# APB Modeling in Vertical Oil Wells

Gleide K. M. Lins[1], Catarina N. A. Fernandes[1], Thiago B. da Silva[1], João Paulo L. Santos[1], William W. M. Lira[1], Charlton O. de Souza[2], Emilio C. C. M. da Silva[3]

[1]*Laboratory of Scientific Computing and Visualization, Federal University of Alagoas*
*Av. Lourival Melo Mota s/n, Cidade Universitária, CEP: 57072-900, Maceió/AL, Brazil*
*gleidekarolayne@lccv.ufal.br, catarina@lccv.ufal.br, thiago.barbosa@lccv.ufal.br, jpls@lccv.ufal.br, william@lccv.ufal.br*
[2]*Petróleo Brasileiro, CENPES/PDIDP/EPOCOS/PERF*
*Av. Horácio de Macedo, 950, Cidade Universitária, CEP: 21941-915, Rio de Janeiro/RJ, Brasil*
*charlton@petrobras.com.br*
[3]*Petróleo Brasileiro*
*Rua do Senado, 115, 16º andar, 20231-003, Rio de Janeiro, Brazil*
*emiliosilva@petrobras.com.br*

**Abstract.** The paper presents a robust and automated approach for modeling Annular Pressure Buildup (APB) in vertical oil wells using ABAQUS software and Python scripts, providing an efficient and accurate alternative for studying this phenomenon. APB is caused by temperature variations in confined fluids, increasing pressure in the annular spaces and creating significant stress on the casings. To ensure structural integrity, APB effects must be considered in equipment design. Given the complexity of APB and the lack of analytical solutions, computational tools are essential. The proposed strategy encompasses the following steps: i) generation of the model geometry; ii) assignment of materials; iii) definition of modeling stages; iv) description of boundary conditions; v) application of loads; vi) definition of fluids in the annular spaces; vii) generation of the finite element mesh; and (viii) submission of numerical analysis and visualization of results. Python scripts were developed to automate these tasks, streamlining the workflow and reducing the potential for human error. ABAQUS's fluid cavity interaction is employed to simulate the annular fluid behavior. The developed strategy shows up to 10% relative errors in APB prediction, attributed to simplifications in reference calculations and assuming constant temperature in the fluid cavity. Despite these differences, the study provides a valuable tool for APB studies, enhancing modeling speed and accuracy.

**Keywords:** APB, fluid cavity, Python scripting in ABAQUS

## 1 Introduction

In recent years, most of Brazil's oil production has come from offshore wells. In 2023, 95% of national extraction was derived from deep and ultra-deepwater wells [1]. However, production in these locations is characterized by deeper wells and large water depths, leading to high costs and significant operational challenges. Among the complications are drill string sticking, well collapse, and phenomena caused by thermal processes [2].

In this context, a phenomenon that has caught the attention of the oil industry and the academic community is Annular Pressure Build-up (APB). This phenomenon is characterized by an increase in pressure in the confined annuli of a well due to temperature variations in the drill string during the drilling phase or the production/injection string (COP/COI) during the well's production lifecycle [3]. There is a heat exchange between the produced or injected fluid and the casing strings, which in turn exchange heat with the fluids present in the annuli. Due to the confinement of the fluids, pressure variation occurs in the annular space, causing significant changes in the loading conditions of the casings.

The literature documents some cases of well failures where APB was the main cause, such as the incidents in the Marlin A-2 and Pompano A-31 wells in the Gulf of Mexico [2]. Hence, it is crucial that well designs consider the effects of APB to maintain the well's structural integrity and prevent economic losses and environmental damage. However, the calculation of pressure build-up is complex and lacks an analytical solution, necessitating the use of computational tools.

In this regard, recent studies have sought strategies to estimate APB with the aid of computational modeling,

such as those by Vasconcelos et al. [4], Almeida [5], and Santos et al. [6]. However, these strategies are local, hindering their widespread use in the academic community, or they do not allow for the calculation of thermal expansion with a single software, requiring intermediate manual calculations in spreadsheets.

Given these issues, this work aims to present a strategy for computational modeling of APB using ABAQUS [7], a robust commercial finite element software widely used in the academic community. Furthermore, to facilitate the reproduction of the strategy, Python scripts are employed to automate the modeling workflow.

By achieving the proposed objective, the main contribution of this work is to provide a robust strategy for APB modeling, assisting studies related to the phenomenon and the prediction of consequent stresses on the casings.

## 2 Methodology

The APB modeling developed in ABAQUS is performed using a plane axisymmetric approach of the model, due to the geometric and loading symmetry conditions of the problem. Thus, to achieve the proposed objective, the strategy developed is based on the following steps: i) generation of the model geometry; ii) assignment of materials; iii) definition of modeling stages; iv) description of boundary conditions; v) application of loads; vi) definition of fluids in the annular spaces; vii) generation of the finite element mesh; and (viii) submission of numerical analysis and visualization of results.

The construction of the model, step (i), begins with the creation of its parts. In this approach, the entire model is built as a single part, as shown in Fig. 1 (a), avoiding the need to define interactions between different regions and allowing the entire structure to deform together. A single rectangular geometry is constructed, and using the "partition face: sketch" tool, the model is divided, as illustrated in Fig. 1 (b). Subsequently, the faces related to the annular spaces are removed, as shown in Fig. 1 (c). Additionally, to enable the closure of the annulus at its top, the vertical dimension of the model is increased by half a meter, creating a "cap" at the top of the model, which practically represents the wellhead.

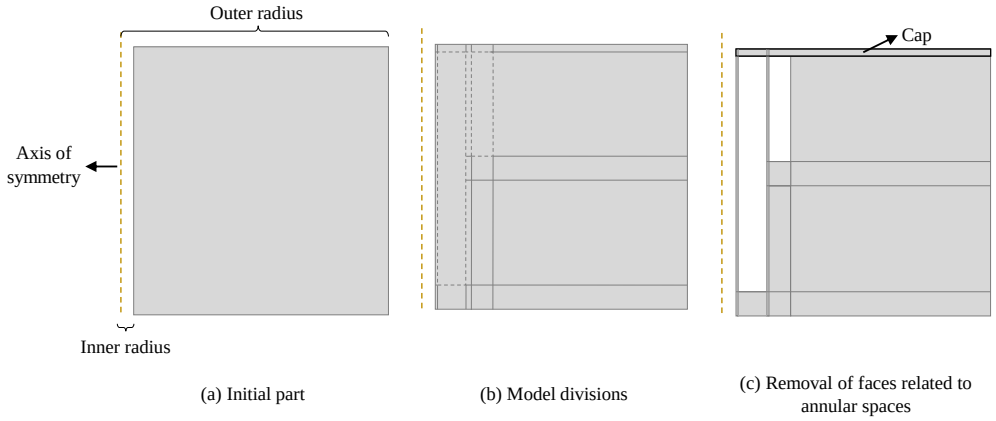



Figure 1. Construction of model geometry

After dividing each region of the model, step (ii) involves creating the materials and associating them with the corresponding regions (Fig. 2). Thus, the elastic properties of the solids and the thermal properties of the modeled fluids are defined.

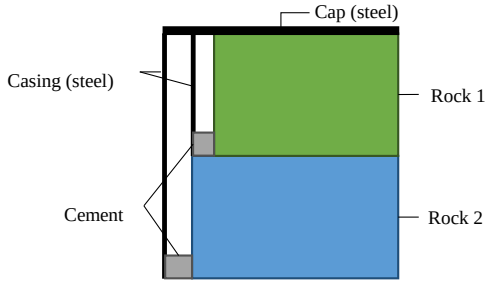
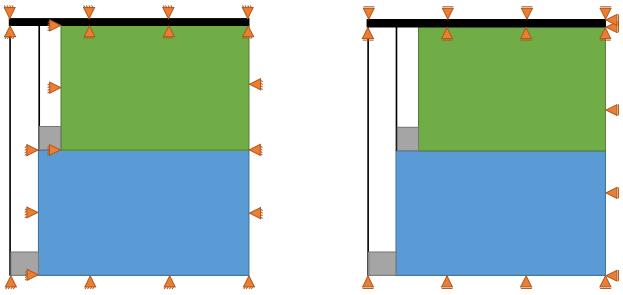



Figure 2. Material assignment for each region of the model

In step (iii), the analysis steps are defined, which are divided into two moments: before and after drilling. The pre-drilling moment consists of the initial and geostatic steps. In the initial step, the boundary conditions and the initial geostatic stress of the massif are defined. Subsequently, in the geostatic step, a body force in the massif is defined, which, along with the geostatic stress prescribed in the previous step, determines the internal stress state of the rock. In these two steps, the displacement throughout the model is zero. In the post-drilling moment, static steps are defined. In these steps, the new boundary condition of the model is prescribed, along with the hydrostatic loading on the inner walls of the annular spaces and the temperature of the trapped fluids.

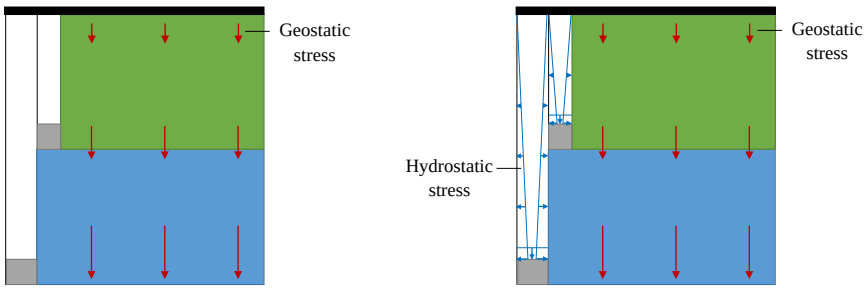
In step (iv), the boundary conditions are described. Before starting the well drilling (initial and geostatic steps), the entire model is undeformed. Second-type supports are prescribed on all boundary edges, as shown in Fig. 3 (a). After drilling, the rock is free to move in the drilled region, i.e., along the well walls. Thus, the displacement restriction in the vertical direction is maintained only at the top and bottom of the model, including the base of the "cap" and in the horizontal direction only at the outer edge of the model, Fig. 3 (b).



(a) Initial (b) After well completion

Figure 3. Boundary conditions of the model

In step (v), the model loads are applied. Before drilling begins, the loading in the model consists solely of geostatic stress, which acts vertically on the rock mass due to the weight of the water column, overlying and the rocks self weight. Figure 4 (a) illustrates this loading. After drilling the well, hydrostatic fluid pressure also acts on the inner walls of the filled annuli. The hydrostatic column formed presses against all surrounding walls. Thus, there is a distributed pressure both on the rock and on the casings in contact with the fluid, as indicated in Fig. 4 (b).




(a) Before starting the drilling (b) After starting the well drilling

Figure 4. Loading applied to the model

The definition of fluids, along with their corresponding properties, occurs in step (vi). In the developed approach, the modeling of thermal expansion of fluids is carried out using the Fluid Cavity interaction. This interaction models a closed cavity completely filled with hydraulic or pneumatic fluid and assumes that the pressure and temperature of the fluid in the specific cavity are uniform at any point within the fluid.

Subsequently, in step (vii), the necessary characteristics for generating the finite element mesh are defined. The distribution of elements is designed to allow greater refinement near the inner wall of the well, gradually reducing as it approaches the outer edge of the well. Figure 5 illustrates the refinement strategy adopted for the model.

Finally, in step (viii), the model is submitted for numerical analysis. At the end of the analysis, the results

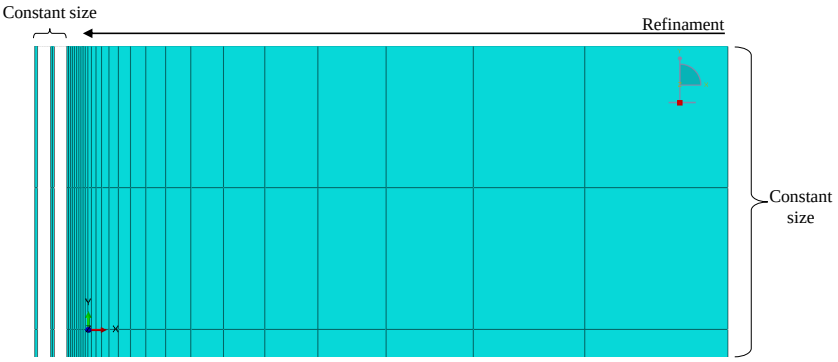



Figure 5. Refinement adopted for the finite element mesh

module is opened, and the pressure increases and volumetric variation for each annulus are observed.

## 3 Python scripting for ABAQUS APB modeling

The entire APB modeling process is initially conducted manually using the ABAQUS/CAE (Complete Abaqus Environment) graphical user interface (GUI). Upon establishing the most suitable modeling strategy, the process is automated through Python scripting, facilitating the execution of all modeling stages, from geometry creation and material assignment to numerical analysis and results visualization.

The initial manual modeling generates an ABAQUS macro file (`.jnl` extension) containing Python commands corresponding to the executed steps. This macro file is subsequently converted into a Python script (`.py` extension) and generalized to accommodate arbitrary parameters and input data.

To enhance code structure and modularity, auxiliary Python routines are developed. The resulting structure of the automated program comprises:

1. **JSON file:** Stores the geometric data and material properties of the model components (rock, casing, and cement).
2. **Main Python file:** Contains the thermal profiles of the annular space, fluid properties, and numerical simulation parameters (e.g., simulation time, initial estimates, and mesh refinement). It is responsible for importing the JSON file, executing auxiliary routines, and initiating the modeling process via the command `abaqus cae script=main_file_name.py`.
3. **Auxiliary file "well.py":** Defines auxiliary classes for code structuring and organization of model data.
4. **Auxiliary file "apb_abaqus.py":** Leverages the ABAQUS Python API to execute all modeling steps. These steps include geometry creation, material definition and assignment, application of loads and boundary conditions, step definition, mesh generation, job creation, and simulation execution.

This modular structure and the use of Python scripts provide greater flexibility, efficiency, and reproducibility to the numerical modeling process of APB, allowing for the analysis of different scenarios and parameters in an automated and streamlined manner.

## 4 Results and discussions

To verify the proposed strategy, the synthetic scenario used by Almeida [5] in his studies is adopted. The model has two annuli filled with fluids (Annuli A and B) and one completely filled with cement (Annulus C). Figure 6 describes the study scenario, and its geometric information is presented in Table 1. The outer radius of the model is 5 meters. This length was adopted by Almeida [5] after conducting some tests to determine the smallest possible model size that would provide a response without boundary interference.

The properties of the rocks are assumed to be constant along the depth of the well. Additionally, only the elastic regime of the salt rock is considered, disregarding its creep effect. The material used for the well casing columns is steel, with the same properties applied to all three casings. Given that Almeida [5] does not mention the properties of the cement, the same properties as the adjacent rock are adopted. The elastic properties of each material are presented in Table 2.

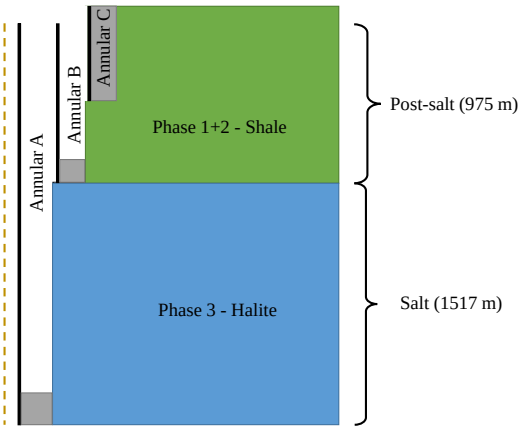



Figure 6. Scenario presented in Almeida [5] and used in the present study

Table 1. Geometric parameters of the study scenario

| Phase | 1 | 2 | 3 |
|---|---|---|---|
| Well diameter [in] | 36 | 26 | 14.75 |
| Top depth [m] | -2,169 | -2,253 | -3,444 |
| Base depth [m] | -2,253 | -3,444 | -4,961 |
| Outer casing diameter [in] | 30 | 20 | 10.75 |
| Casing thickness [in] | 1.5 | 1 | 0.797 |
| Top cement depth [m] | -2,169 | -3,040 | -4,456 |

Table 2. Elastic properties adopted for the regions of the study scenario

| Region | Shale | Halite | Casing (Steel) |
|---|---|---|---|
| Young's Modulus [GPa] | 7.5 | 20.4 | 210 |
| Poisson's Ratio | 0.25 | 0.36 | 0.30 |
| Density [kg/m³] | 1,900 | 2,200 | 7,860 |
| Lateral Stress Coefficient | 0.87 | 1.00 | - |

Table 3 presents the physical and thermal properties of the fluids for each of the annuli. Additionally, the initial stress and temperature conditions of the domain are shown in Table 4.

Table 3. Physical and thermal properties of the fluids in the annuli of the study scenario

| Annulus | Weight [lb/gal] | Density [kg/m³] | Bulk Modulus [Pa] | Linear Thermal Expansion Coefficient [°C$^{-}$1] |
|---|---|---|---|---|
| A | 11.5 | 1,378.0 | $1.50\times10^{9}$ | $2.67\times10^{-}4$ |
| B | 8.6 | 1,030.5 | $1.50\times10^{9}$ | $2.67\times10^{-}4$ |

Table 4. Initial stress and temperature conditions of the study scenario

| Depth [m] | In-situ Stress [MPa] | Temperature [°C] |
|---|---|---|
| -2,169 | -21.69 | 4.00 |
| -3,444 | -45.68 | 47.45 |
| -4,456 | -67.43 | 60.98 |

With regard to the finite element mesh, the same refinement used by Almeida [5] is adopted. The radial

refinement features elements of 3 mm in the inner region of the well wall and 1 m at the model's outer edge. The elements in the vertical direction have a constant size of 1 m. Furthermore, the mesh used for the numerical simulation of the model consists of bilinear quadrilateral axisymmetric elements. In ABAQUS software, this element is referred to as CAX4.

To obtain the updated temperature distribution for a desired time during the production period, Almeida [5] uses WellCat$^{TM}$ software (WELLCAT, 2006), considering an oil flow rate of 10,000 bpd (barrels per day) at 63.64°C. In his studies, Almeida [5] finds that after 4,096 hours, the heating of the fluids in the annuli converges to equilibrium, becoming close to the temperature profile at an infinite time.

In this analysis, the study is conducted only at time 0 (start of production) and after 4,096 hours (time for the temperature equilibrium of the fluids). Since the Fluid Cavity interaction does not allow for defining a temperature profile for the annulus, the average value of the temperature profile presented by Almeida [5] is calculated. Thus, at the start of production, the temperatures of Annulus A and Annulus B are, respectively, 37.96°C and 19.74°C, and after 4,096 hours, they assume values of 54.47°C and 38.19°C.

It is noteworthy that, in the approach adopted by Almeida [5], the thermal expansion of the fluid is calculated by injecting a mass flow rate into the annulus proportional to the temperature increase experienced. Therefore, the finite element model in ABAQUS calculates the system deformations and determines the pressure increase. This calculation is performed with the aid of an Microsoft Excel$^{TM}$ spreadsheet and considers the fluid density constant.

The comparison of results is done through the relative percentage difference DPR (%), taking the results obtained by Almeida [5] in his elastic approach as a reference. Thus,

$$\text{DPR}(\%) = \left( \frac{x - x_{\text{ref}}}{x_{\text{ref}}} \right) \times 100\%, \tag{1}$$

in which $x$ is the value obtained with the presented strategy and $x_{\text{ref}}$ is the value obtained by Almeida [5].

The study results are presented in Table 5, which compares the volumetric variations ($\Delta V$) and the pressure increments (APB) obtained by Almeida [5] and those derived from the approach presented in this work.

Table 5. Comparison of pressure increment (APB) and thermal expansion ($\Delta V$) results

| Approach | Annular A | | Annular B | |
|---|---|---|---|---|
| | $\Delta V$ [m$^3$] | APB [MPa] | $\Delta V$ [m$^3$] | APB [MPa] |
| Almeida [5] | 1.20 | 17.82 | 1.33 | 10.80 |
| Adopted | 0.86 | 16.02 | 0.85 | 10.85 |
| DPR (%) | -28.6% | -10.1% | -35.9% | 0.5% |

It is noted that the strategy presented in this work results in relatively lower values, both in pressure variation and volume, except for the pressure increase obtained in Annulus B.

An important point that may explain these discrepancies is the methodology used by each strategy. In Almeida's strategy [5], the total simulation time is divided into smaller intervals. In each period, the corresponding temperature profile is obtained until reaching the equilibrium temperature at 4,096 hours. For each time step, the corresponding unrestricted volumetric variation (isobaric expansion) is calculated, along with the equivalent mass flow rate. In this strategy, to determine the mass flow rate, the integration of the temperature profile variation and the fluid density at that instant is used. Thus, the calculated mass flow rate is injected into the annulus, and ABAQUS numerically calculates the system deformations and reconciles the pressures.

In the present strategy, all modeling is conducted in ABAQUS. Only one time step is used, in which the fluid is heated from the initial temperature (time 0) to the final temperature (after 4,096 hours). Furthermore, it is not possible to define a temperature profile for the fluid (limitation of the Fluid Cavity interaction), with a single value being employed along the entire length of the annulus. Furthermore, since Almeida's model [5]is not available, it is not known what boundary conditions and loads are used, nor other details of his strategy, which may cause significant differences in the results.

## 5 Conclusions

This work proposed a strategy for modeling pressure buildup in confined annuli of vertical oil wells using ABAQUS. Due to the symmetrical characteristics of the well and its loading, a 2D axisymmetric formulation was adopted. The Fluid Cavity interaction was used to model the thermal expansion of fluids in the annuli, assuming constant pressure and temperature within the cavity.

Automation of the modeling process was achieved through Python routines, allowing for the efficient execution of all analysis steps in ABAQUS. Validation was conducted using a synthetic scenario from the literature, involving two fluid-filled annuli. The results show a 10% discrepancy in one of the annuli and 0.5% in the other between the APB obtained with the adopted strategy and the reference bibliography. Additionally, there was an approximate 30% difference in volume variation in the annuli.

The observed divergences can be attributed to the methodological differences between the two strategies. The reference study calculates the thermal component decoupled from ABAQUS, considering intermediate heating steps until reaching the final temperature profile. Furthermore, its strategy allows for fluid temperature variation along the depth.

On the other hand, the strategy developed in this work models the entire phenomenon in ABAQUS and uses a single temperature step, taking the fluid from the initial temperature (time 0) to the final temperature (after 4,096 hours). Moreover, it is not possible to specify a temperature profile for the fluid, with a single value being adopted along the entire length of the annulus.

Despite the observed differences, the developed strategy is a valuable tool for APB modeling, providing efficient model generation and enabling quick estimates of pressure and volume variations in the annuli.

Future work can explore new scenarios to better understand the observed discrepancies, verifying whether they are repeated in other models and investigating the reasons why the two methodologies result in slightly different outcomes. In particular, it is important to assess whether the discrepancies are strictly related to the limitations of the Fluid Cavity's thermal profile or if they are also influenced by the decoupled calculation of thermal effects used by the reference study.

**Acknowledgements.** The authors would like to thank PETROBRAS for the financial and technical support related to the development of the research project "Desenvolvimento de produtos digitais para projeto e gestao de integridade de estrutura de poço" (ANP code 23599-4).

**Authorship statement.** The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

## References

[1] Agência Nacional do Petróleo, Gás Natural e Biocombustíveis. Anuário estatístico brasileiro do petróleo e do gás natural 2023. Accessed: 2024-06-18, 2023.

[2] T. S. Vasconcelos. Modelagem termomecânica do crescimento de pressão em anulares confinados em poços de petróleo na presença de evaporitos. Dissertação de mestrado, Universidade Federal de Alagoas, Maceió, Brazil. Programa de Pós-Graduação em Engenharia Civil, 2019.

[3] A. S. Halal and R. F. Mitchell. Casing design for trapped annular pressure buildup. *SPE Drilling & Completion*, vol. 9, n. 02, pp. 107–114, 1994.

[4] T. S. Vasconcelos, R. S. E. Filho, and E. N. Lages. Analysis of apb in vertical wells with evaporite layers: A 1d axisymmetric multilayer thermomechanical model. *Mechanics Research Communications*, vol. 139, pp. 104304, 2024.

[5] L. F. M. Almeida. Modelagem termomecânica do crescimento de pressão em anulares confinados, frente a formações salinas. Dissertação de mestrado, Pontifícia Universidade Católica do Rio de Janeiro, Rio de Janeiro, Brazil. Mestrado em Engenharia Mecânica, 2016.

[6] J. P. L. Santos, E. T. L. Junior, L. P. Gouveia, and S. R. D. Junior. Desenvolvimento de metodologia para gerenciamento de projeto de revestimento de poço de petróleo. *Exacta*, vol. 19, n. 3, pp. 609–622, 2021.

[7] M. Smith. *ABAQUS/Standard User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp, Providence, RI, 2009.