TURBULENT MIXING OF HOT UPWARD FLOW AND COLD DOWNWARD FLOW INSIDE THE CHIMNEY OF A POOL TYPE RESEARCH REACTOR

Samiran Sengupta[1], P. K. Vijayan[2], K. Sasidharan[1], V. K. Raina[3]
[1]Research Reactor Design & Projects Division, Bhabha Atomic Research Centre, Mumbai, INDIA-400085
[2]Reactor Engineering Division, Bhabha Atomic Research Centre, Mumbai, INDIA-400085
[3]Reactor Group, Bhabha Atomic Research Centre, Mumbai, INDIA-400085
E-mail of corresponding author: samiran_sengupta@yahoo.co.in

ABSTRACT

Numerical simulation was carried out to study the turbulent mixing behaviour of hot upward flow and cold downward flow inside the chimney of a pool type research reactor. Open pool type research reactors often use this type of chimney structure to prevent mixing of core outlet water directly into the pool in order to keep the radioactivity level at the pool top to a lower limit. The chimney design facilitates guiding of the radioactive water from the reactor core towards the side outlet nozzles and simultaneously allows sucking of water from the reactor pool through the chimney top opening. Thus the cold water from the reactor pool mixes with the hot water from the reactor core inside the chimney. The mixing flow behaviour depends on the flow ratio and temperature difference between these hot and cold streams. The cold downward flow from the pool is compensated by providing a core bypass flow to the pool. CFD simulations are carried out using commercial software PHOENICS for various core bypass flow (0%, 5%, 10% and 15%). Velocity & temperature distribution inside the chimney and stagnation depth (the depth at which upward flow velocity of the core outlet water becomes zero with respect to top opening edge of the chimney) are predicted. The effect of variation of core bypass flow on suppressing the radioactive coolant entering into the pool is analysed. The effect of temperature difference on the stagnation depth is also computed. The details of CFD simulation such as nodalisation, boundary conditions adopted, turbulence modeling, solution procedure and the results are presented. The results indicate that the increase in bypass flow, increases the stagnation depth and try to keep the radioactivity well within the chimney region. On the contrary, the temperature difference between the hot upward fluid and cold downward fluid tries to reduce the stagnation depth. It is observed that if sufficient bypass flow is provided, no water from the core will reach the reactor pool through the chimney top opening.

INTRODUCTION

One of the simplest types of nuclear research reactor is pool type research reactor. The present work aims at studying the mixing behaviour of upward flowing hot fluid and downward flowing cold fluid inside the chimney of a pool type research reactor. The reactor core is cooled by upward forced flow of water through the core. After passing through the core, the coolant becomes radioactive due to formation of various radio-nuclides with neutrons present in the core. Because of the higher flow velocity, these radio-nuclides have a tendency to reach towards the pool top due to its inertia and thereby, radiation level at the pool top increases. Since, pool top radiation level should be limited during normal operation; these reactors often use an open chimney structure at the reactor core outlet in order to prevent radioactive coolant reaching the pool top. A typical example is High Flux Research Reactor (HFRR) being developed at BARC [1]. A simplified flow diagram of the primary coolant system of the HFRR is shown in Fig. 1. The core outlet water being radioactive is passed through delay tanks to decay down the activity level mainly caused by N^{16} radio-nuclides and subsequently it is sent through the heat exchangers where heat is transferred to the secondary coolant. Cold primary coolant water from heat exchanger outlet is fed back to inlet plenum at the bottom of the reactor core inside the pool. Therefore, under normal operating condition, the bottom of the chimney sees a strong upward flow from the core outlet, while at the open top a downward flow only is allowed. Both upward flow from the bottom and downward flow from the top mixes together just before the side nozzles of the chimney and is sucked out of the chimney with the help of pumps. The downward flow into the chimney from the pool is compensated by providing a core bypass flow discharged to the pool at the core inlet. A delicate balance between the core flow, the bypass flow, nozzle opening area & its inclination and height of the chimney are required to prevent the radioactive hot coolant mixing with the bulk pool water. Turbulent mixing behaviour inside this chimney being complex in nature needs to be understood considering the effects of these parameters. Accordingly, a

1/6th scaled test facility [2] has been designed to simulate the mixing behaviour inside the chimney. A schematic diagram indicating the mixing phenomena taking place inside the chimney is shown in Fig. 2.

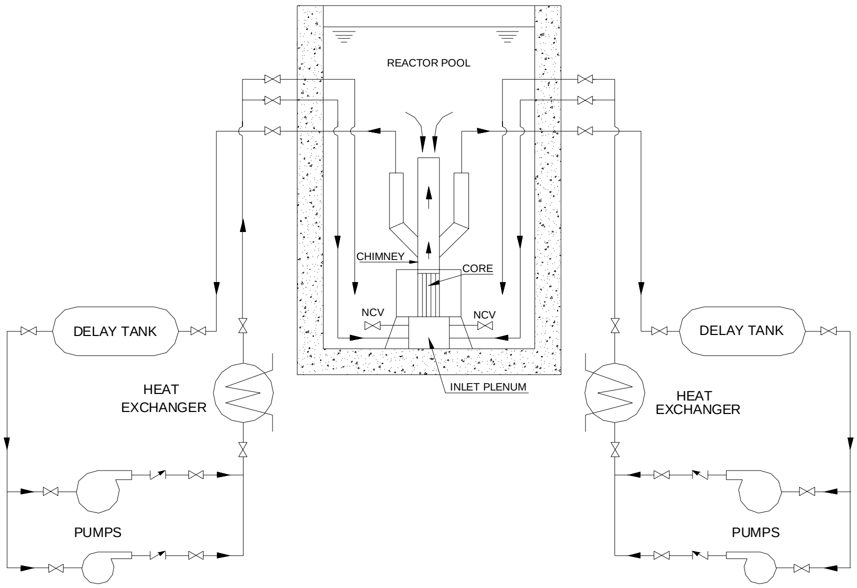


Fig. 1: Simplified flow diagram of primary coolant system of HFRR

This paper describes the CFD simulation carried out for the full scale chimney structure. The chimney height considered here is 3000 mm and diameter is 600 mm. The inclination angle considered here is 45°. Side outlet nozzle dimension is 600 mm × 300 mm. The temperature of hot upward fluid from core outlet is 49°C. The temperature of the core bypass flow is 40°C. The effect of buoyancy on the mixing behaviour is analysed for various core bypass flows by comparing results with a similar case considering equal fluid temperature (at 40°C) for both the upward flow and the downward flow. The objective of these simulations is to study the effect core bypass flow on the stagnation depth to understand the relative importance of inertia force, viscous force and buoyancy force. Accordingly various dimensionless numbers are defined as described in the following section.

DIMENSIONLESS NUMBERS

The dimensionless numbers describing the turbulent mixed convection inside the chimney are defined considering the relevant parameters shown in Fig. 3. The total core bypass flow is W_b, which is distributed through two inlets into the pool in the simulation. The core outlet flow velocity is U_{in} and fluid temperature is T_{in}. Pool water temperature is T_∞. The core bypass flow inlet temperature is also T_∞..

The Reynolds number (Re) shows the significance of the inertia force to viscous force. Richardson number (Ri) shows the importance of natural convection relative to the forced convection. In addition, two more dimensionless numbers (Re* and Ri*) is considered to take into account the ratio between core flow and bypass flow as defined below. Re* signifies the affect downward inertia force in suppressing the upward flow jet. Ri* signifies the more pronouncing effect of buoyancy force against the downward forced flow. The dimensionless numbers occurring for the simulation carried out in the present work is shown in Table 1. In addition to these simulations, four more analyses were carried out considering T_{in} and T_∞ as equal to understand the effect of buoyancy.

Table 1 : Computational data matrix

Sr. no.	Bypass flow (%)	Re	Gr	Ri	Re*	Ri*
1	0 %	1.44×10^7	3.34×10^{12}	0.016	-	-
2	5 %	1.44×10^7	3.34×10^{12}	0.016	7.18×10^5	6.475
3	10 %	1.44×10^7	3.34×10^{12}	0.016	1.44×10^6	1.619
4	15 %	1.44×10^7	3.34×10^{12}	0.016	2.15×10^6	0.719

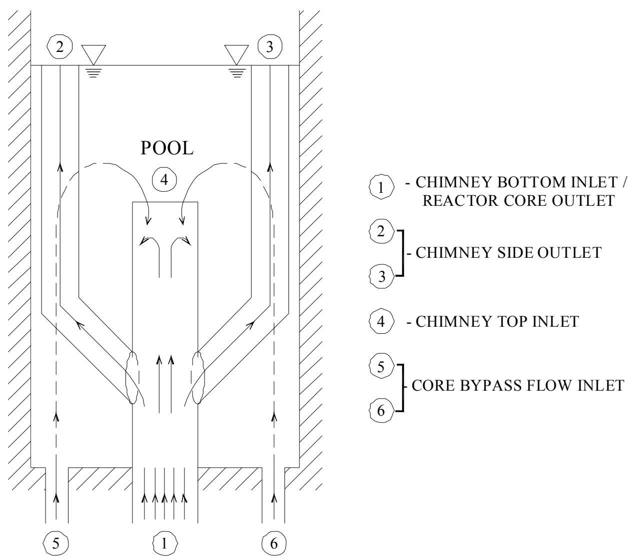


Fig. 2 : Schematic diagram of mixing of hot upward flow and cold downward flow inside the chimney

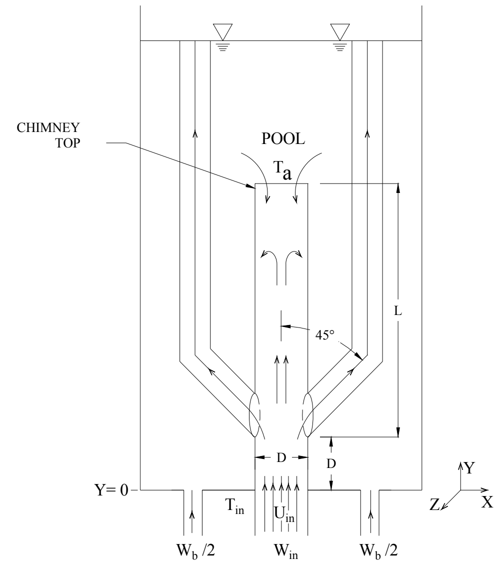


Fig. 3 : Parameters considered for the CFD simulation

MODELLING

The turbulent mixing of the flow of hot upward fluid and cold downward fluid inside the chimney is modelled using control volume based computational fluid dynamics software PHOENICS version 3.6 [3]. In this code, the solution domain is subdivided into a number of control volumes, each associated with a grid point, where the scalar variables such as pressure, temperature, concentration etc. are stored. The control volumes for the velocity are staggered in relation to the control volumes for the scalar variables. The convection terms in the momentum and energy equations are discretised using the hybrid scheme. The discretised equations are solved in a semi-coupled manner by a variant of the well-known SIMPLE algorithm. The solution is considered converged when the following criterion has been met for all dependent variables between sweep n and n+1. Reference value is represented as ϕ_r. Whole field residuals are also checked to ensure that the converged solution satisfies the governing equations within a prescribed error.

$$\max\left|\frac{\phi^{n+1}-\phi^{n}}{\phi_r}\right| \le 10^{-3} \tag{1}$$

Turbulence Modelling

Various turbulence models are developed to determine the eddy viscosity (ν_t) of which the standard k-ε model has been widely used due to its robustness, economy and reasonable accuracy for various industrial flow problems. The standard k- ε model was proposed by Harlow and Nakayama [4]. PHOENICS provides the standard high Reynolds number form of k- ε model as presented by Launder and Spalding [5]. It is a semi-empirical model and the derivation of the two additional transport equations for the turbulent kinetic energy (k) and the turbulence dissipation rate (ε) relies on phenomenological considerations and empiricism. The constants used in the standard form k-ε model are shown in Table-2. This standard k-ε model is applicable in the regions where the turbulence Reynolds number is high. Near walls, where the Reynolds number tends to zero, the model requires the application of so-called 'wall functions'. The following equilibrium log-law wall functions are used in the simulation.

$$\frac{U_r}{U_\tau} = \frac{1}{\kappa}\ln\left(Ey^{+}\right) \tag{2}$$

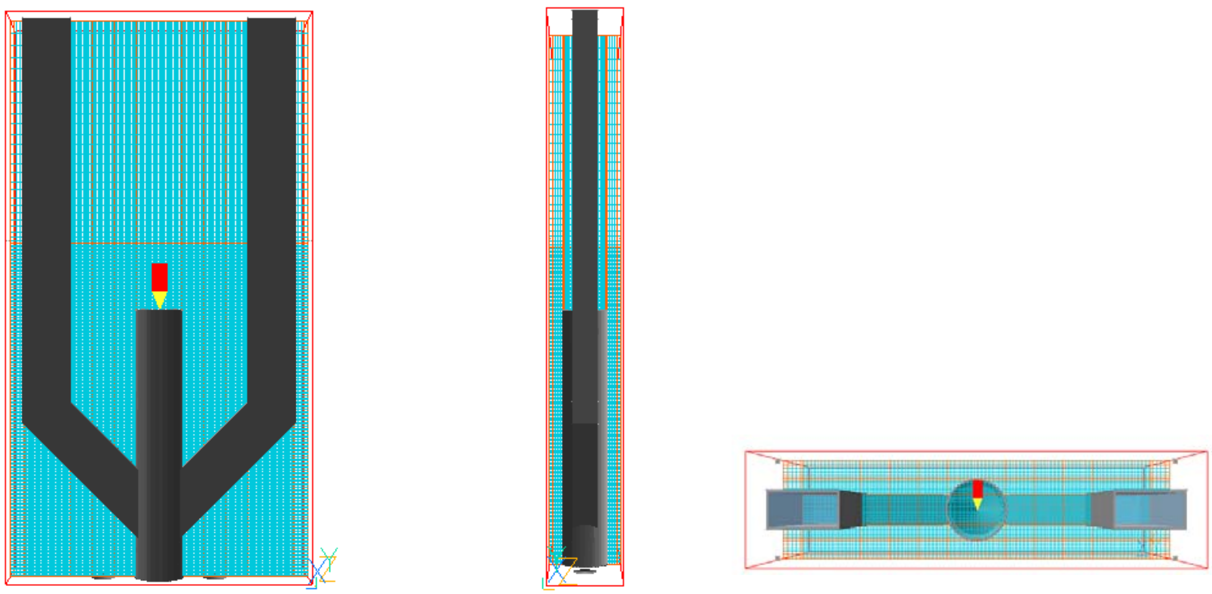
Where, κ is von Karman's constant and E is an integration constant that depends on the roughness of the wall. The values of κ and E considered here is 0.41 and 8.6 respectively.

Table 2: Values of constants in the k-ε model

c_{μ}	$c_{1\varepsilon}$	$c_{2\varepsilon}$	σ_k	σ_{ε}
0.09	1.44	1.92	1.0	1.3

Numerical Discretisation

The computational domain considered for the simulation includes the pool as well as the chimney. The mesh used for the computation domain (4 m × 7.5 m × 1.2 m) is shown in Fig. 4. Calculation is done with (190 × 200 × 40) grid points. The chimney geometry simulated is similar in dimensions as that of the full scale size. Free surface modeling at the interface between pool water and air at the top is not modelled considering larger pool water depth above the chimney top.



a) Mesh in x-y plane b) Mesh in y-z plane c) Mesh in z-x plane

Fig. 4 Mesh used for computation domain

A uniform velocity profile has been set at the inlet. As a boundary condition, values of k and ε are specified at the inlet. The k and ε is computed using the following relation.

$$k = (IU_{\alpha})^2 \quad \text{and} \quad \varepsilon = 0.1643\frac{k^{3/2}}{l_m} \tag{3}$$

Here, I is the turbulent intensity, U_{α} is the bulk inlet velocity and l_m is the mixing length. As a general practice, mixing length is of the order of 10% of the characteristic inlet dimension (i.e., hydraulic radius for the inlet pipe). In all these simulations, I is assumed to be 5%.

RESULTS AND DISCUSSION

The results of the CFD simulation with 45000 lpm (2700 m^3/hr) core flow considering 45° chimney inclination angle is presented here. The centre line upward flow velocity for various core bypass flow is shown in Fig. 5. The stagnation depth is defined as the location where upward flow velocity crosses zero line from the top edge of the chimney. The stagnation depth for 0%, 5%, 10% and 15% core bypass flow is found to be 1.65 m, 1.86 m, 1.96 m and 2.02 m respectively from the top of the chimney. When no bypass flow (i.e., 0%) is sent into the pool, the minimum stagnation depth occurs. As bypass flow increases, the stagnation point shifts towards the bottom, thereby causes increase in stagnation depth. However, effectiveness of core bypass flow on increasing stagnation depth (i.e. suppression of upward jet) gradually diminishes as bypass flow is increased from 0 to 5%, from 5 to10% and from 10 to 15%.

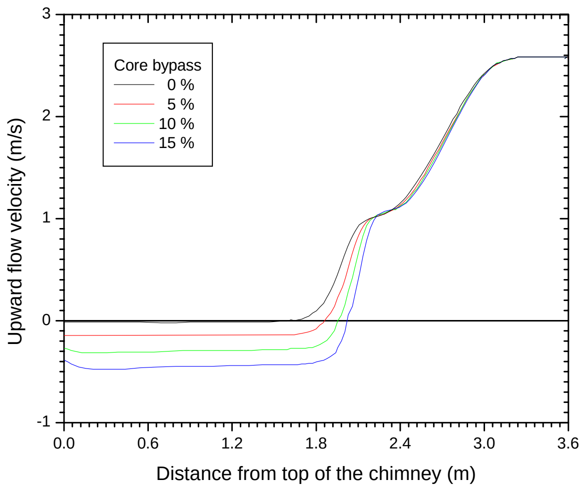


Fig. 5 : Variation of centre line upward flow velocity along the height of the chimney (ΔT=9°C)

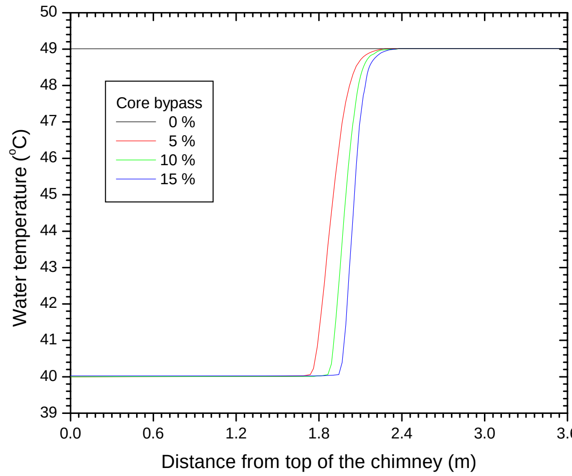


Fig. 6 : Variation of centre line water temperature along the height of the chimney (ΔT=9°C)

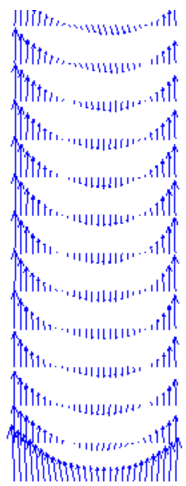


Fig. 7 : Velocity vector plot on x-y plane in chimney top region (core bypass flow=0%, ΔT=9°C)

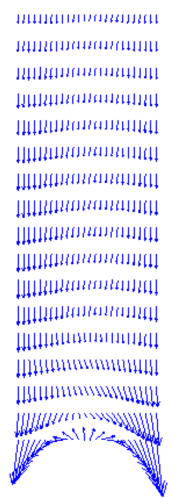


Fig. 8 : Velocity vector plot on y-z plane in chimney top region (core bypass flow=0%, ΔT=9°C)

Variation of water temperature at centre line of the chimney structure along the height is shown in Fig. 6. When core bypass flow is 0% (i.e. no bypass flow), water temperature is found to be equal to the core outlet temperature (49°C). The stabilised water temperature establishes at the core outlet temperature because hot water form the core outlet reaches the chimney top opening through the peripheral region. The velocity vector plot (in x-y plane) inside the chimney clearly shows upward motion of the fluid in the peripheral region (Fig. 7). Since no net flow is sent through chimney top, clear downward motion is observed in the y-z plane (Fig. 8). On the contrary when some net flow is provided by means of core bypass flow into the pool through two nozzles at 40°C, the pool water is sucked inside the chimney. Thereby, mixing of cold downward flow from the chimney top and hot upward flow from the core outlet takes place. As core bypass flow % increases, more amount of cold fluid is available from the chimney top and the pool water temperature front (i.e., 40°C) gradually reaches deeper inside the chimney. For 5%, 10% and 15% bypass flow the temperature front of pool water is observed at depth of 1.68 m, 1.78 m and 1.84 m respectively from the top of the chimney (Fig. 6). The temperature contour plots for 5% and 10% core bypass flows in x-y plane and y-z plane are shown in Fig. 9 to Fig. 12. The temperature of the probe point (here the centre point at the top of the chimney) is also indicated in the figures.

The velocity vector plots for 5% and 10% bypass flow are shown in Fig. 13 to Fig. 16. The turbulent mixing behaviour of hot and cold fluid for these cases is different from that observed for the 0% bypass flow case.

The velocity vectors at the top part of the chimney show a complete downward flow now. Therefore, no flow moves out of the chimney region through the top opening. Figure 14 shows the circulation region due to mixing of upward flow and downward flow. The jet behaviour of upward flow is clearly observed in Fig. 14 and Fig. 16. It is also observed that the height of the recirculation region reduces as core bypass flow increases due to increase downward flow velocity in the upper part of the chimney.

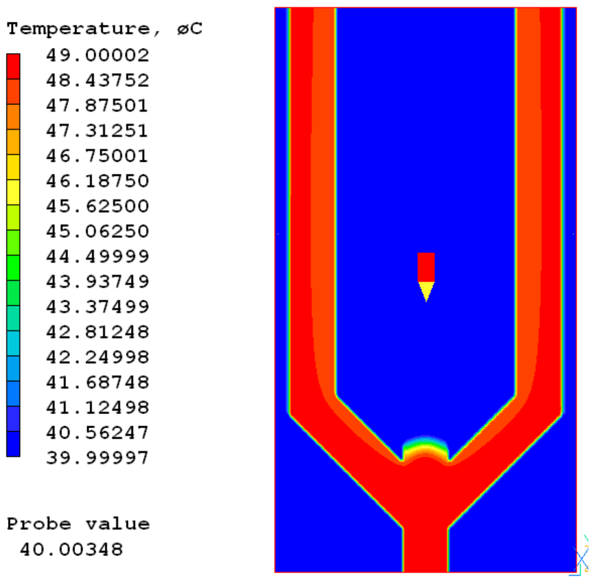


Fig. 9 : Temperature contour plot on x-y plane (core bypass flow=5%, ΔT=9°C)

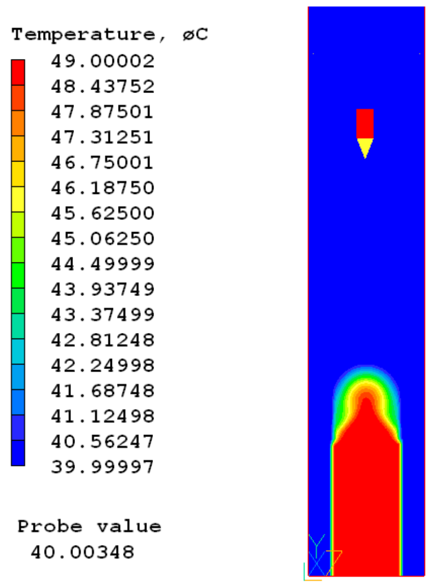


Fig. 10 : Temperature contour plot on y-z plane zoom view (core bypass flow=5%, ΔT=9°C)

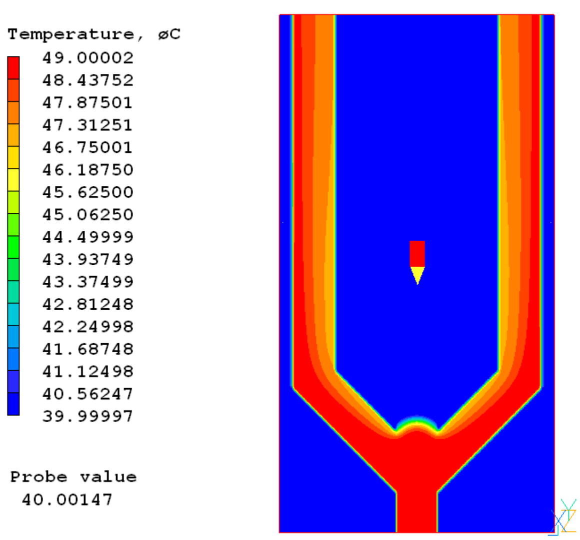


Fig. 11 : Temperature contour plot on x-y plane (core bypass flow=10%, ΔT=9°C)

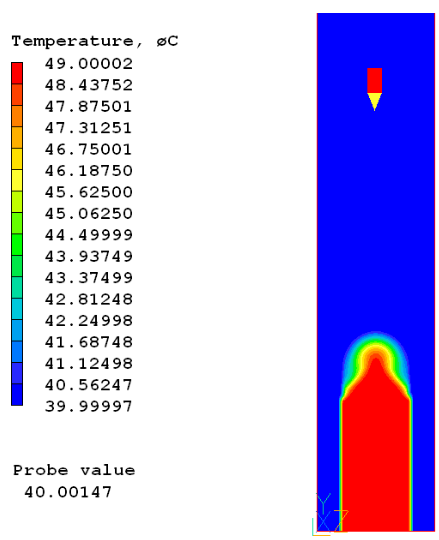


Fig. 12 : Temperature contour plot on y-z plane zoom view (core bypass flow=10%, ΔT=9°C)

To understand the effect of temperature difference on the stagnation depth, computations are done considering no reactor power. Core outlet water temperature and pool water temperature is assumed to be equal. The results for these cases are shown in Fig. 17 and Fig. 18. It is observed from Fig. 17 that the stagnation depth is about 0.1 m to 0.2 m more than that observed for the cases described in Fig. 5. Velocity vector plot inside the chimney for 0% core bypass flow is shown Fig. 18. Similar results showing upward flow through the peripheral region of chimney is also observed here but the magnitude is less than the earlier case.

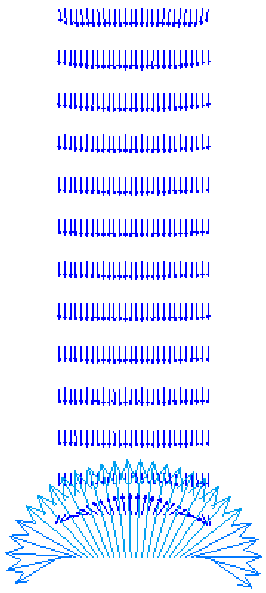


Fig. 13 : Velocity vector plot on x-y plane in chimney top region (core bypass flow=5%, ΔT=9°C)

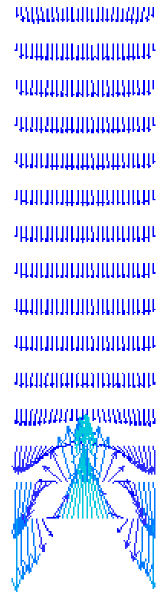


Fig. 14 : Velocity vector plot on y-z plane in chimney top region (core bypass flow=5%, ΔT=9°C)

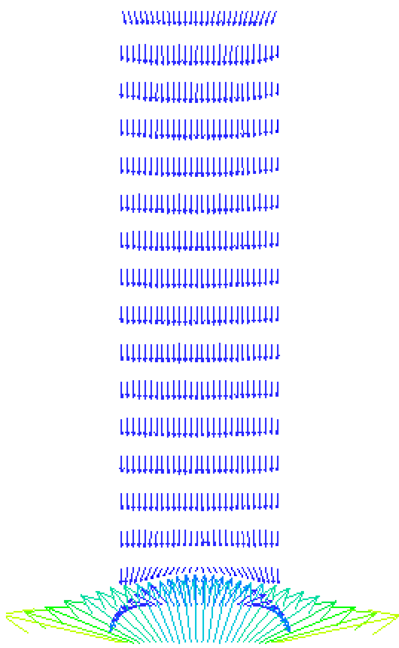


Fig. 15 : Velocity vector plot on x-y plane in chimney top region (core bypass flow=10%, ΔT=9°C)

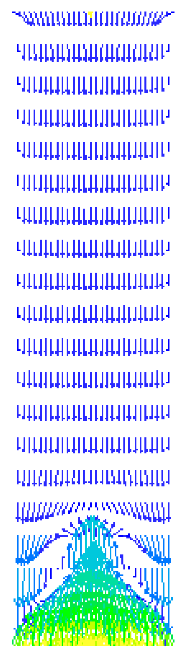


Fig. 16 : Velocity vector plot on y-z plane in chimney top region (core bypass flow=10%, ΔT=9°C)

CONCLUSIONS

The turbulent mixing behaviour of hot upward fluid and cold downward fluid inside the chimney of a pool type research reactor is studied using PHOENICS code. The effect of core bypass flow and temperature difference on stagnation depth is computed. It is observed that provision of core bypass is a necessity to suppress the core outlet water coming out of the chimney.

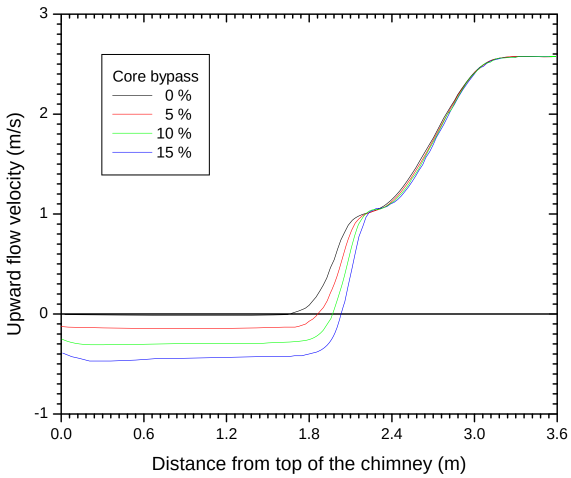


Fig. 17 : Variation of centre line upward flow velocity along the height of the chimney (when ΔT=0°C)

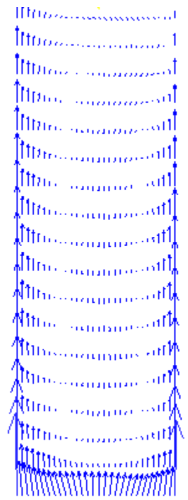


Fig. 18 : Velocity vector plot on x-y plane in chimney top region (core bypass flow=0%)

REFERENCES

[1] Chafle, S. B., Sengupta, S., Mukherjee, P., Sasidharan, K., Raina, V. K., "High Flux Research Reactor", *Peaceful Uses of Atomic Energy-2009,* New Delhi, India, October, 2009.

[2] Sengupta, S., Vijayan, P. K., Chafle, S. B., Sasidharan, K., "Design of a scaled test facility to simulate turbulent mixing inside the chimney of a pool type research reactor", *37th National & 4th International Conference on Fluid Mechanics and Fluid Power,* Chennai, India, FMFP2010-292, December, 2010.

[3] Ludwig, J. C., *Concentration, Heat and Momentum Limited* (CHAM), POLIS: Phoenics On-line Information System, London, 2004.

[4] Harlow, F. H., Nakyama, P. I., "Transport of turbulent energy decay rate", LA-3854, Los Alamos Science Lab, U. California, USA, 1968.

[5] Launder, B. E., Spalding, D. B., "The numerical computation of turbulent flows", *Comp. Math .in Appl. Mech. & Eng.*, Vol 3, pp269, 1974.

NOMENCLATURE

Gr Grashof number, $\dfrac{g\beta(T_{in}-T_{\alpha})L^3}{\upsilon^2}$

g acceleration due to gravity (m/s^2)
k turbulent kinetic energy (J)
L length scale of chimney (m)

Re Reynolds number, $\dfrac{\rho U_{in} L}{\mu}$

Re* = $\dfrac{\rho U_{in} L}{\mu}\left(\dfrac{W_b}{W_{in}}\right)$

Ri Richardson number, $\dfrac{g\beta(T_{in}-T_{\alpha})L}{U_{in}^2}$

Ri* = $\dfrac{g\beta(T_{in}-T_{\alpha})L}{U_{in}^2}\left(\dfrac{W_{in}}{W_b}\right)^2$

T temperature (K)

U_{α} reference velocity (m/s)

U_{τ} resultant friction velocity (m/s), $\sqrt{\dfrac{\tau_w}{\rho}}$

W mass flow (kg/s)

y^+ dimensionless wall distance, $\dfrac{U_{\tau} y}{\nu_l}$

Symbols
ρ fluid density (kg/m^3)
β volumetric expansion coefficient (/K)
ε turbulence energy dissipation rate (W)
μ viscosity (Ns/m^2)
υ kinematic viscosity (m^2/s)
υ_t turbulent kinematic viscosity (m^2/s)

Subscripts
α bulk fluid in pool
in inlet