Computational Fluid Dynamics Simulation of Kartini Reactor with Plate Type Fuels

Tri Nugroho Hadi Susanto^{*1}, Taxwim¹, Puradwi Ismu Wahyono¹, Muhammad Subchan¹

¹ Center for Accelerator Science and Technology, Yogyakarta, 55281, Indonesia

ARTICLE INFO

Article history:

Received: 14 August 2018

Received in revised form: 01 June 2019

Accepted: 01 June 2019

Keywords:

CFD

UDF

Narrow rectangular channel

Plate fuel.

ABSTRACT

The purpose of this study is to determine the characteristics of the cooling system on the new design of the Kartini Reactor plate fuel based on numerical calculations (Computational Fluid Dynamics). The fuel plate model was simplified and made in 3D with dimensions are 17.3 mm x 68 mm x 900 mm. The space between the two plates called the narrow rectangular channels has a gap of 2 mm. On these simulations a heat flux of 10612,7 watt/m² was used which was obtained from the MCNP calculation program. Simulations were conducted in a steady state condition and single-phase model laminar flow of an incompressible fluid through the gap between the two fuel plates. This simulation uses UDF (User Define Function) to approach heat flux behavior that follows the neutron distribution in the reactor core. The simulation results show that the maximum temperature that occur at a flow rate of 0.01 m/s was 43.5 °C.

© 2019 IJPNA. All rights reserved.

1. INTRODUCTION

Kartini Reactor is a TRIGA Mark II type nuclear reactor that uses cylindrical type fuel. Based on the calculation of neutron multiplicity factors by using TRIGAP program, type of the fuel element will be available only until the year 2031, with the assumption that it critical operation for 360 hours / year. Meanwhile General Atomic, the producer of TRIGA Reactor fuel element (UzrH), consider to stop their production. Therefore, modifying a reactor core with fuel plate element (U3Si2-Al) is one of the solution to overcome fuel scarcity in future. Therefore, it is necessary to analyze the safety of a reactor, one aspect of which is the cooling of the fuel as it can affect fission reactions that generate thermal energy. Neutronic aspect analysis using MCNP program should be done to obtain heat distribution in the reactor core. The results of the heat flux distribution from MCNP program will be taken

as input data for thermo-hydraulic analysis using CFD (computational fluid dynamic).

Kartini Reactor is designed to use fuel use of cylinders for the continuity of operations in the future as an alternative source to resolve the lack of availability of fuel so the fuel plate type will be used. The main concern on the fuel plate is the narrow channel which is formed between the two plates. Information about the flow and heat distribution is needed to analyze the core design with the plate fuel.

Zheng et al (2006) has conducted research on the flow velocity profile with a rectangular microchannel of 54 µm x 19 µm x 28 mm. This experiment was conducted and compared to a full 3D theoretical solution. The velocity profile at the z-o-x direction has a tapered shape while the z-x direction has a blunt shaped.

Srivastava (2009) has conducted research using numerical simulation on the flow in narrow channel single phase. With variation of the gap size are 200 µm, 300 µm, and 500 µm.

¹ * Corresponding author. E-mail address: nugh_hs@batan.go.id

The result show that the use of CFD for analysis in the narrow gap with a good modeling approach deserves to be accepted.

Syam et al (2010) has conducted a study on numerical heat transfer characteristics of an interim storage of nuclear spent fuel. The result show that water temperature increases linearly with the higher the position of water in the fuel canal with a maximum temperature of 307 K.

Subekti et al (2013) has done a validation of FLUENT 6.3 CFD models for the calculation of the coolant flow in the narrow channel with dimensions of 1100 x 50 x 2.25 mm and made of stainless steel plate walls that are in the air environment. The model was validated using Heating-02 test equipment. Variation in parameters are flow rate and the temperature of water in the channel input. The results of the simulations and experiments showed that the maximum temperature difference at the plate wall is 6.54%.

Wahyono (2013) has conducted a simulation using the relap program to calculate the thermo-hydraulics parameters of the reactor core. In his research, a heat flux profile followed by neutron distribution in the reactor core that the distribution of neutron follows cosine function.

Ghione, et al (2016) has completed experiments with a narrow rectangular channel that has a high heat flux and variations in the cooling flow rate. The channel size is 1.509 x 47.2 x 599.8 mm and 2.161 x 47.15 x 599.7 mm. The cooling flow rate is between 0.5 to 18 m / s, the pressure ranges from 0.2 to 0.3 MPa, and the heat flux varies between 0.5 to 7.5 MW / m². This study simulates the Cathare code which compares the experimental results with simulations. The result of experimental cooling along the narrow channel shows that the coolant temperature rises linearly along a narrow channel.

2. MATERIALS AND METHODS

The model is a domain area calculation in the simulation, that refers to the case to be simulated. The model created does not have to be the same as the actual form but is made as simple as possible according to the needs of the user but already represents the real condition. For the case in this study, the model is made in

reference to the fuel plate design to be used to replace the cylinder fuel.

The size of the fuel device is 72 x 60 x 600 mm with the size of each plate is 56 x 1.3 x 600 mm. The number of fuel plates in one fuel device is 20 and the number of fuel devices inside the Kartini reactor core is 16. The configuration of the fuel plate in the fuel design of the Kartini Reactor is shown in Figure 1.

In the fuel configuration on the reactor core there is a fuel device equipped with 2 neutron absorbers. The number of fuel plates in the device are 13 pieces. There are 4 pieces of fuel device that has an absorber so that the number of regular fuel devices are 12 pieces.

The model that will be simulated as a computational domain is the simplification of 1 fuel plate device from the 21 cooler channels into 2 channels. The model size and control volume are shown in Figure 2 and 3 respectively.

Fig 1. Fuel plate configuration in the reactor core

Fig 2. Size and coordinate of model (a) vertical z direction (b) A-B cross sectional

Fig 3. Control volume plate fuel

The generation of heat in the fuel plates were calculated using MCNP program with the reactor power of 100 kW. Based on the results of MCNP calculations, it is known that the greatest power plate is equal to 713.1734 watt. While the two sides of the surface area of the fuel plate with neglecting the thickness is 0.0672 m². The value of the heat flux can be calculated as follows:

$$q'' = \frac{q}{A} = \frac{713,1734 \text{ watt}}{0,0672 \text{ m}^2} = 10612.7 \text{ watt/m}^2$$

The heat flux profile occurring in the reactor fuel follows the neutron flux and the neutron flux profile in the axial direction of the fuel plate following the cosine function. Hence UDF (user define function) is required to run the CFD fluent. The heat flux profile on the fuel plate is taken from the Wahyono (2013) research reference. Based on the data in Table 1, it can be made with a function that is curve fitting that is inserted to UDF program.

This function is used for UDF program. The simulation is done with 3 different heat flux input modes to show that the UDF program has been working in accordance with the intended purpose.

Data collection is done by making variation of coolant flow velocity on fuel plates. The temperature distribution and flow profile obtained from the simulation are taken as the study material.

3. RESULTS AND DISCUSSION

3.1 User Define Function

As mentioned, the profile of heat flux in the reactor follows the neutron distribution. hence the CFD simulation using User Define Function (UDF). The simulation is done for 3 different

heat flux input methods in order to validate that the UDF program works in accordance with the desired objectives i.e. CFD simulation without UDF, using UDF and manual method. For the manual method, the axial fuel plate is divided into 15 segments and filled with heat flux accordance to Table 1. The comparison of calculations, results in the graph form, which can be seen in Figure 4.

The temperature profile of the narrow channel without UDF shows that the temperature trend formed is linear along the gap at starting position 100 mm to 600 mm. Maximum temperature reached is 58 °C with a cooling flow rate of 0.1 m/s. In another case, the temperature profile of the narrow channel with UDF when observed at maximum temperature, was found not exceeding 44 °C with a lower cooling flow rate of 0.01 m/s while the heat flux was equivalent. This data indicates that a neutron flux profile (heat flux) that forms a cosine function provides cooling at lower temperatures in the case of a narrow channel.

Table 1. Fraction of heat flux distribution of axial direction of fuel plate (Wahyono Reference)

No. segment	Axial Heat Flux Distribution (fraction)
1	0,00000000
2	0,00104885
3	0,03418148
4	0,06592868
5	0,09318864
6	0,11410607
7	0,12725311
8	0,13173755
9	0,12725311
10	0,11410607
11	0,09318864
12	0,06277745
13	0,03418148
14	0,00104885
15	0,00000000
Total	1,00000000

Fig 4. Temperature profile of narrow channel of fuel with non UDF, UDF and manual method

It is also found that the temperature profile of the narrow channel of the fuel plate by means of manual method, produces maximum temperature output with distribution patterns almost the same as the UDF method. Thus it can be concluded that UDF applied to this case has been as expected.

3.2 Narrow Channel Flow Profile

Characteristic

Fluid velocity at the interface between the surface of the flat plate at no slip condition and fluid is zero (0) due to effect of viscous force. The effect of the viscous force is smaller if the distance from the surface to the fluid layer is greater. If the effect is lost this means the fluid layer has come out from its boundary layer area. For the case of 2 parallel plates or a narrow gap, the effect of the viscous force arises from both sides. As though in the 2 parallel plates, the viscous force of the narrow rectangular channel is affected from 4 surfaces. The simulation results of velocity profile are showed in Figure 5 and Figure 6. Figure 5 shows that the middle region has the highest velocity. The lowest velocity is located at the interface between fluid and surface.

When comparing between the two profiles at Figure 5 and 6, the narrow channel profile that is parallel to the z-x plane has a parabolic shape (taper), while the narrow channel profile that is parallel to the z-y plane has a blunt shape. The difference in profile is due to the effect of the viscous force on the fluid flow. The velocity profile of the narrow channel which is parallel with z-x plane is affected by the viscous force formed between the narrow gap wall of 2 mm with the fluid. Due to the distance, it is relatively

small, therefore effect of the viscous force still occurs until it reaches the center of the gap so that the profile has a parabolic shape. While the velocity profile of the narrow channel which is parallel to the z-y plane, is influenced by a distance as wide as 56 mm. This makes the viscosity effect only occur at a distance of 3 mm from the right and left wall. Therefore, the narrow channel profile that is parallel to the z-y plane has a blunt shape.

Fig 5. Narrow channel velocity profile in z direction and z-x plane with flow velocity 0.01 m/s.

Fig 6. Narrow channel velocity profile in z direction and z-y plane with flow velocity 0.01 m/s.

3.3 Narrow Channel Coolant Temperature Characteristic

The temperature profile in the case of the two parallel plates (narrow channel) is affected by the viscous force from both sides. Figure 7 shows the temperature profile of the narrow channel in the z-x plane. It appears that at higher temperatures are on the walls of the two sides and the lowest in the midst. In another case, the narrow channel in the z-y plane is shown in Figure 8. It can be seen that the profile is not

taper shaped (parabolic) but bluntly shaped. Both temperature profiles are inversely related to the velocity profile.

Fig 7. Narrow channel velocity profile in z direction and z-x plane with flow velocity 0.01 m/s.

Fig 8. Narrow channel temperature profile in z direction and z-y plane with flow velocity 0.01 m/s.

3.4 Flow Rate Change Effect

Cooling the reactor core is closely related to the flow rate of the coolant. The changes of the flow rate through the coolant temperature fuel plates can be seen in Figure 9. Safety limits are determined using conservative considerations based on the result. TONB (Temperature Onset of Nucleate Boiling) or DNB (Departure from Nucleate Boiling) values can be taken as a starting point for determining the values of the safety limits. TONB/ DNB are above the saturation temperature. The coolant temperature at the flow rate between 0.05 - 0.005 m/s is far from its saturation temperature of 113.35 °C. Figure 9 show that the maximum temperature at the flow rate of 0.005 m/s is approximately 59 °C. The value of the flow rate, 0.05 to 0.005 m/s is equivalent to a volumetric flow rate of cooling of 10.3 to 103.5 liters / minute for all of the fuel

plates in the reactor.

Fig 9. The influence of the flow rate to the maximum temperature

4. CONCLUSION AND REMARKS

Conclusions of the research are : a) The UDF of heat flux applied in the CFD software has been as expected; b) The temperature and flow profile of the fuel plates are appropriate with the theory; c) Variation of flow rate to fuel plate cooling on the narrow channel a nonlinear temperature changes.; d) Cooling water temperature of fuel plates are still far from saturation temperature for the flow rate in the range of 0.05 to 0.005 m/s.

REFERENCES

- Bergman T L., Lavine A S., Incropera F P., DeWitt D P., 2011, *Fundamentals of Heat and Mass Transfer* 6th Edition, John Wiley and Sons.
- Ghione, A., Noel, B., Vinai, P., & Demazière, C. (2016). International Journal of Heat and Mass Transfer Assessment of thermal – hydraulic correlations for narrow rectangular channels with high heat flux and coolant velocity. *International Journal of Heat and Mass Transfer*, 99, 344356. <http://doi.org/10.1016/j.ijheatmasstransfer.2016.03.099>
- Ismu wahyono, P., 2013, *Evaluasi Parameter Termohidrolika Teras Reaktor dengan Relap*, Pusat Sains dan Teknologi Akselerator, Yogyakarta.
- Safety Guide No. NS-G-4.4 *Standards Operational Limits and Conditions and Operating Procedures for Research Reactors*. International Atomic Energy Agency, Vienna, 2008.
- Srivastava, R.R, Schneider, N.M., Kandlikar, S.G, *Numerical Simulation Of Single Phase Liquid Flow In Narrow Rectangular Channels With Structured Roughness Walls*, *Proceedings of the Seventh International ASME Conference on Nanochannels, Microchannels and Minichannels ICNMM 2009*
- Subekti, M., Juarsa, M., Hadi Kusuma, M., 2013, *Validasi Pemodelan Kode Fluent 6.3 Untuk Perhitungan Aliran Pendingin Darurat Dalam Kanal Sempit PLTN-PWR*, *Indonesian Journal of Nuclear Science and Technology* Vol. 15, No. 2, Agustus 2014;
- Syam, N.S., Widiharto, A., Rohmat, T.A., 2010, *Kajian Numeris Karakteristik Perpindahan Panas pada Kolam Penyimpanan Sementara Bahan Bakar Nuklir Bekas MTR*, *SEMINAR NASIONAL Perkembangan Riset dan Teknologi di Bidang Industri Ke-16*, p.34-39.
- Tjiptono, T., 2015, *Kajian Keselamatan Reaktor Kartini Dengan Bahan Bakar Plat U3Si2-Al*, *Laporan Teknis, PSTA, Yogyakarta*.
- US Departemen of Energy, *Doe Fundamentals Handbook Thermodynamics, Heat Transfer, And Fluid Flow Volume 2 of 3*, Washington, D.C. 20585, 1992.
- Versteeg H K., Malalasekera W., 2007, *An Introduction to Computational Fluid Dynamics The Finite Volume Method Second Edition*, Bell & Bain Limited, Glasgow.
- Zheng X., Siber li Z H., 2008, *Measurement of velocity profiles in a rectangular microchannel with aspect ratio a=0.35*, 951–959. <http://doi.org/10.1007/s00348-007-0454-4>