

Complex fluid flow and heat transfer analysis inside a calandria based reactor using CFD technique

P. S. Kulkarni

Computational Mechanics Laboratory, Department of Aerospace Engineering, Indian Institute of Science, Bangalore, 560012, INDIA

E-mail: psk@aero.iisc.ernet.in, pskdhar@gmail.com

Abstract. Series of numerical experiments have been carried out on a calandria based reactor for optimizing the design to increase the overall heat transfer efficiency by using Computational Fluid Dynamic (CFD) technique. Fluid flow and heat transfer inside the calandria is governed by many geometric and flow parameters like orientation of inlet, inlet mass flow rate, fuel channel configuration (in-line, staggered, etc.), location of inlet and outlet, etc.,. It was well established that heat transfer is more wherever forced convection dominates but for geometries like calandria it is very difficult to achieve forced convection flow everywhere, intern it strongly depends on the direction of inlet jet. In the present paper the initial design was optimized with respect to inlet jet angle, the optimized design has been numerically tested for different heat load mass flow conditions. To further increase the heat removal capacity of a calandria, further numerical studies has been carried out for different inlet geometry. In all the analysis same overall geometry size and same number of tubes has been considered. The work gives good insight into the fluid flow and heat transfer inside the calandria and offer a guideline for optimizing the design and/or capacity enhancement of a present design.

1. Introduction

In nuclear reactors heavy water is used as the moderator for both energy extraction (primary fluid) and for cooling (secondary fluid). The reactor assembly mainly consist of calandria, Steam Generator, Emergency core cooling system, primary and secondary moderator circulation system, etc. Calandria is a high pressure reactor vessel consisting of stainless steel structure, which holds the hundreds of horizontal fuel channels. In a calandria based reactor fuel channels are arranged in horizontal fashion, which contains fuel material and the primary fluid circulating inside it. Due to fission reaction huge amount of energy is releases in the form of heat, high pressure primary fluid which is circulating inside the fuel channels carries part of this heat to the boiler where it converts ordinary water into steam for energy generation. The secondary fluid which is circulating around the fuel channels is mainly acts as moderator. During loss of primary fluid and the emergency core cooling the moderator inside the calandria must be capable of removing heat to keep a fuel channels within the safe operating conditions.

In order to understand the flow phenomena inside the calandria there have been many experimental studies on scaled test facilities. 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



thermohydraulic phenomena. Some experimental studies have been conducted for isothermal [2] and non-isothermal cases [3], [4] to identify the effect of inlet mass flow rate and internal heating on the flow and temperature distribution inside the calandria. It is very difficult to carry out experimental studies on the full scale calandria model and the alternative approach is to carry out numerical analysis. A Computational Fluid Dynamics (CFD) approach has been widely used to study the flow behavior and heat transfer inside the calandria. Initial CFD studies [5] have used simplified geometry representation by approximating porous media for the matrix of the calandria tubes inside the calandria. According to Rhee et al. [5] the flow inside the calandria can be classified into three groups namely a. Momentum dominated flow, b. Buoyancy dominated flow and the c. Mixed type of flow. These flow patterns deepens on the heat load and inlet mass flow rate.

Experimental analysis of fluid flow and heat transfer inside a full scale calandria unit is associated with huge cost and dangerous for handling high heat loads, mass flux conditions makes it practically complex. Conducting experiments on a scaled model or numerical simulations helps for the designer to optimize the design and one can do lot of parametric studies. There are many parameters which govern the heat transfer and intern fuel channel temperature distribution inside the calandria during an event such as loss of coolant accident(LOCA). As to optimize the calandria geometry different geometry configurations have to be studied.

The case study has been done for different injection rates (mass flow rates) of the moderator and varying heat flux (case 1). Inlets are symmetrically located at 3 'o' clock and 9 'o' clock position, outlets are located at 5 'o' clock and 7 'o' clock position. Fuel channels are arranged in staggered fashion. Two sets of configurations have been studied to analyze the flow behavior namely (1) Without considering the fuel channel geometry and (2) with considering fuel channel geometry. For the case with fuel channels, constant heat flux was applied as a boundary condition, in the whole analysis computation is limited to laminar flow only. The inclination of inlet is exactly horizontal (90^0 to the vertical plane). Moderator inlet angel, inlet flow rate, fuel channel configuration, outlet location, etc., are the main geometric and the flow parameters which must be optimized for safe the working of reactor and to increase the overall heat transfer efficiency.

As to improve the heat removal capacity, analysis was carried out to assess the effects of the angle at which the coolant is injected into the calandria on the heat transfer (case 2). The study was carried out for an assigned coolant flow rate and heat dissipation conditions so as to establish the effect of mixing and enhancements in the heat transfer behavior in the reactor on the considered parameter. The simulation is carried out for fixed Reynolds number of 2.5×10^6 at the inlet, four different heating loads (300MW, 500MW, 800MW and 1200MW) at fuel channels and for different orientation of injection. The aim was to investigate how the orientation of inlet is affecting the heat transfer inside the calandria based reactor and to arrive at the optimal design considerations for safe working of calandria based reactor.

From the study (case 2), where the brute-force approach has been used to arrive at optimum range of inlet angle for constant inlet flow rate, it was found that the inlet angle between 30^0 - 60^0 degree (with respect to the horizontal) gives optimum heat removal for all considered heat loads. The combination of mass flow rate and the inlet angle leading to lowest maximum temperatures inside the calandria was found to cause the underutilization of the calandria. Hence the latter part of the analysis has been carried out in the direction of improving the efficiency of the calandria by splitting the mass of flow near the inlet (case 3).

2. Geometry details

The nature of the flow inside the Calandria has significant effect on the safe operation of reactor vessel in PHWR's (Pressurized Heavy Water Reactors). Since the considered geometry is symmetry in nature (X Y Plane), only half portion was considered for computational study and is as shown in figure 1a, which is having diameter of 8 m and in third direction the inlets and outlets are identical throughout its length (Z direction) and hence a section has been considered for computational study. The symmetry

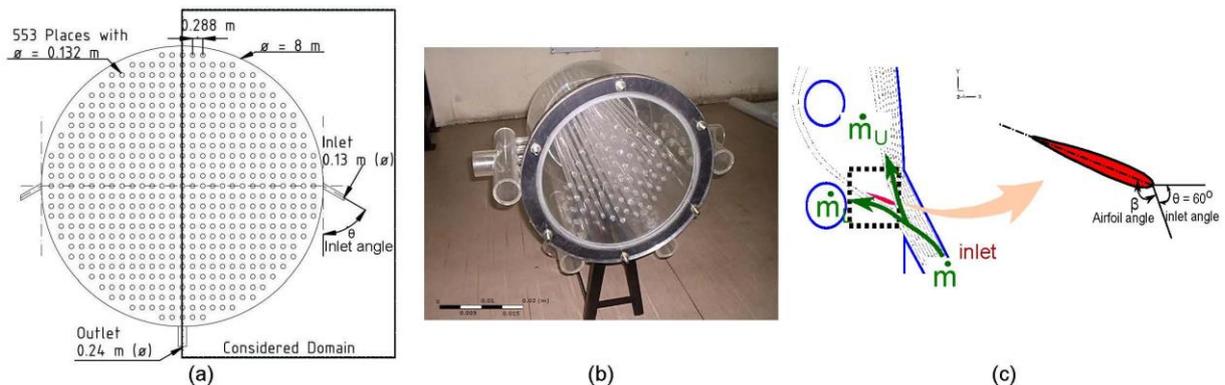


Figure 1. Detailed view of calandria geometry considered for numerical study. (a) considered symmetry model (b) scaled experimental model and (c) Location of airfoil near inlet.

portion of the Calandria contains total 290 fuel channels with diameter of 0.132 m each, one inlet at side and one outlet at the bottom. The diameter of the outlet (0.24 m) is greater than that of the inlet (0.13 m) which is to provide enough pressure difference to flow. The fuel channels are arranged in square form having 0.288 m pitch between them.

A schematic view of the modified design and also a proposed modification near the inlet (placing a splitter) is shown figure 1c. 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 (\dot{m}_L) and from above the splitter (\dot{m}_U) as shown in 1c. There are total 553 fuel channels (diameter = 0.132 m) in the full geometry, equally spaced (square pitch = 0.288 m) 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. Splitter has been added for the case with 1800MW thermal power by placing a NACA0012 airfoil at the center of the inlet and 4m from the calandria center (refer figure 1c). The range of splitter angle (β) studied are from 600 to 1800.

3. CFD model and its validation

3.1. Equations Solved, Boundary Conditions and Mesh details

The steady state flow is simulated by consideration of convection and radiation 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

$$\rho - \rho_{ref} = -\rho_{ref}\beta(T - T_r) \quad (1)$$

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 has been used [6], with default model constants (C_{mu} - 0.09, C1-Epsilon - 1.44, C2-Epsilon - 1.92, TKE Prandtl Number - 1 and TDR Prandtl Number - 1.3). The simulations are carried out for different mass flow rates, different inlet and different splitter angles. Inlet temperature is considered as 44.2°C [7]. The outlet is considered to be at atmospheric pressure. No-slip wall conditions are specified for all the walls.

3.2. Assumption Involved

Because of the geometrical symmetry in the construction of calandria only half portion is considered for numerical simulation and also 0.05 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. The present numerical model is also validated with the non-isothermal experimental work by Koroyannakis et al. [1].

4. Results and discussion

4.1. Optimization with respect to inlet injection angle

Varying inlet angle step by step is an effective parametric approach for arriving to the optimum inlet injection angle. In this optimization process inlet angle varied from 15° (with respect to vertical axis) up-to 90° . The comparison of fuel channel temperatures reaching boiling point temperature for different injection angles and for different power is shown in figure 2. In figure 2d results for only five cases have been presented (i.e. 90° injection angle is not included). The figure shows the area covered by lines for different injection angles and the area under curve shows the region of calandria where temperature reaching boiling point temperature for different considered thermal power. For all considered thermal power and injection angles the value of fuel channel temperatures are less between injection angles 30° to 60° .

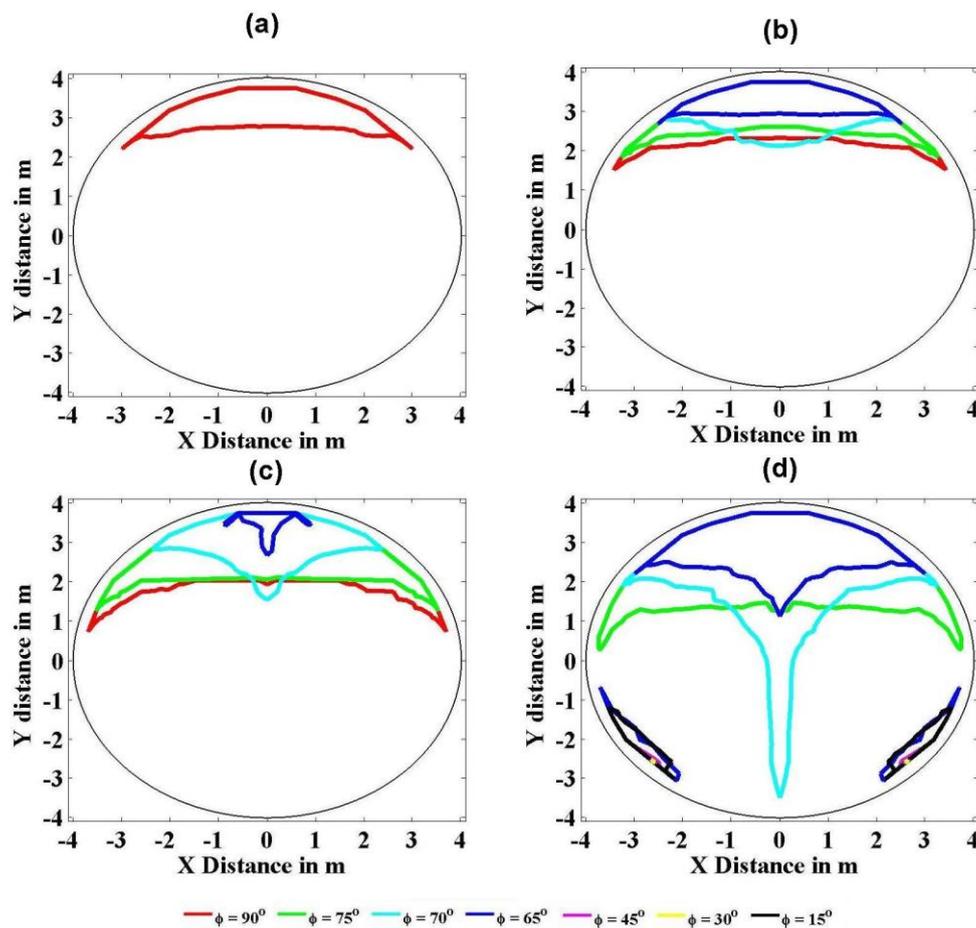


Figure 2. Temperature reaching boiling point temperature for different injection angles (a) 300 MW (b) 500 MW (c) 800 MW and (d) 1200 MW.

Further analysis has been carried out for the case of 1200MW thermal power. The figure 3a shows the variation of temperature for different mass flow rates (dotted lines), the inlet angle is 60° and also it shows variation of temperature for different inlet angles. It was observed that any increase

in the inlet angle above 40° with respect to the horizontal (clockwise) has negligible effects on the resultant temperature. Apart from this, it is also observed that lower mass flow rate cases have higher temperature. But it is important to see the corresponding maximum temperatures of the fuel channel surfaces to prevent any hazardous accidents. The lowest mass flow rates in the present case study (i.e., 60kg/sec and 80kg/sec) leads some of the fuel channels either in the top region or below the inlet close to the calandria wall to attain maximum temperature values greater than 100°C . But the lower mass flow rates ensure low pumping power requirements. In the present work, change of phase has not been considered; hence, further analysis has been carried out for 80kg/sec mass flow rate with inlet angle 60° .

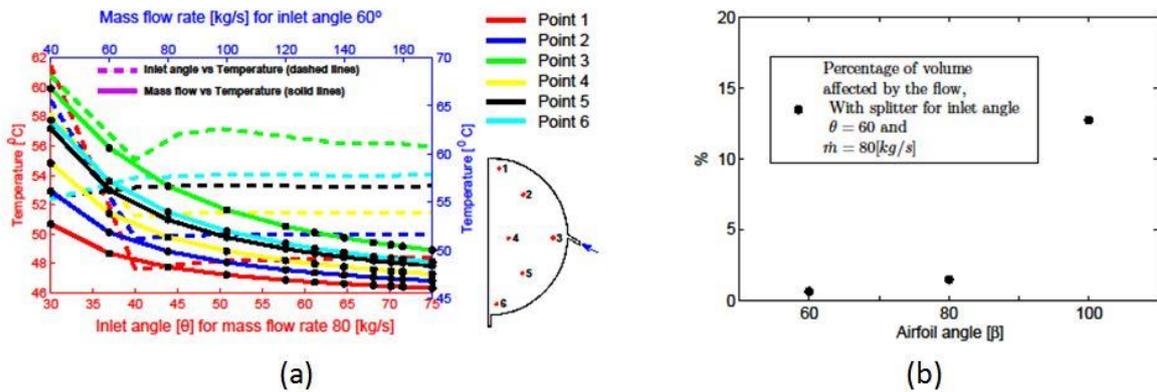


Figure 3. (a) Variation of temperature at few selected locations (points 1 to 6) with respect to inlet injection angle and mass flow rate (b) percentage of volume affected by the flow.

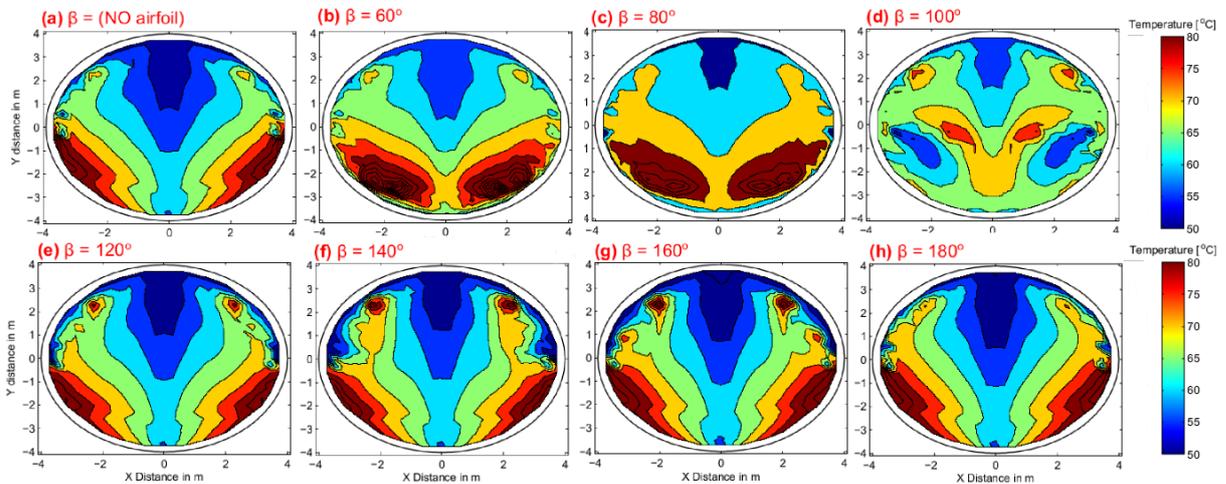


Figure 4. Temperature distribution for different splitter angles.

4.2. Optimizing by introduction of splitter

The idea of using the splitter was to split the mass flow to make the cooler incoming heavy water come in contact with the as many fuel channels as possible, which can ensure enhancement in the heat transfer. The details of splitter and splitter angle studied are explained in the previous section. For splitter angles $\beta = 60^\circ$ and $\beta = 80^\circ$ the mass flux from the bottom of the splitter (\dot{m}_L) is just flowing along the calandria wall where as for higher β values (i.e. $\beta > 80^\circ$) the mass flux (\dot{m}_L) is covering some of the fuel channels. For $\beta > 100^\circ$, the mass flux \dot{m}_L and the mass flux from the top of the splitter (\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. For all considered splitter angles the flow diverting towards the bottom part of the calandria can be seen only in splitter angles $\beta = 60^\circ, 80^\circ$ and

100°. The fuel channel temperature distribution for different splitter angles is as shown in figure 4. The percentage of volume covered by the flow is shown in figure 3b. Even though the percentage of volume covered in the case of $\beta = 60^\circ$, and 80° by the mass flux (\dot{m}_L) having some value, but none of the fuel channels come under this volume.

5. Conclusion

Numerical analysis of the fluid flow and heat transfer has been carried out for the calandria based reactor for different flow parameters, heat load conditions and geometric parameters to ensure safe working. The systematic parametric studies made indicate that the heat transfer is largely influenced by the injection angle. The limiting thermal load evaluation is possible to interpret and for each configuration thereby letting a design optimization for the reactor based on the injection angle, as is demonstrated in the temperature plots for different proven levels. The results reveal a good insight to the complete behavior of the calandria based reactors and is considerable of high significance to the preliminary reactor design. It can be brought out that the detail of the CFD study provide initial information of the flow and heat transfer that is not easy in experimental studies or they could be very expensive and also hazardous. The effect of inlet angle on temperature difference is least for angles greater than 40° and lower mass flow rates resulted in higher maximum temperature and temperature difference;

- For all considered thermal power and injection angles the value of fuel channel temperatures are less between injection angles 30° to 60° ;
- Splitter orientation considerable affects the streamline and temperature distribution within the calandria. Splitter orientation close to 100° with respect to the inlet is found to be optimum.
- In the present analysis, it is found that the mass flux ration required to keep fuel channels within the safe limit is around 50% and it shows that splitter is contributing in an equal amount as that of inlet angle.

Acknowledgment

I thank profoundly Suneel Patil for carrying out numerical simulation and helping me to prepare this paper.

References

- [1] Koroyannakis D, Hepworth R D and Hendrie G 1983 Experimental study of combined natural and forced convection flow in a cylindrical tank *Tech. rep. AECL Report*
- [2] Ravi S D, Rajan N K S and Kulkarni P S 2008 Computational and experimental studies of fluid flow and heat transfer in a calandria based reactor *Computational Fluid Dynamics*, Springer p 233
- [3] Huget R G, Szymanski J K and Midvidy W I 1989 Status of physical and numerical modelling of CANDU moderator circulation *Proceedings of 10th Annual Conference of the Canadian Nuclear Society*, Ottawa
- [4] Huget R G, Szymanski J K and Midvidy W I 1990 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 p 327
- [5] Rhee B W, Yoon C and Min B J 2004 *Journal of the Korean Nuclear Safety* **36** 559–570
- [6] Kim M, Yu S O and Kim H J 2006 *Nuclear Engineering and Design* **236** 1155–1164
- [7] Bajaj S and Gore A 2006 *Nuclear Engineering and Design* **236** 701–722