VETOR

Thermo-hydraulic Simulation of AP1000 Nuclear Reactor Fuel Assembly

Simulação Termo-hidráulica de um Conjunto Combustível do Reator Nuclear AP1000

L. Acosta Martínez[†], C. R. García Hernández, Jesus Rosales García, Annie Ortiz Puentes

Higher Institute of Technologies and Applied Sciences, University of Havana, Cuba.

[†]**Corresponding author:** leo94am@gmail.com

Abstract

One of the challenges of future nuclear power is the development of safer and more efficient nuclear reactor designs. The AP1000 reactor based on the PWR concept of generation III + has several advantages, which can be summarized as: a modular construction, which facilitates its manufacture in series reducing the total construction time, simplification of the different systems, reduction of the initial capital investment and improvement of safety through the implementation of passive emergency systems. Being a novel design it is important to study the thermohydraulic behavior of the core applying the most modern tools. To determine the thermohydraulic behavior of a typical AP1000 fuel assembly, a computational model based on CFD was developed. A coupled neutronic-thermohydraulic calculation was performed, allowing to obtain the axial power distribution in the typical fuel assembly. The geometric model built used the certified dimensions for this type of installation that appear in the corresponding manuals. The thermohydraulic study used the CFD-based program ANSYS-CFX, considering an eighth of the fuel assembly. The neutronic calculation was performed with the program MCNPX version 2.6e. The work shows the results that illustrate the behavior of the temperature and the heat transfer in different zones of the fuel assembly. The results obtained agree with the data reported in the literature, which allowed the verification of the consistency of the developed model.

Keywords

Thermohydraulic • CFD • Nuclear Reactor

Resumo

Um dos desafios da futura energia nuclear é o desenvolvimento de projetos de reatores nucleares mais seguros e eficientes. O reator AP1000 baseado no conceito PWR de geração III + possui várias vantagens, que podem ser resumidas como: uma construção modular, que facilita sua fabricação em série reduzindo o tempo total de construção, simplificação dos diferentes sistemas, redução do investimento de capital inicial e melhoria da segurança através da implementação de sistemas passivos de emergência. Por ser um projeto inovador, é importante estudar o comportamento termo-hidráulico do núcleo aplicando as ferramentas mais modernas. Para determinar o comportamento termo-hidráulico de um conjunto típico de combustível do núcleo do reator AP 1000, foi desenvolvido um modelo computacional baseado em CFD. Foi realizado um cálculo nêutron-termo-hidráulico acoplado que permitiu obter a distribuição da potência axial no conjunto típico de combustível. O modelo geométrico construído utiliza as dimensões certificadas para este tipo de instalação que constam dos respectivos manuais. O estudo termo-hidráulico realizado utilizou o programa ANSYS-CFX baseado em CFD, e considerou um

oitavo da montagem do combustível. O cálculo de nêutrons foi realizado com o programa MCNPX versão 2.6e. O trabalho mostra os resultados que ilustram o comportamento da temperatura e da transferência de calor nas diferentes zonas do conjunto combustível. Os resultados obtidos estão de acordo com os dados relatados na literatura, o que permitiu a verificação da consistência do modelo proposto.

Palavras-chave

Termo-hidráulico • CFD • Reator Nuclear

1 Introduction

Nuclear energy is a reliable way to meet energy demand in an environmentally friendly and sustainable way over a long period of time. The impact that the use of this source has had for more than 60 years has been such that today several countries, mainly developed and also developing, are betting on the use of energy from the fission of the uranium core to diversify its energy matrix.

About 66% of greenhouse gas emissions come from burning fossil fuels for the production of electricity. Despite the challenges it faces today, nuclear energy is considered an effective technology that can be used to mitigate climate change with specific characteristics that underpin the commitment of some countries to maintain it as an option for the future [1].

The data recorded up to the end of 2019 demonstrate the above. On the planet there are 443 reactors connected to the grid in 31 countries, which produce about 393.2 GWe, representing approximately 10.5% of the total electrical energy consumed. Of these, 303 are of the PWR (Pressurized Water Reactor) type, thus being the most widely used technology in the world for the production of electrical core energy. With this capacity, 32% of electricity is obtained free from greenhouse gas emissions, which is equivalent to stopping 2 billion tons of CO_2 into the atmosphere [2].

Westinghouse has designed the AP1000 (Advanced Passive 1000) nuclear reactor. Generation III + reactors based on the PWR concept present different modifications and advantages with respect to the design of the previous generation. Considerable effort has been devoted to simplifying the various systems, significantly reducing the overall size of the installation, the number of auxiliary components, the length of the pipes and the necessary wiring. Likewise, its design allows a modular construction, facilitating its mass production, reducing the total construction time and reducing the initial investment [3].

Due to these advantageous features, the AP1000 design was chosen to perform the thermohydraulic simulation of a typical fuel element. The thermohydraulic study was carried out using the program based on Computational Fluid Dynamics (CFD) Ansys-CFD. This tool allows us to analyze a wide range of variables subjected to different conditions, which will allow us to evaluate the fuel assembly thermohydraulic performance.

CFD is one of the branches of fluid mechanics that uses numerical methods and computational algorithms to study and analyze problems involving moving fluids, heat transfer, and even other processes such as chemical reactions. There are different numerical methods and algorithms that solve the fundamental equations in different ways. In other words, CFD is the art of replacing systems of partial differential equations with algebraic systems of equations that can be solved using computers.

The application of CFD codes in the nuclear field is wide. Currently this application could be divided in four categories: severe accident phenomenology, design of fuel elements, applications to the operation of current reactors and design of new fission / fusion systems [4].

In order to complete the thermohydraulic analysis, it is necessary to know the fuel assembly power distribution. The selection of a Monte Carlo N-Particle (MCNP) type code to in the fuel assembly offers great advantages. The most important of which is that it allows detailed modeling of the geometry in three dimensions, which is a necessity due to the complexity of the geometry and the heterogeneity of the AP1000. In this work, it was used the Monte Carlo N-Particle Extended code (MCNPX) version 2.6e.

The objective of this paper is to develop a computational model that allows studying the thermohydraulic and neutronic behavior in a typical fuel assembly of the AP1000 reactor. The document is organized as follows. Section 1 covers a brief introduction to the state of nuclear energy at the world level. The technical characteristics of the AP1000 reactor, as well as the details of the neutronic and thermohydraulic model are discussed in section 2. While section 3 assesses and analyzes the results. The conclusions are given in section 4.

2 Materials and methods

2.1 AP1000 Nuclear Reactor

The AP1000 is a water-cooled nuclear reactor, which is kept under sufficient pressure to limit the generation of steam at the core exit, and where the large amount of heat produced is transferred to a secondary system through a heat exchanger.

The reactor core is mounted inside a steel vessel, called a reactor vessel. To direct the flow, the core is enveloped by a cylindrical barrel. The reactor vessel has inlet nozzles, which communicate the annular space between the internal wall of the vessel and the exterior of the barrel. The coolant, at an approximate temperature of 288ºC and a pressure of 15.2 MPa, enters this annular space flowing downwards to the inlet chamber formed by the lower plenum of the reactor vessel. At this point the flow changes direction, flowing up through the reactor core to the upper plenum that communicates with the outlet nozzles of the reactor vessel. This flow is illustrated in Fig. 1.

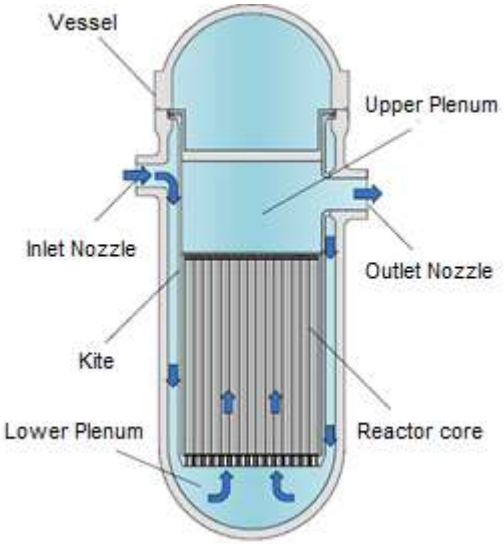


Figure 1: Flow in the PWR reactor vessel.

Now let's see how the core of the AP 1000 rector is shaped. The nuclear fuel consists of cylindrical pellets of slightly enriched uranium dioxide (UO_2) (approximately 3% of U^{235}) with a diameter and length of approximately 1cm. A metal tube from ZIRLOTM called clad with wall thickness of about 0.6mm is filled with pellets to an active length of about 427 cm and sealed to form a fuel rod. Between the outer surface of the pellets and the inner wall of the tube there is a gap which is filled with a gas, generally helium. The metal tube serves as a coating to retain radioactive fission products, as a support for the pellet column, and to protect the fuel from interaction with the coolant. A total of 264 fuel rods are grouped into an assembly that is approximately 20 cm per side, and 157 of these elements are assembled in a roughly cylindrical arrangement to form the reactor core (Fig. 2).

The fuel elements are supported by the internal elements of the reactor. The internal elements of the reactor also direct the flow of coolant through the fuel rods. The coolant, and at the same time moderator, consists of a solution of light water and soluble boron. The fuel, internals, and coolant are contained within the reactor vessel, which is a pressure steel vessel, having an outer diameter of approximately 5 m, a height of 12 m and walls up to 30 cm thick.

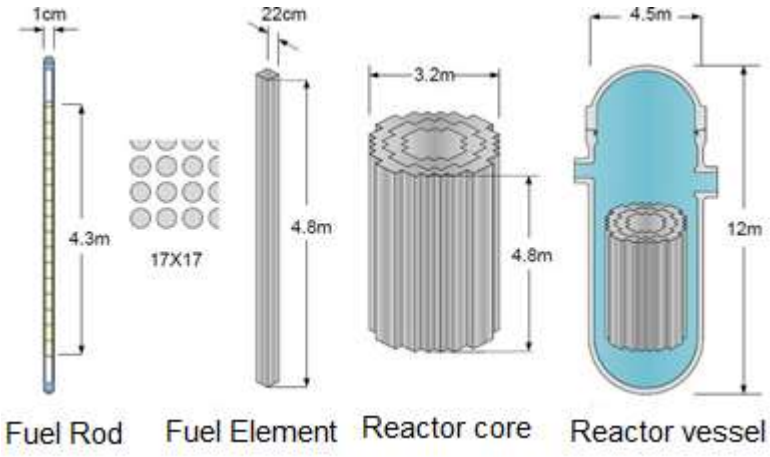


Figure 2: Construction of the reactor core.

2.2 Computational Fluid Dynamics

ANSYS CFX is a software based on the general purpose CFD techniques Navier-Stokes equations in their conservation form. A heat transfer model is chosen to predict the temperature throughout the flow. It includes heat transfer by conduction, convection and turbulent mixed and viscous work.

To carry out a CFD numerical simulation it is necessary to cover three fundamental stages: pre-processing, processing ("solver") and post-processing. During the preprocessing stage, the geometry, meshing and simulation settings are generated. During the processing stage, the fundamental equations are numerically solved and the post-processing stage is defined by the visualization and analysis of results.

As mentioned above, the goal of a CFD code is to solve the conservation equations. In general, these equations are based on three fundamental principles:

a) Conservation of mass.

b) Newton's second law $F = \frac{d\rho}{dt}$ (Conservation of the moment)

c) Conservation of energy.

From these principles the conservation equations are obtained.

2.2.1 CFD model

To carry out the thermohydraulic simulation of the typical AP1000 fuel assembly, first the geometric configuration was stated. Using the dimensions specified above, a geometric model of the fuel assembly based on a square arrangement of 17×17 fuel rods oriented vertically was built. From the total of 289 cylinders, 24 are guide tubes to place control rods or burnable absorbents and they are arranged symmetrically as shown in Fig. 3.

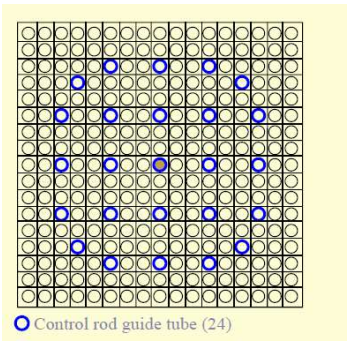
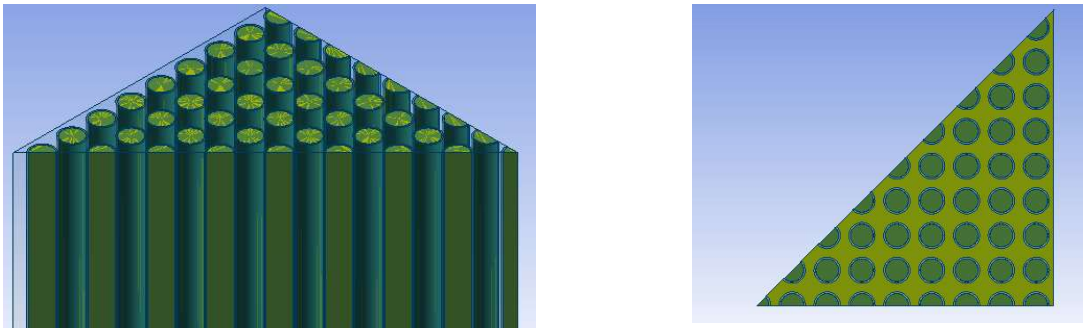


Figure 3: Arrangement of control rods in the fuel assembly [13].

We must clarify that in our model we will assume the initial approximation in wich all cylinders correspond to fuel rods. In order to simplify the calculations and optimize the available computational resources, a simplification of the geometry is made, taking advantage of its symmetry, which allows us to take 1/8 of the complete set, as illustrated in Fig. 4. This part will have the same length as the set complete, but it will have only 28 full fuel rods, 16 rods will be half and one rod will only have one eighth, which equates to a total 4.6 m^2 of heat transfer area.



A B

Figure 4: Eighth part of the fuel assembly. –A) Isometric view -B) Top view.

The generation of the mesh is one of the most important steps in the pre-processing stage. CFD requires subdivision of the domain into a number of small, non-overlapping elements in order to solve the physics of the flow within the geometry that was created. The accuracy of the CFD solution is largely determined by the number and distribution of elements within the domain. In general, a large number of elements results in higher precision. However, the precision is highly dependent on the limitations imposed by computational costs and computational time.

To obtain a numerical solution independent of the mesh size, a mesh sensitivity study was carried out. The methodology used to carry out the study is as follows: first, an initial simulation was carried out with a first mesh with elements of 1 mm in the radial direction and 80 mm in the axial direction with a total of 1,238,000 elements. Once the convergence of the first simulation was obtained, the mesh is refined to obtain smaller elements in the domain. The refinement was carried out in two steps, first it was refined in the axial direction and then in the radial direction. The results obtained in two consecutive simulations are compared using monitors at different points in the geometry. The optimal mesh has a total of 2,780,000, with a cell size of 40 mm by height and 0.8 mm radial spacing. As a variable for comparison, the temperature at different points of the coolant was used. For our analysis, the temperature in the coolant was considered the most important parameter, so that the other zones remain without variation in the mesh. The studied points for comparison cover the entire area of the coolant, emphasizing the area near the wall of the fuel elements where higher rates of heat transfer occur and there is greater turbulence and in points away from this where the average temperature is reached.

Once the study region has been discretized, the boundary conditions were considered, as well as the materials and physical models to be used. The boundary conditions were defined according to the parameters set out in Table 1.

Table 1: Model parameters

Parameters	Value
System Pressure	15.2 MPa
Water Properties	450K-900K, 1MPa-30MPa (Steam 5V)
Coolant Inlet Temperature	288ºC
Coolant Inlet Mass Flow	10.71 kg/s

The coolant inlet temperature of the fuel assembly is taken equal to the core inlet temperature. The properties of water were taken according to the International Association for the Properties of Water, from the library of materials provided by CFX, IAPWS-IF97, which is a formulation of water and properties of steam prepared for industrial uses, where the formulations must be designed for fast and complex calculations. These properties of water are taken in the temperature range 450K-900K corresponding to a pressure between 1000 kPa and 30000 kPa. The total mass flow of the coolant is determined by dividing the total design mass flow entering the reactor core by the total number of fuel assemblies (157).

To predict the temperature throughout the flow, the Total Energy heat transfer model was used, which takes into account the effects of the kinetic energy of the flow. The Total Energy model describes the enthalpy transport through the domain. It differs from the Thermal Energy model in that the effects of the kinetic energy of the flow are included. For cases where viscous heating and the effects of turbulence are important, it is advisable to use the Total Energy model.

Turbulence models have been specifically developed to take into account the effects of turbulence without resorting to an extremely fine mesh. Since the 1970s, the most widely used turbulence models for high Reynolds number flow are the models of two equations of the k-ε type, hence this is the turbulence model that was used. These models close the RANS (Reynolds Averaged Navier-Stokes) system of equations that governs the mean flow by approximating the effect of turbulence using a turbulent viscosity, which varies with position and with time as a function of the kinetic energy and its dissipation rate ε.

2.4 Model in MCNPX

The first step of neutronic calculation consists of generating the geometry, which is divided into 31 zones by height, varying the density in each one of them according to a linear distribution of the change in temperature at the constant pressure of the system. The model takes into account the 264 fuel rods of a set, as well as the 25 guide tubes, each of these was filled with air.

The serial coupling approach is used for the coupling between MCNPX and Ansys-CFX computer codes. In such coupling it is required that the results of a code, previously calculated, be entered as initial data for the calculation in the other code. The parameters that are exchanged between the two codes for the coupling are: energy distribution in the fuel elements obtained with the MCNPX code, distribution of the water density, water temperature distribution and fuel temperature distribution obtained with the Ansys-CFX code.

Figure 5 shows the flow chart of the coupled calculation. In the iterative process of the coupled calculation, first the axial distribution of energy released in the fuel elements is calculated with the code MCNPX, the axial distribution of the average coolant density is assumed for the first iteration according to a linear variation of the temperature from 288 ℃ to 325 ℃.

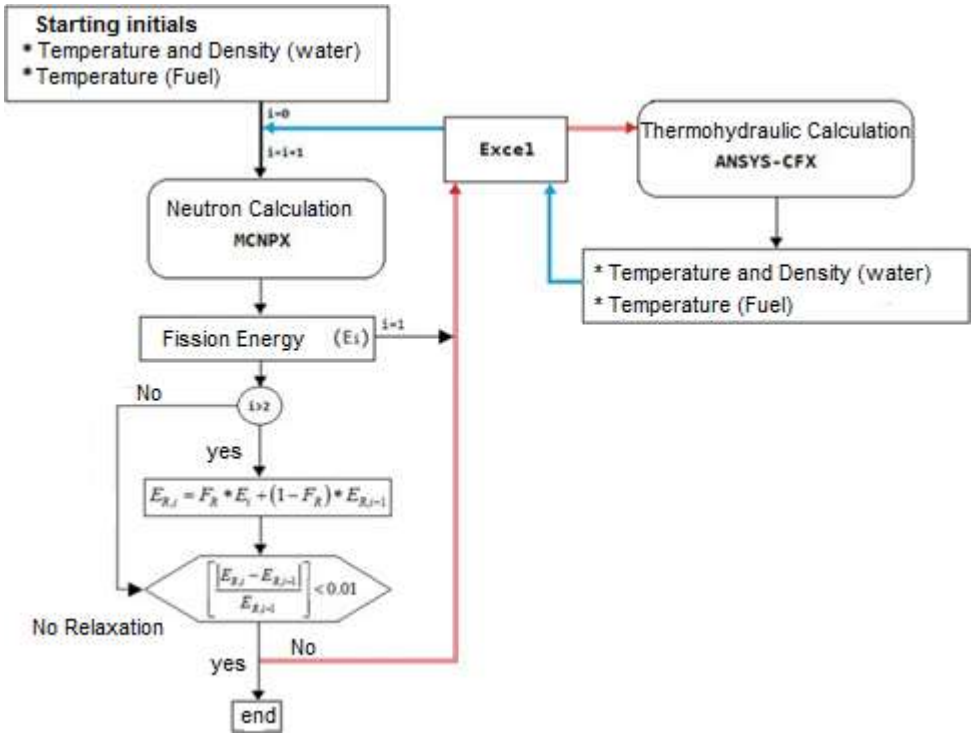


Figure 5: Flow diagram of the coupled calculation.

3 Results and discussions

To perform the thermohydraulic calculation of the typical fuel assembly of the AP1000 reactor, it was necessary to know the axial power distribution. To know it, first a linear increase in the coolant temperature by height was considered. Figure 6 shows the temperature and power profiles for the first iteration.

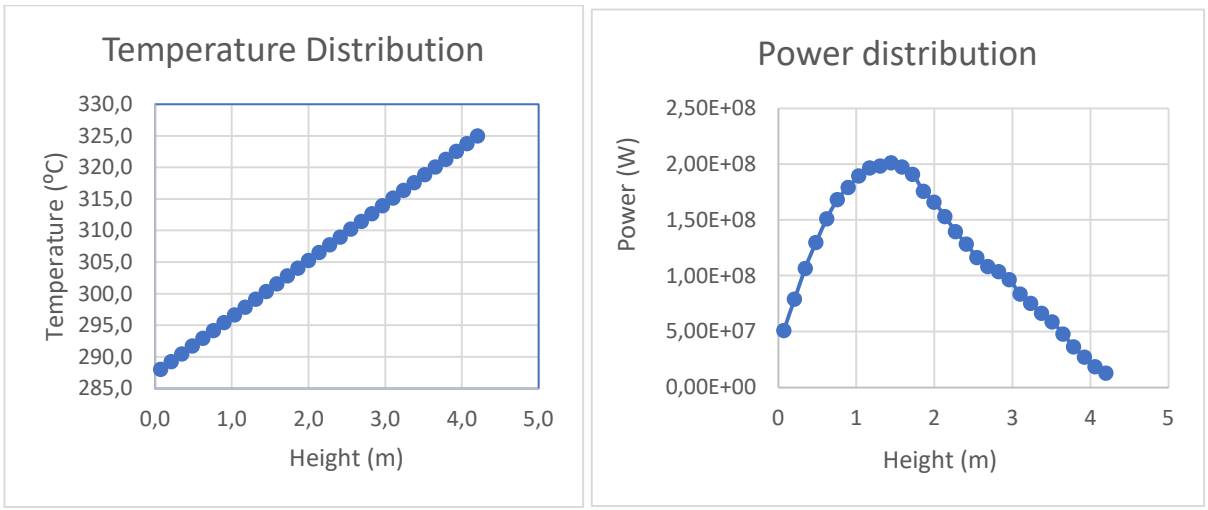


Figure 6: Temperature and power distribution for the first iteration.

To obtain the final profiles of released energy, it was necessary to carry out an iterative neutronic-thermohydraulic calculation. Convergence was achieved after 5 iterations. The final axial power and temperature profiles are shown in Fig. 7.

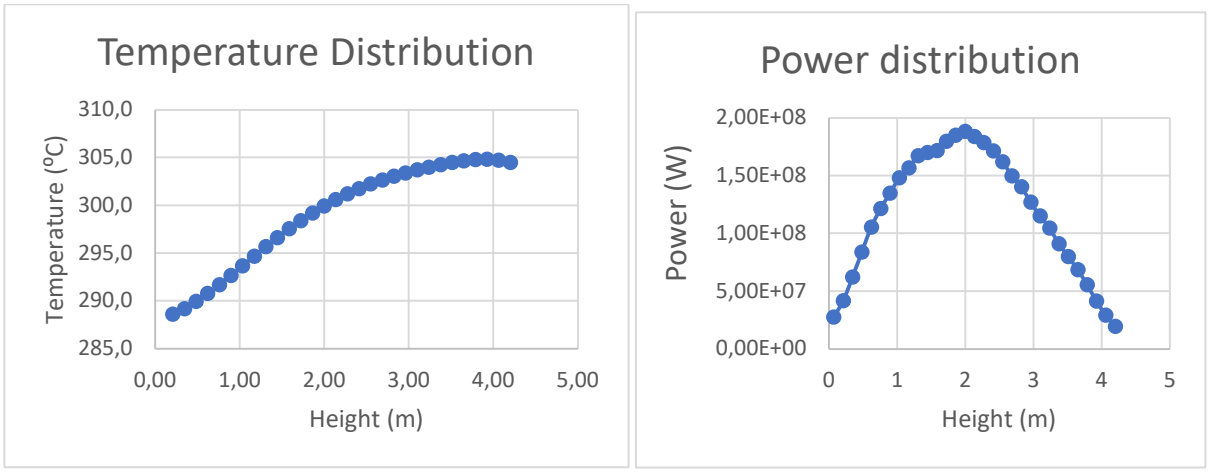


Figure 7: Temperature and power distributions in the last iteration.

Using the axial power profile obtained in the iterative calculation, the average coolant temperature at the outlet was calculated, showing how the radial temperature distribution at the channe outlet varies between 326 ^{0}C and 329 ^{0}C. Figure 8 shows how the maximum temperature values are reached at the vertices of the fuel assembly. This phenomenon is related to the symmetry conditions imposed on the model.

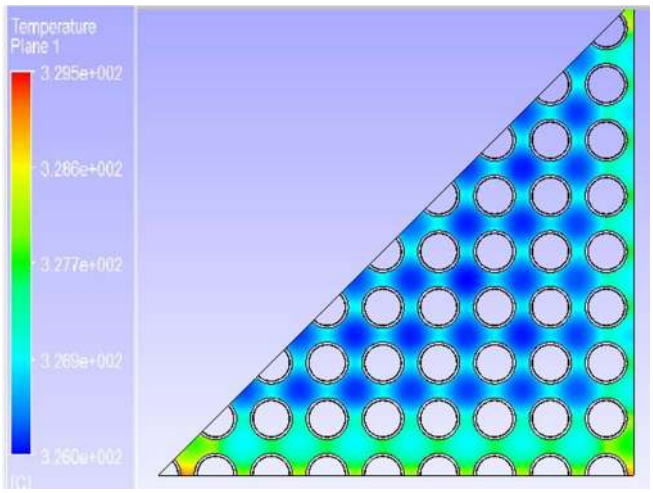


Figure 8: Coolant temperature radial distribution at the outlet.

In the design documentation, the coolant average temperature at the outlet is 325 ^{0}C, the graph plotted in Fig. 9 shows the behavior of the average coolant temperature versus height. It is observed that the average coolant temperature at the outlet is a little higher than 326 ^{0}C, this is in agreement with the design value and permit to validate our model according to the simplifications made.

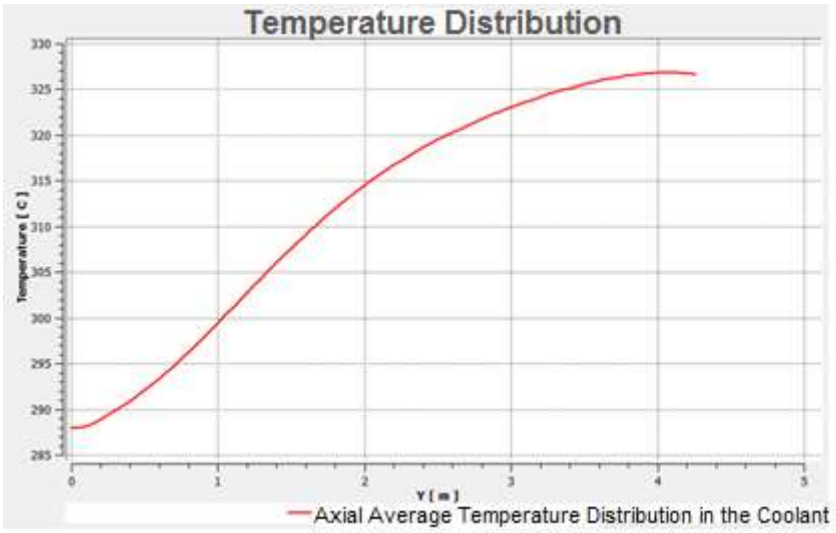


Figure 9: Coolant average temperature versus height.

An important aspect to evaluate the safety is to know the fuel temperature distribution in each fuel rod studied, in order to verify that both the zircaloy clad and the UO_2 tablets do not exceed the design limits of temperature established. Fig. 10 shows the central plane of the line of fuel rods that reach the highest temperature. As can be seen in the graphics, the highest temperature is reached in the center of the fuel because it is the area where there is less heat transfer due to its distance from the coolant. The temperature reached at the hottest point is lower than the design limit temperature for this type of fuel (1600 ^{0}C).

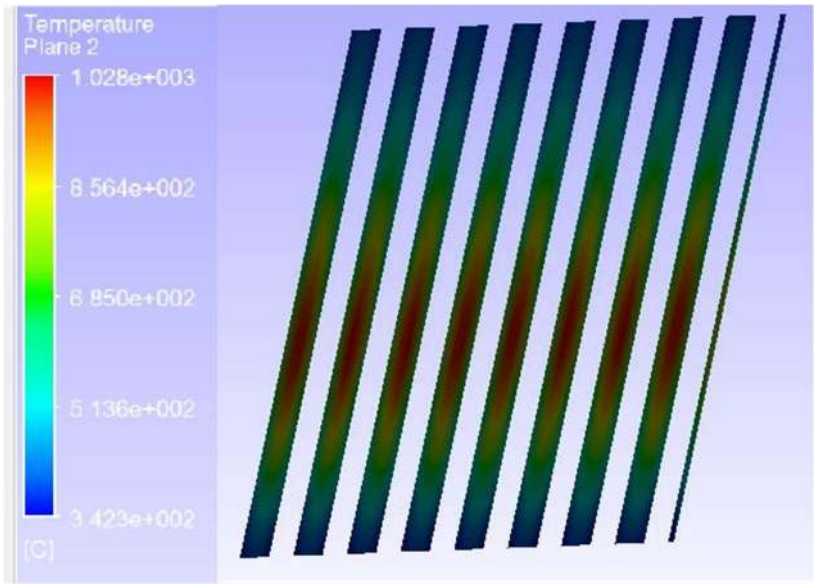


Figure 10: Fuel temperature.

A more detailed analysis of the temperature drop across the fuel cladding will identify the element with the highest temperature. The graph plotted in Fig. 11 shows the radial temperature distribution in the area with the highest energy released.

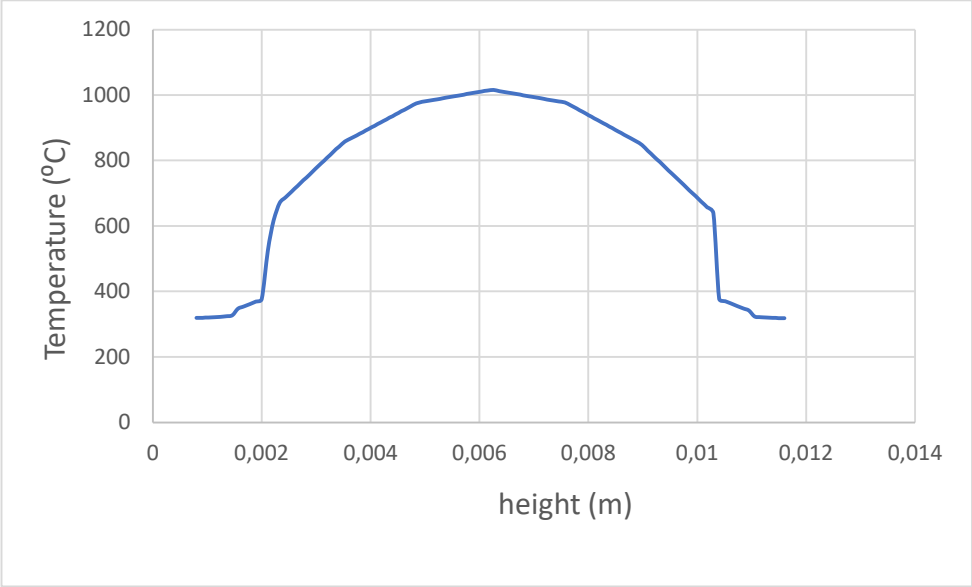


Figure 11: Radial temperature distribution in the zone with the highest energy released.

It can be seen how the temperature of the fuel does not exceed the melting point of UO_2 that is 2865 °C. The maximum temperature that is reached in the fuel rod is 1110 °C, a value that is below 2093 °C at which this type of fuel is designed. The maximum value of temperature obtained in a fuel rod is low. This is due to the fact that the model considered that energy was released in all tubes, including those destined for control rods or burnable absorbents.

Another important result is that the maximum temperature reached in the clad is 427 °C, which is below 1204 °C at which the maximum temperature is permited, since this temperature it oxidizes releasing hydrogen, leading to material degradation.

After demonstrating the consistency of the computational model developed, it is of interest to analyze the thermohydraulic behavior of the typical fuel assembly for steady state with loss of coolant.

To determine the possible states to be analyzed, it is necessary to determine the value of the volumetric flow rate from which our model is no longer valid. The fluid model used is single-phase and as the volumetric flow rate decreases, the coolant temperature increases and it boils at certain points. At the pressure of 15.2 MPa the water changes its phase at the temperature of 343.2 °C according to [41]. The initial mass flow rate is 10.71 kg / s and from this flow rate it gradually decreases as shown in the Fig. 12.

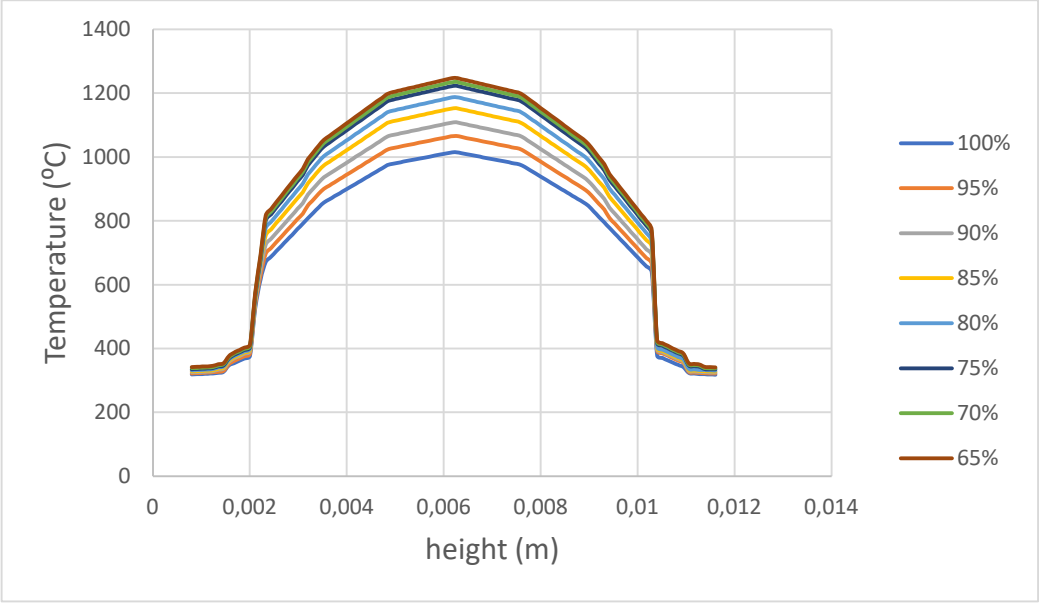


Figure 12. Radial temperature distributions for different volumetric flow rate.

At 65% of the original volumetric flow ratea maximum coolant temperature of 341.7 ℃ is reached. Therefore, with a volumetric flow rate less than this, the computacional model is not valid. The decrease in volumetric flow leads to an increase in the coolant temperature in the model. This causes a decrease in heat transfer in the fuel assembly, so it is of interest to know the state in which it would reach the melting point. In the model presented, this point is not reached. The fuel temperature is always below the melting point.

According to the study made for steady state with loss of coolant, with a coolant volumetric flow rate equivalent to 65% of the nominal value, a maximum fuel temperature of 1256 ℃ is reached in the most loaded fuel rod of the typical fuel assembly (average power), which constitutes an increase in the temperature value of 87 % approximately. This temperature is below 1600 ℃, which is the fuel design limit temperature.

4 Conclusions

In the work, a computational model based on the CFD code was developed and implemented, using the Ansys-CFX software for thermohydraulic simulation, allowing the study of the behavior of temperature and heat transfer in the eighth part of the fuel assembly of the core of the AP1000 reactor.

The axial power distribution in a typical fuel assembly of the AP1000 reactor was determined through a coupled iterative neutronic-thermohydraulic calculation using the codes MCNPX and Ansys-CFX. The average coolant temperature at the outlet of a channel was calculated, which allowed to validate the consistency of the model. It was found that it is close to the design value since the average outlet temperature value in our model is 326.5 ^{0}C, very close to the design value of 325 ^{0}C.

The highest temperature in the fuel rods were identified and the radial temperature distributions in the rod, gap, clad and coolant were calculated. It was observed that temperature design limit values were not exceeded.

The radial temperature distribution of the most loaded fuel rod was compared with that reported in the bibliography and a very good coincidence was found.

A study for steady state with loss of coolant was made. The value of the coolant volumetric flow rate at the inlet of the channel was varied, determining the limit flow rate for the application of the computational model developed and the value of increase in the maximum temperature of the fuel.

References

[1] D. S. Siqueira, J. A. Meystre, M. Q. Hilário, D. H. D. Rocha, G. J. Menon, and R. J. Silva, “Current Perspectives on Nuclear Energy as a Global Climate Change Mitigation Option,” *Mitigation and Adaptation Strategies for Global Change*, vol. 24, no. 5, pp. 749–777, 2019. Available at: https://link.springer.com/article/10.1007/s11027-018-9829-5

[2] Z.-Y. Hung, Y.-M. Ferng, W.-S. Hsu, B.-S. Pei, and Y.-S. Chen, “Analysis of AP1000 Containment Passive Cooling System during a Loss-of-Coolant Accident,” *Annals of Nuclear Energy*, vol. 85, pp. 717–724, 2015. Available at: http://dx.doi.org/10.1016%2Fj.anucene.2015.06.027

[3] G. Chen, Z. Zhang, Z. Tian, L. Li, and X. Dong, “Challenge Analysis and Schemes Design for the CFD Simulation of PWR,” *Science and Technology of Nuclear Installations*, vol. 2017, article ID 5695809, 2017. Available at: https://doi.org/10.1155/2017/5695809

[4] T. L. Schulz, “Westinghouse AP1000 Advanced Passive Plant,” *Nuclear Engineering and Design*, vol. 236, no. 14–16, pp. 1547–1557, 2006. Available at: https://doi.org/10.1016/j.nucengdes.2006.03.049

[5] L. C. Gonzalez, C. G. Hernández, L. R. Mazaira, and C. B. O. Lira, “Metodología para el cálculo acoplado neutrónico-termohidráulico del reactor nuclear de agua ligera de alto desempeño,” *Energética [online]*, vol. 37, no. 2, pp. 124-134, 2016. Available at: http://scielo.sld.cu/scielo.php?script=sci_arttext&pid=S1815-59012016000200006 (in Spanish)

[6] A. Erfaninia, A. Hedayat, S. M. Mirvakili, and M. R. Nematollali, “Neutronic-Thermal Hydraulic Coupling Analysis of the Fuel Channel of a New Generation of the Small Modular Pressurized Water Reactor Including Hexagonal and Square Fuel Assemblies Using MCNP and CFX,” *Progress in Nuclear Energy*, vol. 98, pp. 213–227, 2017. Available at: http://dx.doi.org/10.1016/j.pnucene.2017.03.025

[7] C. Zhao and C. Jianyun, "Dynamic Characteristics of AP1000 Shield Building for Various Water Levels and Air Intakes Considering Fluid-Structure Interaction," *Progress in Nuclear Energy*, vol. 70, pp. 176–187, 2014. Available at: http://dx.doi.org/10.1016%2Fj.pnucene.2013.08.002

[8] X. Wang, H. Chang, M. Corradini, T. Cong, and J. Wang, "Prediction of Falling Film Evaporation on the AP1000 Passive Containment Cooling System Using ANSYS FLUENT Code," *Annals of Nuclear Energy*, vol. 95, pp. 168–175, 2016. Available at: http://dx.doi.org/10.1016%2Fj.anucene.2016.05.014

[9] M.-T. Kao, C.-Y. Wu, C.-C. Chieng, Y. Xu, K. Yuan, M. Dzodzo, M. Conner, S. Beltz, S. Ray, and T. Bisset, "CFD Analysis of PWR Core Top and Reactor Vessel Upper Plenum Internal Subdomain Models," *Nuclear Engineering and Design*, vol. 241, no. 10, pp. 4181–4193, 2011. Available at: http://dx.doi.org/10.1016/j.nucengdes.2011.08.007

[10] J. Tu, G. H. Yeoh, and C. Liu, *Computational Fluid Dynamics. A Practical Approach. RMIT University Australia*, 1st ed. Butterworth-Heinemann, 2008.

[11] R. J. Fetterman, "AP1000 Core Design with 50% MOX Loading," *Annals of Nuclear Energy*, vol. 36, no. 3, pp. 324–330, 2009. Available at: http://dx.doi.org/10.1016%2Fj.anucene.2008.11.022

[12] N. E. Todreas and M. S. Kazami, *Nuclear Systems. Thermal Hydraulic Fundamentals*, Massachusetts Institute Technology, 2001.

[13] E. I. M. San Martín. "Modelación de la transferencia de calor en las varillas de combustible del reactor nuclear PWR," Monograph, Universidad de Chile, 2011. Available at: http://www.tesis.uchile.cl/tesis/uchile/2011/cf-munoz_es/pdfAmont/cf-munoz_es.pdf (in Spanish)

[14] Status report 81. Advanced Passive PWR (AP1000), 2011.