STUDY AND ANALYSIS OF THREE DIMENSIONAL FLOWS THROUGH A PLENUM BOX

Deepak Kumar

Assistant Professor, Dept. of Mechanical Engineering, Amity University Gurgaon, Haryana, India

***

Abstract – *In present work, the time independent three dimensional flow of a fluid has been studied. The fluid had been allowed to flow through a plenum box with a circular inlet at the bottom and an offset circular outlet at the top. A computer program had been developed to solve the equations. Velocity in different directions and flow pattern had been investigated.*

***Key Words*: Plenum box, Velocity Vector, FLEX PDE, Three Dimensional Flow, pressure, Streamlines.**

1. INTRODUCTION

Understanding the complexities of three dimensional flows through a plenum is a problem that has been studied for many years. Researchers in this field have been creative and innovative by introducing several new techniques. In this study problem of three dimensional flows in a plenum box was studied and investigated. Tauveron, N. [1] described a typical study on thermal hydraulic problems on high temperature reactors. It deals with thermal stresses on the core outlet region of a new concept of high temperature reactor. The simulations point out the thermal fluctuations in the nominal state in the fluid and in the solid. First results were presented. They illustrated the complexity of the calculation due to particular geometry and boundary conditions. Qualitative analyses of the simulations reinforce the former evaluations on the oscillating character of the flow, the effects of mixing of different flows, and the consequences on the thermal load on the solid structures. In the future quantitative results can be used as source term for studies of solid mechanics. These calculations need also the computation of the global behaviour of the circuit. Simulations were performed with the TRIO_U/PRICELES code for the 3D-analysis and the CATHARE code for the system modelling. Both were developed by the CEA. Kailash C. Karki et al. [2] investigated a Computational Fluid Dynamics model for calculating airflow rates through perforated tiles in raised-floor data centers. The model was based on the assumption that the pressure in the space above the raised floor is uniform, which allows the calculation to be limited to the space below the raised floor. It used finite-volume method, the k-ε turbulence model, and a multigrid method. The model was applied to a real-life data center. The calculated results for velocity and pressure distributions were discussed. The flow rates through the perforated tiles were shown to be in good agreement with the measured values. Keith G. Condie et al. [3] goal of this study was to provide the data necessary to enable validation of the analysis tools which were planned for use in the analysis of the Very High Temperature Reactor (VHTR) plant. These advanced gas-cooled reactors (AGCRs) for higher efficiency and enhanced safety and for deployable reactors for electrical power generation, process heat utilization and hydrogen generation. While key applications would be VHTRs using the closed Brayton cycle (CBC) for higher efficiency, results of the proposed research should also be valuable in gas-cooled fast-spectrum reactor systems (GFRs) as well as reactors with supercritical-pressure flow. Higher efficiency leads to lower cost/kwh and reduces life-cycle impacts of radioactive waste (by reducing waste/kwh). The outcomes will also be useful for some space power and propulsion concepts and for some fusion reactor concepts as side benefits, but they were not the thrusts of the investigation. Richard W. Johnson et al. [4] report showed the progress of turbulent CFD predictions for a section of the VHTR lower plenum using the NPHASE and FLUENT®codes. To date, the NPHASE simulations have focused on RANS steady-state solutions and the standard k~İ turbulence model with 2 inlet jets operating. FLUENT® simulations have been performed for unsteady flow using the realizable k~İ turbulence model with 4 inlet jets operating. These results should be treated as preliminary and used to guide the final CFD computations and experiments. Areas for further study have been outlined. Comparison with relevant experimental data was necessary to validate the CFD models. Donna Post Guillen [5] analysed advanced gas-cooled reactors which offer the potential advantage of higher efficiency and enhanced safety over present day nuclear reactors. Accurate simulation models of these Generation IV reactors are necessary for design and licensing. One design under consideration by the Very High Temperature Reactor (VHTR) program is a modular, prismatic gas-cooled reactor. In this reactor, the lower plenum region may experience locally high temperatures that can adversely impact the plant's structural integrity. Since existing system analysis codes cannot capture the complex flow effects occurring in the lower plenum, computational fluid dynamics (CFD) codes are being employed to model these flows. The goal of his study was to validate the CFD calculations using experimental data. Nolan Anderson [6] investigated the flow of hot gas in the outlet plenum of the very high temprature reactor using coupled RELAP5 – 3D system code and a CFD code. The very high temperature reactor (VHTR) system behavior should be predicted during normal operating conditions and postulated accident conditions. The plant accident scenario and the passive safety behavior should be accurately predicted. Uncertainties in passive safety behavior could have large effects on the resulting system characteristics. Due to these performance issues in the VHTR, there is a need for development, testing and validation of design tools to demonstrate the feasibility of the design concepts and guide the improvement of the plant components. One of the identified design issues for the gas-cooled reactor is the thermal mixing of the coolant exiting the core into the outlet plenum. Incomplete thermal mixing may give rise to thermal stresses in the downstream components. To provide flow details, the analysis presented in this paper was performed by coupling a VHTR model generated in a thermal hydraulic systems code to a computational fluid dynamics (CFD) outlet plenum model. The outlet conditions obtained from the systems code VHTR model provide the inlet boundary conditions to the CFD outlet plenum model. By coupling the two codes in this manner, the important three-dimensional flow effects in the outlet plenum are well modeled while avoiding modeling the entire reactor with a computationally expensive CFD code. The values of pressure, mass flow rate and temperature across the coupled boundary showed differences of less than 5% in every location except for one channel. The coupling auxiliary program used in this analysis can be applied to many different cases requiring detailed three-dimensional modeling in a small portion of the domain. Chang H. Oh et al. [7] major goal of this 3-year study was to perform air-ingress-related experiments and validate the computer codes, such as computational fluid dynamics (CFD) and GAMMA, so they can be reliably used to predict the consequences of air-ingress in the NGNP. The associated research objectives were as follows: Conduct experiments to supply information needed to validate GAMMA and CFD codes to model important phenomena during air-ingress accidents. These experiments measured: - The effects of density-driven, stratified flow on air ingress into the reactor core - The internal pore area density of nuclear grade graphite, which was an important parameter for determining the oxidation rate - The oxidation and density variation in terms of burn-off in the core bottom structures - The effects of the burn-off on the structural integrity of the core bottom structures. Hassan Yassin et al. [8] goal of this project was to investigate the fundamental physical phenomena associated with internal coolant flow in a prismatic core VHTR vessel during normal operation and under accident scenarios. Previous studies have revealed the importance of complex jet/plume flows in each plenum, with potential to generate recirculation zones that can lead to formation of hot spots within the lower plenum. It was therefore of interest to ensure that adequate mixing is promoted, but the complexity of the internal flow fields (characterized by structures spanning multiple orders of magnitude in time and length scales) makes rational design challenging. These difficulties were further compounded by limited availability of data for validation of predictive models.

2. GOVERNING MATHEMATICAL FORMULATION

The flow dynamics for this type of flow within the flow domain is described by the partial differential equations as given below

div_v = dx(vx) + dy(vy) + dz(vz)

vx: dens*(vx*dx(vx) + vy*dy(vx) + vz*dz(vx)) + dx(p) - visc*div(grad(vx)) = 0

vy: dens*(vx*dx(vy) + vy*dy(vy) + vz*dz(vy)) + dy(p) - visc*div(grad(vy)) = 0

vz: dens*(vx*dx(vz) + vy*dy(vz) + vz*dz(vz)) + dz(p) - visc*div(grad(vz)) = 0

p: div(grad(p)) = PENALTY*div_v

Extrusion z = -high-duct,-high,high,high+duct

3. PROBLEM DESCRIPTION

In this study, the three dimensional flow of fluid through a plenum box as shown in Fig. A was considered

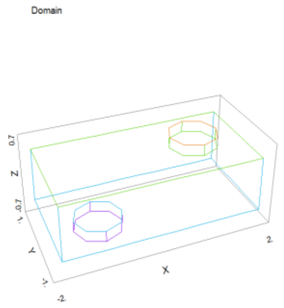

Fig A

4. NUMERICAL SIMULATION

For simulating the stream line patterns, velocity and pressure distributions in the flow regime under study were simulated using CFD computer code Flex PDE (A flexible solution system for partial differential equations byPDE solutions Inc., www.pdesolutions.com).This code utilizes the finite element numerical solver performing the operations necessary to turn a description of a partial differential equations system into a finite element model, solve the system, and present graphical and tabular output of the results. It performs the entire range of functions necessary to solve partial differential equation systems: an editor for preparing scripts, a mesh generator for building finite element meshes, a finite element solver to find solutions, and a graphics system to plot the results.

5. RESULTS AND DISCUSSION

A computer program was developed here for quantitative and qualitative analysis of three dimensional flows through a plenum box.

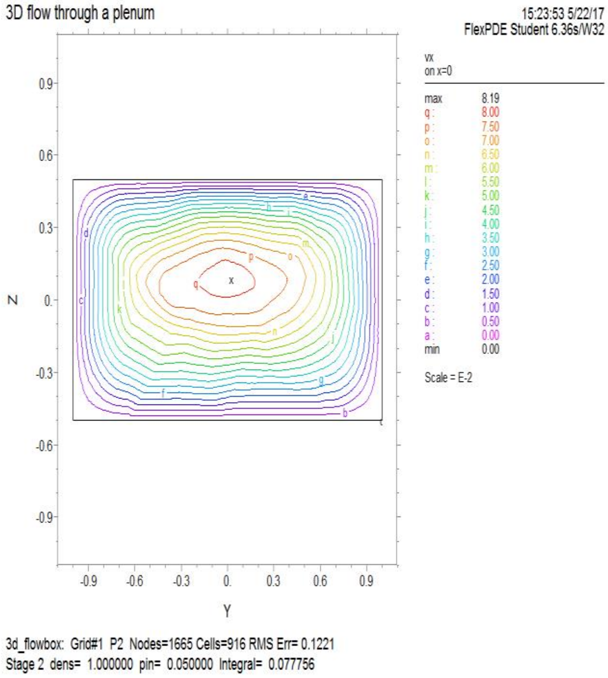

Fig -1: Variation of velocity

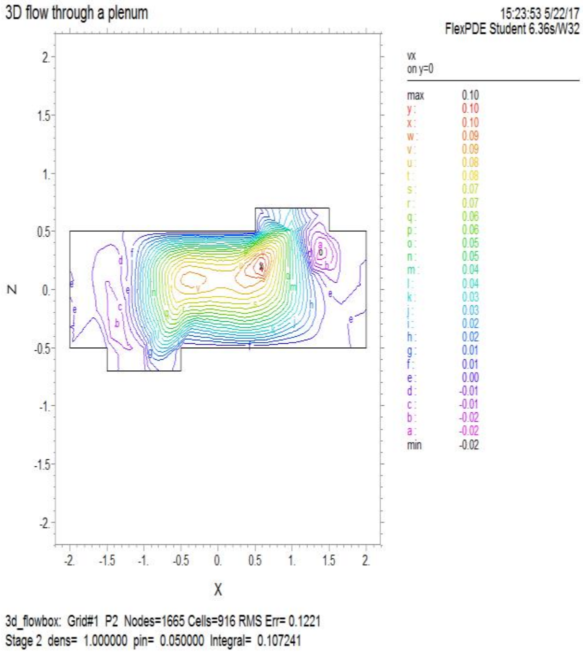

Fig -2: Variation of velocity

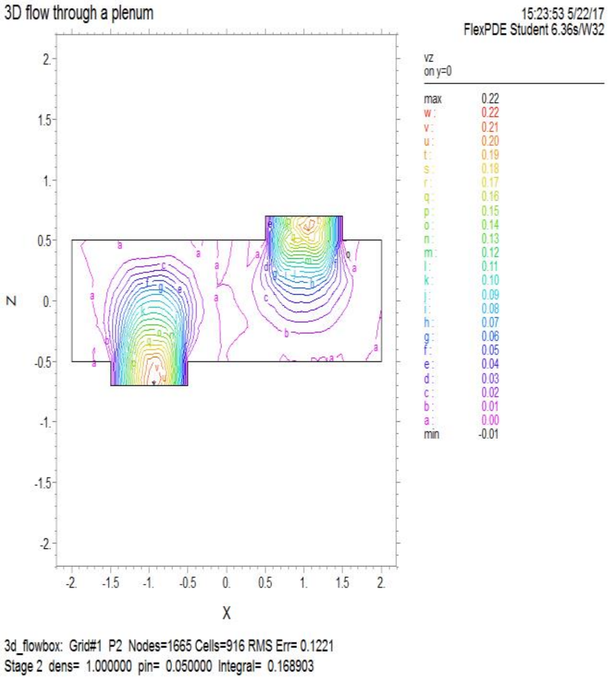

Fig -3: Variation of velocity

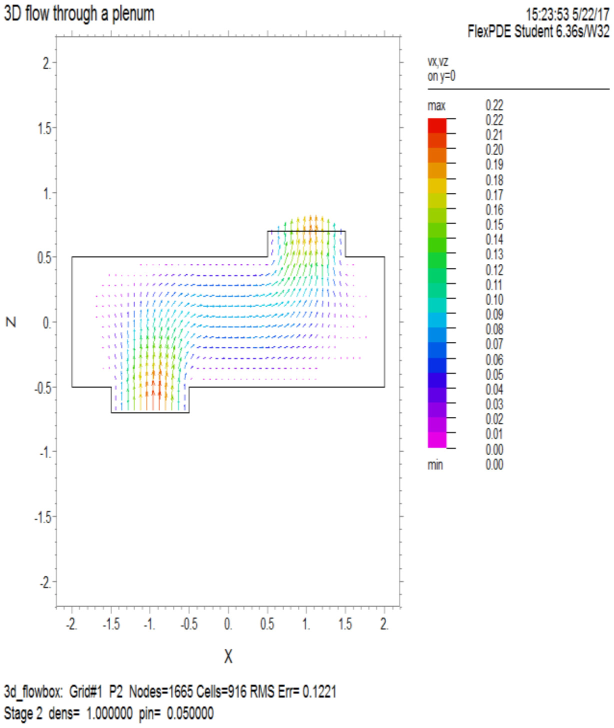

Fig -4: Variation of flow pattern

6. CONCLUSIONS

The Flex PDE software was used for numerical simulation of fluid dynamics to evaluate the flow characteristics of a fluid i.e. water flowing through a plenum box. In this study the variation of velocity in different directions is shown. The variation of flow pattern is also shown.

REFERENCES

[1] Tauveron, N., "Thermal fluctuations in the lower plenum of a high temperature reactor". Nuclear engineering and design,vol 222(2): 2003,pp. 125-137.M. Young, The Technical Writer's Handbook. Mill Valley, CA: University Science, 1989.

[2] Kailash C. Karki, Amir Radmehr, and Suhas V. Patankar "Use of Computational Fluid Dynamics for Calculating Flow Rates Through Perforated Tiles in Raised-Floor Data Centers" International Journal of Heating, Ventilation, Air-Conditioning, and Refrigeration Research, Volume 9, Number 2, April 2003, pp. 153-166.

[3] Keith G. Condie, Glenn E. McCreery, Hugh M. McIlroy, Jr., Donald M. McEligot "Development of an Experiment for Measuring Flow Phenomena Occurring in a Lower Plenum for VHTR CFD Assessment" INL External Report INEEL/EXT-05-00603, (September 2005).

[4] Richard W. Johnson, Donna P. Guillen, Tara Gallaway "Investigations of the Application of CFD to Flow Expected in the Lower Plenum of the Prismatic VHTR" INL External Report INL/EXT-06-11756, (September 2006).

[5] Donna Post Guillen "Computational Flow Predictions for the Lower Plenum of a High-Temperature, Gas-Cooled Reactor" Idaho National Laboratory, P.O. Box 1625, Idaho Falls, ID 83415, Nov. 2006.

[6] Nolan Anderson " Analysis of the hot gas flow in the outlet plenum of the very high temperature reactor using coupled relap5 – 3D system code and a CFD code", Journal of the nuclear engg and design, vol. 238,issue1,jan 2008,pp. 274-279

[7] Chang H. Oh, Eung S. Kim, Hee C. NO, Nam Z. Cho "Experimental Validation of Stratified Flow Phenomena, Graphite Oxidation, and Mitigation Strategies of Air Ingress Accidents" INL External Report INL/EXT-08-14840, (December 2008).

[8] Hassan Yassin, Anand NK, Wolf James "Experimental and CFD Studies of Coolant Flow Mixing within Scaled Models of the Upper and Lower Plenums of NGNP Gas-Cooled Reactors" Reactor Concepts Research Development and Demonstration, Texas A&M University, Project No. 12-3759 (March 2016).

[9] Kumar Deepak,Malk R.K. "Modelling of Flow Dynamics of an Incompressible Viscous Fluid in a Channelwith Solid Obstruction of Variable Length"IJERGS.3(1):117-127.2015.

[10] Kumar Deepak "Analysis of Flow Dynamics of an Incompressible Viscous Fluid in a Channel"IJERGS.4 (4):79-89.2016.