ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR – ABEN
2019 International Nuclear Atlantic Conference - INAC 2019
Santos, SP, Brazil, October 21-25, 2019

Agência Brasileira do ISBN
ISBN 978-85-99141-08-3

ANALYSIS OF STRESSES ACTING ON THE INTERNAL AND EXTERNAL SURFACES OF FUEL ROD OF A PRESSURIZED WATER REACTOR USING COMPUTATIONAL SIMULATION

Leonardo S. Lima[1], Caio Melo[1], Danilo P. Faria[1], José Berretta[1], Amanda Abati[2] and Claudia Giovedi[2]

[1] Diretoria de Desenvolvimento Nuclear da Marinha (DDNM)
Av. Professor Lineu Prestes, 2468
05508-000 São Paulo, SP, Brazil
leonardolima@marinha.mil.br

[2] Instituto de Pesquisas Energéticas e Nucleares (IPEN / CNEN - SP)
Av. Professor Lineu Prestes, 2242
05508-000 São Paulo, SP, Brazil
amanda.abati.aguiar@gmail.com

[2] Analysis, Evaluation and Risk Management Laboratory (Poli / USP)
Av. Prof. Mello Moraes, 2231
05508-000 São Paulo, SP, Brazil
claudia.giovedi@labrisco.usp.br

ABSTRACT

During operation of a Pressurized Water Reactor, the cladding of the fuel rod is subjected to various loads, such as: temperature, internal pressure, and external pressure which generate dimensional and geometric variations in the cladding tube. In the fuel rod, at the operating temperature, the internal pressure comes from the initial pre-pressurizing with Helium gas and the release of fission gases by the UO_2 pellets during the irradiation. The external pressure is assumed to be the same as that of the coolant. In this paper, it was proposed the study of a mathematical model for computational simulation using the Finite Element Method to calculate and analyze the mechanical stresses acting on the internal and external surfaces of the fuel rod, adopting the normal operating condition, at 0 W of power. The boundary conditions, such as temperature and pressure profile, come from a modified version of a fuel performance code, considering as cladding material an iron-based alloy (austenitic stainless steel). The fuel rod was modeled and simulated using the Solidworks and ANSYS softwares, respectively. The values of the stresses acting on the cladding tube obtained by simulation were compared to the values obtained by analytical calculation. Then, it was checked the consistency of the adopted mathematical model, in order to ensure the reliability of the computational simulation as a tool to evaluate the stresses acting on the internal and external surfaces of the fuel rod under a PWR environment.

1. INTRODUCTION

During the operation of a PWR reactor, the fuel rod is subjected to various loads due to the irradiation conditions, such as temperature variation, internal and / or external overpressure, power variation, among others. There are also dimensional and geometric variations of the rod design, such as: dimensional tolerances, ovalization and concentricity.
The stresses resulting from internal overpressure on the internal and external surfaces of the rod vary according to the difference between the internal pressure from the pressurization (predicted in design) and also the gases released by the fission of the fuel pellets and the external pressure, which is properly the pressure of the coolamt fluid. These factors influence the stresses acting on the surfaces of the tube and an analysis of them is necessary to guarantee the integrity of the fuel rod during all cycles of operation of the reactor.
The present article aims to calculate, analyze and validate a mathematical model of computational simulation for the mechanical applications acting on a Series 300 Stainless Steel fuel rod for the zero power case of a PWR reactor. FEM (Finite Element Methods) was adopted using the ANSYS code by comparison with the results obtained by employing open literature formulas.

2. METHODOLOGY

As a basis for the analytical calculations of the stresses acting on the fuel rods, a computational code of stress analysis reproduced in spreadsheets was used. With the analytical results used as reference, a computer simulation was then performed using the Finite Element Method using ANSYS software. The boundary conditions used for this study, admitting a PWR reactor operating at zero power were:

- Geometry of the rod was adopted according to Figure 1, below:

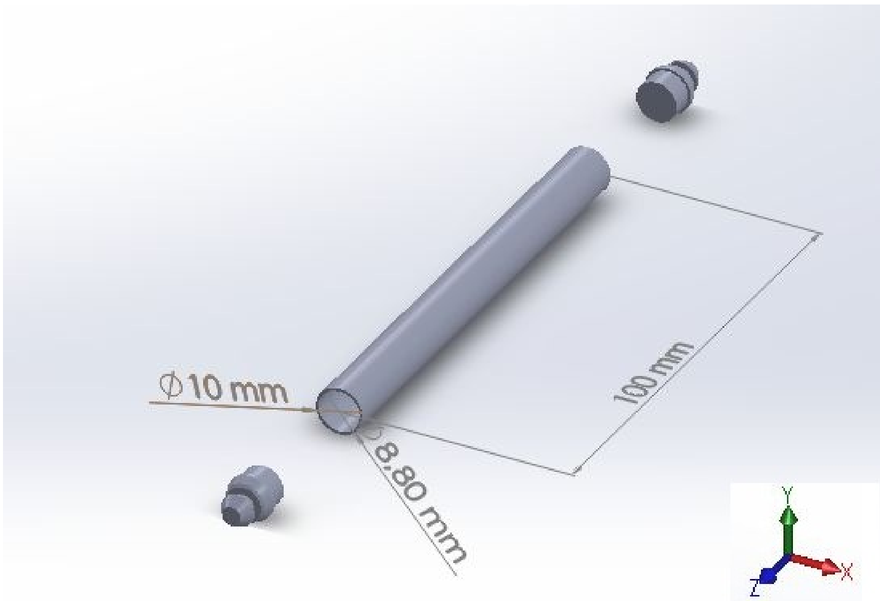

Figure 1: Geometry fuel rod.

- Use of Series 300 Stainless Steel [1], as tube material, subjected to a temperature of 200 ° C.

- The fuel rod is subjected to overpressure on its internal and external surfaces, the external pressure being the same pressure as the coolant fluid. The internal pressure value of the fuel rod was calculated by the simulation with the fuel performance code, FRAPCON [2]

Table 1: Pressures acting on the Fuel Rod

Internal Pressure (MPa)	6,00
External Pressure (MPa)	15,00

2.1. Analytical Calculation

An electronic spreadsheet was used to obtain the stress values due to various loads acting on the rod. The calculations are made with analytical formulas in the elastic region, both for the inner surface and the outer surface of the fuel rod in the radial, axial and tangential directions. As shown in Figure 2.

Radial Stress (Direction of X Axis);
Tangential Stress (Direction of Y Axis)
Axial Stress (Direction of Z Axis)

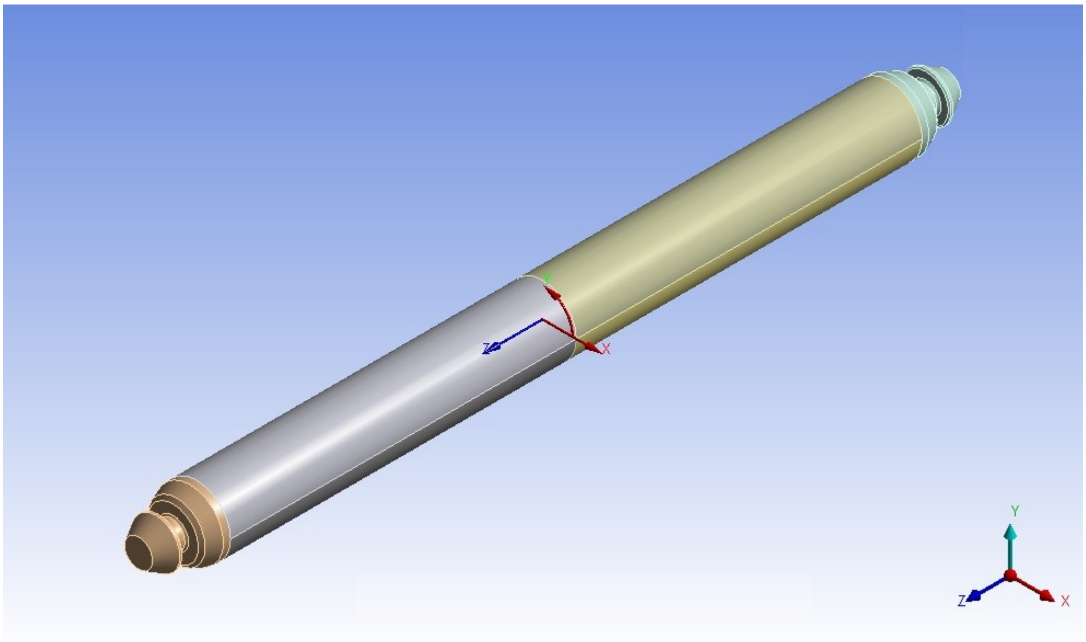

Figure 2: Orientation of the fuel rod Tensions Axes.

It is verified that the fuel rod is a cylinder of the type "thick walls". Thus, the Lamé formulation [3] is used. For a closed cylinder subjected to internal and external pressure, as shown in Figure 4, the Lamé equations are:

Tangential Stress:

$$\sigma_T = \frac{p_i r_i^2 - p_0 r_0^2}{r_0^2 - r_i^2} + \frac{r_i^2 r_0^2 (p_i - p_0)}{r^2 (r_0^2 - r_i^2)} \tag{1}$$

Radial Stress:

$$\sigma_R = \frac{p_i r_i^2 - p_0 r_0^2}{r_0^2 - r_i^2} - \frac{r_i^2 r_0^2 (p_i - p_0)}{r^2 (r_0^2 - r_i^2)} \tag{2}$$

Axial Stress:

$$\sigma_A = \frac{p_i r_i^2 - p_0 r_0^2}{r_0^2 - r_i^2} \tag{3}$$

r_i - internal radius

r_0 - external radius

p_i - internal pressures

p_0 - external pressure

r - radius of the place of interest

By inserting the values of the rod geometry, the internal pressures (provided by the FRAPCON program) and external pressures (foreseen in design) into the spreadsheet, the following stress values due to the overpressure are obtained for the inner and outer surfaces of the fuel rod, outside the welding region, calculated analytically, according to Tables 2 and 3.

Table 2: Stress acting on the fuel rods

Internal Surface	STRESS ANALYSIS CODE
Tangential Stress (MPa)	-85,787
Radial Stress (MPa)	-6,00
Axial Stress (MPa)	-45,894

Table 3: Stress acting on the fuel rods

External Surface	STRESS ANALYSIS CODE
Tangential Stress (MPa)	-76,787
Radial Stress (MPa)	-15,00
Axial Stress (MPa)	-45,894

2.2. Computer simulation

The geometry was initially designed in the software SOLIDWORKS and later the simulation of the model through the ANSYS [4] program. A thermal simulation (Steady State Thermal) was performed to evaluate possible changes in the mechanical properties of the austenitic stainless steel at a temperature of 200º C. A Static Structural analysis was performed to determine the stresses acting on the surfaces of the fuel rod.

2.2.1. Discretization of the Model

The upper and lower plugs of the rod were modeled with solid finite elements of tetrahedral geometry. The fuel rod was modeled with hexahedral finite elements and along its thickness was divided into 3 elements, ensuring a good precision of the calculated stress values in the tangential, radial and axial directions. In all, the resulting mesh presents approximately 210,699 nodes and 92,652 elements, as can be observed in Figure 3.

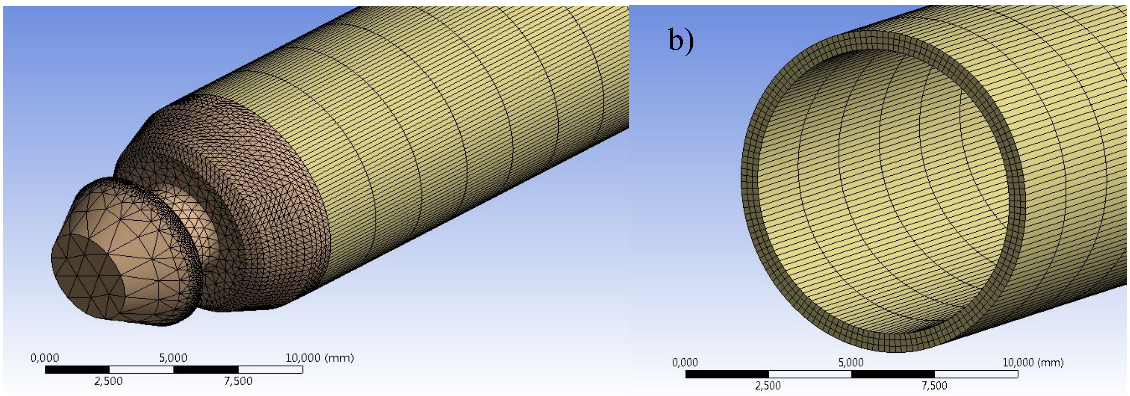

Figure 3: Finite element mesh for: (a) Plug geometry; (b) Fuel rod.

2.2.2. Boundary Conditions

The temperature to which the fuel rod is subjected when the reactor operates at zero power is 200º C. Therefore, this temperature was assigned as constant during the simulation. The internal pressure provided by the FRAPCON code (6.00 MPa) due to the pressurizing of the helium gas, the temperature and the production of the fission gases, besides the external design pressure (15.00 MPa) to which the rod is subjected, were applied uniformly over the whole internal and external surface of the rod. The translation of the fuel rod plugs is constrained in the axial direction (Z axis) through the Displacement bracket according to Figure 10. In the other directions (X and Y axis), movement is free.

2.2.3. Contacts for modeling during simulation

Two types of contact were used: Bonded and Frictionless to represent the surfaces in contact between the fuel rod and the plug. Bonded contact means welding between the plug and the tube: there is no penetration or gap between these surfaces. The Frictionless type contact is used between the inner surface of the tube and the inner part of the plug: this type of contact allows separation of the surfaces contact, as well as tangential slipping without any hindrance. Figures 4 and 5 below show the contacts used**.**

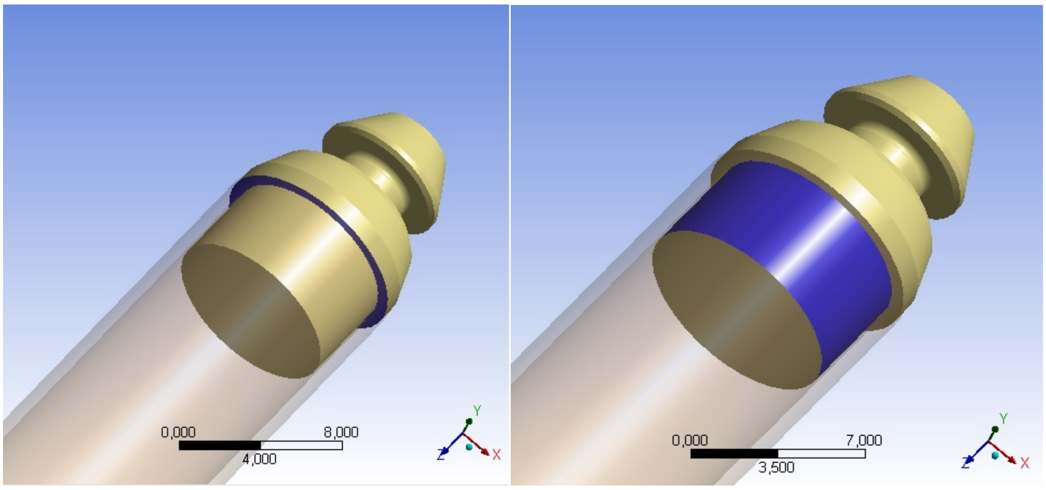

Figure 4: Representation of the contacts between tube and plug adopted in ANSYS computational code: a) Bonded, b) Frictionless

3. RESULTS AND DISCUSSIONS

The loads considered and studied were obtained on the internal and external surfaces of the fuel rod.

3.1. Overpressure stresses, outside the welding range, acting on the inner surface of the rod

To check the overpressure stresses acting on the inner surface of the tube, outside the welding strip, several random points were selected along the fuel rod. Figures 05, 06 and 07 show the axial, radial and tangential stresses of the rod, respectively, obtained by the ANSYS computational code.

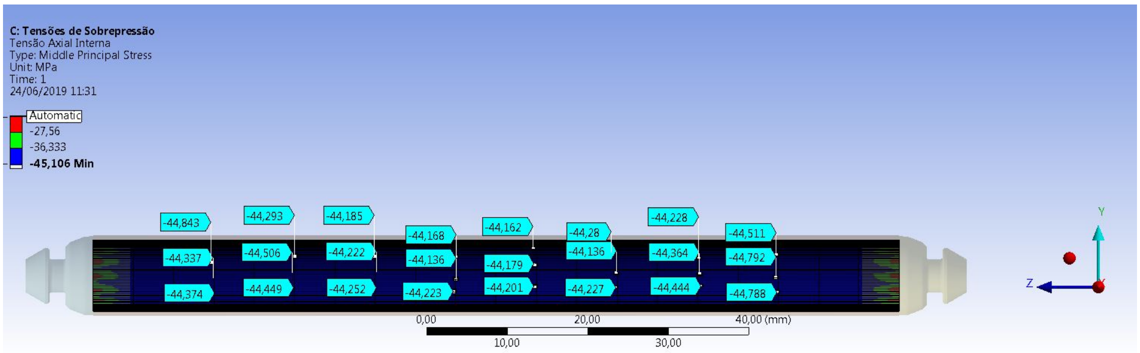

Figure 5: Representation of the axial stresses along the inner surface of the rod, outside the region of the weld.

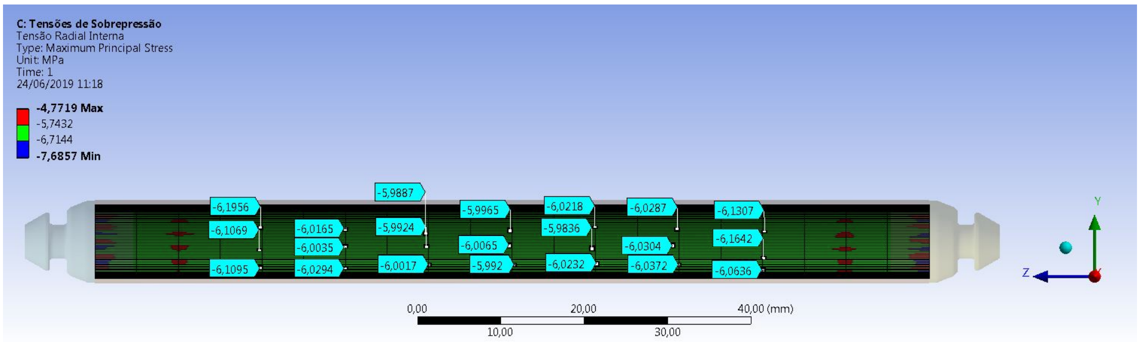

Figure 6: Representation of the radial stresses along the inner surface of the rod, outside the region of the weld.

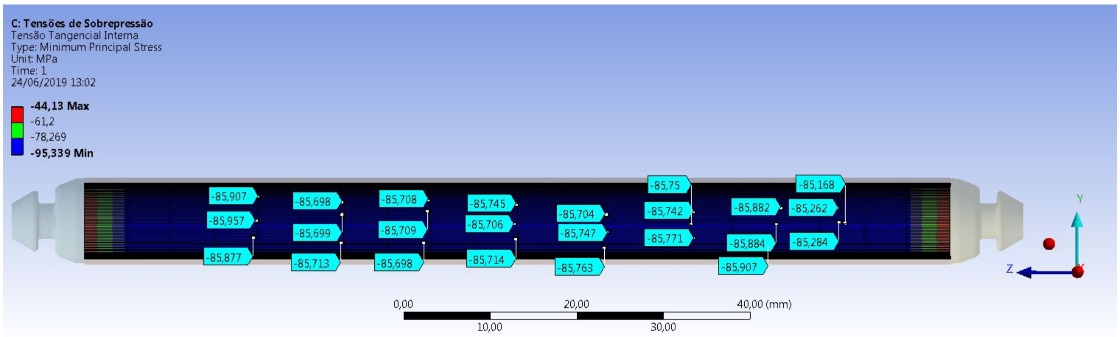

Figure 7: Representation of the tangential stresses along the inner surface of the rod, outside the region of the weld.

Among the overpressure stress values obtained along the inner surface of the rod, for each direction (tangential, radial and axial), small variations can be noted in the values found, shown by the standard deviations, according to Table 4. It can be seen a smaller deviation value for the results in the radial direction compared with the deviations in the axial and tangential directions. This is due to the better discretization of the mesh in this direction.

Table 4: Overpressure stresses in the tangential, radial and axial directions acting on the inner surface of the rod obtained with the ANSYS code.

	Tangential Stress (MPa)	Radial Stress (MPa)	Axial Stress (MPa)
	-85,907	-6,1321	-44,374
	-85,957	-6,0593	-44,337
	-85,877	-6,0992	-44,843
	-85,698	-6,0185	-44,449
	-85,699	-6,0423	-44,506
	-85,713	-6,0722	-44,293
	-85,708	-6,0107	-44,252
	-85,709	-5,9584	-44,222
	-85,698	-6,0118	-44,185
	-85,745	-6,0067	-44,223
	-85,714	-5,9896	-44,168
	-85,704	-5,9784	-44,201
	-85,747	-6,0046	-44,179
	-85,763	-6,0001	-44,162
	-85,742	-5,9945	-44,227
	-85,771	-6,0134	-44,136
	-85,882	-5,9956	-44,444
	-85,884	-6,0424	-44,364
	-85,907	-6,0673	-44,228
Average	-85,775	-6,021	-44,296
Standard Deviation	0,084	0,043	0,161

A comparison made between the average of the overpressure stresses acting on the inner surface of the rod in the three directions, performed by the simulation with the ANSYS code, and their respective values calculated analytically by the stress analysis code, shows a very small difference comparing the two methods , as shown in Table 5 below:

Table 5: Comparison of the values of the overpressure stresses acting on the inner surface of the rod obtained with the ANSYS code and with the values obtained analytically.

INTERNAL SURFACE	ANALYTICAL	ANSYS	DIFFERENCE (%)
Tangential Stress (MPa)	-85,787	-85,775	0,014
Radial Stress (MPa)	-6,000	-6,021	0,354
Axial Stress(MPa)	-45,894	-44,296	3,482

3.2. Overpressure stresses, outside the range of welding, acting on the external surface of the fuel rod

To check the overpressure stresses acting on the outer surface of the rod, outside the weld range, several random points were selected along the rod. Figs. 08, 09 and 10 show the axial, radial and tangential stresses of the rod, respectively, obtained by the ANSYS computational code.

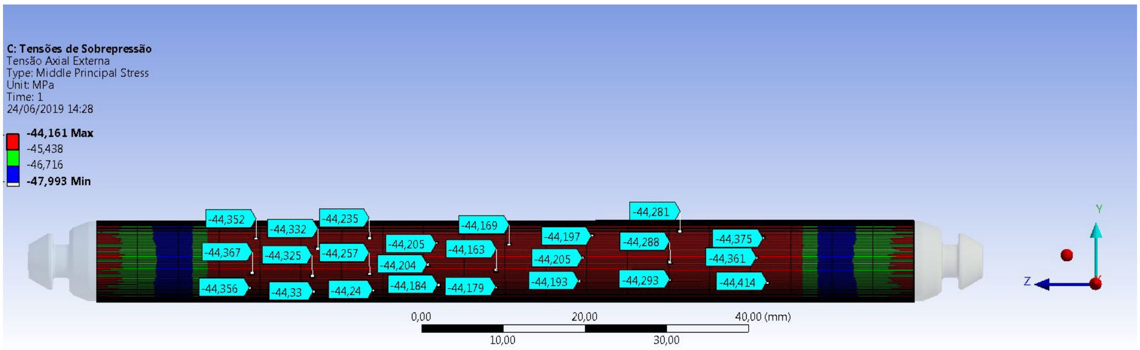

Figure 8: Representation of the axial stresses along the outer surface of the rod, outside the region of the weld.

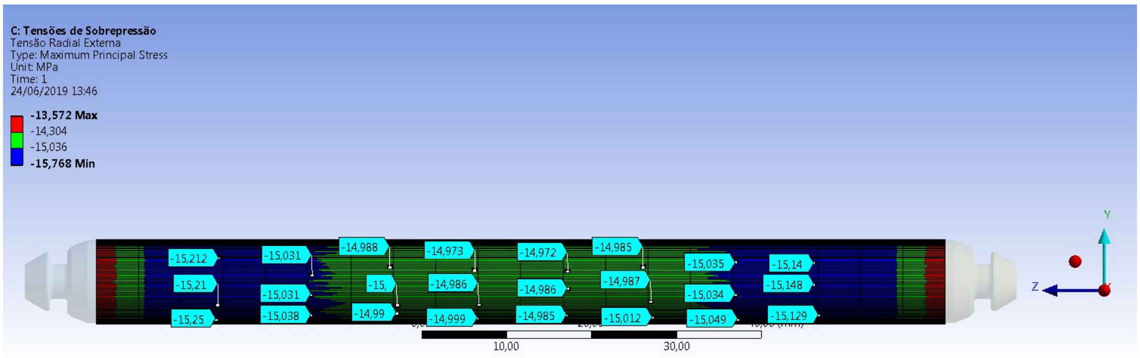

Figure 9: Representation of radial stresses along the outer surface of the rod, outside the region of the weld.

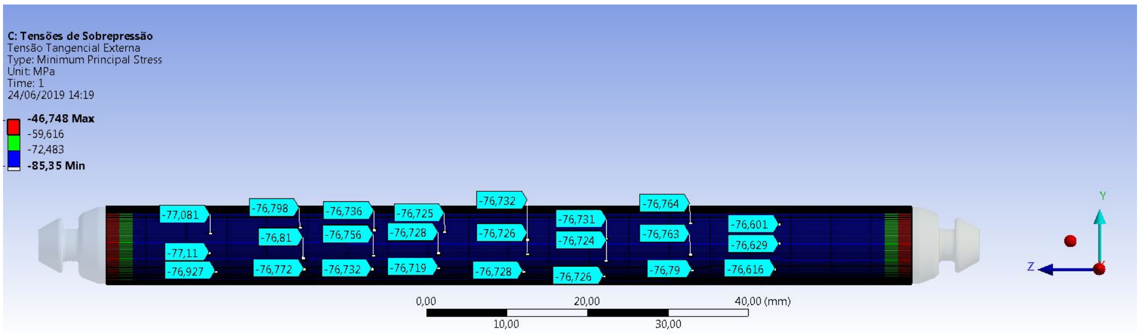

Figure 10: Representation of the tangential stresses along the outer surface of the rod, outside the region of the weld.

Among the overpressure stress measurements performed along the outer surface of the rod, for each direction (tangential, radial and axial), small variations can be noted in the values found, shown by the standard deviations, according to Table 6. It can be seen a smaller deviation value for the measurements in the radial direction compared with the values of deviations in the axial and tangential directions. This is due to the better discretization of the mesh in this direction.

Table 6: Overpressure stresses in the tangential, radial and axial directions acting on the outer surface of the rod, outside the weld region, obtained with the ANSYS code.

	Tangential Stress (MPa)	Radial Stress (MPa)	Axial Stress (MPa)
	-77,081	-15,212	-44,352
	-76,772	-15,038	-44,33
	-76,798	-15,031	-44,332
	-76,736	-14,988	-44,235
	-76,719	-14,999	-44,184
	-76,728	-14,986	-44,204
	-76,725	-14,973	-44,205
	-76,728	-14,985	-44,179
	-76,726	-14,986	-44,163
	-76,732	-14,972	-44,169
	-76,726	-15,012	-44,193
	-76,724	-14,987	-44,205
	-76,731	-14,985	-44,197
	-76,79	-15,049	-44,293
	-76,763	-15,034	-44,288
	-76,764	-15,035	-44,281
Average	-76,789	-15,036	-44,255
Standard Deviation	0,110	0,080	0,067

A comparison made between the mean of the overpressure stresses acting on the outer surface of the fuel rod in the three directions, performed by the simulation with the ANSYS code, and their respective values calculated by the stress analysis code, show a very small difference comparing the two methods, as shown in Table 7 below:

Table 7: Comparison of the mean values of the overpressure stresses acting on the external surface of the rod obtained with the ANSYS code and with the values obtained analytically

EXTERNAL SURFACE	ANALYTICAL	ANSYS	DIFFERENCE (%)
Tangential Stress (MPa)	-76,787	-76,789	0,003
Radial Stress (MPa)	-15,000	-15,036	0,239
Axial Stress(MPa)	-45,894	-44,255	3,571

4. CONCLUSIONS

The results of the stresses acting on the internal and external surfaces, outside the region of the welding of the fuel rod, due to the overpressure obtained through computational simulation were similar to those found analytically by the Code of Analysis of Stresses. In none of the directions of the stress results (axial, radial and tangential) at randomly chosen points along the fuel rod, did the value found exceed 4.00 %, proving the precision of the computational simulation. The divergence of the values is due to the difference in calculation methodology: while the stress analysis code use analytical tools, based on a 2D analysis, the ANSYS uses the finite element method applied to 3D geometry. Given the low deviation of results between the analytical and the simulation, it is verified that the mathematical model for the computational simulation is reliable.

REFERENCES

1. D. PECKNER; I. M. BERNSTEIN, Handbook of Stainless Steels, MacGraw Hill Book Company, 1977..
2. K. J. GEELHOOD; W. G. LUSCHER; C. E. BEYER, M.E. FLANAGAN, *FRAPCON-3.4: A Computer Code for the Calculation of Steady-State Thermal-Mechanical Behavior of Oxide Fuel Rods for High Burnup*, U.S.NRC, NUREG/CR-7022, Vol.1, PNNL-19418-Vol.1, March 2011.
3. E. R. Norton, *Projeto de Máquinas – Uma abordagem Integrada*; editora: Bookman; 2ª edição (2004).
4. ANSYS, User's Maual, Version 15, Swanson Analysis Systems, Inc.