Comparison of Experimental Power Reactor Hot Gas Chamber Geometry for Hot Gas Mixing Optimisation

R Andika Putra Dwijayanto and Muhammad Subekti
Centre for Nuclear Reactor Technology and Safety - BATAN
Building No. 80 PUSPIPTEK Area, Serpong, South Tangerang, 15310, Indonesia
putra-dwijayanto@batan.go.id, subekti@batan.go.id

Abstract. Experimental Power Reactor (Reaktor Daya Eksperimental/RDE) is BATAN’s proposed demonstration nuclear power reactor based on high temperature gas-cooled reactor (HTGR) design. As such, RDE faces an issue concerning severe temperature deviation in its hot coolant gas channel. Hot gas chamber, thus, is a necessary component to reduce coolant gas temperature deviation into acceptable homogenisation level. A preliminary simulation of hot gas mixing in a different hot gas chamber designs using computational fluid dynamics (CFD) method is performed to study the optimum geometry for homogenising the coolant gas. Of the three designs simulated, the “slope” geometry is found to be performing best in homogenising the RDE hot coolant gas. Pressure drop of each design is also discussed.

Keywords: RDE, HTGR, hot gas chamber, temperature homogenisation

1. Introduction
BATAN is proposing to build a demonstration nuclear power reactor, which is called Experimental Power Reactor (Reaktor Daya Eksperimental, RDE). It is a Generation IV reactor based on High Temperature Gas-cooled Reactor (HTGR) design, which characterised by strong passive safety features and operability at higher temperature compared to conventional light water reactors. The RDE employs multi-pass loading pebble bed-type fuel and cooled by helium gas. The coolant gas moves downward from the upper plenum into the core, heated to an average temperature of 700°C and then flows into the steam generator [1-4]. The simulation of RDE could be shown by RDE simulator [5]

Due to the nature of neutron flux within the reactor core, the coolant gas temperature is not evenly distributed. Higher neutron flux in the centre region means the gas is hotter in the centre of the core and relatively colder in the outer region. Subsequently, severe temperature deviation occurs in the heated coolant gas. As reference [6,7] show, the deviation can reach 120-210°C. Such large deviation can compromise plant safety due by damaging heat exchanger and steam generator, owing to alternating thermal loads.

To ensure operational safety as well as components longevity, the temperature deviation must be reduced until acceptable level before entering hot gas duct. It is done by mixing the gas inside a hot gas chamber located on the bottom of reactor pressure vessel [6]. The heated gas enters the hot gas chamber through numerous small holes and subsequently mixed.

Hot gas chamber by itself is an annular flower-shaped block separated into eight unevenly distributed “chamber” sections. Usually, the chamber sections are shaped with arc-like feature to smooth the gas
flow, therefore increasing mixing profile. However, other shapes may also be used in the chamber [6]. What should be considered is its temperature homogenisation level when using certain chamber shape.

A preliminary study is carried out to compare temperature homogenisation performance on different hot gas chamber geometries. This study is conducted using three dimensions (3D) calculation based on computational fluid dynamics (CFD) method to obtain a detailed 3D flow of coolant gas [8]. FLUENT software to analyse temperature homogenisation as well as pressure drop on each of the geometry.

2. Theory
The coolant gas flows downward from the reactor core into bottom reflector through dozens of small holes, where the gas is mixed but not homogeneous enough to be flown directly into hot gas duct. Hot gas chamber works as a secondary gas mixer to further homogenise the coolant gas until it reaches acceptable limit. Partially homogenised coolant gas enters the hot gas chamber through another numerous small holes, where it will be further mixed.

To describe gas temperature homogeneity, we use a dimensionless unit called Temperature Homogeneity Degree (THD), the same unit with our other paper [9]. It is described using the following equation,

\[ THD = 1 - \frac{\Delta T_{out}}{\Delta T_{in}} \] (1)

where \( \Delta T_{in} \) denotes temperature difference between hot and cold gas in inlet channels (from bottom reflector) and \( \Delta T_{out} \) denotes temperature difference between hot and cold gas in the end of hot gas duct, right before entering steam generator. For its range, we use scale 0% to 100%, where 0% implies that there is no temperature homogeneity and 100% implies perfect temperature homogeneity.

From reference [6], it is understood that the temperature deviation must not exceed 15°C. Therefore, the hot gas chamber design must be able to fulfil this safety criterion. There is no fixed translation of this criterion into THD, since it depends on the temperature deviation in inlet coolant channels.

The geometrical shape of hot gas chamber also influences the THD value. Gas mixing behaviour within the “chamber” depends on the gas flow, which itself is influenced by the shape of space where it flows. Annular channel width may influence the mixing behaviour as well, but that is outside the scope of this study.

Another factor in determining THD value is gas flow rate [6]. Fluid flow is a factor of Reynolds number, especially a turbulent flow. Reynolds number is expressed by the following equation,

\[ Re = \frac{\rho v R_h}{\mu} \] (2)

Where \( v \) is mean fluid velocity and \( R_h \) is hydraulic radius of the fluid channel. \( Re < 500 \) is categorised as laminar flow, whilst \( Re > 12500 \) is categorised as a turbulent flow. Between them is considered transition flow [10,11].

Since the other parameters are constant, \( R_h \) may be the deciding parameter in determining Reynolds number, which in return determining THD value. Despite previous scaled-down experiments concluded that Reynolds number makes little significance to temperature homogenisation [6,7,12], more recent CFD simulation showed that it gives larger impact [13]. However, there is yet any CFD simulation to compare influence of geometrical shape of hot gas chamber in accordance to coolant gas temperature homogenisation.

3. Methodology
The code chosen to simulate the CFD problem is FLUENT. It is accompanied by Gambit to design its geometry, creating mesh and defining parameters. The geometrical design is simplified, consisting only the fluid flow.
FLUENT solves the CFD problem using Navier-Stokes equation shown in the following [14,15].

\[
\frac{DV_i}{Dt} = \nabla \cdot \mathbf{v}_i + V_j \frac{\partial v_i}{\partial x_j} = -\frac{\partial \psi}{\partial x_i} - \alpha \frac{\partial \psi}{\partial x_i} + \alpha \frac{\partial}{\partial x_j} \left\{ 2\mu \left[ e_{ij} - \frac{1}{3} \nabla \cdot \mathbf{V} \right] \right\} \\
+ \alpha \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \psi}{\partial x_k}
\]

(3)

Helium is chosen as the working fluid; its properties are adjusted to the operational parameters. Other operational conditions, convergence criteria, initial value of iteration and other input parameters are also determined.

The scope of geometrical design is in accordance with typical HTGR hot gas chamber, albeit slightly simplified. The design consists of inlet coolant channels (both from bottom reflector and pebble fuel channel), the “chambers”, annular alley and hot gas duct. Of these, the “chambers” design will be varied as this section is where gas mixing mainly occurs.

There are three “chambers” designs which gas mixing behaviour will be studied; simple box shape, box with arc-like feature and sloped box. They will be referred as “box”, “arc” and “slope” respectively. “Arc” chambers are usually used in HTGR-type to the arc is supposed to smooth the gas flow [11], thereby increasing gas mixing profile. Here we will attempt to check whether “arc” design is really better than other possible designs or otherwise.

Gambit software is used as a means of designing the geometrical model of hot gas chamber fluid flow. Top view of fluid flow model is shown in the following.

Figure 1. Top view of fluid flow modeling in Gambit.

The mesh for each design was generated using tetragonal type mesh. The mesh number is different on each design: 740 thousand mesh for “box”, 678 thousand mesh for “arc” and 640 thousand mesh for “slope”. All numbers are approximate. First order upwind is set as interpolation format.

The holes above the “chambers” are treated as inlet channel, as well as main pebble fuel channel. It is assumed that the gas flow in pebble fuel channel is not obstructed by pebbles, giving more conservative assumption of gas flow pattern compared to real operational condition. The gas temperature on each inlet channels are set in descending order from the innermost channel to outermost channel. The far end of hot gas duct is treated as output channel. The rest are treated as wall; graphite for hot gas chamber wall and steel for hot gas duct wall.

The temperature deviation between innermost and outermost channel is set at 200°C. This value is still within the limit mentioned in reference [4]. Gas flow leakage is assumed at 14%.

The following table shows the input parameters for the hot gas chamber model. Other parameters not mentioned in this table are determined within FLUENT software.
**Table 1. CFD simulation parameter**

| Parameter                                                      | Value  |
|---------------------------------------------------------------|--------|
| Fluid type                                                   | Helium |
| Pressure (MPa)                                               | 3      |
| Diameter of hot gas duct (cm)                                 | 30     |
| Diameter of “chambers” inlet gas channels (cm)                | 4      |
| Diameter of pebble fuel channel (cm)                         | 50     |
| Fluid flow rate (kg/s)                                       | 4.27   |
| Inlet gas temperatures range (°C)                            | 600-800|
| Turbulence model                                             | k-epsilon |

**4. Result and Discussions**

CFD simulation to determine THD value of each “chamber” design is performed using aforementioned fluid flow-only model. As a comparison, “slope” design reached convergence with the lowest iteration, around 930 iterations. Meanwhile, “box” shape needed the most iteration to reach convergence, around 1950. “Arc” design reached convergence after around 1300 iterations. This difference may be influenced by the gas flow pattern within the “chambers”. For each design, temperature homogeneity and pressure drop will be evaluated based on CFD simulation result.

Comparison of temperature profile within the selected “chamber” designs are shown in figure 2.

Judging from the figure above, it can be seen that the gas flow pattern in “box” design is the least smooth. This may explain why it took the longest iteration to reach convergence, since the flow pattern is relatively chaotic within the “box”. However, this not necessarily implies that “box” design has the lowest THD value, which will be shown later.

All three geometries show remarkable temperature homogeneity. It is interesting that, despite the apparent chaotic flow, the “box” geometry actually performs quite well in mixing the hot and cold gas. Quite the opposite, in spite of apparently smoother gas flow, “arc” geometry performs least well compared to the other two. This may be explained by the existence of arc-like turbulent flow exists within “box” shape.

The hottest gas entering the hot gas chamber from the innermost bottom reflector channel, instead of creating

Bottom view of temperature mixing profile in the selected designs are shown in figure 3.
Despite seemingly huge difference in mixing behaviour shown in figure 3a and 3b, the flow pattern into the annular alley is relatively the same. The major difference is that gas holdup occurs in “box” design, caused by considerably chaotic gas flow owing to its shape. This does not occur in “arc” design, due to its arc-contour smoothen the gas flow. However, it should be noted that the occurrence of gas holup in certain geometry does not necessarily mean its pressure drop is higher.

On the other hand, figure 3c shows that gas flow pattern in “slope” design is even smoother than the other two. Hotter, unmixed gas is less apparent when it enters the annular alley, so does gas turbulence in the alley. The coolant gas is mixed considerably better when entering hot gas duct.

Although gas mixing also occurs in hot gas duct, the mixing is not particularly significant. Since most of mixing has already occurred in both “chambers” and annular alley, only little mixing left to do in hot gas duct.

Cross-sectional temperature profile of each design is shown in the following figures.

**Figure 3.** Bottom view of total temperature profile of selected hot gas chamber design; (a) box; (b) arc; (c) slope.

**Figure 4.** Cross-sectional temperature profile of “box” design; upper region (left), middle region (middle) and lower region (right).
Figure 5. Cross-sectional temperature profile of “arc” design; upper region (left), middle region (middle) and lower region (right).

Figure 6. Cross-sectional temperature profile “slope” design; upper region (left), middle region (middle) and lower region (right).

THD value is determined from the difference between hot and cold gas temperature in the inlet and outlet. Consistent with the apparent figure, “slope” design has the best THD value, reaching 99.85% in the process. Surprisingly, “arc” design is the worst performer, despite the difference with second-best “box” design is so small that can be practically indistinguishable.

All three designs studied have THD value more than 99%, a remarkably high value. However, compared to similar CFD simulation previously performed [11], these values are considerably larger. It should be noted that previous simulation was performed using more detailed design, and consequently more mesh. Interpolation format used may influence the result as well.

It has been theorized in our previous paper that addition of pebble bed channel as gas inlet may result in lower THD value. Here it is proven true, as the “box” design only reach THD value of 99.27%. Slightly lower than THD value of pebble bed channel-less design used in our previous work, albeit not particularly significant.

Temperature profile of each design are provided in Table 2.
Table 2. Calculation result of temperature profile of each hot gas chamber design.

| Design | Channel | Maximum temperature (°C) | Minimum temperature (°C) | THD   |
|--------|---------|--------------------------|--------------------------|-------|
| Box    | Inlet   | 800                      | 600                      | 99.27%|
|        | Outlet  | 700.693                  | 699.231                  |       |
| Arc    | Inlet   | 800                      | 600                      | 99.24%|
|        | Outlet  | 700.71                   | 699.19                   |       |
| Slope  | Inlet   | 800                      | 600                      | 99.85%|
|        | Outlet  | 700.125                  | 699.833                  |       |

Pressure drop is also discussed in this paper. Naturally, significant pressure drop occurs when the coolant gas enters the “chambers” and when it enters the hot gas duct from the alley, as shown in the figures below. No significant pressure drop in the duct, although pressure homogeneity itself is less apparent in the “slope” design.

The “slope” design also has the largest pressure drop when the gas enters the “chambers” dan annular alley, also its pressure profile is less heterogeneous in the hot gas duct. This may be due to its gas mixing profile. Nevertheless, this occurrence does not particularly represent larger pressure drop, since we should account other factors as well.

Figure 7. Total pressure profile of hot gas chamber designs; “box” design (top), “arc” design (middle) and “slope” design (bottom).

Despite apparently larger pressure drop in the “chambers”, the “slope” design has the lowest pressure drop compared to the other designs. Slope “chambers” shape may have a part in lowering the total pressure drop, by directing the flow straight into the alley while simultaneously homogenise the gas well. Meanwhile, “box” design and “arc” design have relatively similar pressure drop, consistent with their similar gas mixing performance.
When compared to experiment conducted in reference [11] with similar gas flow rate, all three designs studied in this paper have lower pressure drop, about one-third of that previous experiment. It should be noted that this paper employs relatively simpler design, less meshing and other different parameters from reference [11], so that future simulations with better defined parameters are necessary.

Table 3. Pressure profile of hot gas chamber designs.

| Pressure       | Box     | Arc     | Slope    |
|---------------|---------|---------|----------|
| Inlet (average)| 4455.193| 4460.175| 4162.09  |
| Outlet        | 979.352 | 980.547 | 965.909  |
| Pressure drop  | 3475.841| 3479.628| 3196.181 |

5. Conclusion
The comparison of hot gas chamber design is meant to understand the temperature homogenisation behaviour of coolant gas within the hot gas chamber. By using FLUENT software, CFD simulation is conducted to figure out their temperature and pressure profiles. The simulation result shows that “slope” design is the better performer to homogenise the coolant gas. It has highest THD value and lowest pressure drop. Meanwhile, “box” design and “arc” design have similar homogenisation performance. This finding may raise the question on necessity of arc-like shape in the hot gas chamber. However, further researches are required to confirm this finding.

The three designs studied in this paper all have large THD value, thereby fulfilling safety criteria. Thus, it may be sufficient to use simple “box” design for RDE hot gas chamber. Although the “slope” design performs best, the difference is not significantly high and well within safety limit. Design improvement and more detailed simulation parameters are needed for the future researches to obtain more precise data.

Acknowledgments
This work presented in this paper was funded by the government of Indonesia through the DIPA of the Centre for Nuclear Reactor Technology and Safety (PTKRN), National Nuclear Energy Agency (BATAN) and research grant from Ministry of High Education and Science Technology of Indonesia.

References
[1] Suwoto, Adrial H, Setiadipura T and Zuhair 2016 Sigma Epsilon 20 pp 64-72
[2] Cahyono A, Handono K, Prasaja S T and Atmoko D F 2016 Jurnal Perangkat Nuklir 10 pp 80-89
[3] Husnayani I and Udiyani P M 2018 Tri Dasa Mega 20 pp 69-76
[4] Setiadipura T, Bakhri S, Sunaryo G R and Wisnubroto D S 2018 AIP Conference Proceedings 1984 020034
[5] Subekti M, Bakhri S and Sunaryo G R 2018 IOP Conf. Series: Journal of Physics: Conf. Series 962 012054
[6] Yao M S, Huang Z Y, Ma C W and Xu Y H 2002 Nuclear Engineering and Design 218 pp 233-240
[7] Damm G and Wehrlein R 1992 Nuclear Engineering and Design 137 pp 97-105
[8] Subekti M 2016 KnE Energy and Physics [S.I.] pp 75-85
[9] Dwijayanto R A P and Subekti M 2019 IOP Conf. Series: Journal of Physics: Conf. Series 1198 022019
[10] Son H M, Yang S H, Park C and Lee B C 2015 Annals of Nuclear Energy 75 pp 44-53
[11] Salama and El Morshedy 2012 Progress of Nuclear Energy 55 pp 78-92
[12] Inagaki Y, Kunugi T and Miyamoto Y 1990 Nuclear Engineering and Design 123 pp 77-86
[13] Zhou Y, Li F, Hao P, He F and Shi L 2014 Nuclear Engineering and Design 271 pp 510-514
[14] Liu C C, Feng Y M and Shih C K 2012 Applied Thermal Engineering 40 pp 389-396
[15] Cham C H and Yoneda T 2012 Methods Application Analysis 19 pp 211-242