iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VI **Month of publication:** June 2017

DOI:

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Variation of Pressure and Velocity in A 3- D Flow Through a Plenum

Deepak Kumar[1]
[1]Assistant Professor, Department of Mechanical Engineering, Amity University Gurgaon, India

Abstract: ***In this study, a plenum box is considered and the fluid is allowed to flow through this plenum box. The three-dimensional flow of the fluid which is not dependent on time has been studied. With the help of FLEX PDE software the governing equations are solved. The variation of pressure and velocity in different directions is analysed.***
Keywords: ***Plenum box, Velocity Vector, FLEX PDE, Three-Dimensional Flow, pressure, Streamlines.***

I. INTRODUCTION

In this study problem of three dimensional flows in a plenum was studied and investigated. Borghi et al. [1] compared different methodologies of CFD analysis, applied to the intake plenum of a turbocharged HSDI Diesel engine. The study was performed by using both an engine cycle simulation code and a 3D-CFD code. Experiments at the engine dynamometer and at a steady flow bench support the theoretical study.The most promising simulation technique presented in the paper was the integrated 1D and 3D-CFD simulation. This numerical approach showed itself to be particularly suitable for analysing complex engine components where the flow patterns are fully transient. Tauveron, N. [2] 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. Glenn E. McCreery [3] addressed experimental modeling of flow and thermal mixing phenomena of importance during normal or reduced power operation and during a loss of forced reactor cooling (pressurized conduction cooldown) scenario. The objectives of the experiments were, 1), provide benchmark data for assessment and improvement of codes proposed for NGNP designs and safety studies, and, 2), obtain a better understanding of related phenomena, behavior and needs. Physical models of VHTR vessel upper and lower plenums which use various working fluids to scale phenomena of interest were presented and the recommended water-flow models were described in more detail. The models may be used to both simulate natural convection conditions during pressurized conduction cooldown and turbulent lower plenum flow during normal or reduced power operation. 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. Richard W. Johnson [5] presented validation simulations for turbulent flow in a staggered tube bank, geometry similar to the lower plenum of a gas-cooled high temperature reactor. Steady 2D RANS results were compared to unsteady 2D RANS results and experiment. The unsteady calculations account for the fact that non turbulent fluctuations (due to vortex-shedding) are present in the flow. The unsteady computations were shown to predict the mean variables and the total shear stress quite well. Previous workers' results indicate that 3D simulations were needed to obtain reasonable agreement; present results indicate 2D is sufficient. Best practices were based on requirements for the ASME Journal of Fluids Engineering. Hugh M. McIlroy, Jr. et al. [6] presented Mean-velocity-field and turbulence data that measure turbulent flow phenomena in an approximately 1:7 scale model of a region of the lower plenum of a typical prismatic gas-cooled reactor (GCR) similar to a General Atomics Gas-Turbine-Modular Helium Reactor (GTMHR) design. The data were obtained in the Matched-Index-ofRefraction (MIR) facility at Idaho National Laboratory (INL) and are offered for assessing computational fluid dynamics (CFD) software. This experiment has been selected as the first Standard

©IJRASET: All Rights are Reserved

Problem endorsed by the Generation IV International Forum. Results concentrate on the region of the lower plenum near its far reflector wall (away from the outlet duct). The flow in the lower plenum consists of multiple jets injected into a confined cross flow - with obstructions. The model consists of a row of full circular posts along its centerline with half-posts on the two parallel walls to approximate geometry scaled to that expected from the staggered parallel rows of posts in the reactor design. The model was fabricated from clear, fused quartz to match the refractive-index of the working fluid so that optical techniques may be employed for the measurements. The benefit of the MIR technique was that it permits optical measurements to determine flow characteristics in complex passages in and around objects to be obtained without locating intrusive transducers that will disturb the flow field and without distortion of the optical paths. An advantage of the INL system was its large size, leading to improved spatial and temporal resolution compared to similar facilities at smaller scales. A three-dimensional (3-D) Particle Image Velocimetry (PIV) system was used to collect the data. Inlet jet Reynolds numbers (based on the jet diameter and the time-mean bulk velocity) are approximately 4,300 and 12,400. Uncertainty analyses and a discussion of the standard problem are included. The measurements reveal developing, non-uniform, turbulent flow in the inlet jets and complicated flow patterns in the model lower plenum. Data include three-dimensional vector plots, data displays along the coordinate planes (slices) and presentations that describe the component flows at specific regions in the model. Information on inlet conditions is also presented. 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. Richard W. Johnson et al. [8] The U.S. Department of Energy (DOE) is promoting the resurgence of nuclear power in the United States for both electrical power generation and production of process heat required for industrial processes such as the manufacture of hydrogen for use as a fuel in automobiles. The project DOE is funding to accomplish this is called the Next Generation Nuclear Plant (NGNP) Project, which is based on a Generation IV reactor concept called the very high temperature reactor (VHTR). The VHTR will use helium as the coolant at temperatures ranging from 450°C to perhaps 1000°C. While computational fluid dynamics (CFD) has not previously been used for the safety analysis of nuclear reactors in the United States, it is being considered for existing and future reactors. It is fully recognized that CFD simulation codes will have to be validated for flow physics reasonably close to actual fluid dynamic conditions expected in normal and accident operational situations. To this end, experimental data have been obtained in a scaled model of a narrow slice of the lower plenum of a prismatic VHTR. The present report presents results of CFD examinations of these data to explore potential issues with the geometry, initial conditions, flow dynamics, and data needed to fully specify the inlet and boundary conditions; results for several turbulence models are examined. Issues are addressed and recommendations about the data are made. Robert D. Knapke et al. [9] addressed an unsteady analysis of the MIT counter rotating aspirated compressor (CRAC) using the Numeca FINE/Turbo 3D viscous turbulent flow solver with the Nonlinear Harmonic (NLH) method. All three blade rows plus the aspiration slot and plenum were included in the computational domain. Both adiabatic and isothermal solid wall boundary conditions were applied and simulations with and without aspiration were completed. The aspirated isothermal boundary condition solutions provide the most accurate representation of the trends produced by the experiment, particularly at the endwalls. These simulations provide significant insight into the flow physics of the aspiration flow path. Time histories and spanwise distributions of flow properties in the aspiration slot and plenum present a flow field with significant temporal and spatial variations. In addition, the results provide an understanding of the aspiration flow path choking mechanism that was previously not well understood and is consistent with experimental results. The slot and plenum had been designed to aspirate 1% of the flow path mass flow; whereas the experiment and simulations show that it chokes at about 0.5% mass flow. Hassan Yassin et al. [10] 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. Kumar Deepak et al. [11] analyzed the time -independent laminar flow of a viscous, incompressible fluid in two - dimensions using Navier –Stokes equations with

©IJRASET: All Rights are Reserved

continuity equation as an incompressibility constraint and vorticity – stream function approach. The fluid was allowed to flow in a rectangular channel obstructed by a solid rectangular plate of variable length with uniform incident velocity. The CFD computer code Flex PDE was used to analyze the flow dynamics. It was observed that by increasing the blockage ratio, the maximum values of the stream function as well as the maximum value of the velocity decreased at the obstruction and the size of vortex increased. It was also observed that with the increase in blockage ratio the region of low pressure increased behind the obstruction. Kumar Deepak [12] studied the time independent three dimensional flow of a fluid. 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.

II. GOVERNING MATHEMATICAL FORMULATION

The governing partial differential equations which are used to analyse the pressure and velocity variation by using the computer program are 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

III. PROBLEM DESCRIPTION

In this study, a plenum box is being considered and the three-dimensional flow of fluid through this plenum box is studied as shown in Fig. A.

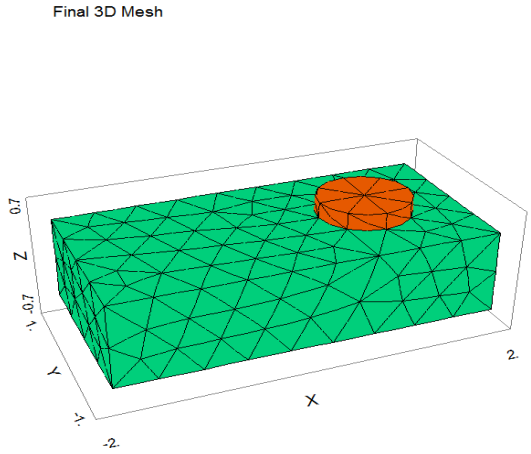


Fig A

IV. NUMERICAL SIMULATION

For analysing the velocity and pressure variations in the flow regime under study were simulated using CFD computer code Flex PDE (A flexible solution system for partial differential equations by PDE 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. Program is made in Flex PDE script and solved with Flex PDE. It is very easy to solve these equations with Flex PDE as there is no need of discretization of partial differential equations. We have to write these equations as it is in the script file of Flex PDE with appropriate boundaries and Flex PDE solve them itself.

©IJRASET: All Rights are Reserved

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

V. RESULTS AND DISCUSSION

With the help of flex pde software the governing mathematical equations are solved and the variation of pressure and velocity is shown as below.

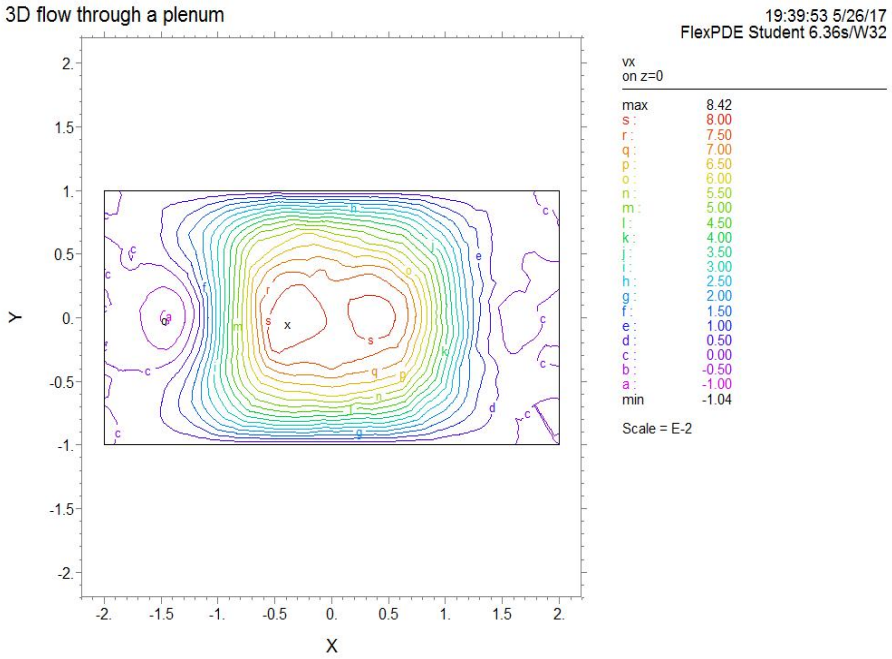


Fig 1 Variation of Velocity

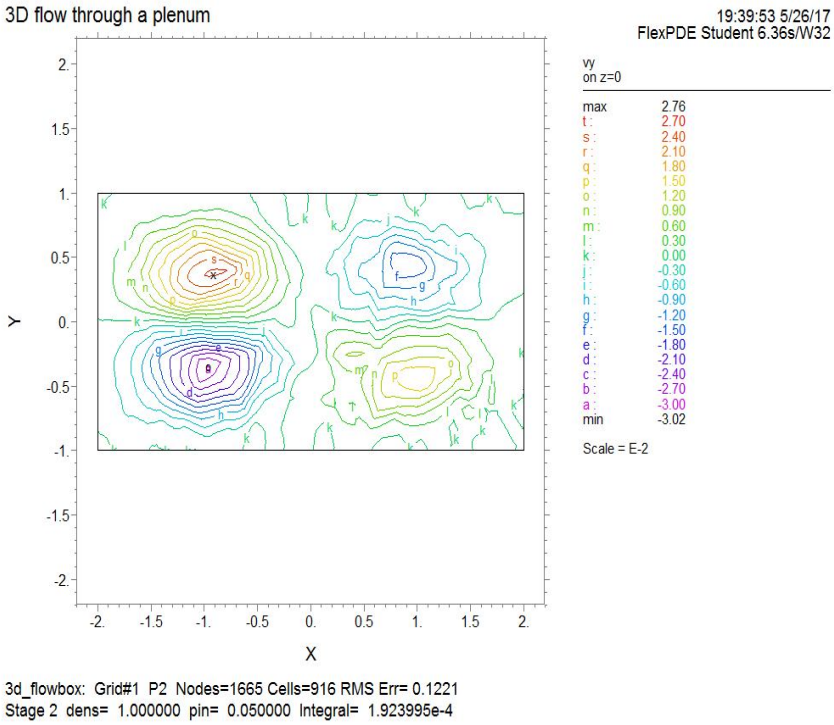


Fig 2 Variation of Velocity

©IJRASET: All Rights are Reserved

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

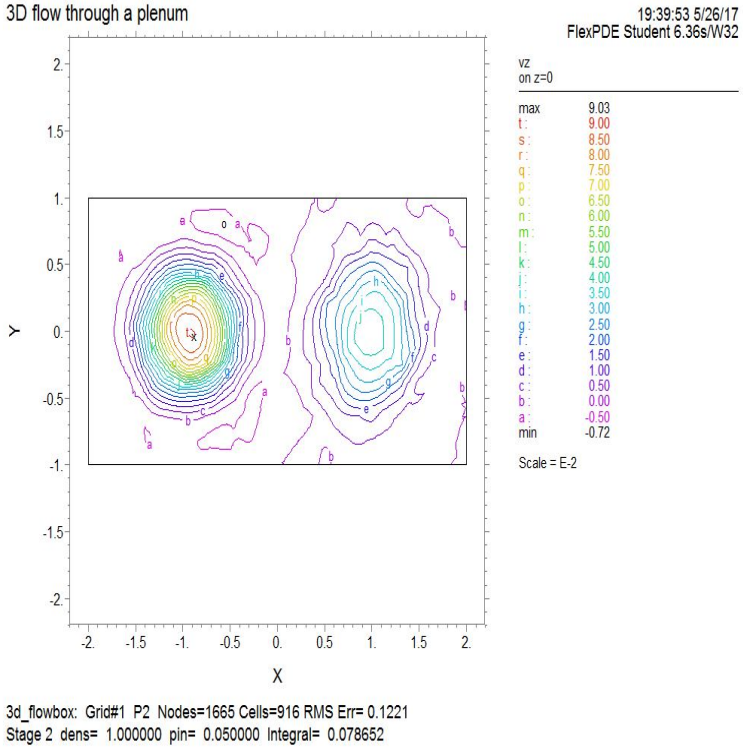


Fig 3 Variation of Velocity

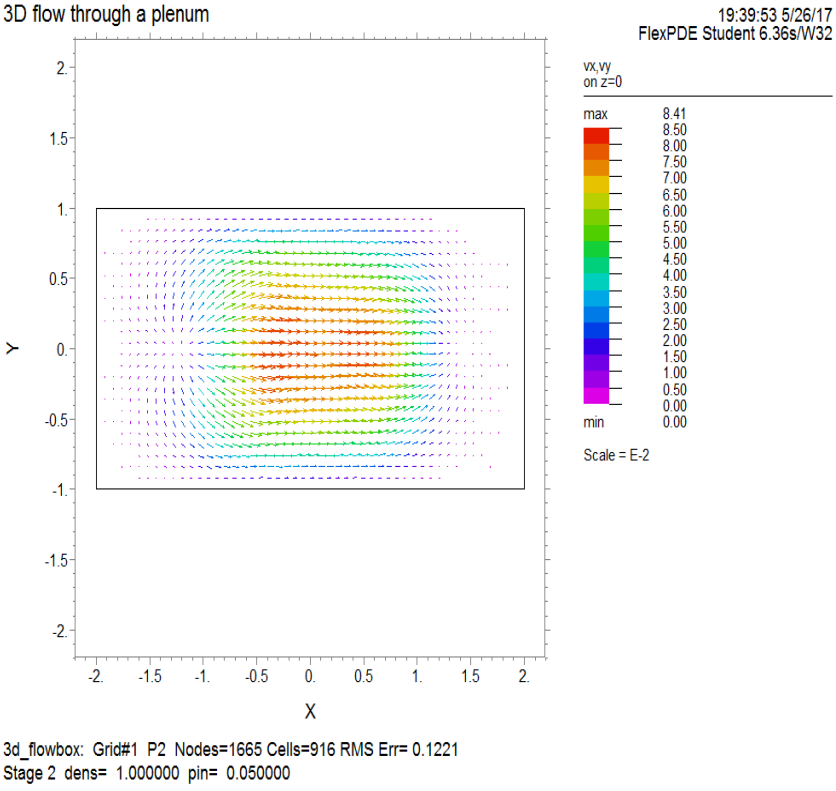


Fig 4 Variation of Velocity

©IJRASET: All Rights are Reserved

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

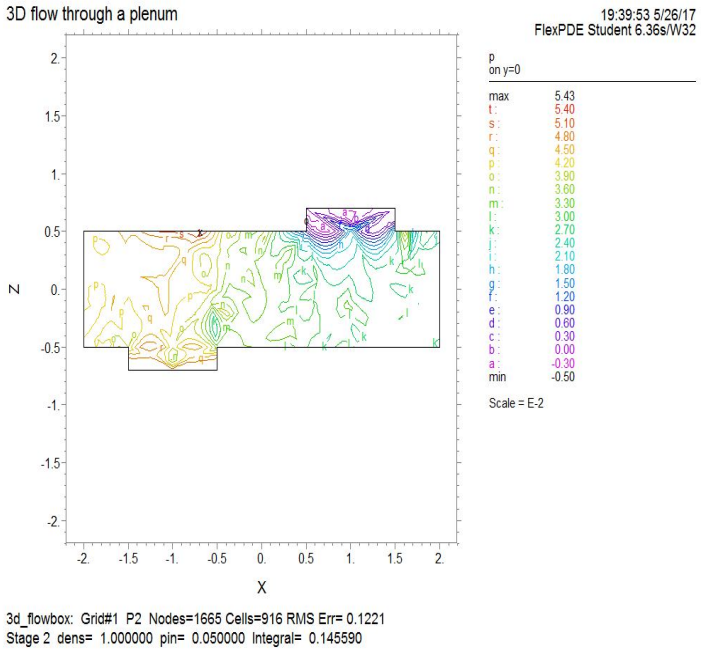


Fig 5 Variation of Pressure

VI. CONCLUSION

The variation of pressure and velocity is investigated and studied by the application of a computer program in which the fluid was flowing through a plenum in three dimensions. This software enables us to solve the partial differential equations with the help of initial and boundary conditions.

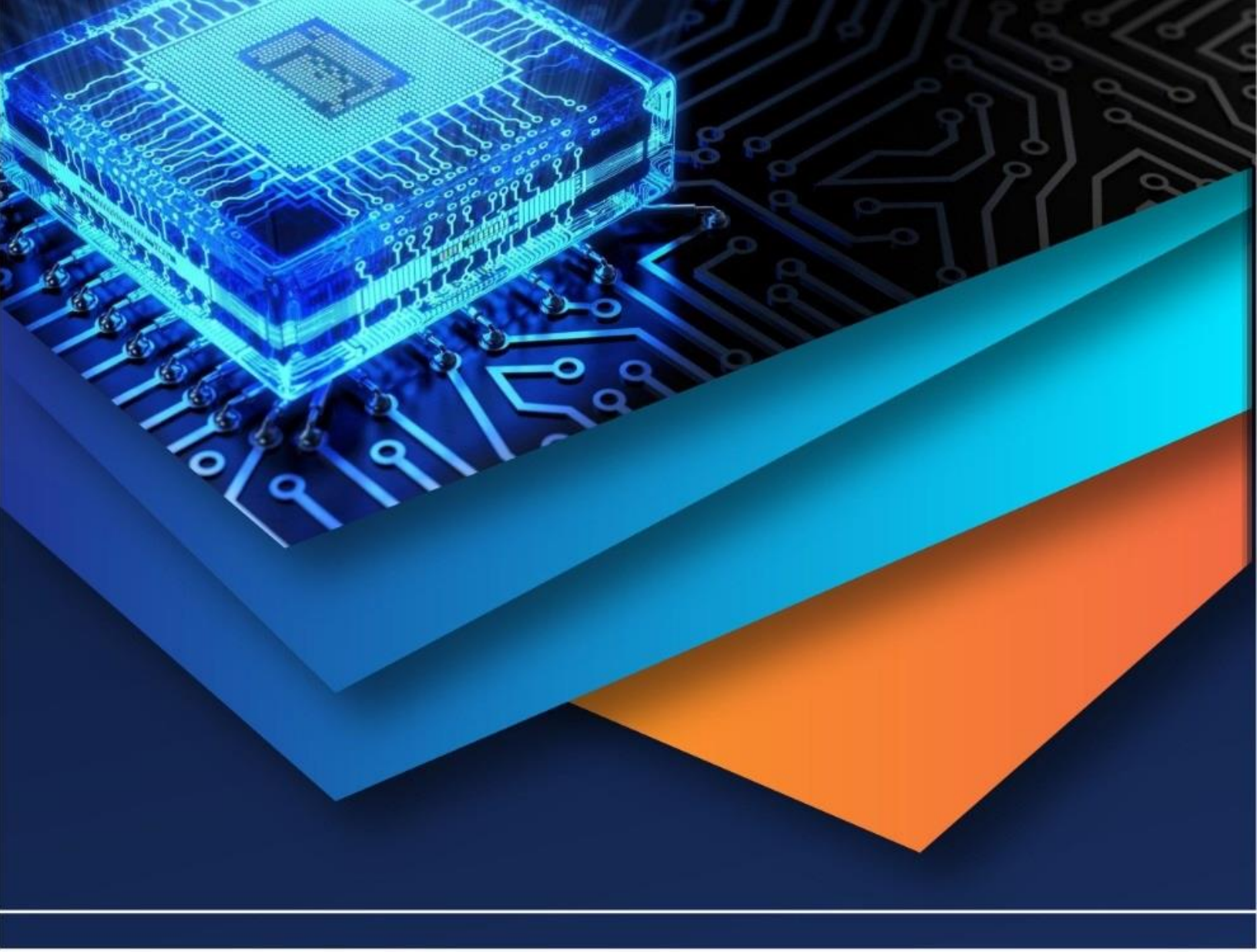
VII. ACKNOWLEDGMENT

I thank our colleagues from Amity University who provided insight and expertise that greatly assisted the research, although they may not agree with all of the interpretations/conclusions of this paper.

REFERENCES

[1] Borghi, M., Mattarelli, E., and Montorsi, L., "Integration of 3D-CFD and Engine Cycle Simulations: Application to an Intake Plenum," SAE Technical Paper 2001-01-2512, 2001, doi:10.4271/2001-01-2512
[2] Tauveron, N., “Thermal fluctuations in the lower plenum of a high temperature reactor”. Nuclear engineering and design,vol 222(2): 2003,pp. 125-137.
[3] Glenn E. McCreery, Keith G. Condie “Experimental Modeling of VHTR Plenum Flows During Normal Operation and Pressurized Conduction Cooldown” INL External Report INL/EXT-06-11760, (September 2006).
[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] Richard W. Johnson “Modeling strategies for unsteady turbulent flows in the lower plenum of the VHTR” Nuclear Engineering and Design,Volume 238, Issue 3, March 2008, Pages 482–491.
[6] Hugh M. McIlroy, Jr. Donald M. McEligot Robert J. Pink “Measurement of Flow Phenomena in A Lower Plenum Model of A Prismatic Gas-Cooled Reactor” Proceedings of the 16th International Conference on Nuclear Engineering ICONE16 May 11-15, 2008, Orlando, Florida, USA.
[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] Richard W. Johnson, Richard R. Schultz “Computational Fluid Dynamic Analysis of the VHTR Lower Plenum Standard Problem” ” INL External Report INL/EXT-09-16325, (July 2009).
[9] Robert D. Knapke , Mark G. Turner “Detailed Unsteady Simulation of a Counterrotating Aspirated Compressor with a Focus on the Aspiration Slot and Plenum” IJRM Volume 2013 (2013), Article ID 857616, 17 pages
[10] 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). 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.
[11] Kumar Deepak. “Study And Analysis Of Three Dimensional Flows Through A Plenum Box”IRJET.4 (5):1672-1675.2017.

©IJRASET: All Rights are Reserved



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)