International journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.2., Issue.4, 2014

RESEARCH ARTICLE

ISSN: 2321-7758

THERMAL ANALYSIS OF CANDU REACTOR MODERATOR USING 'CFD' TOOL

SAHEBAGOUDA S[1*], ABDUL RAZAK.R.K[2], MOHAMMAD SAMEE.A.D[2],

[1]P.G.Student, Department of Mechanical Engineering, P.A.College of Engineering, Mangalore, Karnataka, INDIA

[2]Asst Professor,Department of Mechanical Engineering, P.A.College of Engineering, Mangalore, Karnataka, INDIA

Article Received: 19/08/2014 | Article Revised on: 25/08/2014 | Article Accepted on:27/08/2014

SAHEBAGOUDA S

ABSTRACT

The prime objective of the present study is to perform the simulation of a CANDU Reactor using Ansys Fluent (CFD). In this paper, first focus is given to assessment of the moderator's thermal and hydraulic characteristic in normal operating conditions. The obtained results are compared with experimental results and it is observed that the maximum temperature inside the CANDU vessel is 40^{o}C. Major part of the study focus on fluid temperature inside the CANDU vessel. Graphs are plotted to predict the temperature and velocity distribution inside the Calandria tank. Thefluid flow inside the vessel is due to the interaction between the momentum force generated by the inlet jets and the buoyancy force of heat load to fluid. It has been observed that the obtained results are well agreed with the experimental results. Finally, it is concluded that majorparameters affecting the flow patterns are the inlet flow rate and heat load to the fluid.

Key words: CANDU reactor, Fluent, Turbulent flow, CFD

INTRODUCTION

Nuclear power reactor is a device to initiate and control a sustained nuclear chain reaction for the purpose of power generation. Nuclear power reactors are generally used for generation of electricity and as well for the propulsion of ships. The heat released from nuclear fission is passed on to a working fluid (water or gas). Steam or gas runs through turbines that power either generators or the propellers of ships. Some reactors are used for the production of isotopes for medical and industrial use, while others are used mainly for research purposes.

Environment plays an important role in the overall development of the world. Recognizing the importance of environmental protection and sustainable development, ever-rising demand for energy resulting from global population growth, increased awareness of the aftermath of global warming induced by the use of fossil fuels, sky-rocketing prices of fossil fuels andconcerns over security of energy supply, the present world demands for a huge supply of environment-friendly power generation for which nuclear power plants are a viable alternative. Nuclear power is a significant contributor to the global supply of electricity and continues to be the major source that can provide electricity on a large scale with a comparatively minimal adverse impact on the environment. The nuclear power plants are extolled as engines of growth and development of the world in regard to power generation. Gazy F. Al-Sumaily [1] numerically investigated the Time-dependent forced convection heat transfer from a single circular cylinder embedded in a horizontal packed bed of spherical particles under local thermal non-equilibrium condition using the spectral-element method. The effect of solid boundaries, inertia forces, and the thermal dispersion, are taken into account. The results presented here provide a better understanding of the influences of the presence of a porous medium of spherical particles, and its thermal properties, on the rates of heat transfer and the hydrodynamic and thermal behavior around a heated circular cylinder mounted in a horizontal channel. Manwoong Kim[2] Presented the thermal–hydraulic characteristics of a moderator flow subject to two forces, i.e., the momentum force by inlet jets and buoyancy force by the heat load inside Calandria vessel of CANDU-6, the analysis model has been established through the comparison with the experimental data and a series of the numerical simulation has been performed for the normal operating condition and the transient condition. It is found that the major parameters affecting the flow patterns are the inlet flow rate and heat load to the fluid. With inlet flow rate, heat load, or both, three flow patterns can be predicted, that is, momentum-dominated flow, mixed type flow and buoyancy-dominated flow. Sajida Lafta Ghashim Jassim [3]: In this work, a numerical study is performed to predict the solution of two – dimensional, steady and laminar mixed convection flow over a square cylinder placed symmetrically on a vertical parallel plate. A finite difference method is employed to solve the governing differential equations, continuity, momentum, and energy equation balances. The solution is obtained for stream function, vorticity and temperature as dependent variables by an iterative technique known as successive over relaxation. The maximum crowding of the isotherms is seen on the bottom face, indicating the highest Nusselt number, as compared to other surfaces of the cylinder, since the thermal boundary layer growth starts from this face.Eduardo Schnurr Siqueira [4]carried out a work to implement the power-law model in MFIX, validate the implementation and conduct a case study using the model implemented. With the implementation of a non-Newtonian model to the code, a new possibility for the simulation of multiphase flows of solid-non-Newtonian liquids is opened, as well as there is an increase in the capability of the code regarding the study of single-phase fluid flows of non-Newtonian fluids subject to heat transfer. During investigation it was found that the parameters like Nusselt Number and the Prandtl number are greatly influenced by blockage ratio and power law index.H. P. Rani [5] presented the numerical analysis of conjugate heat transfer and heat generation effects on the transient free convective boundary layer flow over a vertical slender hollow circular cylinder with the inner surface at a constant temperature. An unconditionally stable Crank-Nicolson type of implicit finite difference scheme is employed to obtain the discretized form of the governing equations. Which are further solved by tridiagonal algorithm Numerical results for the transient velocity and temperature profiles, average skin-friction coefficient and an average Nusselt number are shown graphically. In all these profiles it is observed that the time required inreaching the steady-state increases as the conjugate-conduction parameter or heatgenerated parameter increases. E. D. dos Santos[6]presented a work on two dimensional transient heat transfer under forced convection incompressible turbulent flow over a pair of cylinders having different arrangements. Simulation is carried out by using a finite volume method by considering the parameter like Reynolds number and the Prandtl number. From the results obtained it is concluded that transient fluid dynamics and thermal patterns are affected by the configuration of cylinders.

2. METHODOLOGY

To solve the present problem a CFD software is used which involves different steps to achieve the solution. The steps used are:

The steps to obtain a proper solution for the flow of the fluid in FLUENT are:

Pre Processing: consisting of construction of geometry, the generation of mesh on surface or volumes. The geometry is created in CATIA V5.Defining the boundary conditions and other parameters, before starting a

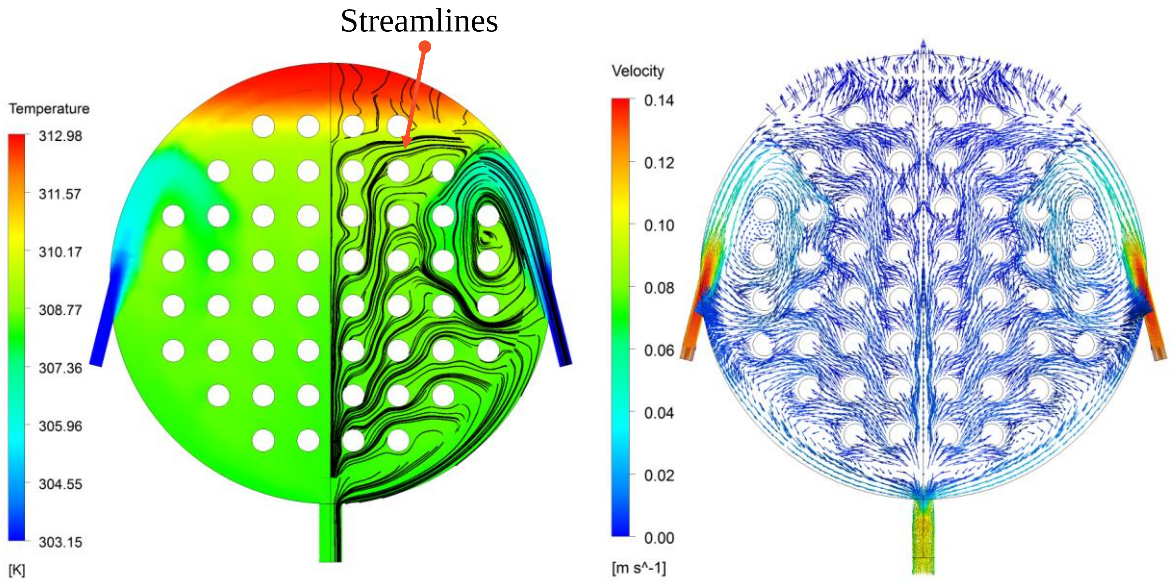
Vol.2., Issue.4, 2014

simulation in FLUENT, the mesh has to be checked and scaled and modified if necessary. The physical models have to be tackled. This includes the choice of compressibility, viscosity, heat transfer considerations, laminar or turbulent flow, steady or time dependent flow. The boundary conditions have to be clear because they specify the information of the state of the flow in the determined zones: walls, symmetries, inlet, air, outlet air etc. Resolution of the problem, which is done through iteration until the convergence of the variables is obtained. First of all, the variables of the flow have to be initialized and set to be computed from certain part specified by the user. At this stage the equations of the flow are solved. The convergence is checked until it reaches the criterion value set by the user.

Post Processing or analysis of the results computed. There are lots of choices: contours, X-Y plots, velocity vectors, path lines. In them, several variables can be analyzed: velocity, pressure, turbulence, forces, density, heat transfer coefficient and others.

3. RESULT AND DSCUSSION

Numerical calculations were carried out for the CANDU vessel via SPEL experiment results, and the present analysis model predicted the fluid temperature reasonably. The maximum temperature inside Calandria-like tank is 40^{o}C which is similar to the other two published results. It is found that three flow patterns are momentum-dominated flow. It is also noticed that the onset conditions of these flow patterns mainly depend on the heat load and inlet flow rate. In the condition of low heat load and high flow rate, the effect of momentum force on the main stream inside Calandria vessel is more dominant than that of buoyancy force.



Contours of Temperature Velocity Vectors

Fig3.1 Contours of CANDU Reactor

Fig. 3.1 shows the contour of temperature and velocity distributions of the Calandria vessel. Moderator with relatively high temperature is placed upper region of Calandria vessel, due to buoyancy force and its temperature decreases gradually along the vertical line. Moreover, the two temperature profiles along the symmetric vertical centerlines are asymmetric each other. Moderator velocity magnitude changes sharply near the wall of Calandria vessel, so the velocity gradient is concentrated on there. The velocity in the core region is relatively small due to the hydraulic resistance, i.e., fuel channels, as well as the force balance between buoyancy and the momentum forces.

3.2 Comparison of Results

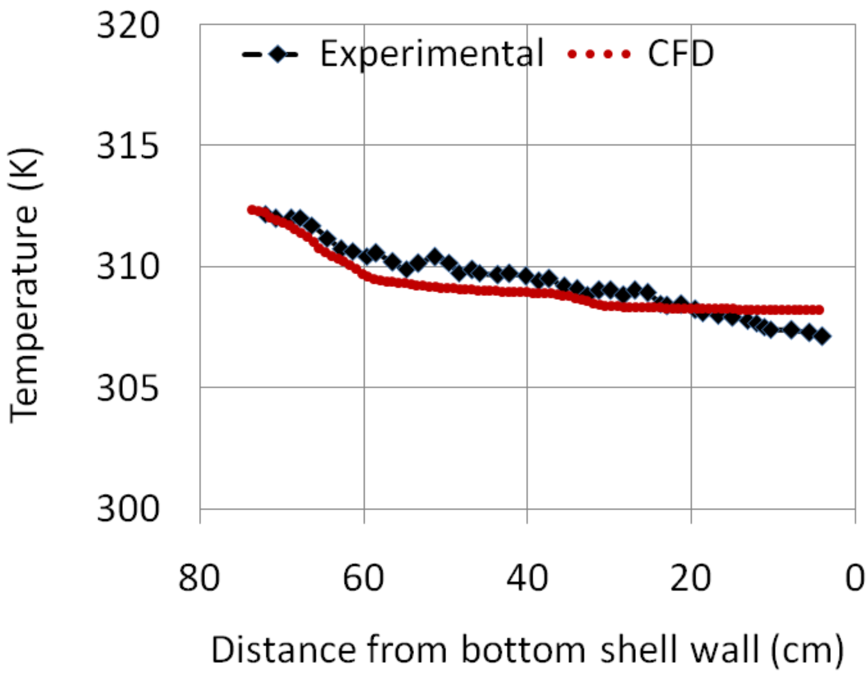


Fig.3.2 Code validation against published results

Fig.3.2 shows the comparisons of present analysis with Prabhakaran, Kim et al., (2006). It shows the maximum temperature inside Calandria-like tank is 40^{o}C, which is similar to the other two published results. It is observed from above fig that the temperature decreases sharply due to the momentum of inlet flow at the regions on the flow passage of fluid. However, the temperature increases slightly due to the heat generation, and decreases again at the bottom region due to the forced convection.

Fig.3.3 shows the comparisons of velocity vectors inlet plane of CANU reactor. The fluid flow inside the vessel is very complex due to the interaction between the momentum force generated by the inlet jets and the buoyancy force by the heat load to fluid. According to the computational results for SPEL geometry, it is found that three flow patterns are momentum-dominated flow. It is also noticed that the onset conditions of these flow patterns mainly depend on the heat load and inlet flow rate. In the condition of low heat load and high flow rate, the effect of momentum force on the main stream inside Calandria vessel is more dominant than that of buoyancy force. Although the entering fluid is heated up by heat generation, the fluid flows toward the outlet due to small amount of heat load and strong inlet jets. Two inlets flows encounter each other in the nearly center region, and discharge through the outlet. In this condition, the location of the high temperature of fluid is the near-bottom of Calandria and the momentum-dominated flow forms.

Typical flow pattern:

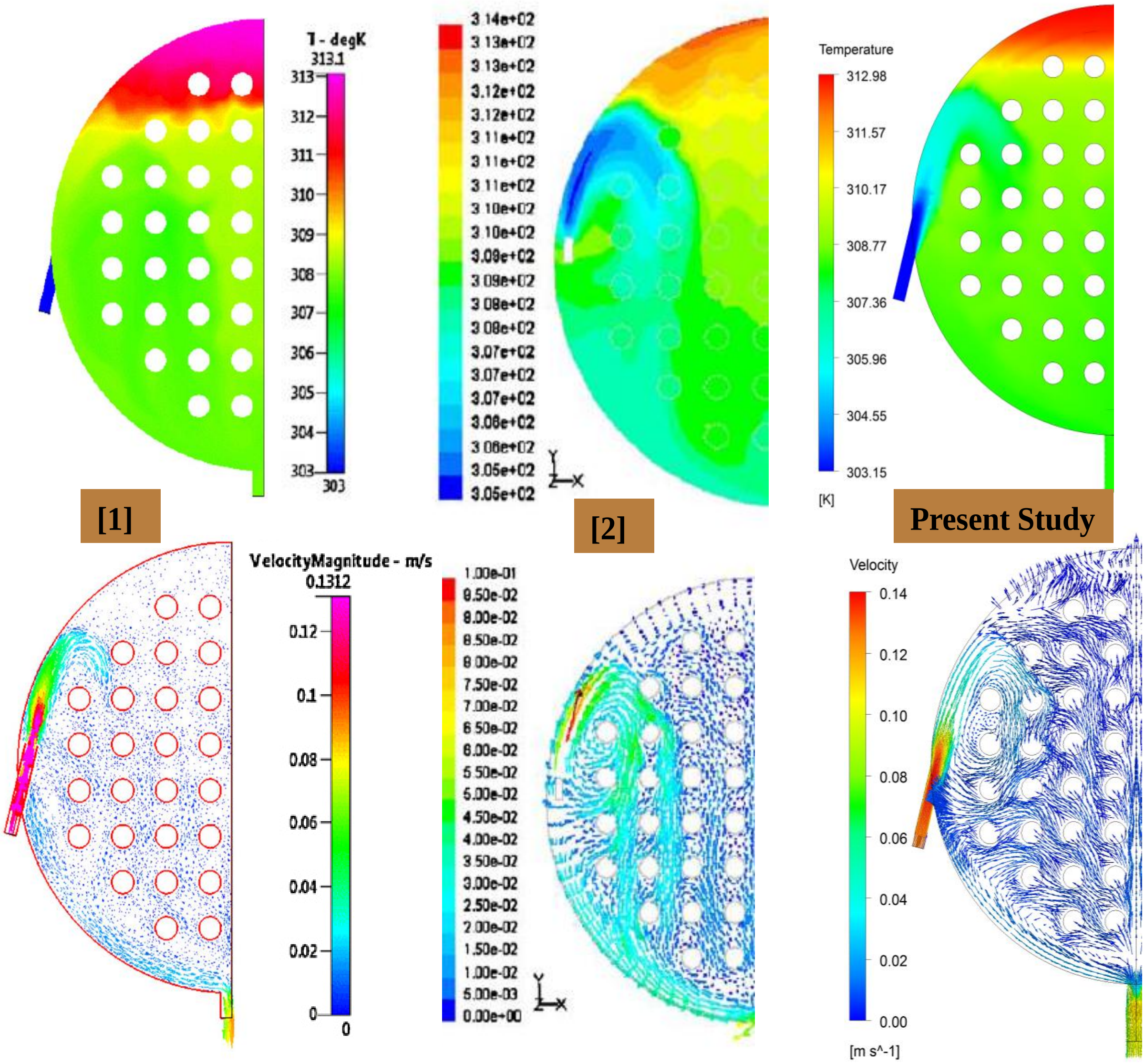


Fig.3.3 Comparisons of Temperature contours and velocity profile

The temperature increases slightly due to the heat generation, and decreases again at the bottom region due to the forced convection. From the above results, it can be considered that the present model has the capability to properly analyze the fluid flow subject to the combined buoyancy and momentum forces simultaneously. The fluid flow inside the vessel is very complex due to the interaction between the momentum force generated by the inlet jets and the buoyancy force of heat load to fluid. The entering fluid is heated up by heat generation, the fluid flows toward the outlet due to small amount of heat load and strong inlet jets. Two inlet flows are encounter each other at the nearly center region, and discharge through the outlet. In this condition, the location of high temperature of fluid is the near-bottom of Calandria, and the momentum-dominated flow forms. In the condition of high heat load and low flow rate, however, the effect of buoyancy force on the main stream becomes more significant than that of momentum force. Due to the strong heat generation, the entering cold fluid is heated up enough to flow upward by density gradient, and it flows along the inner wall of vessel and discharges through the outlet, producing the secondary flows in the vicinity of the

Calandria wall and heaters. The location of the high temperature of fluid is the upper region of Calandria, and the buoyancy-dominated force is found. In the condition of certain heat load and flow rate, due to the force balance of momentum and buoyancy forces, the mixed type flow forms and high temperature of fluid is placed in the center region.

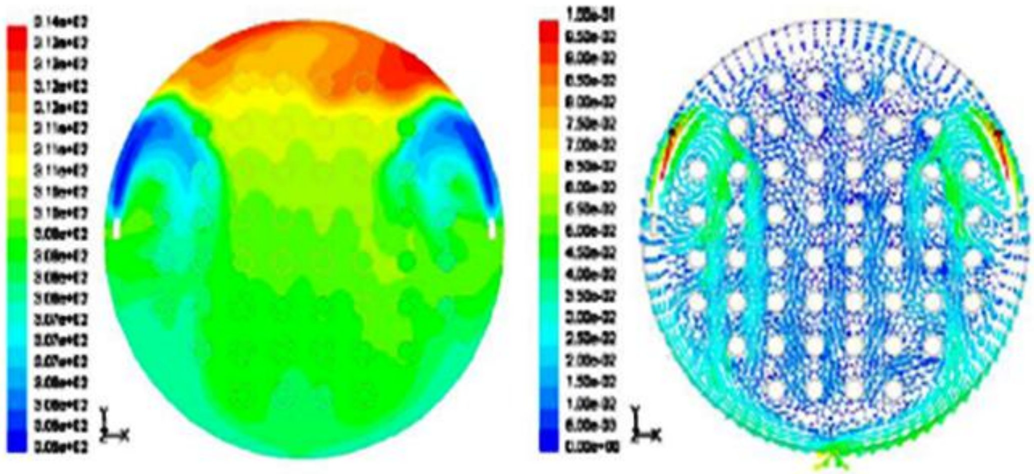


Fig.3.4 Buoyancy dominated flow (V_{in}=0.13m/s, T_{in}=30^{0}C, Heatload 10Kw)

4.CONCLUSION

Canadian Deuterium Uranium (CANDU) nuclear reactor is a pressurized water reactor. The study presents the assessments of moderator thermal–hydraulic characteristics in the normal operating condition of CANDU reactor. The comparison with the SPEL experimental data show that the present model can reasonably predict the temperature distributions of moderator, which means that the present model has the better capability to properly analyze the fluid flow subject to the buoyancy. Finally it is concluded that the major parameters affecting the flow patterns are the inlet flow rate and heat load to the fluid.

REFERENCES

[1]. Gazy F. Al-Sumaily, John Sheridan, Mark C. Thompson “Analysis of forced convection heat transfer from a circular cylinder embedded in a porous medium” International Journal of Thermal Sciences 51 (2012) 121-131.

[2]. Manwoong Kim, Seon-Oh Yu, Hho-Jung Kim “Analyses on fluid flow and heat transfer inside Calandria vessel of CANDU-6 using CFD” Nuclear Engineering and Design 236 (2006) 1155–1164.

[3]. Sajida Lafta Ghashim Jassim “Numerical study of the mixed convection flow over a square cylinder” Iraqi Journal of Chemical and Petroleum Engineering Vol.11 No.1 (March 2010) 29-45 ISSN: 1997-4884.

[4]. Eduardo Schnurr Siqueira, Flavia Zinani, Maria Luiza, Sperb Indrusiak “Numerical study of forced convection flow of a power-law fluid across a square cylinder”, 22nd International Congress of Mechanical Engineering (COBEM 2013), November 3-7, 2013.

[5]. H. P. Rani, G Janardhana Reddy “Conjugate Transient Free Convective Heat Transfer from a Vertical Slender Hollow Cylinder with Heat Generation Effect” Applied Mathematics. 2011; 1(2): 90-98.

[6]. E. D. Dos Santos, F. M. V. Da Silva,I. C. Acunha Jr, M. M. Galarça, L. A. Isoldi, And L. A. O. Rochac “Numerical Investigation of Turbulent Forced Convective Flows over a Pair of circular cylinders. Engenhariatermica (Thermal Engineering), Vol. 11, June and December 2012, pp77-84.