PRELIMINAR MECHANICAL EVALUATION OF THE STRUCTURE OF A NUCLEAR PLATE-TYPE FUEL ELEMENT

Marcelo Moraes dos Santos, Miguel Mattar Neto e Javier González Mantecón

Nuclear and Energy Research Institute, IPEN – CNEN/SP
Av. Prof. Lineu Prestes, 2242 – Cidade Universitária
05508-000 São Paulo, SP
marcelommds1003@gmail.com
mmattar@ipen.br
javier.mantecon@ipen.br

ABSTRACT

The improvement in the efficiency and safety aspects of compact nuclear reactors is directly linked to innovations in fuels and in the geometry of fuel elements (F.E), as is the case of plate-type fuel elements. From the mechanical viewpoint, to ensure that the structure of a fuel element is safe to operate in a compact PWR reactor is important to confirm that it meets the functional design requirements for structures of this type and application, present in ANSI/ANS-57.5-1996 and, also, that the stresses resulting from the loads imposed are less than the permissible mechanical limits for their structural materials, in accordance with ASME III, division 1, subsection NB. In order to develop a methodology of mechanical analysis to verify compliance with the criteria of the cited standards, a numerical model of a plate-type fuel element was developed, taking into consideration the main active loads admitted from the full power operation event belonging to the normal operating condition of a compact PWR type nuclear reactor. The results of the analyses demonstrated that the fuel element designed did not show signs of mechanical failure with respect to the modes of plastic collapse and excess of mechanical deformation.

1. INTRODUCTION

The nuclear fuel that makes the fission process possible in a reactor is confined inside the cladding, that can be a rod or a plate, generally. The set of this fuel rods or plates arranged in a single structure is called fuel element, which is, by definition, a structure that has the function of providing means to keep nuclear fuel in position and safety during the operation of a reactor. In addition to the safety aspect of nuclear fuel, the other reasons for using fuel element structures are:

- In the event of a fuel element failure, only the self-structure can be removed in a simple manner, not the entire core of a reactor.

- These structures make easier the process of fuel management: fuel element containing partially spent fuel material can be relocated to maintain the equilibrium of reactivity throughout the reactor core.

- These structures include a relatively simplified fabrication.

- The handling and storage of the fuel elements are relatively simple.

The main objective of this article is to present the development of a methodology of structural evaluation directed to a plate-type fuel element for the operation event at full power, belonging to the normal operation of a compact PWR-type reactor. In order to meet the general objective mentioned above, the three following specific objectives are achieved:

1. A mechanical design of a plate-type fuel element is shown, which is used as modeled geometry for analysis in the Ansys® software.

2. An analysis methodology is presented that involves the hydraulic, thermal and structural areas, using the CFX®, Steady-State Thermal® and Static Structural® analysis systems, respectively, all of which belong to the Ansys® software.

3. The proposed methodology is applied to the presented fuel element design.

According to Kaufmann [1], a fuel element should basically contains: cooling channels, handles for its handling, clamping springs, pins for the contact with the internal support structure of the fuel element in the pressure vessel core, connections for the instrumentation and a channel for the control element. In addition, a fuel element should be a structure small enough to be easily transported. The basic arrangement for the construction of a fuel element consists, essentially, in the joining (by means of welding) of the so-called main components, which are a top nozzle, four fuel-plate sets and a bottom nozzle, as will be shown later in this paper.

2. METHODOLOGY

The methodology proposed in this work consists in the use of two numerical computational methods: the finite-element method (used in Steady-State Thermal® and Static Structural® analysis systems) and the finite volume method (used in CFX® analysis system).

The finite-element method principle is the attendance to the equilibrium equations. In this method a modeled geometry is subdivided into small parts (called elements), which now represent the problem analysis region. The division of the geometry modeled into small elements allows solving a complex problem by subdividing into simpler problems. Such divisions (or elements) may present different forms such as triangular, quadrilateral, among others, depending on the type (linear and non-linear/ dynamic and static) and the dimension of the model. The finite elements are connected to each other by points (which are called *nodes* or *nodal points*) and, to the set of all these *elements* and *nodes*, is called the *mesh*. The precision of this method is dependent, basically, on the number of nodes and elements, size, types and quality of the elements present.

The finite-volume method differs from the finite element method in one simple way: this method has the principles of mass conservation, momentum and energy as the basis for its mathematical modeling. The formulation of integral conservation equations for mass, momentum and energy is applied in these control volumes.

It is considered, in the present work, that during the event of operation at full power of a compact PWR reactor the fuel element is subjected to five loads, these being the self-weight, the force of the fixing springs, the hydrostatic thrust, the hydraulic drag and the temperature field. The values of these loads will be shown later.

The analysis of the fuel element is divided in two parts. The first concerns a thermal-hydraulic analysis in the central fuel plate of the fuel assembly (there are eighteen fuel plates in one fuel element assembly), through which the values of hydraulic drag loads and the temperature field are obtained. These values are accepted for all eighteen fuel plates. The second part of the analysis of the fuel element consist of a thermomechanical analysis, in which all the loads mentioned above are considered, in the form of successive combinations of the loads, as will be shown later.

In order to be able to analyze the fuel element, it is necessary to integrate the active loads by means of combined and successive computational analyzes. A thermal-hydraulic analysis is performed in the *CFX*® analysis system for the calculation of temperature distributions and drag force on a fuel plate, with representative volumes of a fuel core, a cladding and the coolant passing on the sides of this plate, as will be shown in the analysis model in the section 2.2. Through this analysis it is possible to map the temperature field of the fuel plate and the cooling channel. It was admitted that these values are valid for all other plates present in a fuel element. For this analysis, the thermal-hydraulic and physical properties are configured. As input data, are determined: the ambient temperature (30°C), the power density in the core ($1{,}81.10^8$ W/m³) and the thermo-hydraulic properties of the coolant (present in section 2.7.). The result of this analysis is a graph with the temperatures acting on a fuel plate as a function of its height at specific locations (fuel center, the interface between the fuel and the inner cladding, external cladding, and coolant) along its thickness, as shown in Figure 1. Through this analysis, the hydraulic drag force value acting on this fuel plate is also obtained (4.91 N) and assumed for each plate of the fuel element.

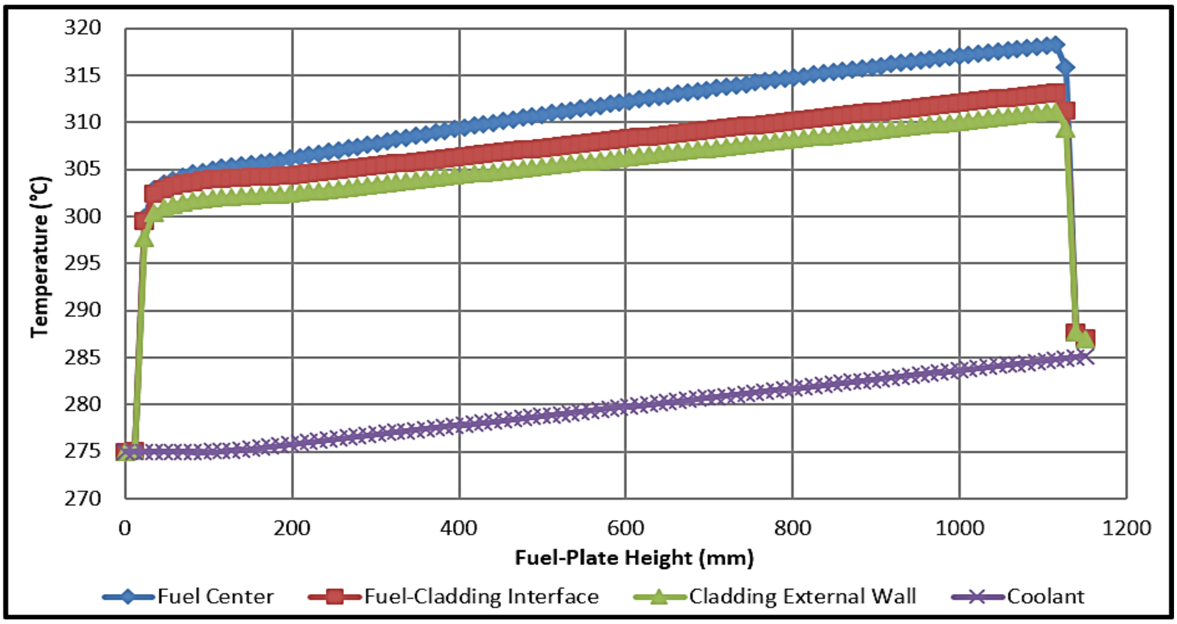

Figure 1 - Temperatures acting on a fuel plate.
Source: M. M. Santos [2].

The second part is performed in the *Static Structural*® module and consist of the use of the same geometry mentioned above, but with a focus on mechanical loads. In this stage, the loadings of the self-weight, the fixing springs acting on the upper nozzle, the thrust acting on the entire structure, the hydraulic drag acting on the fuel plates and side plates, and the thermal field acting on the entire structure (analyzed in *Steady-State Thermal*®), were configured in the mechanical analysis model. The analyzes of the loads were performed in steps of combinations of loads, as will be shown in section 2.4, so that it is possible to evaluate the influence of each of these. The applied restriction condition was the restriction of the vertical movement on the face of the lower nozzle and the lateral movements on the faces referring to the location of the symmetry region, as will be shown in section 2.5.

The use of the numerical methods requires that parameters presented in the following topics be defined so that the analyses of the loading situations in a component represent more accurately the real conditions of a possible existing model.

2.1. Actual geometry

The designed fuel element model is shown in Figure 2, and the main components data referring to this structure are shown in Table 1.

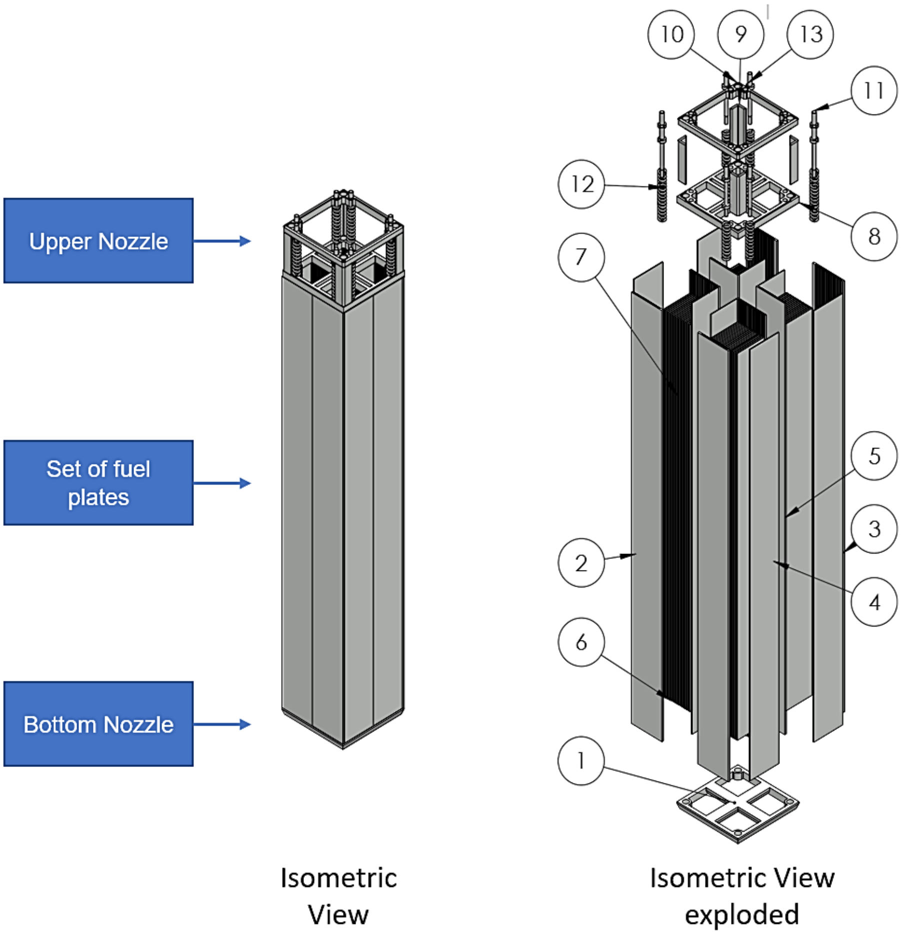

Figure 2 - Fuel element structure: isometric view and isometric view exploded.
Source: M. M. Santos [2].

Table 1 - Fuel element components data.

Component number	Component name	Material	Quantity
1	Bottom nozzle	ASTM B351 (Zircaloy-4)	1
2	Plate A	ASTM B352 (Zircaloy-4)	4
3	Plate B	ASTM B352 (Zircaloy-4)	4
4	Plate C	ASTM B352 (Zircaloy-4)	4
5	Plate D	ASTM B352 (Zircaloy-4)	4
6	Cladding	ASTM B352 (Zircaloy-4)	72
7	Fuel Core	U-10Mo	72
8	Base B	ASTM B351 (Zircaloy-4)	1
9	Corner	ASTM B351 (Zircaloy-4)	4
10	Base A	ASTM B351 (Zircaloy-4)	1
11	Support pin	ASTM B351 (Zircaloy-4)	8
12	Securing spring	ASTM B637 (Inconel 718)	8
13	Tab	ASTM B351 (Zircaloy-4)	8

Source: M. M. Santos [2].

2.2. Modeling Geometry

The initial step in the use of numerical methods is the elaboration of a "calculation scheme" or, in other words, a "calculation model" or a "modeled geometry". It is a geometric computational model whereby the relevant regions of the problem under study are identified and allow hypotheses for the analysis to be defined. The modeled geometry was elaborated in the SolidWorks® software and was later exported to the analysis systems pertinent to each analysis performed.

For the thermal-hydraulic analysis, the analysis model consists of a fuel plate surrounded by two cooling channels, as shown in Figures 3 and 4. This model counts on with three volumes of control: coolant, cladding and fuel.

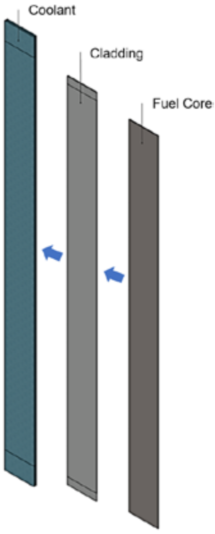

Figure 3 - Thermal-hydraulic modeling geometry volumes of control.

Source: M. M. Santos [2].

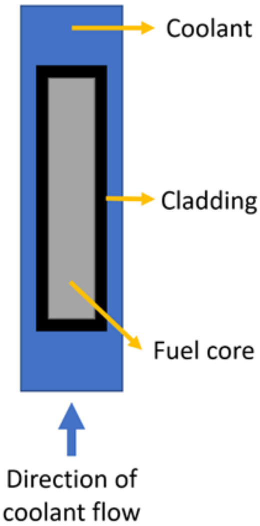

Figure 4 - Thermal-hydraulic modeling geometry scheme.

Source: M. M. Santos [2].

For the thermomechanical analysis, the analysis model consists of a quarter of the real geometry. In other words, a double symmetry condition is defined in this model, as shown on the right side of Figure 5. Virtual springs were configured, suppressing the need to model a finite element spring. In this way, Base A, Corner, Tabs and Support Pins were suppressed in the modeled geometry, as a way to simplify the model and to focus the analyzes mainly on the fuel plate assemblies. The main geometric data referring to this structure are shown in Table 2.

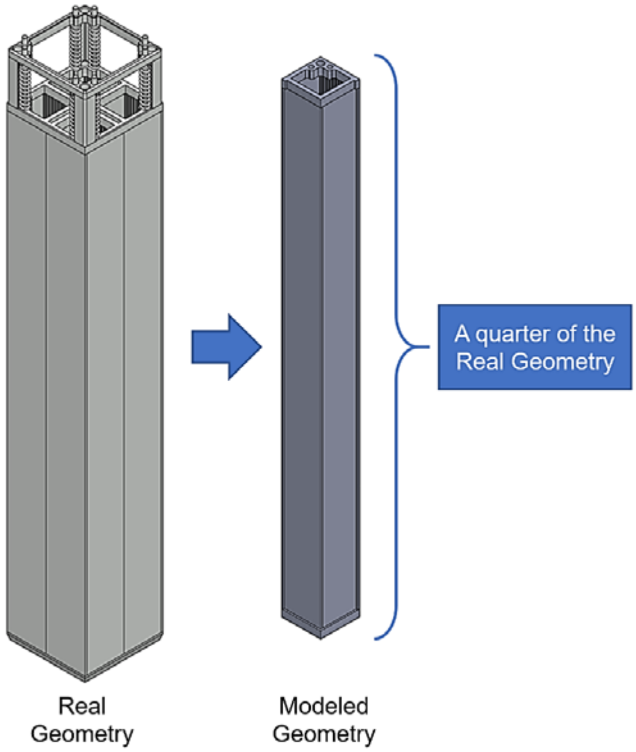

Figure 5 - Real Geometry and Modeled Geometry.
Source: M. M. Santos [2].

Table 2 - Main dimensions of fuel element

Description	Values	Unit
Active plate height	110.5	cm
Active plate width	9.435	cm
Cladding thickness of the fuel plate	0.020	cm
Cooling channel hydraulic diameter	0.581	cm
Cooling channel thickness	0.300	cm
Cooling channel width	9.435	cm
Fuel core volume	211.6	cm^3
Fuel plate thermal exchange area	2085	cm^2
Number of fuel plates per fuel element	72.	-
Thickness of the core of the fuel plate	0.203	cm
Transversal area of the fuel core of the fuel plate	1.915	cm^2
Transversal area of a cooling channel	2.830	cm^2

Source: Adapted from C.S. Andrzejewski *[2]*.

2.3. Finite Element Mesh

The modeled geometry is subdivided into several smaller elements that form a mesh of elements, so that the mathematical solution of the analyzed problem is obtained. The types of elements in this mesh and their characteristics, such as number of nodes and degrees of freedom per node, are configured in this step.

The verification of the meshes is based on a refinement analysis [3]. This check consists of observing the values of a variable in at least three different mesh refinement settings for the same analysis model, as shown below in Table 3 and Table 4. The purpose of this verification was to know if the refinement of the mesh interfered in the obtained results.

Table 3 - Thermo-hydraulic analysis verification.

Number of elements	Maximum coolant outlet temperature (°C)	Processing time
500.000	288,2	7 min 49 s
900.000	287,3	12 min 14 s
1.400.000	287,0	18 min 31 s

Source: M. M. Santos [2].

Table 4 - Thermomechanical analysis verification.

Number of elements	Plate 1 – Maximum Stress Intensity (MPa)	Processing time
215.000	20,79	67 min
100.000	20,80	28 min
70.000	20,79	2 min

Source: M. M. Santos [2].

For both analyses, the intermediary meshes were selected.

2.4. Active Loads and Their Combinations

In this step the defined (or pre-established) loads are inserted in their respective places of application in the modeled geometries, as forces, pressures and temperatures.

For the thermo-hydraulic analysis, it was considered a thermal load of $1{,}81.10^{8}$ W/m³ in the fuel core of the fuel plate. This heat source (or power density) corresponds to the nominal power of the reactor, 58 MW, divided by the total fuel volume present in the reactor.

For the thermomechanical analysis it was considered that during the event of operation at full power of a compact PWR reactor the fuel element assembly is subject to five active loads, as said before. The values of the active loads referring to this structure are shown in Table 5, and their considered combinations are shown in Table 6.

Table 5 – Values of the active loads.

Load	Value	Method of calculation
Self-Weight	3412 N	SolidWorks®
Hydrostatic thrust	281 N	Analytical method
Hydraulic drag	353,5 N	Analytical method
Temperature field	0,181 W/mm³	Ansys CFX®
Force of the fixing springs	1012 N	Ansys CFX®

Source: M. M. Santos [2].

Table 6 - Load combination.

Load Combination (L.C.)	Overloaded Uploads	Analysis System
1	Self-Weight Load + Load of the fixed springs	Static Structural®
2	L.C.1 + Hydrostatic thrust	
3	L.C.2 + Hydraulic Drag	
4	L.C.3 + Thermal Expansion	Steady-State Thermal® + Static Structural®

Source: M. M. Santos [2].

2.5. Restrictions

For the thermo-hydraulic modeling geometry, the faces of all bodies for which no loading or interfaces were assigned are defined as "*adiabatic walls*". All external walls of the model, except for the inlet and outlet faces of the coolant, are also defined as adiabatic walls and represent the geometric limits of the performed analysis.

For the thermomechanical model, the boundary conditions related to the restriction to the movement of the model in the three cartesian directions are configured, basically, in form of fixed supports and imposed displacements. The modeled geometry is constrained to represent the actual constraint condition of a fuel element, as shown in Figure 6. For this, the following conditions are imposed on this geometry:

- A restriction condition for the vertical movement of the fuel element was defined on all nodes present on the lower face of its lower nozzle (U(y)=0).
- On the symmetry faces of the modeled geometry located to the left of Figure 6 a constraint condition was defined with respect to the horizontal axis Z (U (z) = 0).
- On the symmetry faces of the modeled geometry located to the right of Figure 6, that will be shown below, a constraint condition was defined with respect to the horizontal axis X (U (x) = 0).

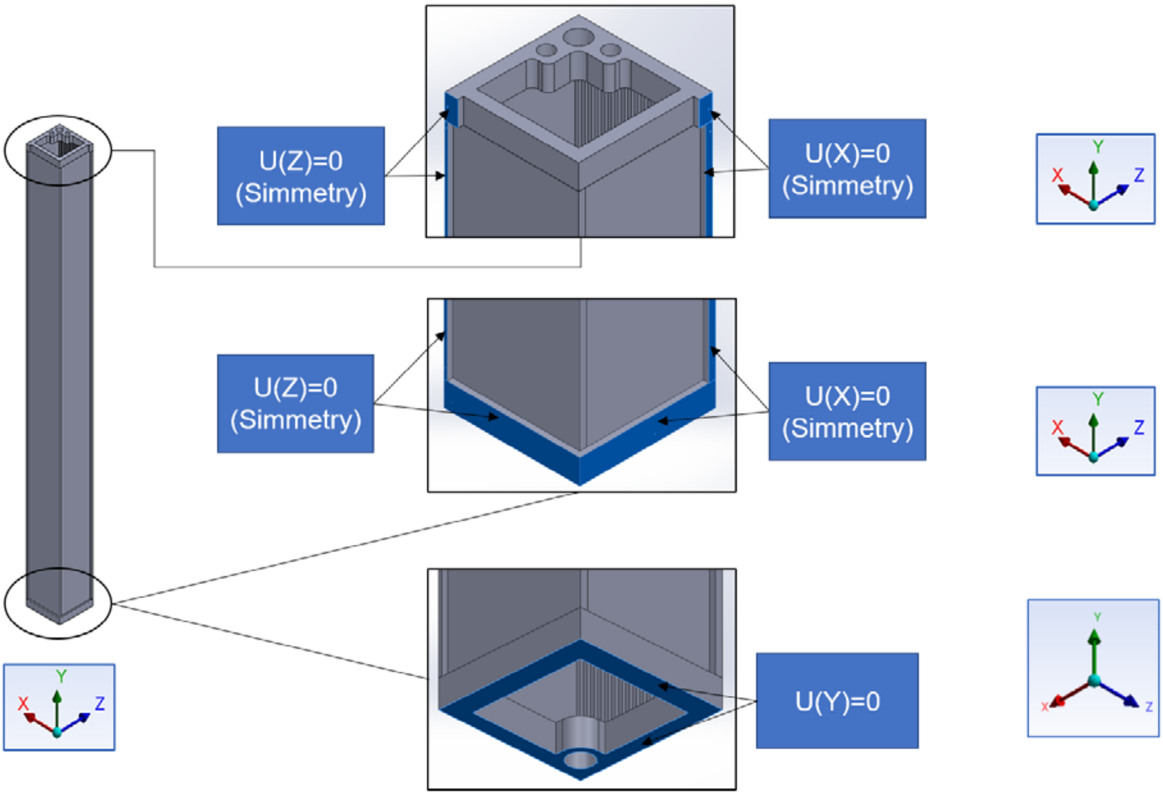

Figure 6 - Restriction conditions.
Source: M. M. Santos [2].

2.6. Contacts

For the thermo-hydraulic modeled geometry, made in the *CFX*® analysis system, the fuel core and cladding faces that are in contact with each other are defined as “*solid-solid*” interfaces, which is the actual boundary condition between these two bodies. The faces of the volumes of the cladding and the coolant which are in contact with each other have been defined as “*fluid-solid*” interfaces.

For the thermomechanical modeled geometry, that was made in the Steady-State Thermal® and in Static Structural® analysis systems, two types of linear contact with the modeled geometry were configured: *Bonded* and *No Separation*. *Bonded* refers to a contact in which there is no movement allowed between body faces. *No Separation* refers to a contact in which there is a sliding motion between body faces, without separation between them.

2.7. Properties of Materials

The considered properties of the materials in the analysis are shown in Tables 7, 8, 9 and 10.

Table 7 - Considered properties of Zircaloy-4.

T (°C)	ρ (Kg/m³)	k (W/m.k)	α (°C^-1)	E (GPa)	σ_e(MPa)	ν
25	6551	13,78	0,00E+00	92,4	941	0,36
100		15,30	1,26E-05	88,4	877	0,37
200		17,39	1,26E-05	82,9	788	0,38
300		19,48	1,26E-05	77,4	700	0,38
400		21,57	1,26E-05	71,9	612	0,39

Source – References *[1]*, *[4]* and *[5]*.

Table 8 - Considered properties of U-10Mo.

T (°C)	ρ (Kg/m³)	k (W/m.k)	α (°C^-1)	E (GPa)	σ_e(MPa)	ν
25	17130	12,11	1,19E-05	125	165	0,326
100	17060	14,35	1,31E-05	119		
200	16970	17,12	1,45E-05	105		
300	16880	20,06	1,60E-05	92,4		
400	16800	23,18	1,75E-05	79,3		

Source – References *[1]* and *[6]*.

Table 9 - Considered properties of Inconel 718.

T (°C)	ρ (Kg/m³)	k (W/m.k)	α (°C^-1)	E (GPa)	σ_e(MPa)	ν
20	17130	11,4	1,30E-05	200	1450	0,327
100	17060		1,31E-05	196		
200	16970		1,40E-05	190		
300	16880		1,44E-05	185		
400	16800		1,45E-05	179		

Source – Reference *[7]*.

Table 10 - Considered properties of water.

Material	T (°C)	Pr (MPa)	ρ (Kg/m³)	α (°C^-1)	k (W/m.k)	c (J/kg.°C)
Water	275	13	763,6	2,57E-04	0,5832	4181

Source – Reference *[8]*.

3. RESULTS

The values shown in Table 11 refer to the maximum stress intensity, and the values of Table 12 refer to the maximum displacement. These values refer to all components considered in the analyses performed, that considered the four load combinations showed in Table 6.

The Figure 7 shows the stress distribution in the fuel element, including the points where it is possible to find the maximum and minimum values.

Table 11- Maximum Stress Intensity (MPa).

Load Combination (L.C.)	Upper Nozzle	Plate A	Plate B	Plate C	Plate D	Cladding	Lower Nozzle
0	0,016	1,281	0,796	1,441	0,912	0,778	0,646
1	10,719	9,821	7,756	3,331	1,970	1,800	1,426
2	10,719	9,822	7,756	3,222	1,900	1,740	1,374
3	10,720	8,982	7,756	3,083	1,812	1,664	1,308
4	25,788	94,232	87,696	92,742	52,286	151,15	12,161

Source: M. M. Santos [2].

Table 12 - Maximum displacement in the fuel element (m).

Cartesian Axes	Upper Nozzle	Plate A	Plate B	Plate C	Plate D	Cladding	Lower Nozzle
X	3,67E-04	3,67E-04	3,61E-04	3,37E-04	3,48E-05	1,15E-04	3,50E-04
Y	4,60E-03	4,54E-03	4,53E-03	4,54E-03	4,54E-03	4,34E-03	6,46E-05
Z	4,54E-04	4,80E-04	3,64E-04	1,41E-04	4,58E-04	4,72E-04	3,51E-04

Source: M. M. Santos [2].

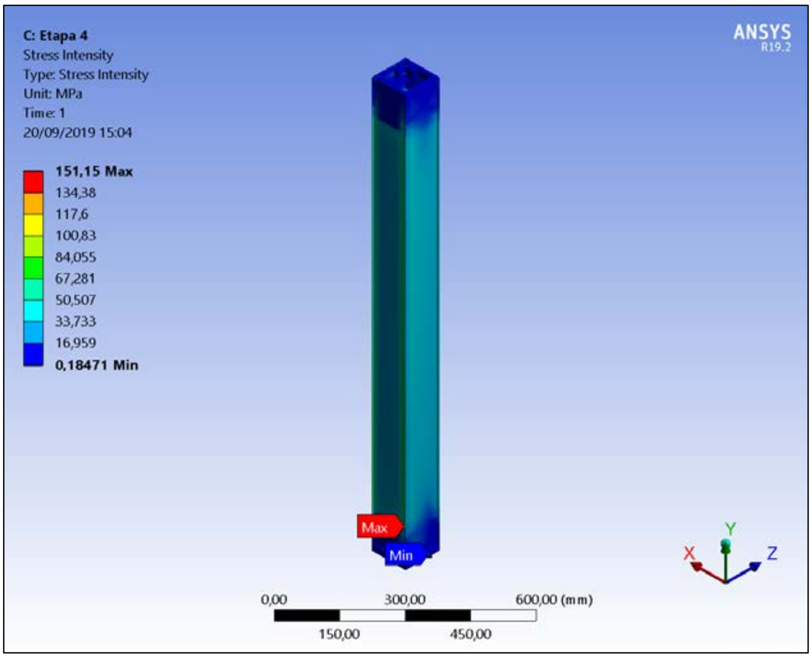

Figure 7 - Stress Intensity in the fuel element
Source: M. M. Santos [2].

4. ANALYSIS AND DISCUSSION OF THE RESULTS

According to the procedure of the design by analysis of *ASME III, division 1, subsection NB* [9], the permissible limits of the materials are compared with the Stress Intensity values resulting from the analyses performed, and are found in the *ASME II, Part D*. The values of the permissible limits for the materials that form the components of the fuel element analyzed, however, are not included in this standard. In this way, the admissible limits that will be shown below in Table 13 are based on the yield stress limits of the materials, that were shown in the Tables 7, 8 and 9, obtained by external references to ASME, and are correlated with their limits as follows:

- Sm: considered to be equivalent to $^2/_3$ of the material yield stress.
- 1,5 Sm: considered to be equivalent to the material yield stress.
- 3 Sm: considered as equivalent to twice the material yield stress.

According to *ASME III, division 1, subsection NB* [9], compliance with the Sm and 1,5 Sm (mechanical limits) assure that a component will not undergo plastic collapse, and compliance with 3 Sm limit ensures that a component will not suffer excessive deformation.

Plastic collapse is the failure of a component when the resulting mechanical stresses exceed the value of its yield strength limit. Excessive deformation occurs when the cycling of stresses in plastic regime (including the ones from thermal loading) causes the material to accumulate plastic deformation. It should be noted that all this is true for materials considered to be perfectly elastoplastic.

For all combinations of loads the fuel assembly must meet the limits of Sm, 1,5 Sm and 3 Sm. It should be noted, however, that in the load combinations 1, 2 and 3 there are only mechanical loads, and because of that it was assumed that the maximum mechanical limits in these cases are Sm up to 1,5 Sm. In the load combination 4 the fuel assembly is subjected to mechanical and thermal loads, and so it was assumed that the maximum stress limit in this case is up to 3 Sm.

Table 13 - Admissible limits.

Components	Materials	Admissible limits		
		Sm (MPa)	1,5 Sm (MPa)	3Sm (MPa)
Upper Nozzle	Zircaloy-4 (ASTM B351)	70	105	210
Plate A	Zircaloy-4 (ASTM B352)	66,7	100	200
Plate B	Zircaloy-4 (ASTM B352)	66,7	100	200
Plate C	Zircaloy-4 (ASTM B352)	66,7	100	200
Plate D	Zircaloy-4 (ASTM B352)	66,7	100	200
Cladding	Zircaloy-4 (ASTM B352)	66,7	100	200
Lower Nozzle	Zircaloy-4 (ASTM B351)	70	105	210

Source: M. M. Santos [2].

By comparing the values shown in Table 11 and Table 13, respectively corresponding to the maximum values of stress intensity and the values of the permissible limits, it is possible to verify that the fuel element is not damaged due to plastic collapse nor due to excessive deformation. The resulting values from the load combinations 1, 2 and 3 remained far from the Sm limit, which is the most conservative limit adopted for these three load combinations. The resulting stress intensity values from the load combination 4 are also shown to be lower than 3 Sm permissible limit of the materials.

Of all the components, in all load combinations, the cladding of one fuel plate presents the highest stress result value: 151,15 MPa. In general, no structural problems are observed in the fuel element when exposed to the full power operation event, belonging to the normal operating condition of a compact PWR reactor.

The heating of the fuel cores reaches all the components of the fuel element, causing their thermal expansion, which is not free, since there are vertical and lateral restrictions as to their movement in the core of the reactor. Herewith, there are the arising of stresses especially in the cladding of the fuel plates, which are in contact with the core. Therefore, considering the mechanical effects of thermal expansion of the fuel element is important since:

- The axial or vertical expansion of the fuel element cannot be greater than an assumed distance of 15 mm gap between the upper nozzle and the upper support structure of the reactor core, under risk of contact between these structures causing damage to both.

- The transverse expansion of the fuel element must not be such as to allow contact between the neighboring fuel elements. It is assumed a distance of 1 mm between them.

According to the results presented in Table 12, it is possible to observe that the value of the vertical expansion of the fuel element assembly is not greater than 15 mm, considered between its upper nozzle and the upper support structure of the core inside the pressure vessel. In the same way, it is observed that the values of the resulting horizontal displacements, corresponding to the X and Z axes, meet the limit of 1 mm of distance considered among the neighboring fuel element, which is to say that one fuel element does not come into lateral contact with neighboring fuel element in the reactor core.

5. CONCLUSION

From the results here presented, it is concluded that the developed methodology allows the mechanical dimensioning and the structural evaluation of the fuel element considering the main loads coming from the operation event at full power (of the normal operating condition) of a reactor compact PWR.

The developed methodology allowed to analyze the mechanical integrity of one Plate-Type Fuel Element structure through simulations of the application of the loads in a computational conceptual model of this, through which it was possible to make preliminary studies without the need to build an experimental set-up, contributing to reduce the costs of a possible project. Through this numerical study, it was possible to conclude that the conceptual structure of the parallel plate assembly could be an alternative substitute of rod-type fuel elements in compact PWR reactors.

ACKNOWLEDGEMENTS

The authors are grateful for the technical support provided by IPEN-CNEN/SP, CTMSP and USP.

REFERENCES

1. KAUFMANN, ALBERT R. **Nuclear Reactor Fuel Elements**. New York-London: John Wiley and Sons, Inc., 1962.

2. SANTOS, M. M. D. **Avaliação estrutural de um elemento combustível do tipo placa para um reator nuclear compacto.** São Paulo: Instituto de Pesquisas Energéticas e Nucleares - IPEN/USP, 2019.

3. ANDRZEJEWSKI, C. D. S. **Avaliação de alternativas de combustível tipo placa para reatores de pequeno porte**. São Paulo: IPEN - Instituto de Pesquisas Energéticas e Nucleares, 2005.

4. KWASNIEWSKI, L. **Application of grid convergence index in FE computation**. Warszawa: Bulletin of the polish academy of sciences, v. 61, 2013. 123-128 p.

5. INTERNATIONAL, A. **ASTM B351/B351M - Standard Specification for Hot-Rolled and Cold-Finished Zirconium and Zirconium Alloy Bars, Rod and Wire for Nuclear Application.** West Conshohocken, PA. 2017.

6. INERNATIONAL, A. **ASTM B352/B352M - Standard Specification for Zirconium and Zirconium Alloy Sheet, Strip and Plate for Nuclear Application**. West Conshohocken, PA. 2017.

7. LABORATORY, I. N. **Thermophysical Properties of U-10Mo Alloy**. Idaho: Idaho National Laboratory, 2010.

8. INTERNATIONAL, A. W. **Technical Datasheet AWS 013 Ver.1**. [S.l.]: [s.n.].

9. KLEIN, S. A. **Engineering Equation Solver**. Suwon Kyungki-Do: Samsung Electronics Co. Living Systems R & D Center, 1992-2003.

10. ASME. **Asme Boiler and Pressure Vessel Code - Section III**. Nova York: The American Society of Mechanical Engineers, 2015.