.264	625	4.6
Rotated triagular 60^o Staggered array	60^o	0.264	0.229	741	4.8
Rotated square 45^o stagered array	45^o	0.373	0.187	627	4.6
Rotated Triangular 30^0 stagered array	30^o	0.457	0.132	741	5.0

2.1 Details of Considered Geometry

A schematic view of the considered calandria model is shown in Figure 1. there are more than 600 fuel channels (exact number depends on type of fuel channel arrangement) with diameter of 0.132 m, arranged in different configurations and are submerged in the moderator. The diameter of the calandria is 8m and has 8m length in Z-direction. Inlets are located diametrically opposite sides and outlet is located at the bottom. Four different tube arrangement were considered namely, 1. Square pitch 90^o In-line array, 2. Rotated triangular 60^o Staggered array, 3. Rotated square 45^o staggered array and 4. Rotated Triangular 30^0 staggered array refer Figure 1. The details of X and Y pitch, rotation angle, total number of tubes and mesh size is given in Table1.

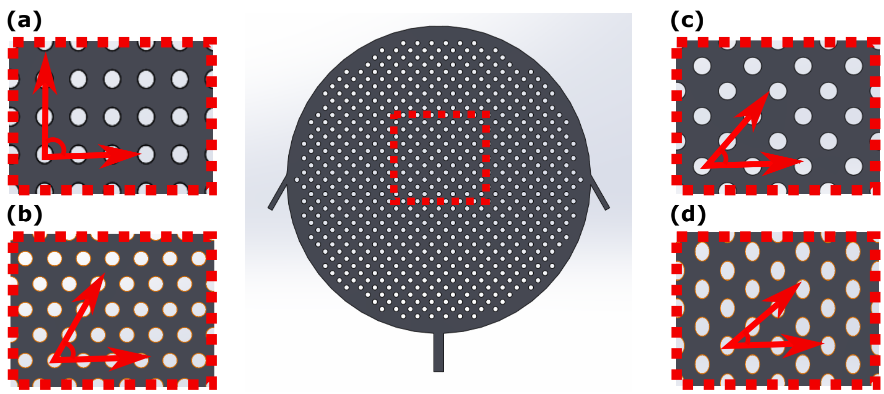

Figure 1: Detailed view of calandria geometry considered for present study and inset are, (a) Square pitch 90^o In-line array, (b) Rotated triangular 60^o Staggered array, (c) Rotated square 45^o staggered array and (d) Rotated Triangular 30^0 staggered array.

2.2 Equations Solved, Boundary Conditions and Mesh details

The steady state flow is simulated by consideration of convection inside the calandria. Reynolds Averaged Navier Stokes equations (RANS) are solved for convection heat transfer to the water, buoyancy term is added to the y-momentum equation using a Boussinesq model. The Boussinesq model is used when density variation is driven by temperature variation. In Boussinesq model a source term is added to the momentum equation (parallel to the gravitational direction) to account for density variation and by keeping constant reference density in all other equations. The buoyancy source term is approximated as

$$\rho - \rho_{ref} = -\rho_{ref}\beta(T - T_{ref}) \tag{1}$$

Where, ρ is the density of the fluid, ρ_{ref} is reference density, β is thermal expansion coefficient, T is temperature of the fluid and T_{ref} is the reference temperature.

A standard k-ϵ turbulence model [7], [12] has been used with default model constants. Based on Root Mean Square (RMS) residual in the variables such as density, velocity, temperature, etc. solution of equations

Table 2: Details of applied boundary conditions

Boundary type	Assigned condition	Value
Inlet	Mass flow	140 kg/s
Outlet	Static pressure	'0' atm (Relative pressure)
Fuel channels	No slip, with Heat Load	600MW
Calandria wall	No slip	Adiabatic wall

are obtained. The RMS residual is targeted bellow 10-4 for all the equations. Heat flux boundary condition is specified for calandria tube surface and the value depends on the number of calandria tubes for 600MW thermal load. Inlet temperature is considered as 44.2^oC [13] and the outlet is considered to be at atmospheric pressure and temperature. No-slip wall conditions are specified for all the walls. In this study, unstructured tetrahedron grid is generated using ANSYS Workbench Meshing, in which the computational domain is discretized into a finite number of control volumes. Higher grid density regions are created near the walls with the help of prism layers. To study the dependency of results on the number of control volumes, calculations are repeated for five different grid sizes (control volumes). The maximum fuel channel temperature and temperature at three selected sections is observed for any variation due to increase in number of elements and it is found that there is 1^oC increase in the maximum temperature for increasing elements from 3.5 to 4.6 millions. The solution process is an iterative and the solution starts with an initial condition as constant temperature through the domain i.e. 44.2^oC. The details of applied boundary conditions are given in Table 2.

2.3 Assumption Involved

Because of the geometrical symmetry in the construction of calandria, only half portion is considered for numerical simulation and also 0.2 m symmetry section is considered in length wise direction by assuming inlet and outlets are spanning throughout its length in Z-direction. The working fluid (water) is assumed as incompressible single phase flow. The heating of the fuel channels is assumed to be uniform throughout the considered fluid domain.

2.4 Validation

The numerical procedure is validated with isothermal and non-isothermal literature experimental work. Velocity and temperature distribution along the centerline are compared. The Figure 2a shows the comparison of vertical velocity variation along the vertical center line of the calandria between present simulated results

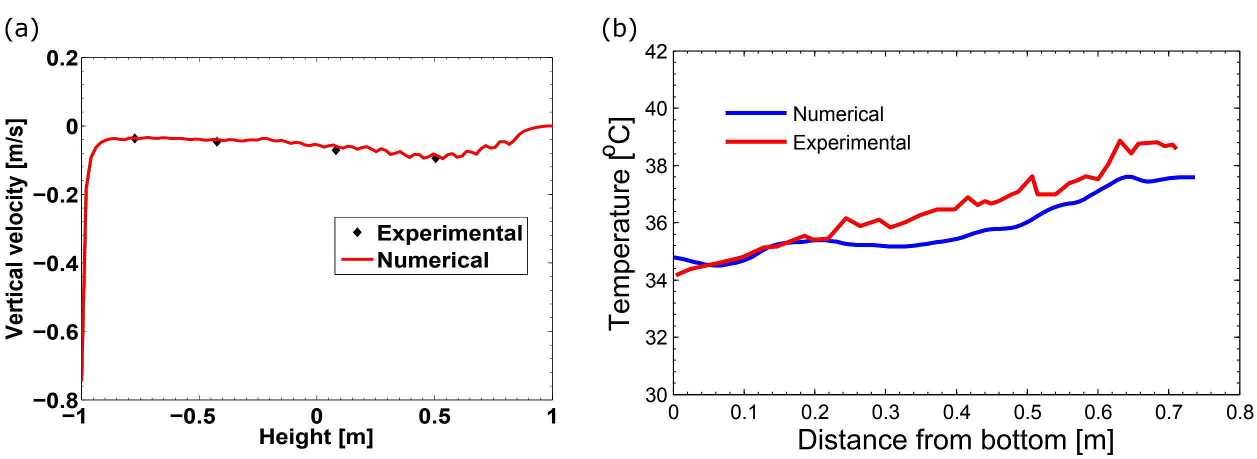

Figure 2: Validation of numerical model with the literature experimental work. a. Vertical velocity comparison along a vertical center line [3] and b.Temperature comparison along a vertical center line [1]

and the isothermal experimental work by Huget et al. [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 very well with the experimental work. The present numerical model is also validated with the non-isothermal experimental work by Koroyannakis et al. [1], where the heat load is 100 kW and mass flow rate is 2.4kg/s. The temperature comparison shown in Figure 2b is deviates as the height increases, this may be due to collapsing of flow symmetry [14] and this may lead to the mixed type of flow [7] as explained earlier.

3 Results and Discussion

The steady state flow is simulated by solving Reynolds Averaged Navier-Strokes Equation (RANS) inside the calandria. A standard k-ϵ turbulence model has been used with scalable wall function to model near wall effects. The simulation are carried out for different fuel channel configurations (Refer Figure 1), no-slip wall conditions is imposed on the walls. The maximum fuel channel temperature, pressure drop and the fuel channel temperature distribution are the main observable parameters. All considered configurations are for inlet angle 45^0 and heat load of 600MW.

Figure 3 shows the diffusion of inlet flow along its length. For 90^0 inline array 3a, most of the inlet flow

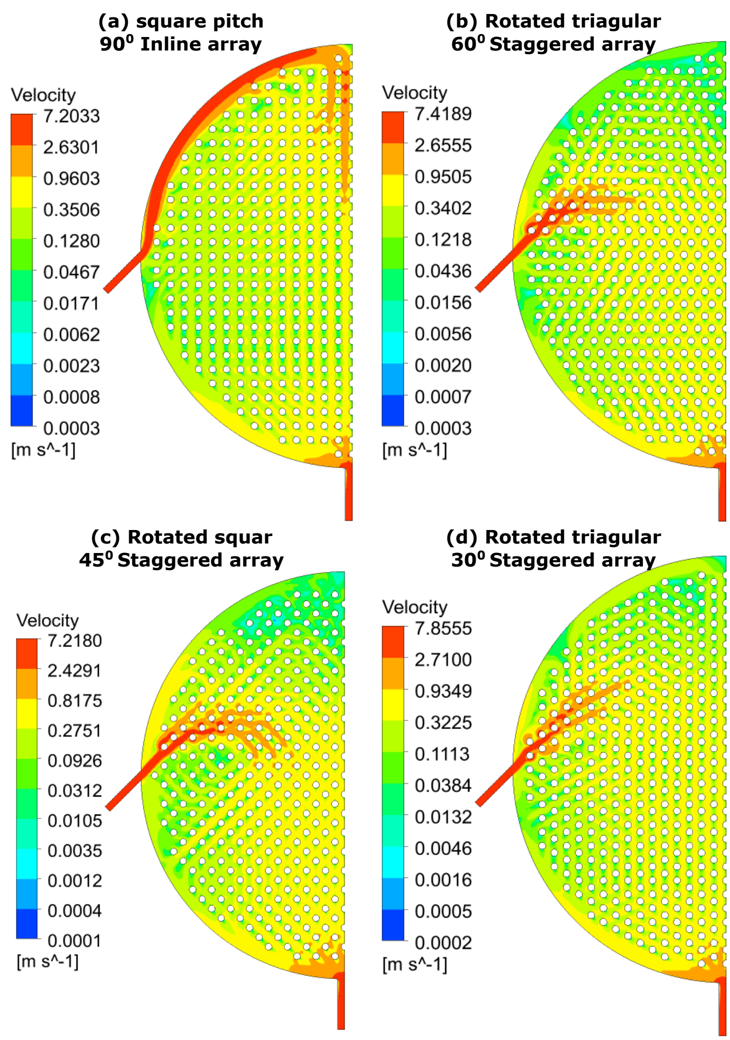

Figure 3: Velocity contour at the mid-section for different configurations.

is attached with the calandria wall and the maximum temperature reached in this case is well below the boiling point temperature. The increase in temperature at the outlet is around 14^0C which indicates that there is an occurrence of effective heat transfer between calandria tube and the working fluid. For staggered array configuration the inlet flow is diffusing in-between the calandria tubes and the tubes which are in the upper part not receiving sufficient flow from the inlet. For all staggered array configurations the flow is similar in the core region of the calandria, there were no recirculation zones observed as compared to previous work [10]. For all considered geometries the reflector region was having same dimension, the arrangement of calandria tubes affected the inlet diffusion length and it was varying across the considered cases. The tubes which are in direct contact with the inlet flow are absorbing most of the inlet energy. There is not much difference in the pressure drop between inlet and outlet for all considered configurations. The pressure drop was highest in triangular staggered array configuration as compared to square staggered array.

As discussed earlier most of the calandria based reactors will operate in the mixed type of flow regime, where buoyancy and momentum forces will make flow to be mixed. Figure 4 shows the temperature dis-

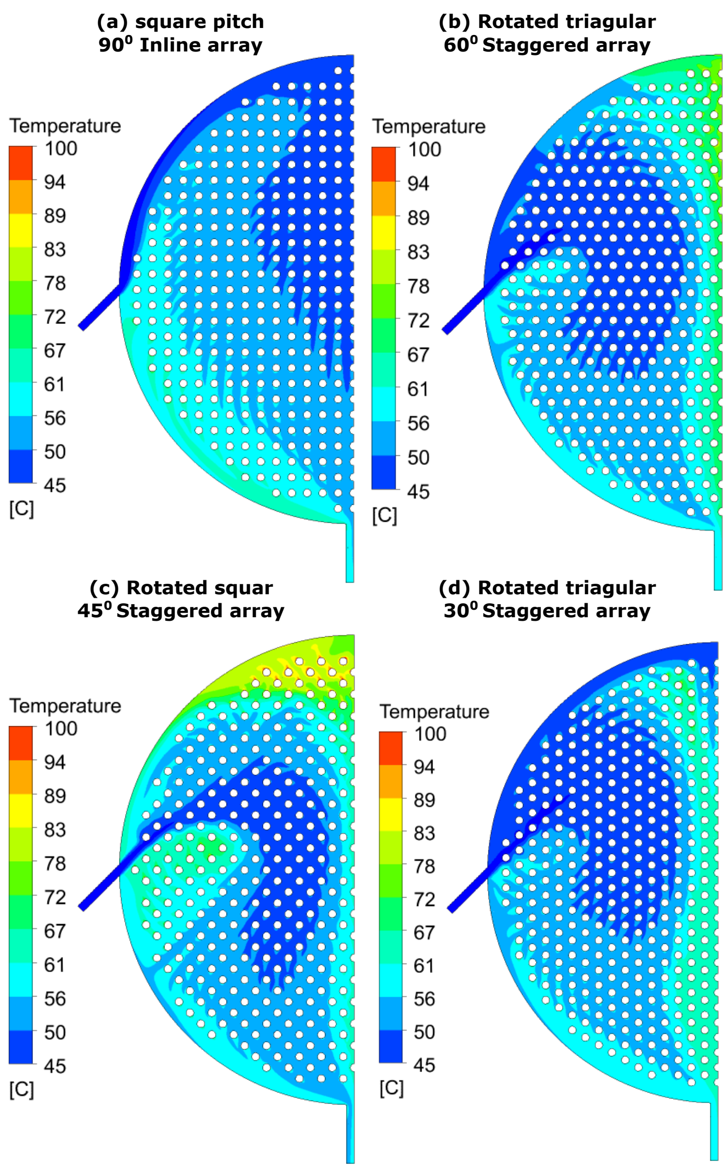

Figure 4: Temperature contour at the mid-section for different configurations.

tribution for all the considered cases. For square pitch 90^0 inline array configuration (refer 4a) the low temperature region is covering top part of the calandria where as for staggered 45^0 configuration the top part is filled with high temperature fluid and this is the region where buoyancy dominated flow likely to occur. In the square 90^0 configuration the maximum calandria tube temperature is well below the boiling point temperature and it is reaching highest value in rotated square 45^0 configuration (refer 4c).

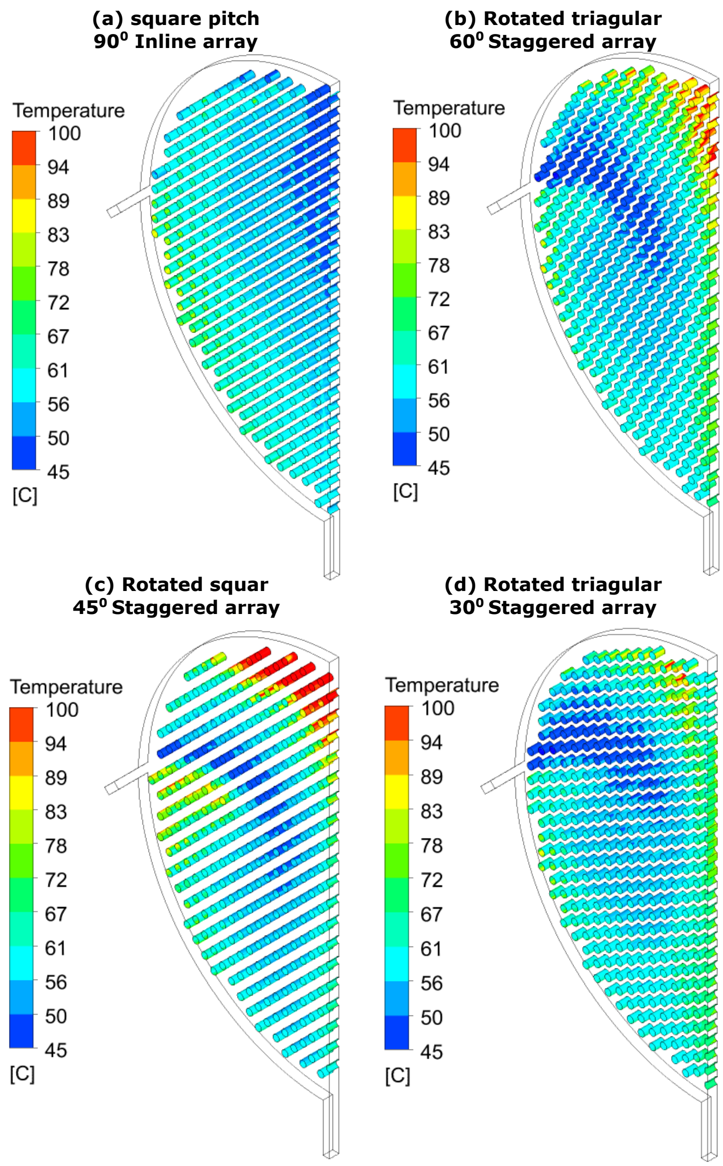

Figure 5: Calandria tube surface temperature for different configurations.

The pressure tube temperature will be dependent on the local convective heat transfer coefficient, if the fresh inlet water diffuses up-to the core then there will be high heat transfer from the calandria tubes which intern reduces the surface temperature. Due to fuel channel arrangement it is difficult to diffuse up-to the core of the calandria. Figure 5 shows the calandria tube surface temperature, as the flow is approaching fro the top (refer figure 5a) most of the upper portion of the tubes receiving low temperature fluid from the inlet where as for rotated square 45^0 staggered array the inlet flow is turning midway leaving top portion under cooled.

4 Conclusion

Numerical analysis of the fluid flow and heat transfer has been carried out for the calandria based reactor to ensure safe working of the calandria based reactor during events like loss of coolant by accident and/or failure of emergency core cooling. Four different calandria tube arrangements were studied for 600MW thermal power.

- Numerical and literature experimental results compared favorably, flow visualization and calandria tube temperature distribution for different tube configurations were obtained by present numerical simulation.
- The maximum calandria temperature was observed in rotated square 45^0 staggered calandria tube configuration and the number of tubes crossing boiling point temperature is highest in this arrangement.
- The difference between pressure drop is less across all considered configurations, highest pressure drop was observed in triangular staggered array configuration.

Table 3: Details of outlet, maximum pressure tube temperature and number of tubes crossing boiling point temperature for different considered configuration

Configuration type	Outlet Temp ^{0}C	Max PT Temp ^{0}C	Number of tubes crossing boiling point Temp
Square pitch 90^0 Inline array	58.05	88.50	0
Rotated triangular 60^0 staggered array	58.48	127.90	17
Rotated square 45^0 staggered array	56.04	137.09	22
Rotated triangular 30^0 staggered array	59.20	114.58	2

- In the core the hear transfer stabilizes due to flow diffusion for all considered configurations, lower heat transfer rate has seen in the upper part of the calandria.

References

[1] D Koroyannakis, R D Hepworth, and G Hendrie. Experimental study of combined natural and forced convection flow in a cylindrical tank. Technical report, AECL Report, 1983.

[2] R G Huget, J K Szymanski, and W I Midvidy. Status of physical and numerical modelling of CANDU moderator circulation. In *Procedings of 10th Annual Conference of the Canadian Nuclear Society, Ottawa*, 1989.

[3] R G Huget, 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. In *Procedings of 9th International Heat Transfer conference*, volume 3, page 327, 1990.

[4] P S Kulkarni, N K S Rajan, and Ravindra S Tupake. A study and analysis of fluid dynamic and heat transfer behaviour in a calandria based reactor. *Computational Fluid Dynamics Journal*, 13:79–84, 2004.

[5] Churl Yoon, Bo Wook Rhee, and Byung-Joo Min. 3d cfd analysis of the candu-6 moderator circulation under normal operating conditions. *Journal of the Korean Nuclear Society*, 36(6):559–570, 2004.

[6] S D Ravi, N K S Rajan, and P S Kulkarni. Computational and experimental studies of fluid flow and heat transfer in a calandria based reactor. In *Computational Fluid Dynamics, Springer*, page 233, 2008.

[7] Manwoong Kim, 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*, 236(11):1155–1164, 2006.

[8] Araz Sarchami, Nasser Ashgriz, and Marc Kwee. Three dimensional numerical simulation of a full scale candu reactor moderator to study temperature fluctuations. *Nuclear Engineering and Design*, 266: 148–154, 2014.

[9] M H Dinesh, S D Ravi, G Premkumar, and N K S Rajan. Limmiting thermal dissipation in a calandria based nuclear reactor. In *Proceeding of the International Conference on Advances in Mechanical Engineering, Gujarat, India*, 2008.

[10] N K S Rajan, P S Kulkarni, and M P Suneel. Numerical investigation of the performance of a calandria based reactor under varied orientation of injection. In *The Eighth International Conference on Computational Fluid Dynamics, China*, 2014.

[11] P S Kulkarni, Vijesh Joshi, M P Suneel, and N K S Rajan. Analysis of the flow inside a calandria based reactor and the proposed geometric modification for better efficiency. In *Proceedings of the 23rd National Heat and Mass Transfer Conference, Thiruvananthapuram, India*, page 1273, 2015.

[12] C Yoon, BW Rhee, and BJ Min. 3-d moderator circulation simulation of CANDU6 nuclear reactor using CFX-4.3. 2001.

[13] SS Bajaj and AR Gore. The indian PHWR. *Nuclear Engineering and Design*, 236(7):701–722, 2006.

[14] Jae Ryong Lee, Sang Gi Park, Han Young Yoon, Hyoung Tae Kim, and Jae Jun Jeong. Numerical study for CANDU moderator temperature prediction by using the two-phase flow analysis code, CUPID. *Annals of Nuclear Energy*, 59:139–148, 2013.