Analysis of coolant flow distribution to the reactor core of modified TRIGA Bandung with plate-type fuel

V I S Wardhani1, J S Pane1 and S Dibyo2

1Center for Applied Nuclear Science and Technology - National Nuclear Energy Agency (BATAN) Jl. Taman Sari No. 71, 40132, Bandung, Indonesia
2Center for Nuclear Reactor Technology and Safety - National Nuclear Energy Agency (BATAN), Puspiptek Area, Gd.80 Setu, Tangerang Selatan, 15310, Indonesia

E-mail: vero@batan.go.id

Abstract. A conversion of the TRIGA reactor Bandung has been planned by replacing fuel from rod type to plate type. The fuel replacement causes the fuel arrangement or reactor core configuration should be changed as well as the cooling system. Currently, the reactor cooling system takes place by natural convection with the flow direction from the bottom to the top of the core. The replacement of plate-type fuel with a small distance between plates causes the core cooling process to decrease, therefore to make more effective the forced convection cooling system with the direction of upward flow is required. It is expected that the cooling process will increase. The ability of the cooling system depends on flow distribution in the core at the time of reactor in operating, therefore it is necessary to analyze the distribution of coolant flow and the velocity distribution of the cooling system from the top reactor pool to the reactor core to determine that the coolant flow from the inlet pipe mostly goes to the core. The purpose of this research is to analyze the distribution of coolant flow and flow velocity to the reactor core. The analysis was carried out in a simulation using computational fluid dynamics FLUENT software by creating reactor core modeling. The result shows that the cooling water flows from above along the reactor pool, then the stored energy is reduced so that the flowing slowdown. In this case, resulting in evenly flow distribution to the reactor core.

1. Introduction

The cooling process in the TRIGA reactor core takes place by natural convection, the natural convection process between the fuel element and the coolant flow in the core occur because the fuel arrangement of the core consists of cylindrical fuels that have relatively wider flow area of cooling channels [1]. The planning to change fuel from cylindrical type to plate type will change the cooling system as well as the reactor core cooling channel [2]. The higher cooling flow velocity is required to obtain better heat transfer in the core. Therefore, the core cooling system uses the downward flow of forced convection cooling mode [3]. The direction of the coolant flow changes from the bottom of the core upward to the downward flow. In this case, the analysis of coolant flow distribution of and flow velocity to the reactor core is important to be carried out. The ability of the cooling system depends on flow distribution in the core at the time of reactor in operating, therefore it is necessary to analyze the distribution of coolant flow and the velocity distribution of the cooling system from the top reactor pool to the reactor core to determine that the coolant flow from the inlet pipe mostly goes to the core.

The pattern of coolant flow distribution and flow velocity requires the analysis of fluid dynamics so that the coolant flow distribution to the core can be predicted [4, 5]. This simulation will be conducted...
to determine the distribution of coolant flow from the primary cooling inlet pipe to the reactor core. It is all of the flow distribution that will enter the reactor core in evenly flow and the cooling effect is expected. The reactor core consists of a plate arrangement in the fuel bundle, between one plate and another having a narrow gap. The process of forced convection heat transfer that occurs will result in increased flow velocity after passing through the reactor core, therefore it is necessary to analyze fluid dynamics including flow distribution and fluid velocity distribution. The analysis will be done by simulating the coolant flow in the reactor pool as well as the reactor core using the computational fluid dynamics FLUENT software [6, 7]. The results of this simulation are expected that flow patterns will be obtained that can describe the condition of the movement of the cooling fluid from the inlet pipe to the reactor core, so that it can be seen that the reactor core cooling system is effective or not.

2. Theory

Every stream that occurs in nature must fulfill the principle of mass survival, according to this principle the mass cannot be destroyed or created and always a balanced calculation can be made and is called mass balance. The process of heat transfer is inseparable from the movement of fluid flow around the surface of the heat transfer, both laminar and turbulent flow [8, 9]. To find out the cooling inside the reactor core, a thermal-hydraulic analysis is required. An analysis to determine the phenomenon of flow distribution that occurs at the top or bottom of the reactor core, the distribution of the coolant flow velocity is difficult to do because there are no measuring devices that can be placed inside the reactor core, therefore it is necessary to simulate using a Computational program Fluid Dynamics with mathematical solutions in the formulation of fluid dynamics equations that are used to describe the phenomenon of fluid flow [10]. The Computational Fluid Dynamics program is able to predict fluid dynamics, heat transfer, chemical reactions and other phenomena by solving mathematical equations in particular solving Navier-Stokes equations [11, 12].

\[
\frac{DV_j}{Dt} = \frac{\partial V_j}{\partial t} + V_j \frac{\partial V_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} - \alpha \frac{\partial P}{\partial x_i} + \alpha \frac{\partial}{\partial x_i} \left\{ 2\mu \left[ e_{ij} - \frac{1}{3} (\nabla V) \right] \right\} 
\]

(1)

The Navier-Stokes equation is a non-linear second-order partial differential equation for fluid dynamics, this equation can be solved using Computational Fluid Dynamics software using three conservation laws, namely conservation of mass (continuity), conservation of momentum and conservation of energy.

- Equality of mass (continuity)

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = S_m 
\]

(2)

- External momentum equation

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_i + F_i 
\]

(3)

- Energy conservation equation

\[
\frac{\partial}{\partial t} (\rho h) + \frac{\partial (\rho u_i h)}{\partial x_i} = \frac{\partial}{\partial x_i} (k + h) + \frac{\partial}{\partial x_i} \sum_j h_j + \frac{Dp}{Dt} \frac{\partial u_i}{\partial x_i} + S_h 
\]

(4)
In solving the Navier-Stokes equation, this software is able to create a form of modeling that is appropriate to the actual situation for analysis.

3. Methodology
The performance of the cooling system in the reactor core is one of the factors that must be considered to guarantee safe operation of the reactor, therefore it is necessary to analyze the prediction of the distribution of coolant flow starting from entering the reactor pool to the reactor core by taking the heat energy from the fuel element. In this analysis, a Computational Fluid Dynamic is used, this program helps to solve mathematical equations that formulate the process of fluid dynamics in describing the phenomenon of fluid flow that occurs by making a modeling geometry that matches the actual state of both the shape and dimensions for simulation. The results obtained in this simulation are in the form of data, images and curves that show predictions of system reliability in modeling. As a first step, a modeling image of the reactor tank and reactor core will be made with a 1981 mm tank diameter and 6377 mm height as shown in Figure 1. The reactor core is composed of plate-type fuel bundles with 868 mm length, 353 mm width and 402 mm thickness. While the cooling fluid enters through the inlet pipe with a diameter of 6 inches (152.4 mm) as shown in Figure 2.

![Figure 1. Reactor pool.](image1)

![Figure 2. Reactor core.](image2)

The next step is to do the analysis by making modeling geometry for simulation, in order to obtain accurate computational results and approaching the actual conditions it is necessary to make meshing by dividing into small elements in which each element is considered to have the same physical properties. Figure 3 and Figure 4 depicts the geometric data modeling, Figure 5 and Figure 6 describe the meshing. The size of the mesh must be arranged in such a way as to obtain precise results and the required computing power is not too large.
Figure 3. Plate type fuel geometry.

Figure 4. Geometry modeling.

Figure 5. Point meshing.

Figure 6. Meshing.

The calculation begins by providing a boundary condition in the form of a coolant flow rate from the primary cooling pump and enters the reactor pool tank through the inlet pipe. Besides that, in this case the boundary conditions, physical properties of the material and physical properties of the water used are pure water.

4. Results and discussion

The analysis was performed using computational fluid dynamics software. As explained in the previous description, an analysis of the flow model, especially focused on the coolant flow to the reactor core. In this way, the distribution and flow velocity in the geometry of the test model can be obtained. This is intended to see the trend of fluid flow distribution so that it can be estimated how effective the inlet is cooling flow used in the reactor core. The limitation of the problem carried out in this analysis is that the geometry of the modeling is still limited to using cladding where the plates type fuel, while it does not involve the existence of the plate fuel itself. The next limitation of the problem is that it does not involve heat exchange in the process of coolant flow. In this case the coolant used is pure water with the characteristics as stated in the software, which includes the fluid density of $1,225$ kg/m$^3$ with a viscosity of $1.7894 \times 10^{-5}$ kg/m-s. The parameter of the inlet coolant mass flow rate is $50$ kg/s. The modeling geometry as shown in figure 4. By looking at the modeling geometry which is quite complex, the type of grid/meshing used was the unstructured grid. The grid points and grid shapes in the model geometry can be seen in Figure 5 and Figure 6. The number of iterations as residual functions can be shown in figure 7.
Coolant flow condition and velocity distribution in the geometry model can be seen in figure 8 below.

Furthermore, coolant flow conditions and flow rates in more detail from upper to the lower reactor tank (reactor core) geometry of the model, can be seen in both figure 9 and figure 10.

Figure 7. Curve of iteration vs residual scale.

Figure 8. Velocity distribution.

Figure 9. Velocity distribution at upper reactor pool.
Coolant fluid which in this case is pure water, enters through the flow area of 6 inc (152.4 mm) diameter inlet pipe and inlet mass flow rate of 50 kg/s. The inlet pipe enters the reactor tank, with a depth of 1577 mm from the surface of the reactor pool. When the coolant flows exits from the pipe, the stored energy of the fluid is still large enough so that the flow velocity still high, approximately 4 m/s. Meanwhile the coolant flows spread in the reactor pool.

The cooling water flows down along the reactor pool, reducing the stored energy, the flow will slow down and resulting in the uniform flow to the reactor core. as shown in Figure 10, as it flows through the reactor core the flow will slightly increase because of the changing of the cross-flow area. The fluid flow velocity will greatly reduce when it flow in the tank under the reactor core and the fluid reverse flow will occur at the cross-section of the outlet.

5. Conclusion
Cooling flow simulation has been carried out in the reactor using computational fluid dynamics FLUENT software. Flow simulations do not involve the presence of fuel plates and do not consider heat exchange. Simulation results show that with the coolant mass-flow rate of 50 kg/s in the inlet pipe at 1577 mm below the surface of the reactor pool, backflow still occurs at the outlet side. The cooling water flows from above along the reactor pool, then the stored energy is reduced so that the flow is a slowdown. In this case, resulting in evenly flow distribution to the reactor core. Furthermore, the velocity will be a slight increase as the coolant flows through the reactor core because of the changing in because the changing in crossflow area in the core. For this reason, it is necessary to study further for another geometry modeling with the lower end of the inlet pipe so that the coolant flows into the reactor core is more effective with an indication of no back fluid flow at the outlet side. Further modeling simulations can be carried out by involving the presence of plate fuels and the occurrence of heat exchange in the reactor core.

References
[1] Amgad Salama, Salah El-Din El-Morshed. CFD simulation of flow blockage through a coolant channel of a typical material testing reactor core. Annals of nuclear energy. 2012; 4: 26-39.
[2] Bruce R. Munson, Donald F. Young, Theodore H. Okiishi, Wade W. Huebsch. Fundamentals of fluid mechanics. Internat. Student version. N. J. John willey & son Inc. 2010: 211-214.
[3] C.C. Liu, Y.M. Firing, C.K. Shih. CFD evaluation of turbulence models for flow simulation of the fuel road bundle with space assembly. J. Thermal engineering. 2012; 40: 389-396.
[4] David A. Randall. Navier stokes equation. Departement of atmospheric science colorado state university. ©copyright 2000: 193-206.
[5] Daxin Gong, Shanfang Huang, Guanbo Wang and Kan Wang. Heat transfer calculation on plate-type fuel assembly of high flux research reactor. Science and Technology of Nuclear Installation. 2015; 15: 1-13.

[6] Dehbi A. and Martin S.,"CFD Simulation of Particle Deposition on An Array of Spheres Using an Euler/Lagrange Approach", Jurnal Nuclear Engineering and Design, Vol.241, pp.3121–3129 (2011).

[7] Forced convection: laminar flow over on isothermal plate. Available from: http://www.efunda.com/formulae/heat_transfer/convection_forced/isothermalplate_lamflow.cfm. Accessed April 2013.

[8] Genick Bar – Meir, Ph.D. Basic of fluid mechanics version 0.3.4.0. 2013: 145-218. [8].Hyung Min Son, Soo Hyang Yang, Cheol Park, Byung Chul Lee. Transient thermal-hydraulics analysis of complete single channel blockage accident. J. Annals of nuclear energy. 2015; 75: 44-53.

[9] V.Indriati Sri Wardhani, Henky P Rahardjo, Surip Widodo. Analisis Distribusi Temperatur Kanal Terpanas Teras Reaktor TRIGA Bandung Berbahan Bakar Pelat menggunakan Program CFD. Prosiding Seminar Nasional Teknologi Energi Nuklir. 2017, 63-65.

[10] S Dibyo, K.S Sudjatmi, S Sihana, I.D Irianto, Simulation of Modified TRIGA-2000 with Plate-Type Fuel under LOFA Using EUREKA2/RR-Code, Atom Indonesia 44 (1), 31-36.

[11] C.H. Chamand, T.Yoneda, On possible isolated blow-up phenomena and regularity criterion of the 3D Navier-Stokes equation along the streamlines. Methods Appl. Anal.2012; 19: 211-242.