# STUDY TO IMPLEMENT 3D STRESS-STRAIN ANALYSIS AT VARIOUS TEMPERATURES FOR STANDARDIZED TENSILE TESTS ON GEN IV STRUCTURAL MATERIALS

*DENISA TOMA [1)], NICU BIZON [2)], ALEXANDRU TOMA [1)], ALEXANDRA JINGA [1)], VALENTIN OLARU [1)], MIRELA MATEI [1)], AMALIA POPA [1)], GRAŢIELA BĂDULESCU [1)]*

[1)] Institute for Nuclear Research (RATEN ICN) Piteşti, Romania
[2)] National Univ. of Science and Technology POLITEHNICA Bucharest, Piteşti Univ. Centre, Romania
*e-mail:* denisa.toma@nuclear.ro

*Abstract*

Gen IV reactors are designed to operate at higher temperatures compared to Gen III reactors, and the problem of the behaviour of structural materials at high temperatures becomes of great importance. The mechanical degradation induced by the liquid metal can be either of the embrittlement type or environmental-assisted degradation, and the material properties can be affected. Experimental tests carried out for the characterization of structural materials can take a long period, and the optimization of the working time is considered by using some simulation codes. The paper develops a first approach to use the finite element modelling of uniaxial mechanical stresses of a typical 316L stainless steel sample in the ANSYS Workbench working environment by creating a typical tensile model. It was obtained the field of mechanical stresses and deformations. The results will be used as input for ongoing modelling of 316L stainless steel creep for specimens tested in various environments.

***Keywords:*** *Finite element analysis, 316L stainless steel, Generation IV nuclear reactors*

## Introduction

The Advanced Lead Fast Reactor European Demonstrator (ALFRED) integrates new design concepts for use in power plants to provide viable technical solutions to simplify the qualification and licensing process [1, 2].

The topic of the behavior of structural materials at high temperatures has gained enormous importance since Generation IV reactors are designed to operate at higher temperatures than Generation III reactors [3, 4].

Experiments designed to characterize the behavior of structural materials under mechanical loads are known as mechanical tests. Optimization of working time is taken into consideration, given that the actual testing required to characterize structural materials might be laborious. This can be accomplished by utilizing certain simulation codes. In this regard, finite element simulation of mechanical testing will be performed using the ANSYS code [5, 6].

Through a preliminary investigation on the stress-strain states seen in traditional tensile testing, this work intends to emphasize the behavior of structural materials under conditions similar to those corresponding to the operation of Generation IV nuclear reactors. In order to conduct this investigation by using the ANSYS code, the finite element modeling approach was applied to a tensile sample made of 316L stainless steel. By processing the results, the mechanical stress-strain curves in the elastic and elasto-plastic domains were obtained, being produced as ″.xls″ files after the simulations were performed.

## Tensile testing of materials and characteristic curve of steel

The mechanical stress-strain curve illustrates how a material behaves under different loads and is derived from laboratory tests conducted on tensile, compression, torsion, or creep, among other materials.

To define the behavior of the material, its characteristic curve must be drawn (see Fig. 1), which expresses the connection between the stress σ and the specific deformation ε. This diagram characterizes the behavior under loads of material as follows:

- point A: the limit of proportionality given by the stress $\sigma_p = \frac{P_p}{A_0}$, up to which the material obeys Hooke's law;
- point B: the limit of elasticity given by the stress $\sigma_e = \frac{P_e}{A_0}$, up to which no permanent deformations remain upon unloading;
- point C: the yield stress given by the stress $\sigma_c = \frac{P_c}{A_0}$, that is the stress for which the deformations increase without the load increasing;
- point E: corresponding to the highest stress that occurs in the sample, $\sigma_{max} = \frac{P_{max}}{A_0}$, after which the deformations increase for decreasing values of the force P.

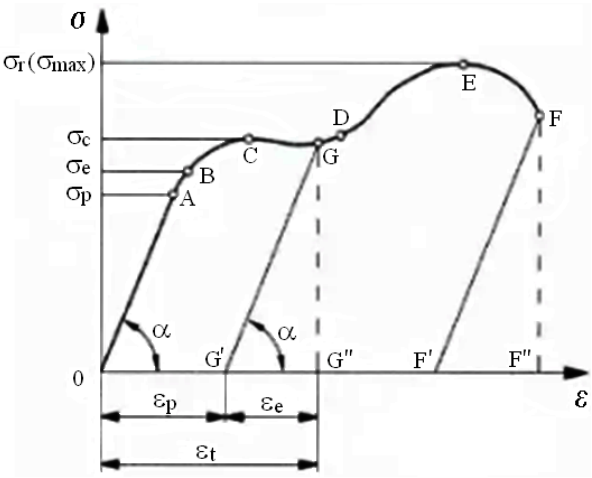



Fig. 1. Material characteristic curve

Although the breaking of the sample occurs at a force $P_r < P_{max}$, that is at a stress $\sigma_r < \sigma_{max}$, it is conventionally considered as the stress corresponding to the break, the stress $\sigma_{max}$.

The longitudinal modulus of elasticity, E, will be equal to the trigonometric tangent of the angle of inclination of the line OA with the axis Oε:

$$E = \text{tg}\,\alpha \quad (1)$$

Beyond the proportionality limit, the modulus E will be variable and will be expressed by:

$$E = \lim_{\Delta\varepsilon \to 0} \frac{\Delta\sigma}{\Delta\varepsilon} \quad (2)$$

If the sample is loaded and unloaded, as long as it is below the limit of elasticity $\sigma_e$, the portion of the characteristic curve BAO is described without permanent deformations remaining in the sample. If the loading produced in the sample stress exceeds the elastic limit (point G in Fig. 1), then upon unloading, the GCBAO curve is no longer described, but the GG' curve, which is approximately a straight line parallel to OA. The total specific elongation corresponding to point G on the curve is $OG'' = \varepsilon_p$, a permanent strain that remains after unloading [7, 8].

**Finite element modelling of the tensile test**

Finite element analysis (FEA) involves three stages of work: data preprocessing, processing, and post-processing.

Nodal coordinates, element connectivity, boundary conditions, material qualities, loads, and data entry and preparation are all included in preprocessing. There are two options for introducing data: reading from a data file or entering data interactively using an intuitive interface. By using FEA modeling, some input data can be loaded from other CAD (computer-aided design) tools. Nodal coordinates, the best way to number nodes, and the specification of element connection, all can be defined using automatic discretization mesh generation.

During the processing phase, the input data are processed by the finite element program, which then computes nodal variables like displacements and temperatures (equation solving), and quantities like stresses and primary variable derivatives (back calculation).

Post-processing stage is focusing on presentation of the results. Tabular listing was utilized by earlier calculators. Most of the applications available today graphically depict the stress distribution, the eigenmodes of vibration's forms, and the deformed configuration [9, 10].

*Geometry creation and sample discretization*

The material properties and the shape modeled using the finite element approach were established after the outcomes of the mechanical tests were evaluated. The model to be built in ANSYS Workbench was selected to be the geometry of a tensile sample (see Fig. 2) that was made in accordance with the ASTM E8 standard [11].

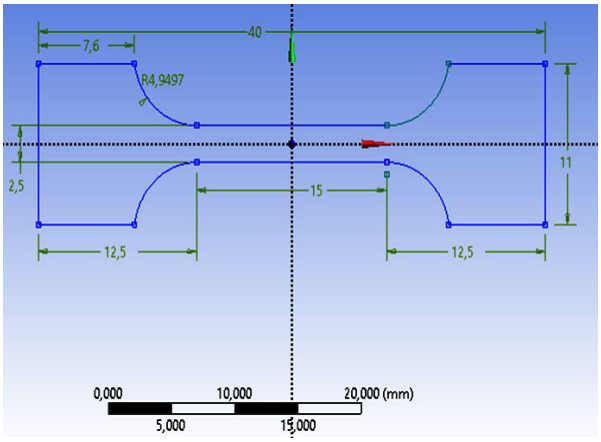



Fig. 2. Tensile sample sketch made using ANSYS code

The 316L steel was selected as the sample's material. The geometry and surface of the usual tensile test were produced, considering the steps needed for modeling with finite elements (see Fig. 3) [10, 12, 13].

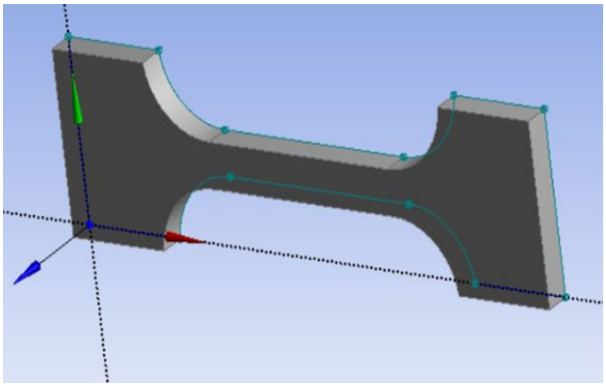


Fig. 3. Typical tensile test surface

The model was conditionally discretized using a mesh size of 0.5 mm, keeping in mind that the mesh is utilized to see the area under tension in detail (see Fig. 4) [14].

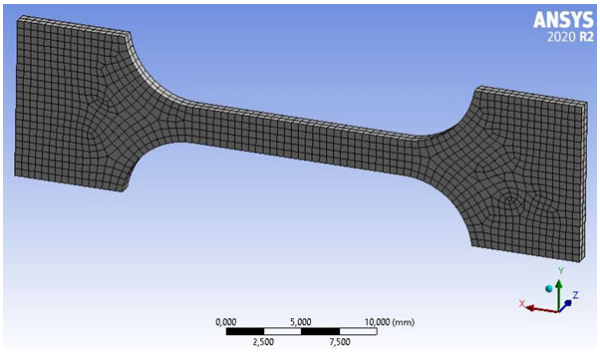



Fig. 4. Conditional discretized model (0.5 mm mesh size)

*Material creation and geometric constraint modelling*

The ANSYS material library can be accessed through two different methods: first, by selecting the desired material from the list, or second, by creating new material that is not already included in the library.

Subsequently, 316L steel was developed as the material for modeling in both the elastic (see Fig. 5), and elasto-plastic domains (see Fig. 6) [15, 16].

Fig. 7 shows the modeling of constraints and geometric loadings considered in simulating the 316L steel tensile sample.

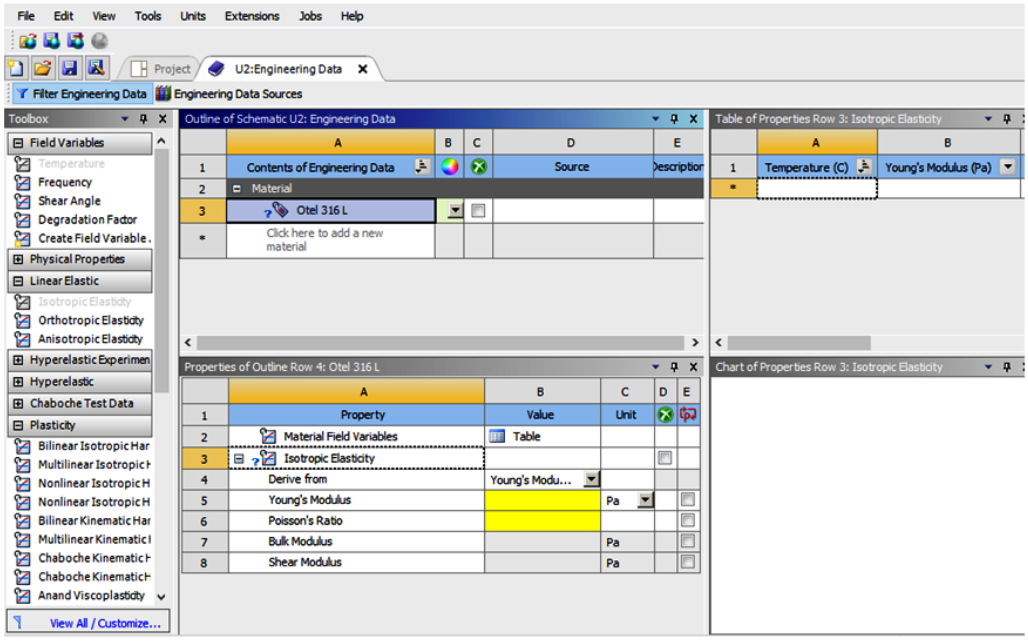



Fig. 5. Creating a new material for the elastic domain

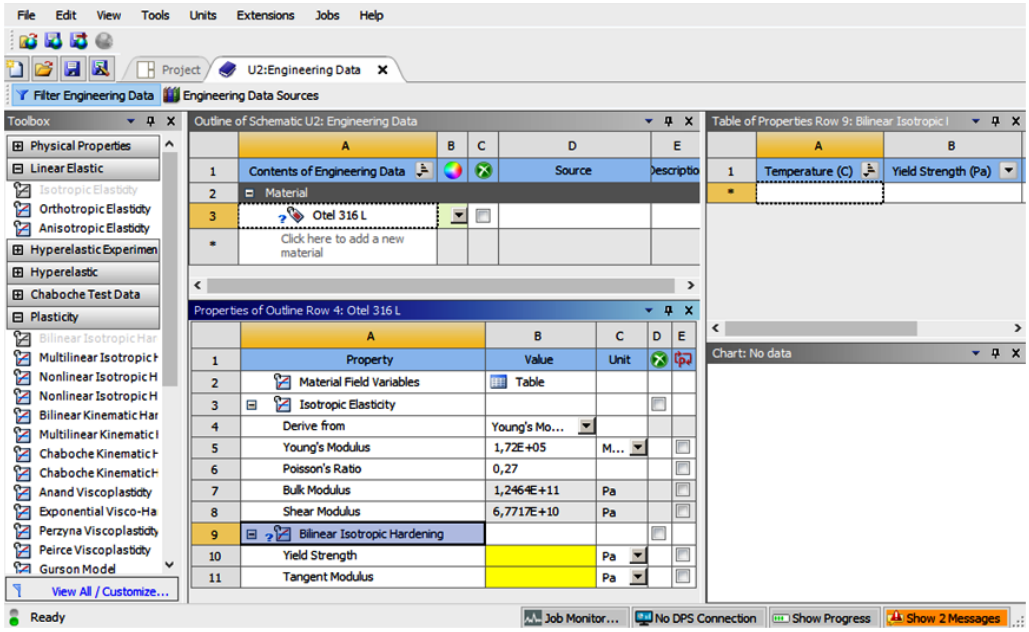



Fig. 6. Creating a new material for the elasto-plastic domain

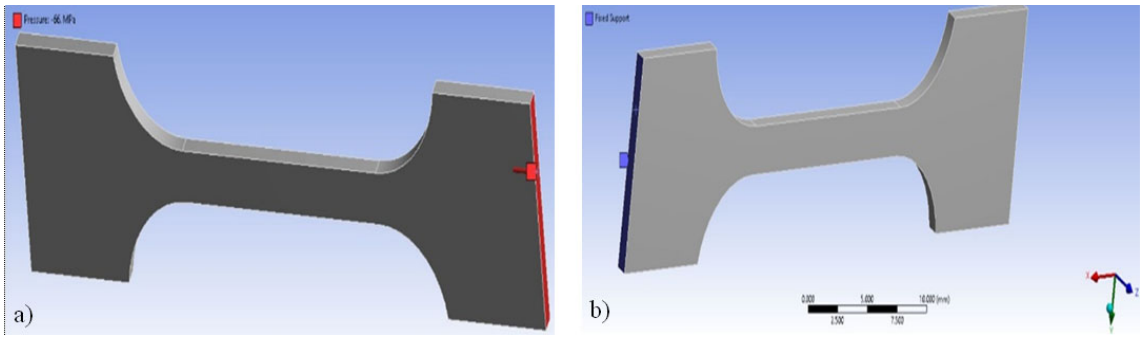



Fig. 7. Modeling of constraints and geometric loadings by: a) applying a force and b) fixing a surface.

**Analysis of the results obtained with the finite element method in the elastic and elastoplastic domain**

The lifespan of components that are prone to corrosion, cracking, and other forms of damage can be predicted by the mechanical loads simulation. Information on the occurrence and development of cracks as well as the material's maximum strength can be found through the tensile stress.

First, the fields of mechanical stresses (see Fig. 8a) and deformations (see Fig. 8b) were observed in the elastic and elastoplastic domains for the usual tensile model [10, 14]. Subsequently, analyses were conducted on several regions of this model (including region from the sample's fixed portion, region in the center, and region within the sample's free portion).

The simulated regions of the sample are displayed in Figs. 9a and 9b. An illustration of the simulated core region, for which stress and strain studies were conducted in the elastic and elastoplastic domains, can be found in Fig.9a. The collected results' stress-strain curves were processed. The stress-normal strain curve in the elastic domain is displayed in Fig.10a, and the stress-strain curve in the elastoplastic domain is displayed in Figs 10.b and c.

Taking into account the fact that the ends of the sample are symmetrical, only one of the side areas will be presented, as in Fig. 9b is an image for the area in the fixed part of the simulated sample, and stress and strain analyses were applied to it in both the elastic and elastoplastic domains.

The stress-strain curves for the obtained results were processed. In Figs. 10a and 11a, the stress-strain curve in the elastic domain is shown. Meantime, in Figs. 10b and 10c, and in Figs. 11b and 11c, the stress-strain curve in the elastoplastic domain is shown.

Following the analysis, the results highlighting the specific behavior of the tensile are obtained, because maximum values of the stress and strains are obtained in the central area of the sample, area that is subject to crack. Obtaining The obtained stresses and strains in the elastoplastic field for steel 316L are in agreement both with ASTM standard and other performd studies from literature.

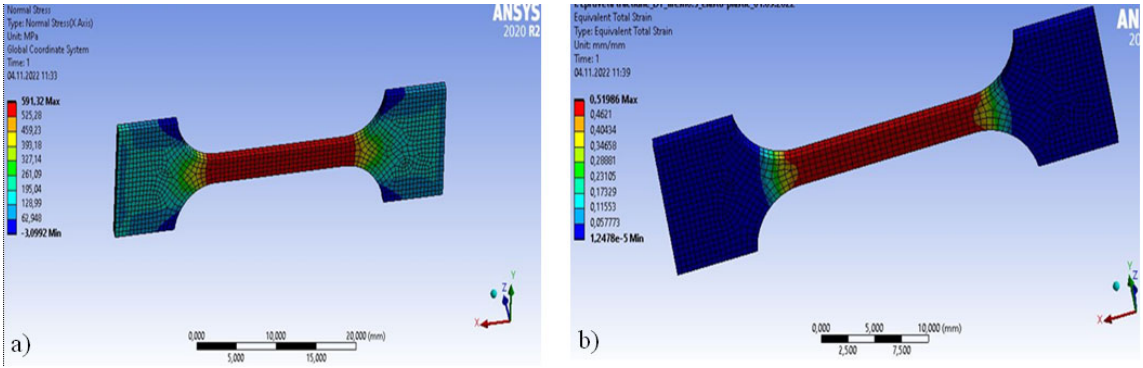



Fig. 8. (a) Normal stress and (b) Total deformations – visualizations for the elasto-plastic domain

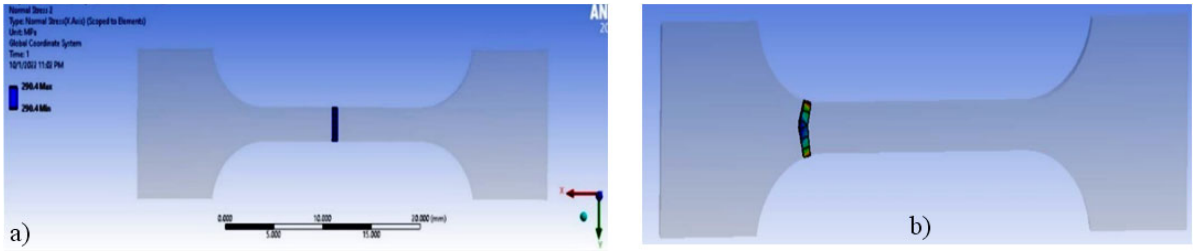



Fig. 9. (a) Simulated central area and (b) Simulated area from the fixed part of the sample

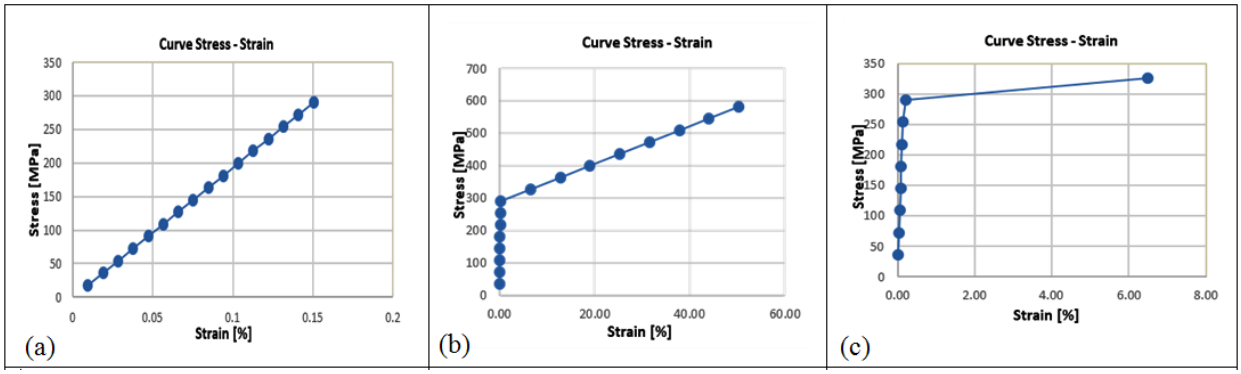



Fig. 10. (a) Curve σ-ε for Stress-Strain in the elastic domain, b) Curve σ-ε for Stress – Strain for the central area of the sample in the elasto-plastic domain; c) Curve σ-ε for Stress – Strain partial representation for ε < 7% for the central area of the sample in the elasto-plastic domain

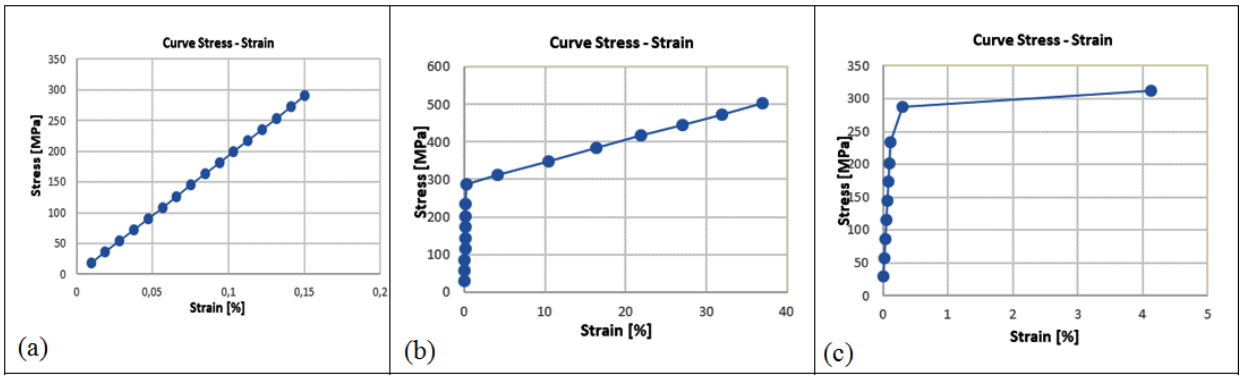



Fig. 11. (a) Curve σ-ε for Stress-Strain in the elastic domain;(b) Curve σ-ε for Stress – Strain for the central area of the sample in the elasto-plastic domain; (c) Curve σ-ε for Stress – Strain partial representation for ε<4.5% for the fixed part of the sample in the elasto-plastic domain

**Conclusions**

This study used the ANSYS-based finite element method to predict the behavior of stainless steel 316L at room temperature in the elastic field, or the elasto-plastic domain of mechanical stress-strain. The main findings are summarized in the following.

• By using the ANSYS Workbench, a typical tensile model was created according to the ASTM E8 standard.

• For this model of the tensile sample were introduced in ANSYS as input, the properties of the material for stainless steel 316L at room temperature (for the elastic and elasto-plastic domain), the type of discretion, and the model of the finished element (the size of the mesh element was chosen. being 0.5mm).

• By simulating analyses of some tensile loading on the sample are obtained the stress-strain curves in the elastic (linear) field, but also in the elasto-plastic (bilinear). For these simulations, 3 distinct areas of the sample were chosen, being selected the finite elements in those areas, for visualizing the stress and strain domain, being useful for initially comparing the curves obtained with those in specialized documents.

• As a result of the maximum stress and strain values being found in the sample's center, which is the region most susceptible to cracking, the findings are obtained that specifically illustrate the tensile behavior.

The analyzes using the finite elements using ANSYS software will further continue, and will consist of modeling 316L stainless steel at other temperatures of interest for Generation IV reactors, in the elastic and elasto-plastic fields. A comparison of the results obtained in simulations with the results given by the experimental tests will be taken into account to completely validate the method of analysis with the finite elements for the structural materials specific to Generation IV nuclear reactors.

**References**

[1] Oproescu, M., Plaiasu, A.G., Bizon, N., Iana, G.V., Toma, D. (2021) ″Oxygen Concentration Monitoring and Control in a Melted Lead Environment, Hardware Concept and Implementation″, *Proc. 13th Int. Conf. on Electronics, Computers and Artificial Intelligence* (ECAI)

[2] Ionescu, V., Toma, D., Niţu, Al., Jinga, A., Olaru, V. (2023) ″Preliminary investigations of the ultrasonic signal attenuation in the liquid lead using waveguide″, U.P.B. Sci. Bull., Series B, Vol. 85(4).

[3] Nitu, Al., Abrudeanu, M., Radu, V. (2018) ″The Study of the Mechanical Properties of Small Tubes Made from Oxide Dispersion Strengthened Steel by Using the Ring Tension Tests″, *Rev. Chim*. , Vol. 69(11), pp.3033-3036.

[4] Nitu, Al., Abrudeanu, M., Radu, V. (2018) ″Investigation of the Fracture Mechanics Properties of Small Tubes from Oxide Dispersion Strengthened Steels″, *Rev. Chim*. , Vol. 69(12), pp.3533-3536.

[5] Butnariu, S., Mogan, G. (2014) *″Analysis with finite elements in Mechanical engineering. Practical applications in ANSYS″* (Analiza cu elemente finite in ingineria mecanica. Aolicatii practice in ANSYS), Ed. Univ. Transilvania, Brasov, Romania (in Romanian)

[6] Toma, D., Nitu, Al., Bizon, N. (2022) ″Generation IV Nuclear Energy Systems - Alternative Solutions to Carbon Emission Energy Sources″, *Proc. 14th Int. Conf. on Electronics, Computers and Artificial Intelligence* (ECAI), pp. 1-7, DOI: 10.1109/ECAI54874.2022.9847457

[7] Atanasiu, C. et al. (1982) *″Material testing″* (Incercarea materialelor), Vol 1, Ed. Tehnica, Bucharest, Romania (in Romanian)

[8] Niţu, Al., Stoica, L., Radu, V. (2020) ″Development of the fracture toughness methodology for the small tubes used in the Generation IV reactors″, U.P.B. Sci. Bull., Series B, Vol. 82(1).

[9] Maksay, S.I., Bistrian, D.A. (2008) *″Introduction in the method of finite elements″* (Introducere in metoda elementelor finite), Ed. Cerni, Iasi, Romania (in Romanian)

[10] Toma, D., Bizon, N., Nitu, Al., Ionescu, V., Radu, V., Stoica, L., Jinga, A., Olaru, V., Matei, M. (2023) ″Study on the finite element modeling of the mechanical stresses of the samples of nuclear structural materials″, *Proc. 15th Int. Conf. on Electronics, Computers and Artificial Intelligence* (ECAI), pp. 1-6, DOI:10.1109/ECAI58194.2023.10194018

[11] ASTM International (2013) ″ASTM E/E8M - Standard Test Methods for Tension Testing of Metallic Materials″.

[12] Stoica, L, Radu, V., Ionescu, V., Niţu, Al., Jinga, A. Toma, D., Matei, M., Olaru, V. (2022) ″The Microstructural Investigations of the Liquid Metal Embrittlement Phenomenon on 316L Specimens tested in the Liquid Lead Environment″, *Journal of Nuclear Research and Development*, Nr. 24, pp. 14-20

[13] Lates, M.T. (2008) *″The Method of finite elements″* (Metoda elementelor finite), Ed. Univ. Transilvania, Brasov, Romania (in Romanian).

[14] Toma, D., Niţu, Al. (2022) ″Mechanical Analysis for Generation IV Structural Materials Using FEA Approach″, *Romanian Journal of Physics*, Vol. 67(9-19), art. No. 906.

[15] Stoica, L, Radu, V., Niţu, Al. (2023) ″Development of the Ramberg-Osgood Mechanical Stress-Strain Curve Using the Artificial Neural Network Method to Evaluate Mechanical Behaviour of 316L Stainless Steel in the Liquid Lead″, *Journal of Science and Arts*, Vol. 23(2), pp. 537-552

[16] Heanjia Super-Metals Co. Ltd. (2015) ″Stainless Steel 316/316L (UNS S31600)″, datasheet, www.super-metals.com.