Id.: EN17

Thermal Analysis of SMR Reactor Reflector using OpenFOAM

Darío M. Godino[1-2], Santiago F. Corzo[2-3], Emanuel Gimenez[4], Damián E. Ramajo[2-3]

[1]UTN - FRP - Universidad Tecnológica Nacional, Facultad Regional Paraná, Argentina
[2]CIMEC Centro de Investigación de Métodos Computacionales (UNL/CONICET), Santa Fe, Argentina
[3]UNL - Universidad Nacional del Litoral
[4]CNEA – Comisión Nacional de Energía Atómica
dmgodino@gmail.com, http://www.cimec.org.ar

Keywords: Reflector; CFD; Nuclear Reactors; OpenFOAM, SMR.

ABSTRACT

Small Modular Reactors (SMRs), producing less than 300 MWe according to the IAEA, emerge as a promising solution for addressing climate change due to their rapid deployment and cost-effectiveness. Safety assessment through thermo-hydraulic studies is critical for experiment preparation, parametric analysis, and regulatory compliance. A key component of SMRs is the radial reflector surrounding the core, aimed at preventing flow diversion between the core and reactor barrel, thereby avoiding deviations from the core's operational point. Additionally, it enhances neutron efficiency and shields the reactor vessel from irradiation. However, the proximity of the reflector to the core leads to gamma radiation-induced heating, necessitating adequate cooling to maintain a temperature compatible with other internal reactor components. This study presents the development of a methodology to evaluate the three-dimensional temperature field of a generic SMR reflector using a coupled fluid-solid approach employing Conjugate Heat Transfer (CHT) and Computational Fluid Dynamics (CFD) with the open-source software OpenFOAM. The objective is to verify peak temperatures under full power operation conditions. Initially, to validate the necessary mesh requirements, a single channel was resolved. Subsequently, a 1/6 scale model of the reflector was developed, addressing the coupling between regions and applying various boundary conditions for the fluid. The simulations underscored the importance of considering the axial temperature profile in the coolant channels for accurate solid calculations. The results obtained facilitated the development of an effective methodology for simulating the reflector behavior in SMR reactors.

1. INTRODUCTION

In the field of nuclear reactors, optimizing neutron utilization is a crucial aspect of maximizing reactor efficiency and safety. The most widely used strategy to achieve this is using neutron reflectors. This consists of a material layer surrounding a reactor core and reflects many neutrons back into the core that would otherwise escape. This process reduces neutron loss and, consequently, improves the reactor's neutron economy, potentially leading to a higher fission rate and more efficient fuel utilization. Another important aspect of neutron reflectors is reducing the core's coolant flow bypass. Common materials used as reflectors include graphite, beryllium and water, each with unique properties that make them suitable for different types of reactors and operational conditions.

Since reflectors in a nuclear power plant are solid components located inside the reactor vessel, they are subjected to the same pressure and temperature conditions as the rest of the core. Moreover, these components can experience temperature increases due to various mechanisms such as absorption of gamma radiation, neutron absorption, inelastic scattering, and the heat transfer from the coolant. Therefore, it is crucial to ensure proper cooling of the different parts of the reflector. To achieve this, holes are made in these components to allow the passage of the cooling fluid, which helps remove the heat generated in the reflectors.

In the literature, numerous studies are addressing the simulation of reflectors in PWR reactors such as SMRs, specifically focusing on neutronic simulations and the effects of reflector composition[1][2][3]. However, studies on the arrangement and size of cooling channels are less common. This work analyzes the heat transfer in a reflector of a Small Modular Reactor (SMR) to assess the capacity of the cooling channels using Conjugate Heat Transfer (CHT) models and Computational Fluid Dynamics (CFD). This paper aims to develop an effective methodology for simulating the reflector behavior in SMR reactors addressing the heat transfer from the core and the outer side while simultaneously addressing the heat source generated internally.

The simulations were carried out in the open-source software OpenFOAM which presents several advantages for this type of problem where the radiation-induced heating is taken into account addressing the spatial distribution. The paper is outlined as follows: First, the governing equation, the boundary conditions, and volumetric sources are detailed, Secondly, the results are presented to evaluate the thermal distribution in the reflector to verify the maximum temperatures. Another important aspect to verify is that the coolant flow does not reach the saturation condition.

2. GOVERNING EQUATIONS

The governing equations of the CHT model are from the chtPimpleFoam solver implemented in the suite OpenFOAM[4]. The chtPimpleFoam solver performs the coupling explicitly in an iterative manner, similar to a segregated solver. This means that chtPimpleFoam solves the energy equation for the solid and fluid separately, but they are linked through conservative boundary conditions. This coupling technique gives several advantages from the point of view of the computational implementation and leads to accurate solutions in a few iterations.

2.1. Fluid domains

The fluid flow is mathematically described by means of the Navier-Stokes equation along with the mass and energy conservation equations. This set of partial differential equations is assembly averaged (Favre average [5]) to reduce the spectra of spatial and temporal scales of the solutions. Finally, the resulting equations are written in terms of the independent variables (spatial coordinates and time) and the mean variables: pressure (p), internal energy ($\hat{u}$) or the enthalpy ($\hat{H} = \hat{u} + p\rho$), velocity (U), and density (ρ). Then, the balance equations for Newtonian fluids with constant properties are expressed as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{1}$$

$$\frac{\partial(\rho\mathbf{U})}{\partial t}+\nabla\cdot(\rho\mathbf{UU})=-\nabla p+\nabla\cdot\tau_t+\rho g \tag{2}$$

$$\frac{\partial(\rho\hat{u})}{\partial t}+\nabla\cdot(\rho\hat{u}\mathbf{U})+\frac{\partial(\rho K)}{\partial t}+\nabla\cdot(\rho K\mathbf{U})= \\ -\nabla\cdot q-\nabla\cdot(p\mathbf{U})-\nabla\cdot(\tau_t\mathbf{U}) \tag{3}$$

where $\tau_t = \tau + \tau R$ is the turbulent stress tensor, τ is the viscous stress tensor, and τR is the Reynolds stress tensor.

Assuming that there is a relation (eddy viscosity hypothesis) between τ, τ_R and the mean velocity U, $\tau_t = \mu_{eff} [\nabla U + (\nabla U\ T)] - 2/3\ \mu_{eff}(\nabla\cdot U)I$ where $\mu_{eff} = \mu + \mu_t$ is the effective viscosity, μ is the dynamic viscosity and μt is the turbulent viscosity. I the identity tensor, K the kinetic energy, g the acceleration of gravity, and κ the thermal conductivity. By introducing the Fourier law, the conductive term in the energy equation takes the following form $q = -\kappa_{eff}\nabla T$.

The k-ω SST model was selected due to its versatility in blending the turbulence in the high turbulence in the bulk (using k-ε [6]) and the low turbulence in the boundary layers (using k-ω [7]). The SST k-ω model developed by Menter et al. [8] is the most recommended for this industrial application where precision and computational cost must be balanced.

2.2. Solid domains

In this case, only the thermal equation is solved and the resulting equation is accomplished from the energy balance and the Fourier law:

$$\partial\ (\rho sCpT)\ \partial t = \nabla\cdot(\kappa s\nabla T) \tag{4}$$

where κs, ρs, and Cp are the thermal conductivity, density, and heat capacity of the solid.

3. COMPUTATIONAL MODEL

In the integrated SMR type, the reactor pressure vessel (RPV) contains the reactor core in its lower part, which is surrounded by a shroud or reflector (See Figure 1). Surrounding the reflector is the barrel, which divides the RPV into two fluid regions: an inner region where the coolant ascends from the core, and an outer region where the coolant descends after passing through the steam generators (SG). The fluid circulates through the core in an upward direction until it reaches the upper area of the SG, passing through it in a downward direction and exchanging heat with the secondary side.

The present work addresses the thermal distribution in the reflector. For this reason, the outer and inner coolant flow was not solved directly. However, proper boundary conditions were adopted to analyze the heat transfer. The domain is divided into two regions: one consisting of the solid and the other of the coolant fluid. The solid, in turn, is composed of three parts: the core shroud, the barrel, and the stagnant water gap between them, as shown in Figure 2. Additionally, in the computational model, the core shroud is divided into two zones to easily apply the thermal power term, which changes after a certain radius. As for the fluid domain, it consists only of the interior of the coolant channels, as boundary conditions are imposed on the remaining surfaces of the solid.

Taking advantage of the symmetry of the reflector geometry, the computational model only addresses a portion of the domain (1/6). For the channels, the mass flow rate is imposed in the inlet sections and represents 5% of the primary flow rate. Table 1 resumes the thermal boundary conditions in the solid zones.

Table 1. Solid boundary conditions

Boundary	**Boundary condition type**	**Boundary condition values**
Inner wall	Robin	hc = 6640 W/m2K T∞ = variable.
Outer wall	Robin	hc = 1420 W/m2K T∞= constant.
Lateral	Periodic	-
Lower wall	Dirichlet	T = constant.
Upper wall	Robin	hc = 1468 W/m2K T∞= constant.
Channels	Continuity	-

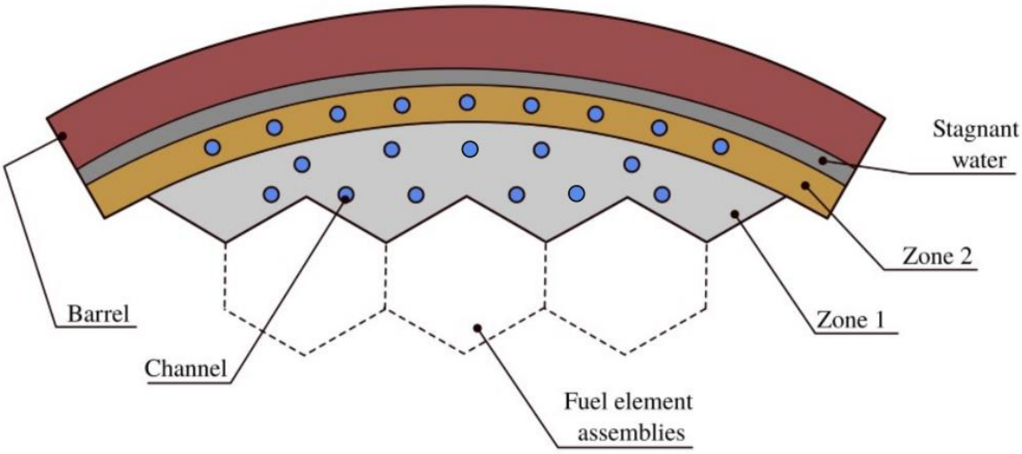

Figure 1: Generic diagram of a core reflector.

Figure 2 shows the computational mesh. Note that there are four solid regions (including the stagnant water gap) and one region for the coolant channels. The computational grid was built with a total of 2,273,103 cells, where 1,530,792 correspond to the solid regions and 742,311 correspond to cells for the fluid zone (cooling channels). Figure 3-a shows the normalized power distribution in solid zones 1 and 2. Only axial distribution is taken into account. The heat generation was imposed as a volumetric source. Finally, Figure 3-b shows the temperature distribution for the coolant flow in the inner face.

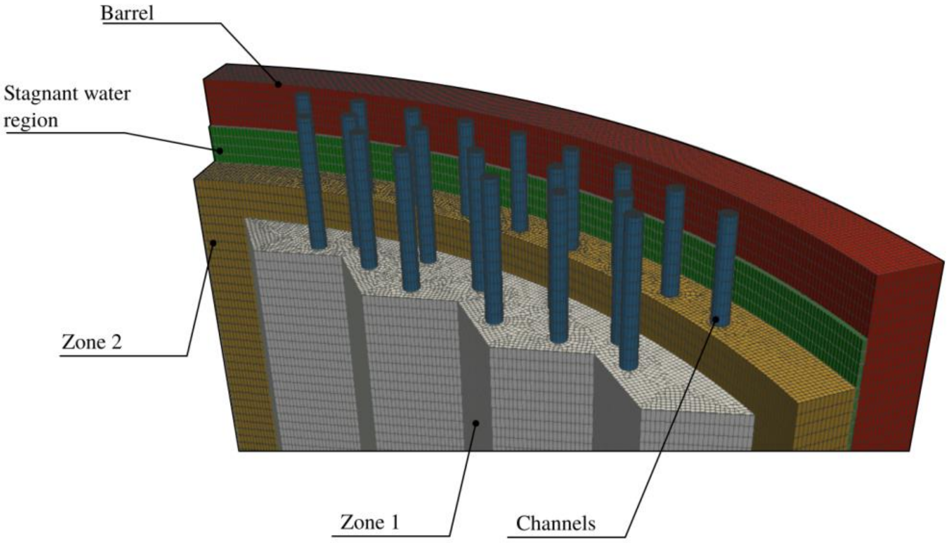

Figure 2. Mesh geometry details.

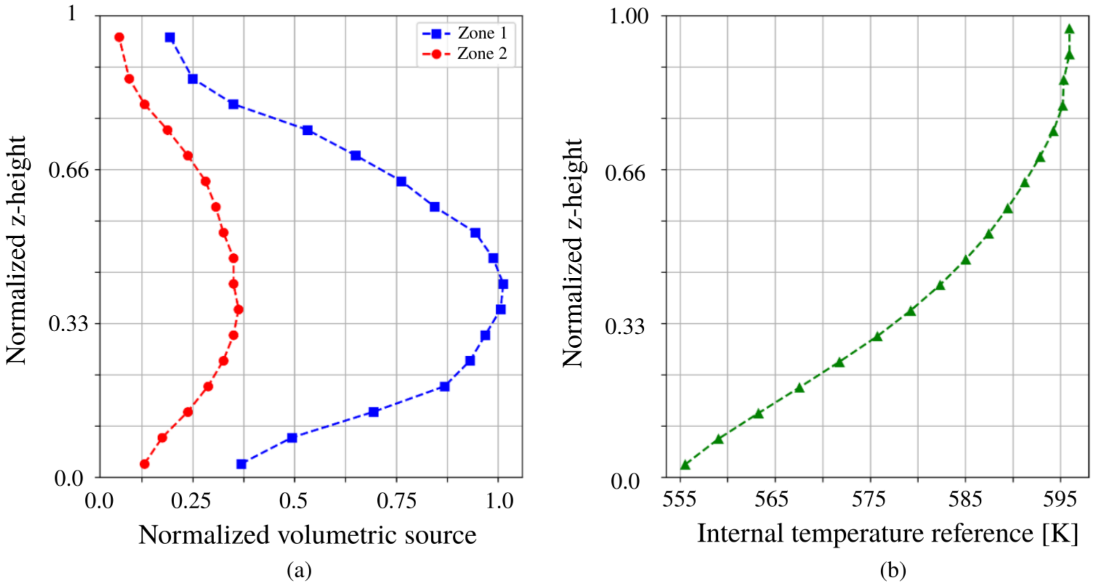

Figure 3: Variable boundary conditions and heat sources: a) Heat power source for solid zones 1 and 2. b) Inner coolant temperature in the reflector.

4. RESULTS AND DISCUSION

The simulation was carried out during a time until reaching the steady state condition (2500s). Figure 4 shows the mean and the maximum temperature for the different regions and Figure 5 shows the outlet channel temperature to understand the heat balance. The heat transferred by neutron flux from the core added to the volumetric heat source leads to a higher temperature increment in zone 1 close to the core. Even though the heat source in Zone 1 is higher than in Zone 2, the mean and maximum temperatures are close due to the heat transfer between these zones and to the coolant channels.

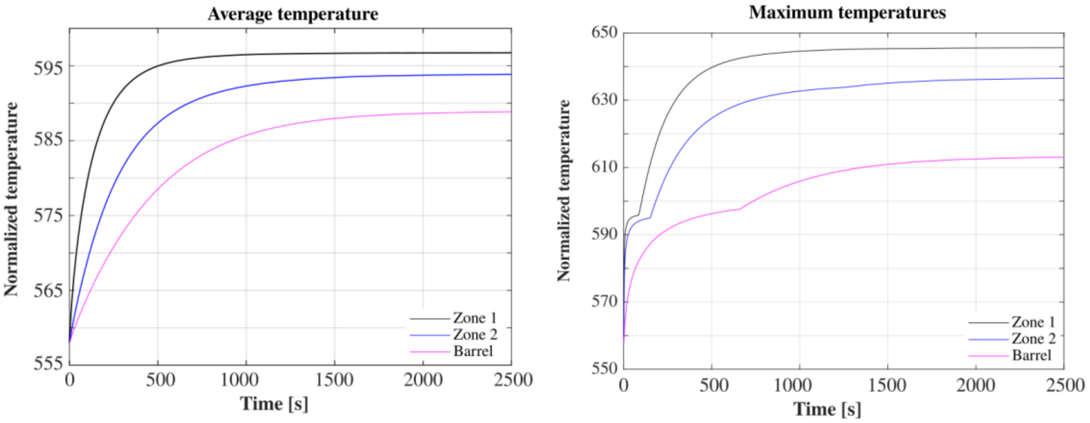

Figure 4**:** Temperature in the different regions: a) Mean temperature, b) Maximum temperature in zones 1, 2 and the Barrel.

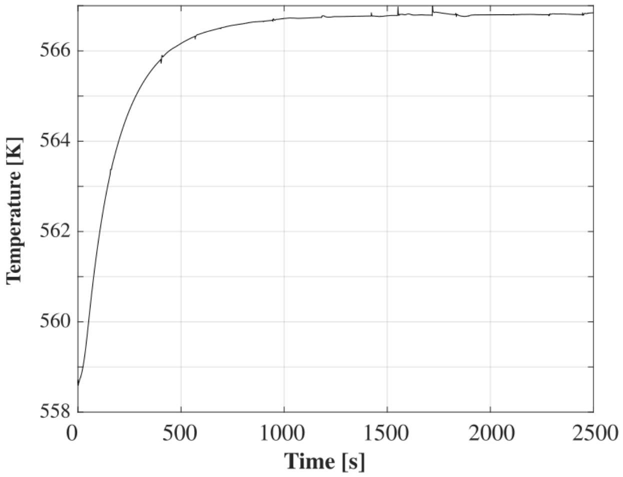

Figure 5**:** Average temperature in the outlet channels.

Figure 6 shows the final temperature distribution of the CRs along with the solid region for the nine cutting planes. The temperature is not uniform across all channels, as it is strongly influenced by the neutronic heat source, as mentioned above. Another important observation is that the coolant temperature does not reach the saturation point. No zones of excessive overheating are observed, and as expected, the regions near the channels exhibit greater cooling of the reflector.

Above the height of 1m, the temperatures decrease rapidly thanks to the cooling provided by the channels and the heat transferred to the outer surface.

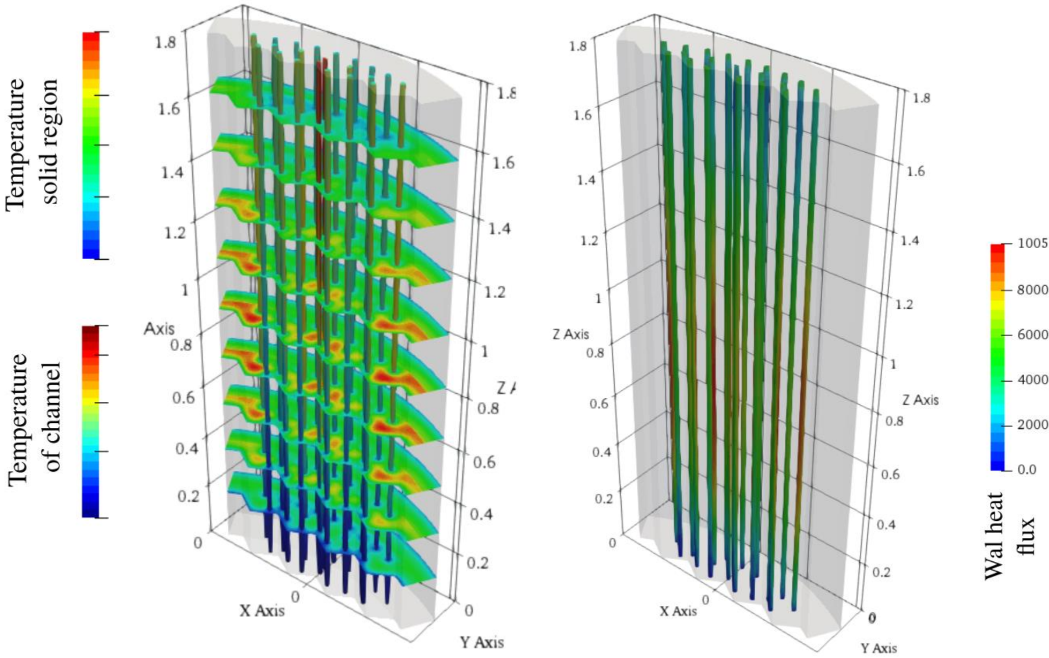

Figure 6: a) Thermal distribution in the solid domain for different cut planes and the coolant channels, b) Heat transfer in the channel walls.*Scales are omitted for confidentiality reasons.

Figure 6 shows the heat flux in the channel walls. The maximum heat transfer from the solid to the coolant flow occurs in the zone of the maximum heat source. The heat transfer decreases close to the outlet due to the temperature reduction.

5. CONCLUSION

This paper conducts a computational simulation of the nuclear reflector in an SMR to evaluate the cooling capacity of the channels within the solid reflector. The study was conducted using a generic SMR reflector model and employed a coupled fluid-solid approach. Conjugate Heat Transfer and CFD simulations were performed using the open-source software OpenFOAM. The results have shown that the configuration of the channels and the established flow rate are appropriate for ensuring that the solid components do not exceed critical temperature limits that could compromise integrity. Additionally, no significant temperature gradients were observed. The conjugate heat transfer model, with volumetric heat sources and appropriate boundary conditions, represents a reliable approach for future thermal evaluation studies, which could further enhance neutron kinetics models.

REFERENCES

[1] Kang, J. K., Hah, C. J., Cho, S. J., & Seong, K. B. The effects of stainless steel radial reflector on core reactivity for small modular reactor. In AIP Conference Proceedings (Vol. 1704, No. 1). AIP Publishing, (2016).

[2] Durán-Vinuesa, L., García-Herranz, N., & Cuervo, D. Cobaya improvements for reflector modeling of SMR, 47ª Reunión Anual de la Sociedad Nuclear Española, (2022).

[3] Optimización térmica del reflector de un reactor nuclear Alan Erik Pérez Winter. (2023).

[4] Greenshields, C. OpenFOAM v12 User Guide, The OpenFOAM Foundation", London, UK, (2020).

[5] Favre, A. Equations des gaz turbulents compressibles. J. de Mecanique, 4(3), (1965).

[6] Launder B., Spalding D., The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, vol. 3 , nº 2, p. 269–289, (1974).

[7] WilcoxD. C., Formulation of the k–ω Turbulence Model Revisited, AIAA Journal, vol. 46, nº 11, pp. 2823-2838, (2008).

[8] Menter, F. R., Kuntz, M., & Langtry, R., Ten years of industrial experience with the SST turbulence model. Turbulence, heat and mass transfer, 4(1), 625-632, (2003).