THE 5TH ASIAN COMPUTATIONAL FLUID DYNAMICS
BUSAN, KOREA, OCTOBER 27 ~ 30, 2003

Numerical Analysis of Heat and Mass Transfer in a Calandria Based Reactor

Ravindra S Tupake*, PS Kulkarni[1] and NKS Rajan[2]
1. psk@aero.iisc.ernet.in, 2. nksr@cgpl.iisc.ernet.in
Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India

ABSTRACT

Numerical investigations are carried out to study the mass flux and temperature distribution in a calandria using a 3-D RANS code. The computations made for simulations of flow and convective heat transfer with near–to working conditions. The work provides an estimate of the safe working limits of the heat dissipation by virtue of prediction of the 'hot spots' in the calandria. The work assumes significance for preliminary designs of the reactors and for detailed critical parametric analysis that would be otherwise more expensive.

1. INTRODUCTION:

In the typical application, a matrix of tubes that contain nuclear fuel channels are stacked in a Pressurized Heavy Water Reactor (PHWR) vessel known as calandria as shown in Fig. 1. The moderator around the tubes receives the heat generated and is cooled externally. The flow and temperature distribution in the calandria is important in the safety analysis. The literature in this area is very sparse partly due to the classified nature of the studies. Investigations [1,2,3,4] have been carried out by considering downward pointing inlets [1] and upward pointing inlets [2] at different hand positions of a clock to study the temperature field. Although a few more studies [3,4,5] have been reported, the studies are not exhaustive enough to fully understand many of the controlling parameters and temperature distributions at the design stage and the experimental investigations are hazardous and cost–prohibitive. A three dimensional parametric study has been carried out using an industry-standard code CFX-TASCflow.

2. CONFIGURATION USED, GRID GENERATION AND THE APPROACH:

In one of the typical configurations, considered for the present study, the inlets and outlets are symmetrically located at 3 and 9 o'clock positions and 5 and 7 o'clock positions respectively. The fuel channels considered are arranged in a staggered fashion. Two multi–block structured grids have been optimally generated, one with and the other without fuel channels as shown in Figs. 2 and 3.

The Reynolds–Averaged Navier–Stokes Equations are solved for steady, single–phase, incompressible and viscous flow. Boussinesq approximation is used to model the effects of buoyancy. The effect of fuel channels is modelled by using block–off approach. For all the computations, fluid considered is water with constant fluid properties. The Reynolds Number (Re) in this study is referenced to the diameter of the calandria and the injection velocity. The boundary conditions include:

- No slip, impermeable and adiabatic walls – $u_i n_i|_b=0$, $\dot{m}_{ip}=0$ and $\left.\frac{\partial T}{\partial n}\right|_b=0$, $Q_{ip}=0$
- Constant and Uniform heat flux at the outer surface of the fuel channels in the case of hot flow = Q_{spec}
- The inlet jet velocity and temperature of the fluid entering the calandria is set as an assigned boundary value, $u_i|_{inlet-b} = u_{spec} n_i$ and $T_b = T_{spec} = 328\text{K}$
- The outlet boundary condition is to set with matching mass flux with the inlet, $\dot{m}_{out} = \dot{m}_{spec}$.

4. RESULTS AND DISCUSSION:

Computations for isothermal and non–isothermal cases are made and the significant flow structure and temperature distribution have been captured spatially. To cover a wider range of analysis,

---

* M E (Mech) Student, Dept. of Mechanical Engg., Govt. College of Engg., Pune, India.

cases for different Re for isothermal flow with and without the fuel channels are studied (Figs. 4, 5 & 6). Expectably, with increasing Re, the jets penetrate deeper and the recirculation pattern gets stronger, as also could be seen from the figures. The large velocity gradients, lead to complex flow pattern. The set of grids providing stable solution was extended for isothermal flow with fuel channels (Fig. 3). The fuel channels is found to be diffusing the strong circulation zones. The weaker recirculation zones are shifted towards the wall (Figs. 4 & 5). The flow patterns (Figs. 6 & 7) in the in the non–isothermal and isothermal cases being closer, indicate that the flow is controlled by the momentum than the buoyancy. This is also supported with the temperature distribution remaining nearly unchanged at different levels (Figs. 8 & 9). The hot–spots are located in the recirculating zones, attributable to low fluid flux, leading to lower heat dissipation (Fig. 10). With the capturing of all the salient features of the flow and the heat transfer, the work provides a good CFD tool for the design and safety considerations of the reactor that prove to be hazardous and expensive otherwise.

REFERENCES:

1. Carlucci, L.N., Agranat, V., Waddington, G.M., Khartabil, H.F., and Zhang, J., "Predicted and Measured Flow and Temperature Distribution in a Facility for Simulating In-reactor Moderator Circulation", 8th Annual conference of the CFD Society of Canada, Montreal, June 2000.

2. Marathe, A.G., Agrawal Lokesh, Mandal J. C., "Numerical Computations of Moderator Flows Inside Calandria of a Pressurized Heavy Water Reactor", Proceedings of ASME FEDSM'01, ASME 2001 Fluids Engineering Division Summer Meeting, New Orleans, Louisiana, May 29 - June 1, 2001.

3. Dharne, S.P., and Gaitonde, U.N., "Analysis of the Moderator Flow and Temperature Distribution Inside the Calandria of Indian PHWRs", 3rd ISHMT_ASME Heat and Mass Transfer Conference, December 1997.

4. Collins, W.M., Garceau, M., Harwary, M.El., Application and Validation of the PHOENICS Code for Moderator System Conditions in a CANDU Nuclear Generating Station", 3rd Int. Conf. On Simulation Methods in Nuclear Engineering, Montreal, 1990.

5. Hassan, Fath, S., and Hussain. M.A., "Moderator Circulation in CANDU Reactors: An Alternative Approach for the Tube Matrix Simulation", Nuclear Technology, Vol.88, pp 307-318, 1989.

6. Kulkarni, P.S., "3D Euler codes based on kinetic upwind schemes and their verification & Validation", Proceedings, ICCFD2, July 15-19, 2002, Sydney.

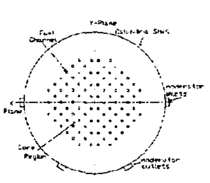

Fig. 1, The typical arrangement of the calandria

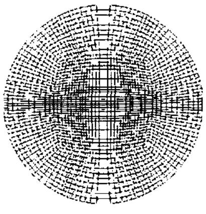

Fig. 2, Grids for calandria without fuel channels

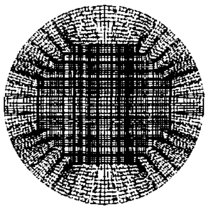

Fig. 3, Grids for calandria embedded with fuel channels

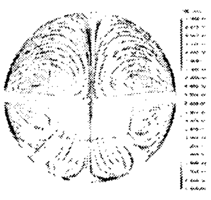

Fig. 4, Isothermal flow pattern for Re 1000, without fuel channels

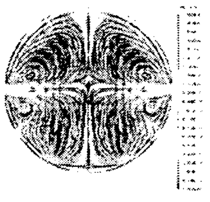

Fig. 5, Isothermal flow pattern for Re 1000, with fuel channels

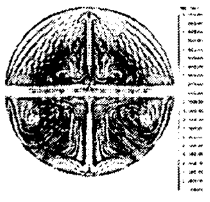

Fig. 6, Isothermal flow pattern for Re 23000, with fuel channels

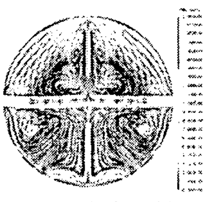

Fig. 7, Non-isothermal flow pattern for Re 23000, with fuel channels

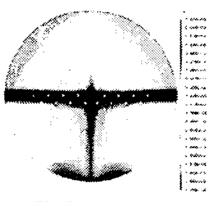

Fig. 8, Temperature distribution at Re 23000 with 0.14 MW thermal dissipation

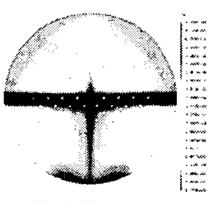

Fig. 9, Temperature distribution at Re 23000 with 0.45 MW thermal dissipation

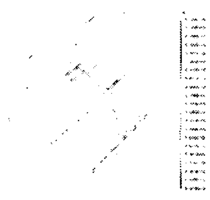

Fig. 10, Hot-spot mapping at Re 23000 with 0.45 MW thermal dissipation