2005 유체기계 연구개발 발표회 논문집, pp. 244~249
창원대학교 국제교류센터 및 정보전산원, 2005. 12. 1~2

Pressure and Flow Distribution in the Inlet Plenum of a Pebble Bed Modular Reactor (PBMR)

Imteyaz Ahmad[*], Kwang-Yong Kim[*]

Abstract-- Flow distribution and pressure drop analysis for an inlet plenum of a Pebble Bed Modular Reactor (PBMR) have been performed using Computational Fluid Dynamics. Three-dimensional Navier-Stokes equations have been solved in conjunction with k- ε model as a turbulence closure. Non-uniformity in flow distribution is assessed for the reference case and parametric studies have been performed for rising channels diameter, Reynolds number and angle between the inlet ports. Also, two different shapes of the inlet plenum namely, rectangular shape and oval shape, have been analysed. The relative flow mal-distribution parameter shows that the flow distribution in the rising channels for the reference case is strongly non-uniform. As the rising channels diameter decreases, the uniformity in the flow distribution as well as the pressure drop inside the inlet plenum increases . Reynolds number is found to have no effect on the flow distribution in the rising channels for both the shapes of the inlet plenum. The increase in angle between the inlet ports makes the flow distribution in the rising channels more uniform.

Key Words : Pebble Bed Modular Reactor(PBMR), Inlet Plenum, Rising Channels, Mass-Flow Distribution, Pressure drops, Computational Fluid Dynamics

1.Introduction

A High Temperature Gas Cooled Reactor (HTGR) [1, 2] is one of the renewed reactor designs to play a vital role in the nuclear power generation. This reactor design concept is currently under consideration and development worldwide including Korea. Since the HTGR concept offers inherent safety, has a very flexible fuel cycle with capability to achieve high burn up levels, and provides good thermal efficiency of power plant, it can be considered a further development and improvement as a reactor concept of generation IV. The combination of coated particle fuel, inert helium gas as coolant and graphite moderated reactor makes it possible to operate at high temperature yielding a high efficiency. The current Pebble-Bed Modular Reactor (PBMR) [3, 4] design under development by Korea Atomic Energy Research Institute (KAERI)[5] belongs to the class of generation IV reactors satisfying the goal ofHTGR. Its power conversion unit is based on the thermodynamic Brayton cycle. The Helium gas traverses through the inlet plenum, rising channels and outlet plenums also called riser plenums before entering into the core from the top of the reactor at a temperature of about 500℃ and at a pressure of about 7 MPa. The coolant exit temperature of the PBMR is 900℃ under normal operating conditions.

For optimum performance of the core, uniform cooling is required as non-uniformity may cause severe temperature gradients inside the pebble core. Also, there is a limit to the maximum temperature rise inside the core as the higher temperature inside the core increases the probability of neutron absorption by U-238 atoms thereby, reducing the number of neutrons available for U-235 fission which in turn, will result in reduced power output. These entail the requirement of uniform coolant flow distribution in the rising channels. Another important phenomenon of concern is the pressure drop inside

---

[*] CFE Lab, Inha University
Email: imteyaz@inhaian.net

the inlet plenum, rising channels and outlet plenums since higher losses will result in decrease in overall efficiency of the power plant. Thus, good knowledge of the flow details in the inlet and outlet plenums as well as in the rising channels are necessary for optimum design.

This study deals with the flow analysis inside the inlet plenum and rising channels using Computation Fluid Dynamics (CFD). A ring type inlet plenum, shown schematically in Fig. 1, is proposed for use as Inlet Plenum for Korean Pebble Bed Modular Reactor.

CFD has extensively been used in different engineering applications including nuclear science. This provides details of flow phenomena at a relatively lesser cost. Cesar Frepoli [6] used FLOW3D, a commercial CFD code, to simulate turbulent flow occurring in the reactor pressure vessel (RPV) inlet plenum of the simplified boiling water reactor (SBWR). Chander et. at. [7] investigated experimentally as well as numerically the core flow and pressure distributions in the outlet header of a pressurized heavy water reactor (PHWR). Boyd et. al. [8] carried out full scale inlet plenum mixing analyses during a PWR severe accident using Fluent 6.0.

Green and Moumtzis [9] analyzed an inlet manifold for combustion applications for a range of flow rates and exit pressure drops using computational fluid dynamics. Maharudraya et. al. [10] analyzed flow distributions and pressure drops in fuel cell stalks for a wide range of flow conditions using CFX 4.3. Jian wen and Yanzhong Li [11] studied flow distributions on the header of a plate-fin heat exchanger using Fluent 6.0. Kim et. al [12] analyzed numerically the effect of header shapes on the flow distribution in a manifold for electronic packaging applications. In general, the authors are convinced of the utility of numerical methods in predicting wide range of flows and where there are available experimental data's, they got reasonable agreements between experimental and numerical predictions.

In this work, three-dimensional flow distributions and pressure drop in the inlet plenum and rising channels are predicted. Two different shapes of the inlet plenum namely, rectangular and oval shapes are analyzed for resulting flow distributions in the rising channels. Parametric studies are performed to assess the effect of different geometrical parameters on the flow distribution in the rising channels and pressure drops.

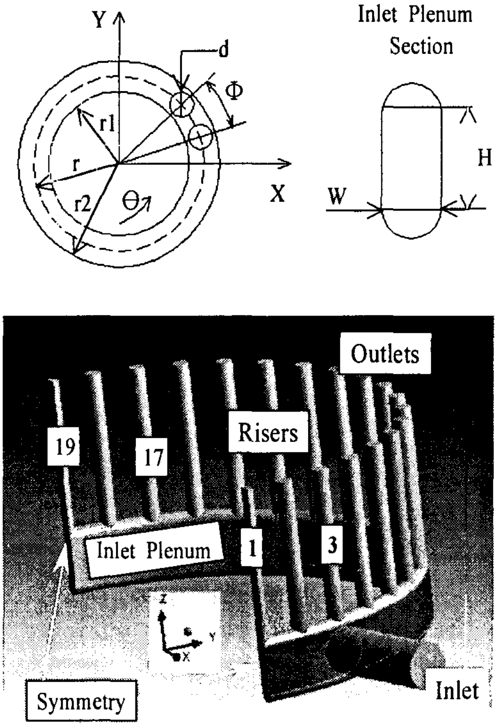

Fig. 1 Ring-Type inlet plenum

2. Governing Equations and Numerical Solution

The present calculation is based on CFD simulations in which the fundamental equations governing the flow, namely the conservation of mass and momentum for an incompressible isothermal flow are solved numerically. A commercial CFD code, CFX 5.7.1 [13] which uses unstructured grid, is used for the calculations. It is a widely used code for a variety of flow situations and contains a number of application specific models such as turbulence, chemical reactions, combustion and radiation. These models introduce a degree of empiricism into the governing equations, the severity of which depends on the application and on the model chosen.

Since the flow in the inlet plenum under operating conditions is always turbulent and so the use of appropriate turbulence model is necessary for numerical solution accuracy. Since the standard K-ε model has been widely used as an engineering turbulence model for many years and has the advantage of simplicity and economy and hence, we have adopted this model in the present study.

The reference case dimensions are as follows; Inlet Plenum Width, W=0.22m, Height of the plenum, H=0.60m, Diameter of the rising channels, d=0.20m, Inlet port diameter, D=0.577m, Angle between rising channels, $\varphi=10^0$, Angle between the inlet ports, $\alpha=40^0$, Inner radius of the plenum, r1=3.0m and outer radius of the plenum, r2=3.22m. Reynolds number of the calculation based on inlet port diameter was 75609 and the fluid used in the calculation is He (Ideal gas) at 500℃. All the calculations are performed under isothermal condition. Since the rising channels and inlet port lengths affect the flow distribution to some extent and hence, their effect is tested first and the resulting lengths 2.25m and 1.5m for the rising channels and inlet port, respectively show a best compromise between calculation time and accuracy. The following boundary conditions were adopted:

- Normal velocity specified as the mean flow velocity at the inlet
- Constant pressure at outlet
- No-slip and adiabatic conditions are used at all the wall boundaries
- Symmetry condition (zero normal gradient) is specified on the symmetry plane.

A systematic study of grid independence was carried out to verify the grid independency of the numerical solution. From the results, grid having 331,500 nodes is selected as an optimum grid. The mesh structure for the optimum grid is shown Fig. 2. All the simulations were conducted by means of a segregated method using the SIMPLE scheme [14] for pressure-velocity decoupling. Nominally second order-accurate schemes were selected for the discretization of the governing equations. A residual reduction factor of 10^{-6} for the mass conservation equation was used to monitor the convergence of the iterative solution.

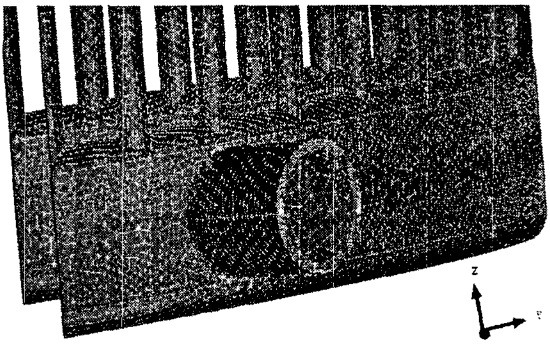

Fig. 2 Mesh Structure

3. Results and discussion

The numerical simulation is performed for the reference case first. The result obtained is analyzed carefully as the flow distribution in the rising channels and pressure drops inside the inlet plenum are sensitive to the inlet plenum dimensions. Since this study is concentrated towards the achievement of uniform flow distribution in all the rising channels, the reference case analysis was helpful in finding the range of variables for parametric studies as well as in the early achievement of uniform flow. The following parameters have been defined to express the results in physical terms:

$$m_c = \frac{m_i}{M} \tag{1}$$

$$S_i = \frac{m_i - m_{avg}}{m_{avg}} \tag{2}$$

$$C_p = \frac{p - p_{in}}{p_{in}} \tag{3}$$

Where, m_i is the mass-flow rate through a rising channel i (i = 1, 2,..,19), M is inlet mass flow rate, m_{avg} is the average of mass flow rate through all the rising channels, S_i is the relative flow mal-distribution parameter, p is the static pressure, p_{in} is the inlet static pressure, and ρ is the fluid density.

The streamlines for the reference case in the inlet plenum is shown in Fig.3. The flow coming out of the inlet pipe directly impinges on the plenum wall and changes its direction. Much of the flow goes to the larger space of the plenum. Static pressure distribution at mid- plane of the plenum is shown in Fig. 4. The pressure near side of the symmetry shows much higher values than the opposite. In this region, the symmetry prevents the inlet flow from expanding and makes the pressure build-up. Due to the high-pressure, flow rates through the two riser channels (1 and 2) are higher than the others.

The relative flow mal-distribution parameter, S_i as shown in Fig. 5, shows the magnitude of difference between the flow rates in different rising channels from the average value. As it is clear from the figure, the mass flow distribution is strongly non-uniform in this case. Fig. 6 shows the distribution of pressure coefficient C_p at a distance just below the rising channels (z = 0.4m). Fluctuating pressure distribution is observed since the flow stagnates on the plenum wall as it turns into the rising channels.

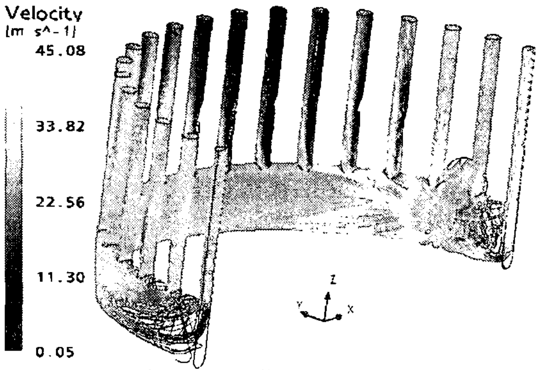

Fig.3 streamlines in the inlet plenum

Fig.4 Static pressure distribution at the mid-plane

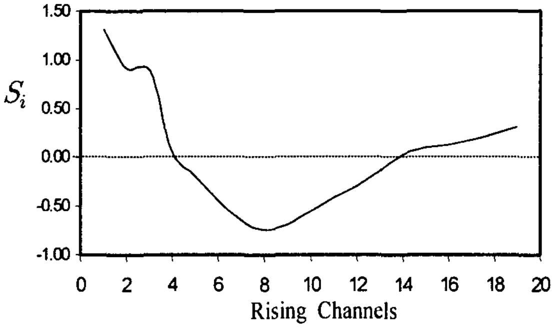

Fig.5 The relative flow mal-distribution parameter variation

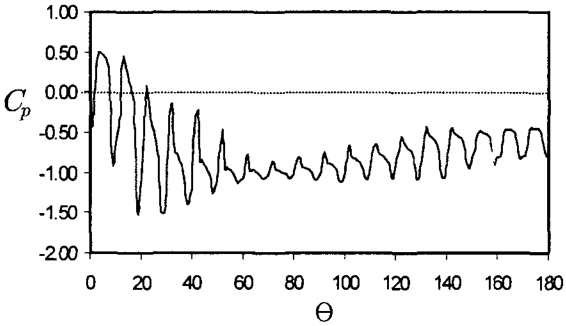

Fig.6 Non-dimensional static pressure distribution inside the inlet plenum(z=0.40m)

3.1 Effect of Rising Channels Diameter

The effect of rising channels diameter is tested in the range, 0.07m < d < 0.20m. Fig. 7 shows the relative flow maldistribution parameter, S_i variation for different rising channels diameter. This figure shows that as rising channels diameter decreases, the individual rising channel flow rates come close to the average value of all the rising channels. This happens due to increased uniformity in the pressure distribution inside the plenum. The pressure drop, Δp including the turning losses at inlet and exit of the inlet plenum for different rising channels diameter is shown in Fig.8. As it is obvious from the figure, the pressure drop increases as the diameter, d decreases which is caused by the increase in velocity. For, d = 0.08 m, the maldistribution parameter, S_i was within 5 percent of the average mass-flow rates of all the riser channels. Hence, this value of riser channels diameter was chosen for assessing other parameters effects.

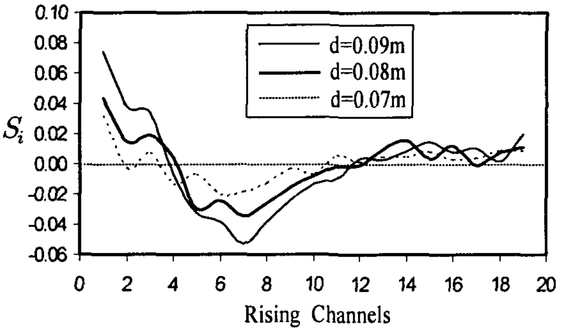

Fig.7 The relative flow mal-distribution parameter variation for different rising channels diameter

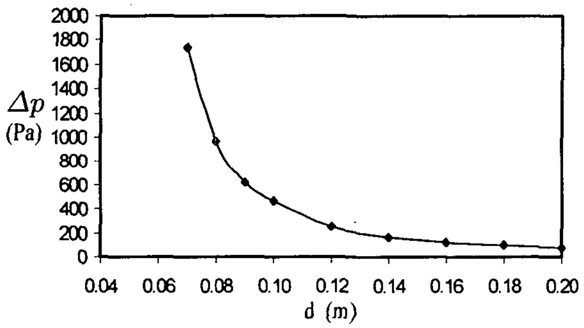

Fig.8 Effect of rising channels diameter on the pressure drop inside the plenum

3.1 Effect of different shapes of the inlet plenum and Reynolds number

The increased uniformity in flow distribution below a certain value of rising channels diameter is achieved at too greater a cost (i.e. increased pressure drop) to be a viable option. Hence, exploring other parameters and shapes effect become inevitable. In this section we have analyzed two different shapes of the inlet plenum, namely Oval-type and Rectangular-type for rising channels diameter, d = 0.08m keeping other geometrical parameters and inlet mass flow rate constant, i.e. same as the reference case. Also, tested in this section is the effect of Reynolds number i.e. the change in the inlet mass flow rate on the flow distribution parameter in the rising channels. The relative flow mal-distribution as shown in Fig. 9 clearly shows that the oval type plenum is a better choice. However, we have observed that the Reynolds number doesn't affect the flow distribution in the rising channels in both the cases.

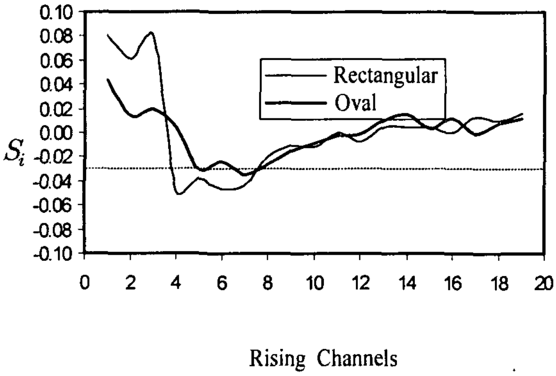

Fig.9 The relative flow mal-distribution parameter variation for different shapes of the plenum

3.2 Effect of Angle between inlet ports, α

The angle between inlet ports will also have some effect on the flow distribution and pressure drops and so we performed calculations for α in the range, $0^0 < \alpha < 180^0$. Angle zero corresponds to only one inlet port while higher values of angle imply two different inlet ports. The relative mal-distribution parameter for different angles between inlet ports is shown in Fig. 10. It is clear from the figure that as the angle α is increased, flow uniformity also increases. The pressure drop inside the plenum is found to decreases as the angle between inlet ports is increased.

4. Conclusion

The CFD analysis has been perforemed to investigate the flow distribution and pressure drop in

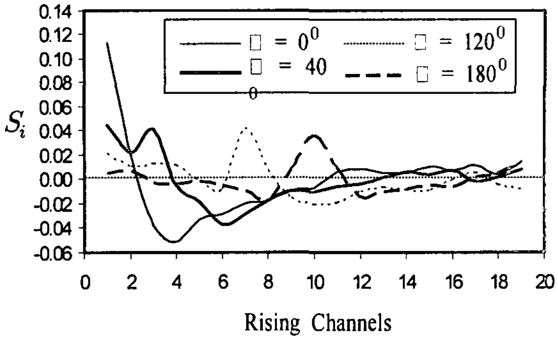

Fig.10 The relative flow mal-distribution parameter variation for different α

the inlet plenum of a PBMR. The flow distribution for the reference case is found to be strongly non-uniform. Decrease in the value of rising channels diameter is found to increase the flow uniformity in the rising channels. however, this also led to increase in the pressure drop inside the plenum. Oval shape of the inlet plenum is found to be a better choice as compared to the rectangular shape. The flow distribution in the inlet plenum is found to be independent of the inlet mass-flow rate indicating the robustness of design under varying operating conditions. As the angle between the inlet ports is increased, the flow uniformity in the rising channels is found to increase.

References

[1] S. Shiozawa, S. Fujikawa, T. iyoku, K.Kunitomi and Y. Tachibana, Overview of HTTR Design Features, Nuclear Engineering and Design, vol. 233, pp. 11-21, 2004.

[2] K. Kunitomi, S. Katanishi, S. Takada, X. Yan and N. Tsuji, Reactor Core Design of Gas Turbine High Temperature Reactor 300, Nuclear Engineering and Design, vol. 230, pp. 349-366, 2044.

[3] G. P. Greyvenstein and P.G. Rousseau, Design of a physical model of the PBMR with the aid of Flownet, Nuclear Engineering and Design, vol. 222, pp. 203-213,2003.

[4] S. Ion, D. Nicholls, R. Matzie and D. Matzner, Pebble Bed Modular Reactor The First generation IV reactor To Be Constructed, World Nuclear Association Annual Symposium, 3-5 September 2003, London.

[5] http://www.hydrogen.re.kr

[6] Cesare Frepoli, Numerical Simulation of the Inlet Plenum Flow Field for a LWR Reactor Pressure vessel, Applied Mathematic Modelling, vol. 20, pp. 252-261, 1996.

[7] D.K. Chandraker, N. K. Maheshwari, D. Saha and V. Venkat Raj, Experimental and Analytical investigations on core flow distribution and pressure distribution in the outlet header of a PHWR, Experimental Thermal and Fluid Science, vol. 27, pp. 11-24, 2002.

[8] C. F. Boyd, D. M. Helton and K. Hardesty, CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident, U.S. Nuclear Regulatory Research (NUREG-1788), Washington DC 2055-0001, May-2004.

[9] A.S. Green and T. Moumtzis, Case- Study: use of inlet manifold design techniques for combustion applications, Applied Thermal Engineering, vol. 22, pp. 1519-1527, 2002.

[10] S. Maharudraya, S. Jayanti and A.P. Deshpande, Flow distribution and pressure drop in parallel-channel configuration of plannar fuel cells, Journal of Power Sources, 2005.

[11] J. Wen and Y. Li, Study of flow distribution and its improvement on the header of plate-fin heat exchanger, Cryogenics, vol. 44, pp. 823-831, 2004.

[12] S. Kim, E. choi, and Y. I. Cho, The effect of header shapes on the flow distribution in a manifold for electronic packaging application, International Communications in Heat and Mass Transfer, vol. 22, pp. 329-342, 1995.

[13] CFX-5.7.1 Solver Theory, Ansys inc., 2005

[14] S. V. Patankar, Numrical Heat Transfer and Fluid Flow, Hemisphere, Washington DC, 1980.