Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30974

COMPUTATIONAL ANALYSIS OF FLUID FLOW IN PEBBLE BED MODULAR REACTOR USING DIFFERENT TURBULENCE MODELS

Akshay Gandhir
Department of Mechanical Engineering
Texas A&M University
College Station, Texas 77840
Email: akshaygandhir@neo.tamu.edu

Dr. Yassin Hassan
Department of Nuclear Engineering
Texas A&M University
College Station, Texas 77840
Email: y-hassan@tamu.edu

ABSTRACT

A steady state computational study was done to obtain the pressure drop estimation in different packed bed geometries, and describe the fluid flow characteristics for such complex structures. Two out of the three Bravais lattices were analyzed, namely, simple cubic (symmetric) and body centered cubic (staggered). STARCCM+ commercial CFD software from CD-ADAPCO was used to simulate the flow.

To account for turbulence effects standard k-epsilon and realizable k-epsilon models were used. Various cases were analyzed with Modified Reynolds number ranging from 10,000 to 50,000. Each model showed different results as far as the velocity and flow structure is concerned. However, for each case the flow structure showed similar features such as vortex formation downstream and between pebbles due to complex flow separation [1]. The pressure drop obtained from each model was found to be in reasonable agreement with the existing data.

KEYWORDS

CFD; Pebble Bed Modular Reactor (PBMR); pressure drop; turbulence; turbulence models; fluid flow

INTRODUCTION

High Temperature Gas-cooled Reactor (HTGR) is the Generation IV reactor concept in the nuclear industry. Pebble Bed Modular Reactor (PBMR) is a HTGR with enriched uranium dioxide fuel inside graphite shells (moderator). The PBMR concept is being designed with Helium gas as the coolant which converts the thermal energy from nuclear fission of Uranium into electrical energy. The pebble bed reactor design was first introduced in 1985 by Sefidvash. His initial design was a modular light water reactor fluidized bed [2]. Currently, there are two Pebble Bed designs that are being developed in the world. A 10 megawatt prototype reactor in China and a modular pebble bed reactor in South Africa with a rated capacity of 165 MWe. The core of PBMR consists of approximately 360,000 spherical fuel pebbles distributed randomly. The PBMR design is being recognized for commercial energy usage worldwide due to its specified inherent safety features and high operating temperature.

In the PBMR the fluid flow inside the core is strongly dependant on the packing of the spheres which is random. This makes it hard to predict the flow structure in the PBMR because of unknown sphere distribution and the area of contact among the spheres [3]. In this study two existing turbulence models are used and analyzed for estimation of fluid flow in packed bed geometries. The pressure drop data from each of the methods is compared with the existing empirical/semi-empirical correlations. Finally, the fluid flow distribution obtained from the models is analyzed because it is determined to be extremely important in the PBMR core from a safety perspective [4].

MESH GENERATION

In this study two different designs are considered, simple cubic and body centered cubic. For both the designs only fluid flow analysis is done. The mesh for the fluid region in the simple cubic design is shown in Fig.1. The symmetric design consisted of a 5x5x5 grid of pebbles of 60 mm diameter. For grid independence study six different meshes were considered, out of which the mesh with 10 million cells was used for flow and data analysis**.**

For the Body Centered Cubic (BCC) design a 3x3x4 grid of pebbles was used as shown in Fig.2. The corresponding fluid region for the BCC design is shown in Fig.3

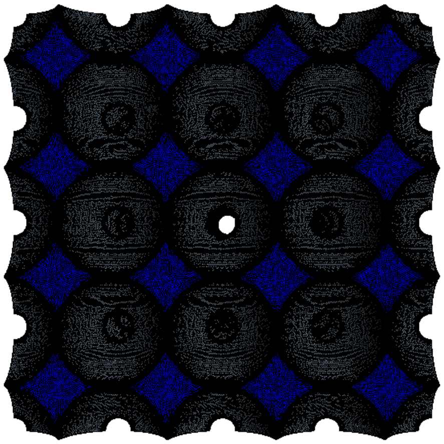

Figure 1: MESH SCENE FOR SIMPLE CUBIC GEOMETRY

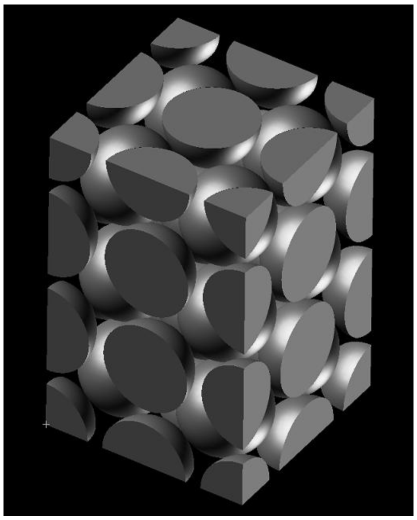

Figure 2: BODY CENTERED CUBIC GEOMETRY

The small circles in the fluid region seen in Fig.3 represent the sphere contact points; this was done for meshing convenience. A total of three meshes were developed for the BCC geometry. Table.1 shows the number of cells in each mesh divided in two parts, namely, fluid region and extrusion. Fig. 4 shows both the parts in a mesh scene of the geometry. The fluid region represents the region containing spheres (top) and the extrusion in the extension of the outlet face (bottom). Extrusion was added in the BCC design in order to avoid reversed flow at the outlet.

The reason for using three meshes with a significant difference in the total number of cells was to obtain the ratio of 1.3 or more between the average cell size of consecutive meshes. The number 1.3 is required in order to use Richardson extrapolated result in comparison with the result from different meshes [5]. The data from this analysis is used for uncertainty estimation due to discretization in CFD calculations. In this experiment the ratio of 1.3 between consecutive meshes was not achieved since produced meshes were unstructured. Nonetheless, for mesh sensitivity the guidelines specified by Celik were followed [5].

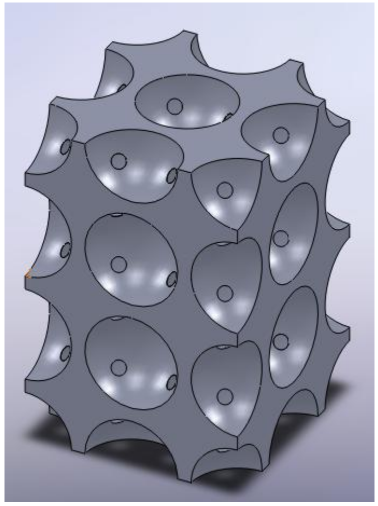

Figure 3: FLUID REGION FOR BCC GEOMETRY

Table 1: BCC MESH SPECIFICATIONS

Mesh	Fluid Region	Extrusion	Total Number of Cells
1	4,829,682	799,980	5,629,662
2	7,862,464	2,625,000	10,487,464
3	14,483,694	3,882,720	18,366,414

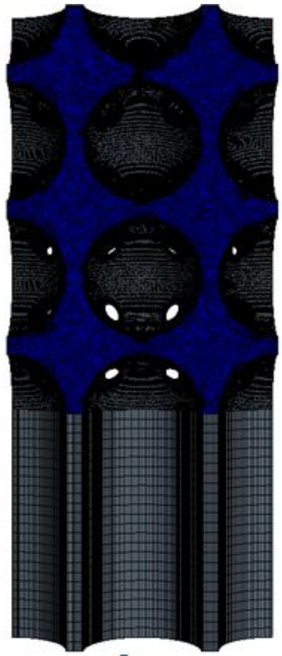

Figure 4: BODY CENTERED CUBIC MESH

Mesh sensitivity study was done at the highest Reynolds number studied in this analysis, which is 50,000. As mentioned earlier the small circles that are seen in Fig. 1, 3 and 4 are the sphere contact points. Fig. 5 shows the mesh scene of these contact points. The reason for this modification in meshing was to make meshing easier near the sphere-sphere contact point.

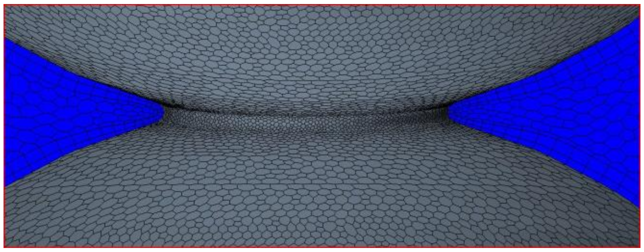

Figure 5: MESH SCENE OF SPHERE CONTACT POINT

PHYSICS CONTINUUM

The CFD modeling for all the cases used the assumption of isothermal conditions. Water with constant properties was used as the active fluid in this analysis. Realizable κ-ε Two Layer, All y+ model was used for turbulence modeling which was compared with Standard κ-ε, High y+ model.

Modified Reynolds numbers varying from 10,000 to 50,000 were simulated. Each geometry had a different porosity which was used with the modified Reynolds number to calculate the corresponding velocity for each case. Porosity is a measure of void space in a structure. In this study it is the measure of fluid volume in the geometries considered. Eq. 1 defines porosity and Eq. 2 defines the modified Reynolds number.

$$\eta = \frac{V_f}{V_t} \qquad \mathbf{1}$$

$$\mathrm{Re}_m = \frac{\rho v_s D}{\mu(1-\eta)} \qquad \mathbf{2}$$

The superficial velocity calculated from Eq. 2 can be used to calculate the fluid inlet velocity for both the cases using Eq. 3.

$$v_s A_s = v_i A_i \qquad \mathbf{3}$$

Table. 2 shows the porosity, superficial and inlet velocity for simple cubic (symmetric) geometry and Table. 3 shows the same data for BCC geometry.

Table 2: SIMPLE CUBIC DATA

Modified Reynolds Number	Superficial Velocity [m/s]	Inlet Velocity [m/s]
10,000	0.082	0.43
20,000	0.163	0.86
30,000	0.245	1.29
40,000	0.327	1.71
50,000	0.409	2.14
Porosity		0.45

Table 3: BODY CENTERED CUBIC DATA

Modified Reynolds Number	Superficial Velocity [m/s]	Inlet Velocity [m/s]
10,000	0.010	0.258
20,000	0.21	0.516
30,000	0.31	0.774
40,000	0.41	1.03
50,000	0.52	1.29
Porosity		0.303

RESULTS & DISCUSSION

The CFD simulations of each geometry show similar flow characteristics. Fig. 6 illustrates a schematic of the vortices formed in the symmetric geometry. This result is vital from a nuclear design perspective. Since this would lead to formation of hot spots in the core.

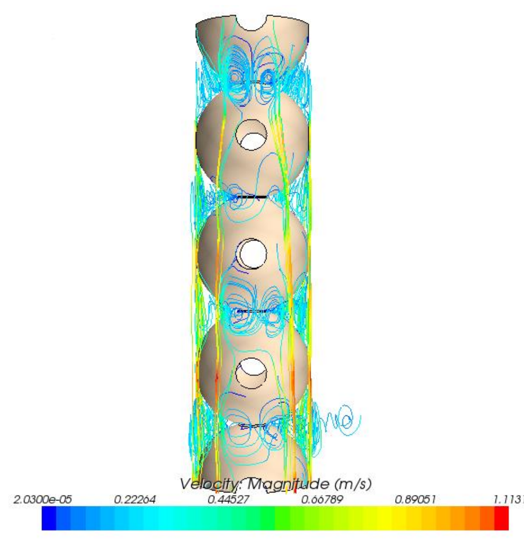

Figure 6: VORTICES FORMED IN SIMPLE CUBIC GEOMETRY

Similar phenomenon was observed in the BCC geometry. It was also observed that the vortex formation increased with the increase in the Reynolds number of the flow.

In BCC, geometry recirculation near the outlet of the simulated geometry, was encountered. In order to avoid any numerical errors due to reversed flow at the outlet, extrusion volume was added to the body centered cubic geometry, shown in Fig. 4. This was not the case for simple cubic geometry, thus no extrusion was added for the symmetric case.

Pressure drop in the reactor core is another important parameter when it comes to nuclear reactor design. Various empirical and semi-empirical correlations exist for such geometries. Fig. 7 compares the pressure drop obtained from computational analysis with the KTA correlation and Choi correlation [6].

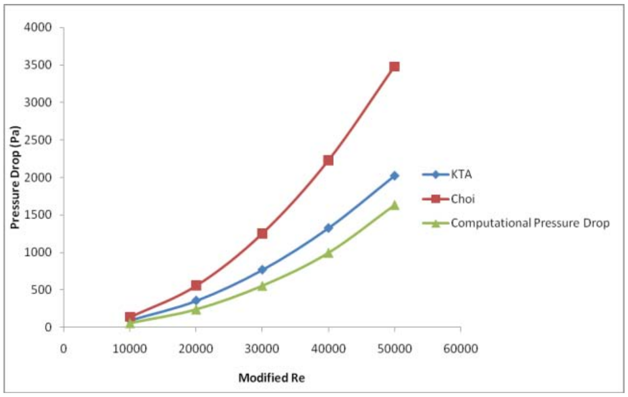

Figure 7: SYMMETRIC PRESSURE DROP DATA

The actual PBMR design being developed in South Africa uses the KTA correlation for pressure drop estimation in the reactor core. For the simple cubic geometry the computational data has similar trend as the KTA correlation. Fig. 8 shows the pressure drop for the staggered geometry with KTA and Choi correlation, here apparant discrepancies were seen between various correlations and the computational result.

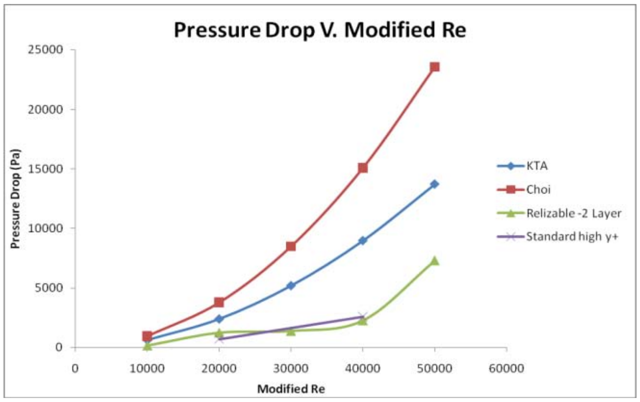

Figure 8: STAGGERED PRESSURE DROP DATA

In Fig. 8 Standard κ-ε model is also compared with the Realizable κ-ε model for the flow rate where Realizable κ-ε model tends to plateau. However, the fluid flow flow structure for both cases was different. Fig. 9 and Fig. 10 compare the outlet velocity profile for Standard κ-ε and Realizable κ-ε.

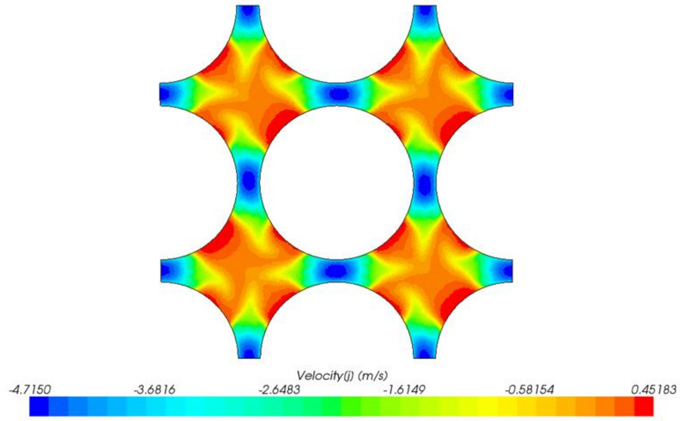

Figure 9: OUTLET VELOCITY FOR STANDARD κ-ε MODEL

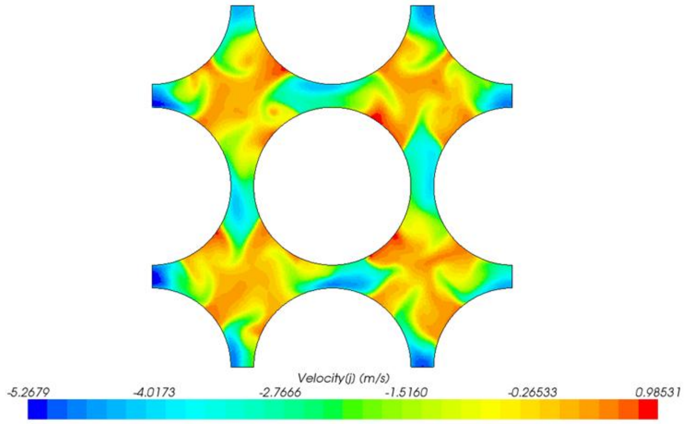

Figure 10: OUTLET VELOCITY FOR REALIZABLE κ-ε MODEL

As seen from the Fig. 9 and 10, the outlet velocity profile for the two models is not similar. In fact, for the Realizable κ-ε model the outlet velocity keeps fluctuating. This is because the y+ for the 18 million mesh size was between 7 and 9 for both the cases. This value of y+ resulted in a convergence issue with the Realizable Two layer model. Therefore, for the Standard κ-ε model high y+ model was chosen, this resulted in steady and symmetric outlet velocity profile. Although, the Standard κ-ε seems to show better velocity profile and converges to a constant value at the outlet, it does not attempt to resolve all the scales of turbulence. In the Realizable κ-ε Two layer model the coefficient of dissipation is calculated from the flow characteristics whereas in Standard κ-ε the coefficient is assumed to be a constant. Thus, from a physics standpoint the Realizable κ-ε can be expected to predict the flow better than Standard κ-ε. The velocity profile in the Realizable case does not converge because it is trying to resolve for all the scales of turbulence up to the y+ value of less than 5 (viscous sub layer).

Both the κ-ε models assume isotropic turbulence, which is not true aswell. The y+ value between 7 and 9 is not good for analysis because it is in the log-law of the wall region. Thus, mesh refinement is needed to reduce the y+ value in order to resolve the velocity profile completely. This could not be done due to limited computer resources.

CONCLUSION

The presence of vortices in the geometries considered indicates existence of hot spots in the actual PBMR design. This is important because vortices can lead to phenomenon like hot spots and particle deposition. The κ-ε model gives a reasonable approximation of the pressure drop, but for a detailed analysis it is not appropriate. Better turbulence modeling such as Reynolds Stress Transport modeling or LES is required to achieve realistic results. The meshed geometry needs to be refined, especially near the wall. This would help reduce the value of y+ which would help resolve more scales of turbulence.

NOMENCLATURE

η	-	Porosity
V_f	-	Volume of Fluid [m^3]
V_t	-	Total Volume [m^3]
Re_m	-	Modified Reynolds Number
ρ	-	Coolant Density [kg/m^3]
v_s	-	Superficial Coolant Velocity [m/s]
v_i	-	Coolant Inlet Velocity [m/s]
A_s	-	Superficial Surface Area [m^2]
A_i	-	Flow Inlet Area [m^2]

ACKNOWLEDGMENTS

I would like to thank Angelo Frisani for his guidance on certain technical details.

REFERENCES

[1] W. K. In, W. J. Lee, Y. A. Hassan, “CFD Simulation of a Coolant Flow and a Heat Transfer in a Pebble Bed Reactor”, HTR2008-58334, September 28-October 01, Washington, DC, USA.

[2] SAFIDESH, FARHANG, “Status of the small modular fluidized bed light water nuclear reactor concept”, Nuclear Engineering and Design, Vol. 167, 203–214, 1996.

[3] M. H. Kim, H. S. Lim, W. J Lee, “Computational Fluid Dynamics Assessment of the Local Hot Core Temperature in a Pebble-Bed Type Very High Temperature Reactor. Journal of Engineering for Gas Turbines and Power, Vol. 131, 012905-1, 2009.

[4] E. E. DOMINGUEZ-ONIVEROS, C. ESTRADA-PEREZ, Y. A. HASSAN, “Measurements of Flow Modification by Particle Deposition Inside A Packed Bed Using Time-Resolved PIV”, HTR2008-58330, September 28-October 01, Washington, DC, USA.

[5] CELIK, I.B., GHIA, U., ROACHE, P.J., FREITAS, C.J., COLOMAN, H., RAAD, P.E., “Procedure of Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications”, Journal of Fluids Engineering, Vol. 130, Issue 7, 2008.

[6] CHOI, YONG SOEK, KIM, SUNG JIN, and KIM, DUCKJONG, “A Semi-empirical Correlation for Pressure Drop in Packed Beds of Spherical Particles”, Transport in Porous Media 75(2), 133–149, November 2008.