Effect of Inlet Jet Injection Angle on a Calandria Based Reactor – An Investigation with Numerical Analysis of Heat & Mass Transfer for an Optimum Reactor Design

NKS Rajan, PS Kulkarni* and MulaVenkataRamana Reddy
Corresponding author: psk@aero.iisc.ernet.in
Department of Aerospace Engineering, Indian Institute of Science, Bangalore-INDIA 560012

Abstract: Heat and mass transfer studies in a calandria based reactor is quite complex both due to geometry and due to the complex mixing flow. It is challenging to devise optimum operating conditions with efficient but safe working range for such a complex configuration. Numerical study known to be very effective is taken up for investigation. In the present study a 3D RANS code with turbulence model has been used to compute the flow fields and to get the heat transfer characteristics to understand certain design parameters of engineering importance. The angle of injection and of the coolant liquid has a large effect on the heat transfer within the reactor.

Keywords: 3D RANS equations, Turbulence Modeling, Calandria, Heat and Mass transfer.

1. Introduction:

The present study is to analyze the fluid flow and temperature distribution and to establish an adequate theoretical basis for the reactor model under consideration and thereafter to investigate the optimum design parameters of a calandria based reactor using the advanced CFD tools. In an earlier numerical investigation extensive study of fluid flow has been carried out and the findings have been validated with the results obtained from the laboratory experimental model[1]. Further to this validation, in the present study, numerical experiments have been carried out to investigate the optimum design parameters that could affect the performances with the constraint of safe working limits of the practical reactor configuration. During initial study, it was identified that there are two possible flows inside the calandria vessel influencing the convection, buoyant flow [2, 3] formed by internal heating and momentum driven convective flow created by the jet from the inlet nozzles. It is well established that enhanced heat transfer is achieved when forced convection is dominating indicating that the coolant injection should be adequately large. However the higher jet velocities that could act as 'hammers' crossing the safe pressure limits on the fuel channels set the working limits of the jet velocities. In the present study, nominally strong jet, chosen within such working limit but delivers far higher forced convective fluxes than the natural connective fluxes. This study has identified another parameter as the jet inlet angle and effect of this parameter is studied in this investigation.

2. Problem Statement:

A section of the reactor that encompasses a single inlet jet that is a repetitive structurally over the length of the reactor (Fig.1a &b) is considered as the basic control element of study for the computational domain. The fuel channels that are embedded in the reactor and submerged in the coolant fluid provide a disturbance to the free flow of the jet offering enhanced mixing and thereby enhance the heat transfer. The fluid is considered as incompressible and is in single phase. The flow is simulation uses the standard N-S equations for mass, momentum and energy and the turbulence is simulated with the k-omega model known to have given the most acceptable results in the validation made in the earlier investigation [1]. A no-slip and adiabatic wall conditions are imposed at the wall of the calandria vessel. On the walls of the fuel channels, no-slip associated with a present heat positive flux (enabling heat flow from the fuel channel to the surrounding coolant fluid) is imposed. The numerical code uses an industry standard CFD code solving 3D RANS equations with a second order upwind scheme. The solution sought is for steady state and a diminishing residuals falling by decades is set as the criteria for convergence. For a clarity the main equations considered for the solution is provided below.

The Continuity Equation:
$$\frac{\partial \bar{U}_j}{\partial x_j} = 0$$

The Momentum Equation:
$$\frac{\partial}{\partial t}(\rho \bar{U}_i) + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \rho \overline{u_i'' u_j''})$$

The Energy Equation:
$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_i'' h'})$$

3. Results and Discussion:

The solutions are obtained for different parametric cases. As described in the previous sectionthe parameter studied is the variation in the angle of the inlet jet. The angular shifts of 15° is provided, starting from one extreme position that makes the inlet jet radially injecting the coolant while the other limiting being 14° off from being tangential to the calandria vessel. The plot shown in Fig. 2 gives a temperature distribution over a horizontal line taken over center of the

calandria for a same injection rate of the coolant but with different angles of injection. The inlet temperature of the coolant is kept constant for all the cases. Oscillatory behavior is clearly due to the increased heat flux that occurs closer to the fuel channels. The differential temperature at each of the crests indicates the heat transfer occurring spatially and could be easily interpreted due to the intensity of convective flux of the coolant at that point. One can see that this differential temperature is shown to have tendency of being larger at the extreme angles and is lowest near the 45° of the jet inlet injection angle. It can also clearly made out that the differential temperature is lowest in the case of jet angle 45°. In Fig. 3, this aspect is brought out more clearly. From the plots in Fig.3 the normalized peak temperature in the entire domain of computation as a function of injection angle, shown at different assigned conditions of thermal dissipation, bring out that at the jet injection angle around 45° the lowest peak temperature is noticed indicating the most effective heat transfer between the fuel channels and the coolant fluid. In another parametric analysis, a study made with change of positions and number of outlets was made and it was found that the normalized peak temperature had little impact as could be seen from Fig. 4 and is explainable since outlets being the downstream elements of the flow have much less impact the flow structure in the calandria and hence on the heat transfer. The observations summarizes to point the strong optimization parameter turns out to be injection angle and the study suggests that the optimal angle for injection to be 40°-45° that could maximize the utilization of the reactor for its capacity.

4. Conclusions and Future work:

The paper provides the summary of an extensive numerical investigation that shows up an important design optimization parameter with an optimum angle injection of the coolant that enhances the capacity utilization of the reactor by a large factor with no perceptible increase in the cost. The work could be continued with different positioning of the inlet ports for further optimization studies.

References:

1. MulaVenkataRamana Reddy et al,. Numerical model for the analysis of the Thermal-Hydraulic behaviors in the Calandria based reactor. Proceedings of ICCFD6, St Petersburg, Russia, 669-676, 2010
2. Wenxian Lin, and S.W. Armfield. Natural convection cooling of rectangular and cylindrical containers. International Journal of Heat and Fluid Flow, 22: 72-81, 2001
3. E. Buyruk. Numerical study of Heat transfer characteristics on Tandem cylinders, inline and staggered tube banks in cross-flow of Air. Int. Comm. Heat Mass Transfer, Vol.29, No.3, 355-366, 2002
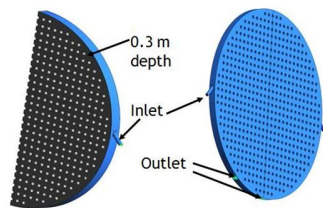
Fig. 1(a) Section of the calandria reactor selected for computational domain
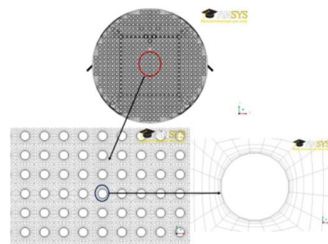

Fig. 1(b). Samples of Mesh generated, along with blown-up views near the fuel channel
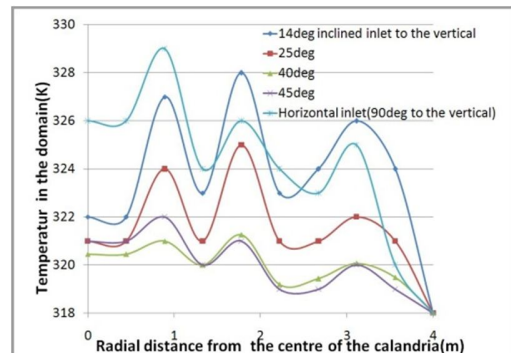

Fig. 2. Temperature distribution over horizontal line passing the center of the reactor
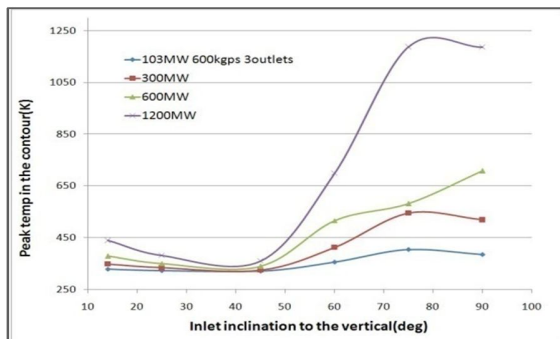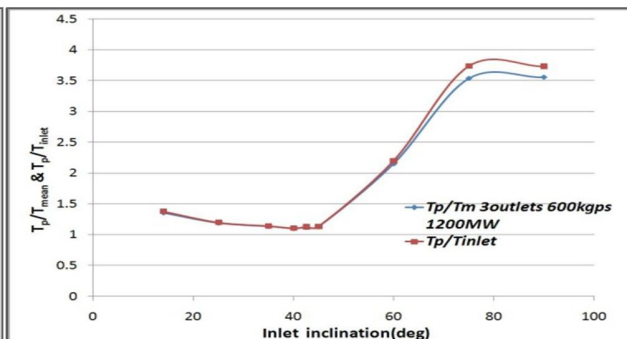

Fig.3 Inlet inclination Vs Peak Temperature for different heat loads

Fig.4 Inlet inclination Vs Non dimensionalised Temperature