**ICMIEE20-260**

# Analysis of Thermal Hydraulics Behavior of Coolant through a Sub Channel in Fuel Assembly of a Pressurized Water Reactor (PWR)

*Tariqur R Shaon, Samsus S Ahmed*, Rashem M Obaidi, Arif Hossain, Md. Ghulam Zakir*
Department of Nuclear Science and Engineering, Military Institute of Science and Technology, Dhaka-1216, BANGLADESH

**Abstract**

Nuclear power is the most promising energy resource to meet the future energy demand. Bangladesh has been reached to the new era of nuclear power generation by building Rooppor Nuclear Power Plant (RNPP). Computational fluid dynamics (CFD) is the most essential technique to measure different parameters for thermal hydraulics analysis inside the reactor. CFD methodology can be applied in investigating the detailed thermal–hydraulic characteristics of coolant through a sub channel within a fuel assembly. This paper represents the simulation of heat transfer and fluid flow in sub-channel of fuel assembly used in pressurized water reactors (PWR). Using a multiphysics software a model of sub-channel of PWR fuel assembly has been developed and different parameters such as temperature profile, velocity profile, isothermal contours and variation of Reynolds number have been analyzed as well. These CFD methodology is considered on steady-state condition inside a PWR fuel assembly.

**Key Words**

Pressurized water reactor, Subchannel. CFD, thermal hydraulics.

## 1. Introduction

Thermal hydraulics is indeed a hydraulic flow analysis of thermal fluids. A typical example of this is the flow of coolant through a sub-channel in fuel assembly, the generating of steam in power stations and the associated conversion of energy to mechanical motion and therefore the changing of water status when undertaking this entire process. Pressurized water reactor contains two coolant circuits. The primary coolant circuit contains light water at 15MPa and 548 k (315 °C; 599 °F) temperature to avoid boiling [1-3]. Heat is passed from higher pressure primary coolant to a low pressure secondary coolant where the coolant turns into pressurized steam. Standard PWR has fuel assemblies of between 200 and 300 rods each. Typically, the fuel bundles consist of 14 × 14 to 17 × 17 fuel rods [4-5]. Coolant is used to remove heat from the nuclear reactor core and transfer it to turbines and the environment and also serves to maintain manageable pressures within the core. This runs first from bottom up to the top between the fuels. The effective way of handling the heat is clear to the protection of the reactor. Rod bundles are by far the most critical component of the reactor [6-7]. All nuclear fuel are indeed accompanied by a cooling system. The inspection of the thermal hydraulic design within the rod bundles also plays an important role in protecting of the reactor. The coefficient of heat transfer through forced convection in the sub-channel shall be regulated by the heat capacity of the fluid and also with the base fluid (e.g. nano fluids to increase heat transfer rate) [8, 10]. By knowing the characteristics of coolant, flow in the sub-channel, thermal hydraulics behavior can be understand. This helps to design such a fuel assembly that helps to increases the overall efficiency of the flow and efficiency of entire reactor [9]. Multiphysics it's self is bridge to finite element evaluation and to investigate thermal hydraulics characteristics. The multiphysics software has been used to analyze the thermal hydraulics through a sub channel. In this analysis Comsol multiphysics 5.3a software version and Finite Volume Method (FVM) has been used to evaluate the coolant characteristics in different region along the sub channel. This analysis represents different thermal hydraulics parameter span which are favorable in steady state condition in PWR.

## 2. Methodology

Computational fluid dynamics (CFD) simulation has been investigated in thermal hydraulic activity of coolant in a particular sub channel. CFD is often a subset of fluid dynamics to analyze and interpret problems involving fluid motion.Uranium-235 is assumed as heat source. By splitting uranium-235 a huge amount of energy is released which converts water into steam. In this case, we did not calculate the energy from fission rather we assigned and assumed specific temperature from the heat source to avoid the neutronic calculation. Table 1 represents the boundary condition used in this analysis.

* Corresponding author. Tel.: +88-01982603947
E-mail addresses: potterh274@gmail.com

**Table 1** Table for boundary condition

| Name Of Parameter | Value of the parameter |
|---|---|
| Pressure | 25kPa |
| Fluid temperature | $293.15^0$C |
| Heat flux from heat source | 1000(W/m$^2$) |

2.1 Geometry Modelling

Geometrical design of a sub-channel has been developed which is presented in figure 1. There has been total 5 domains. Frist a block has been built along with 4 cylinders. With difference command a structure of sub-channel has been formed. The width, depth and height of the block geometry has been considered as 10mm, 10mm and 42 mm respectively. The radius and height of the cylinder has been considered as 4mm and 42 mm respectively for simulation purposes. Four cylinders has placed on the four corners of the block with co-ordination system by manual input. The pitch is 9.2 mm with 2.3 is considered as pitch to radius ratio.

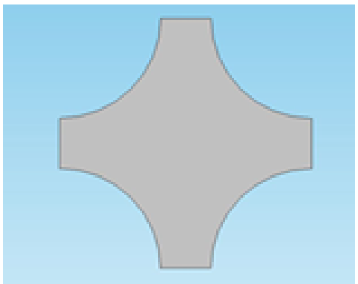


**Fig.1** Design of a sub-channel

2.2 Physics Settings

Turbulent Flow (k-ε) and Heat transfer in Fluids has been added to the model builder as physics input.

2.3 Equations Used For Turbulent Flow

Equations (1-6) has been used for conducting the simulation. These equations represent the conservation of continuity, momentum and energy.

$$\rho\frac{\partial u}{\partial t}+\rho(u.\nabla)u=\nabla.\left[-pI+(\mu+\mu_T)\left(\nabla u+(\nabla u)^T\right)-\frac{2}{3}(\mu+\mu_T)(\nabla.u)-\frac{2}{3}\rho KI\right]+F \text{.......................... ..(1)}$$

$$\frac{\partial\rho}{\partial t}+\nabla.(\rho u)=0 \text{....................................... (2)}$$

$$\rho\frac{\partial k}{\partial t}+\rho(u.\nabla)k=\nabla.\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\nabla K\right]+P_K-\rho\varepsilon \text{..(3)}$$

$$\rho\frac{\partial \epsilon}{\partial t}+\rho(u.\nabla)\varepsilon=\nabla.\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\nabla\varepsilon\right]+C_{\varepsilon1}\frac{\varepsilon}{K}-C_{\varepsilon2}\rho\frac{\varepsilon^2}{K}, \text{.................................................(4)}$$

$$\mu_T=\rho C_\mu\frac{K^2}{\varepsilon} \text{.............................................(5)}$$

$$P_K=\mu_T[\nabla u:(\nabla u+(\nabla u)^T)-\frac{2}{3}(\nabla.u)^2]-\frac{2}{3}\rho K\nabla.u \text{................................................................... (6)}$$

Where, u is water velocity (m/s); μ is dynamic viscosity of water (kg/m.s); ρ is density (kg/m$^3$); p is fluid pressure (N/m$^2$); K is thermal conductivity of water (W/m.k); $\mu_t$ is turbulent viscosity; ε is the rate of turbulent dissipation (m$^2$/s$^3$); $Q_{vhis}$ viscous heat (J); h is heat transfer co-efficient (W/m$^2$k); F is volume force vector (N/m$^3$).In the inlet and outlet the input has been given in the value of pressure in Pa. Inlet and outlet value of the model has been considered as 25000 Pa and 24988 Pa respectively.

2.4 Heat Transfer in Fluids

Generally, heat is mostly distributed through convection, where the flow of the fluid itself brings heat from one location to another. Second form of transmitting heat is through conduction that does not require the movement of a material, but only a flow of energy within a substance. The final way of transferring heat energy is through radiation involving electromagnetic waves. Equation (7) used for heat transfer in fluids:

$$\rho C_p\frac{\partial T}{\partial t}+\rho C_P u=\nabla.(k\nabla T)+Q+Q_{vd}+Q_p \text{..........(7)}$$

where, u is water velocity (m/s); ρ is density (kg/m$^3$);p is fluid pressure (N/m$^2$);K is thermal conductivity of water (W/m.k); $C_p$ is specific heat (KJ/Kg.K); T is absolute temperature (K); $Q_{vh}$ is viscous heat (J); Q is heat work (J); $Q_p$ is pressure work (J).All values were automatically updated from multiphysics library.

2.5 Multiphysics Settings

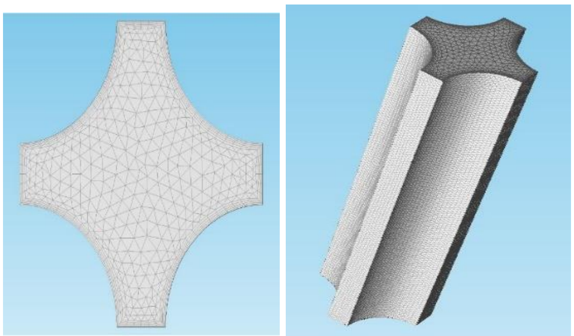
Multiphysics has been added as two different physics were working together. On-isothermal flow applies to fluid flows including temperatures which are not static. Whenever fluid undergoes through a change in temperature, it's certain properties, including such density and viscosity, adjust accordingly.

2.6 Mesh Settings

Meshing has been done for the simulation purpose which is represented is figure 2. Its sequence is physics-controlled mesh with normal element size. The matrix is a non-symmetric matrix. Table 1 represents the details of meshing.

**Table 2** Parameters of Mesh)

| Name Of Parameter | Value |
|---|---|
| Vertex Elements | 16 |
| Edge elements | 552 |
| Number of boundary elements | 6428 |
| Number of elements | 36593 |
| Free meshing time | 1.82s |
| Minimum element quality | 0.2073 |



(a) (b)

**Fig.2** Meshing view of sub channel (a) 2D view and (b) 3D view.

2.7 Study Settings

Study has been solved for Time dependent solver. Time unit is in second. Range is from 0 second to 10 seconds. After setting all those things above simulation has been started by clicking on "compute" command.

## 3. Result

3.1 Temperature Profile

Temperature profile of a sub channel has been simulated in this project. Figure 3 represents the temperature signature in the fuel sub channel.

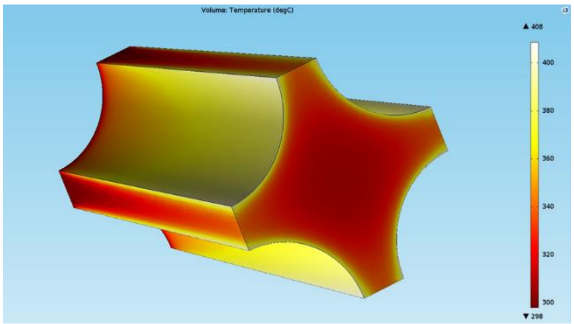


**Fig.3** Temperature signature.

Figure 4 shows that the maximum temperature has found $408^0$C in the coolant close to the cladding surface and in the outlet.

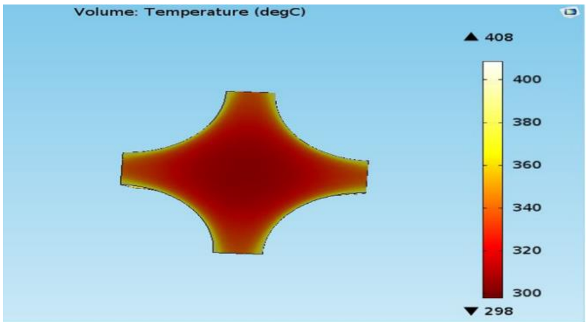



**Fig.4** Outlet temperature.

Figure 5 represents the minimum temperature is $298^0$c found in the inlet of the sub channel.

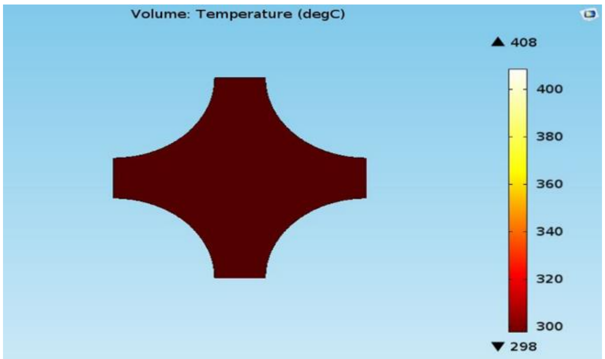



**Fig.5** Inlet temperature.

Water enters into the sub-channel with a $298^0$C temperature that is lower than the outlet temperature as it receives heat and further transfers it out of the sub-channel.

3.2 Heat Flux

Heat flux is an energy flow per unit of area per unit time. Heat flux is plotted against arc length of the sub channel which is represented in figure 6. As like normal condition, it is seen that heat flux is low in the inlet and outlet sides. Heat flux gradually increases with the length and then again gradually starts to go down along the length. The maximum heat flux is found $3.6\times10^8$ $W/m^2$ at 33 mm of the length. There are several restrictions on the heat transfer rate as well as the heat flow from the fuel components to the coolant, so if this thermal conductivity is too high, critical heat flow can be reached resulting to a boiling process. Hence this essence, would result in a rapid rise in the temperature of the clads. Critical heat flux is perhaps the major consideration of the Pressurized Water Reactors (PWR) [11].

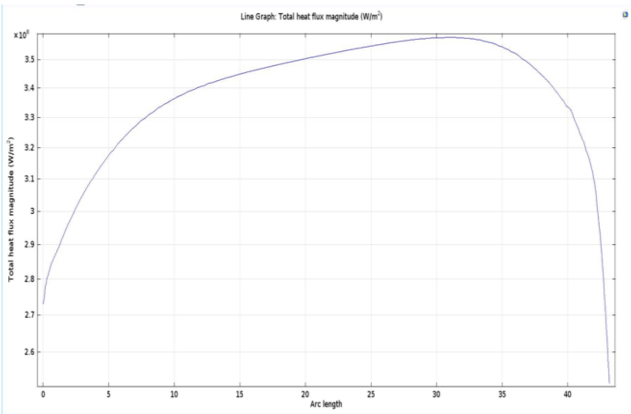



**Fig.6** Heat Flux.

3.3 Velocity Profile

In figure 7, X axis represents Arc length in mm unit and Y axis represents velocity in m/s unit. Left X axis implies input and right y axis represent output analysis. With the increasing of Arc length, velocity decreases up to 0.302 m/s and then it starts to increases gradually. When the Arc length is 0 mm, the velocity is 0.304m/s; at 4mm, velocity is 0.32 m/s.

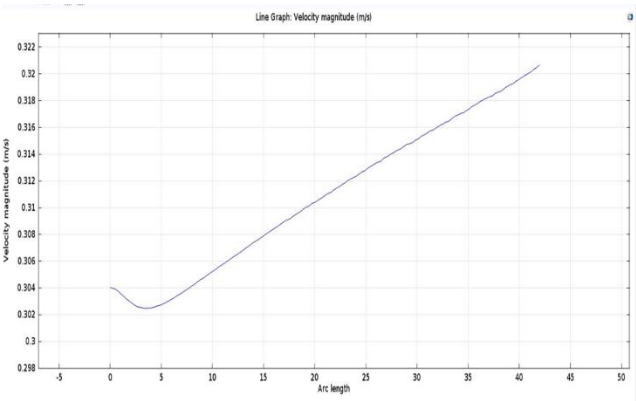


**Fig.7** velocity profile.

3.4 Isothermal Contours

The isotherm is a line linking all points in graph with an identical temperature. Thus every point in a particular isotherm have identical temperature but different Isotherm line have different unique temperatures. This may be the plane segment of the three-dimensional graph for a particular substances like coolant flow in a sub channel. Figure 8 indicates the temperature contour of coolant.

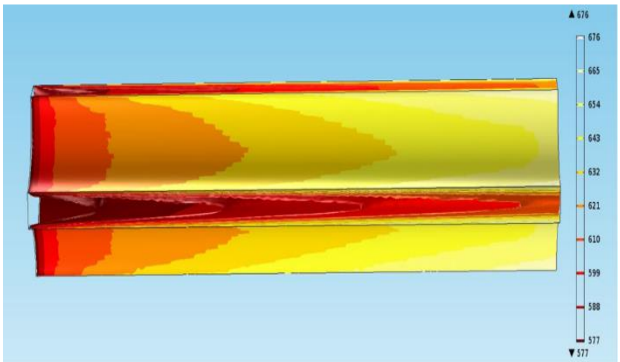


**Fig.8** Isothermal Contour.

3.5 Pressure profile

In the figure 9, X axis represents Arc length in mm unit and Y axis represents pressure in Pa unit. Left X axis implies input and right y axis represent output analysis. With the increasing of Arc length, pressure decreases gradually. When the Arc length is 0 mm, the pressure is 25000Pa; at 40mm, pressure is 24988 Pa.It can be said that the pressure change with respect to Arc length is inadequate.

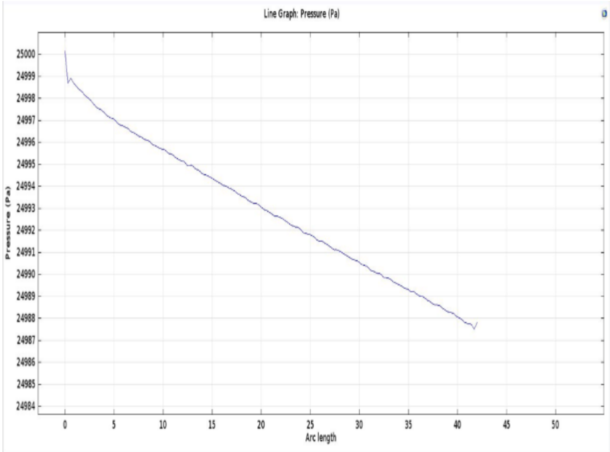


**Fig.9** Pressure Profile.

3.6 Reynolds Number

Reynolds number has been measured which is represented in figure 10.the measured Reynolds number is greater than $10^4$. Therefore it indicates that the flow is turbulent.

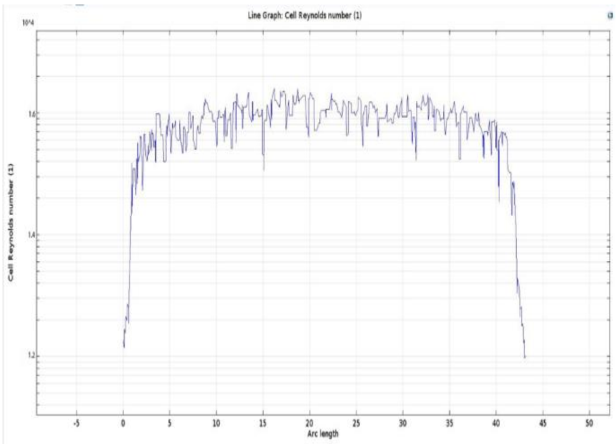


**Fig.10** Reynolds Number.

**4. Conclusion**

The purpose of this study is to investigate the thermal hydraulic activity of the coolant in the fuel assembly sub-channel during the regular operation of the Pressurized Water Reactors. In this process, different parameters such as temperature profile, velocity profile, Reynolds number, heat flux, isothermal contour etc. has been analyzed with the help of a multiphysics software. Sub channel flow is very important in nuclear reactor core. It helps to understand the thermal hydraulics behavior of coolant for entire fuel. The results found at the end of this study were almost close to the accurate results.

As our main focus is to analysis the thermal hydraulics behavior of coolant, neutronics such as neutron flux, core thermal power etc. are not considered while doing this simulation since software doesn't have any feature for neutronics calculations. If neutronics parameters are considered along with thermal hydraulics, more accurate results could be achieved. Therefore, some minor corrections should be introduced in order to get a better result that can be useful in nuclear reactor physics engineering applications.

## 5. Acknowledgement

We would like to take this opportunity to give our deeply appreciations to our supervisor Md. Ghulam Zakir, Assistant Professor, Nuclear Science and Engineering (NSE) Department, Military Institute of Science and Technology (MIST), Dhaka for his kindly help, counsel and high concern. His constant supervision, affectionate guidance and encouragement is of great help for completing this project throughout the study. We are also grateful to Department of Nuclear Science & Engineering, Military Institute of Science and Technology (MIST) for giving us the platform for doing this paper work.

## 6. References

[1] Lamarsh, J.R. and Baratta, A.J., 2001. *Introduction to nuclear engineering* (Vol. 3, p. 783). Upper Saddle River, NJ: Prentice hall.

[2] Glasstone, S. and Sesonske, A., 2012. *Nuclear reactor engineering: reactor systems engineering*. Springer Science & Business Media.

[3] Ginox, J. J. 1978. Two-Phase Flows and Heat Transfer with Application to Nuclear Reactor Design Problems. Hemisphere Publishing Corporation.

[4] Yeoh, G.H., 2019. Thermal hydraulic considerations of nuclear reactor systems: past, present and future challenges. *Experimental and Computational Multiphase Flow*, *1*(1), pp.3-27.

[5] Tong, L.S., 1988. Principles of design improvement for light water reactors.

[6] Baglietto, E., Ninokata, H. 2005. A turbulence model study for simulating flow inside tight lattice rod bundles. Nucl Eng Des, 235: 773–784.

[7] Mishima, K., Ishii, M. 1984. Flow regime transition criteria for upward two-phase flow in vertical tubes. Int J Heat Mass Transfer, 27: 723–737.

[8] Wu, Y. W., Luo, S., Wang, L., Hou, Y., Su, G. H., Tuan, W., Qiu, S. 2018. Review on heat transfer and flow characteristics of liquid sodium (2): Two-phase. Prog Nucl Energy, 103: 151–164.

[9] Yadigaroglu, G. 2014. CMFD and the critical-heat-flux grand challenge in nuclear thermal-hydraulics—A letter to the editor of this special issue. Int J Multiphase Flow, 67: 3–1

[10]International Journal of Applied Engineering Research ISSN 0973-4562 Vol 13, Number 7(2018) pp.5528-5533

[11]Zakir, M.G., Sarkar, M.R. and Hossain, A., 2019. Analysis of Neutronics and Thermal-Hydraulic Behavior in a Fuel Pin of Pressurized Water Reactor (PWR). *World Journal of Nuclear Science and Technology*, *9*(2), pp.74-83.