Comparative Analysis of Numerical Modeling Strategies for APB in Oil Wells

Gabriele K. M. Lins[1,1], Catarina N. A. Fernandes[1], Thiago B. da Silva[1], Gleide K. M. Lins[1], William W. M. Lira[1]

[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
gabriele.lins@ctec.ufal.br, catarina@lccv.ufal.br, thiago.barbosa@lccv.ufal.br, gleidekarolayne@lccv.ufal.br
william@lccv.ufal.br

Abstract. This study examines and compares two numerical modeling strategies for reproducing Annular Pressure Build-up (APB) in vertical oil wells. APB occurs when annular fluids expand due to thermal variations, increasing stresses on well tubing. This is especially critical in offshore environments, where significant temperature and pressure changes can lead to structural failures and major losses. Given the importance of understanding APB and the difficulty of reproducing it analytically, many researchers use finite element software like Abaqus for modeling. In this context, to achieve the proposed objective, the methodology adopted is based on the following steps: a) Define two APB modeling strategies in Abaqus; b) verifying these strategies in a simplified scenario; and c) conducting a case study. The key difference between the strategies is in their approach to thermal expansion: one calculates APB directly from thermal variation, while the other uses an equivalent mass flow. The strategies are compared based on results and accuracy to identify the most effective approach. Both yield similar results, with the latter allowing for modeling fluids with different behaviors.

Keywords: Annular Pressure Buildup, Fluid cavity,Thermal variation, Mass flow, Abaqus.

1 Introduction

The increase in pressure within confined annuli (APB - annular pressure buildup) is a significant concern in the oil industry, especially with the rise of deepwater wells. This phenomenon, driven by the thermal expansion of fluids in the confined annuli of subsea wells, can lead to serious incidents like casing failure or tubing collapse, particularly during the early production phase. Offshore, the problem is exacerbated by limited wellhead access, which restricts pressure release from the annuli. Without adequate space for fluid expansion, internal pressure increases.

This pressure buildup is critical because it can overload well casings, threatening the structural integrity of the installation. Research indicates that APB can cause high collapse loads on the inner tube or burst loads on the outer tube [1]. Therefore, it is crucial to consider these loads during well design, particularly in deepwater environments where managing APB is more challenging [2]. Studying the origins and effects of this phenomenon is vital for ensuring well safety.

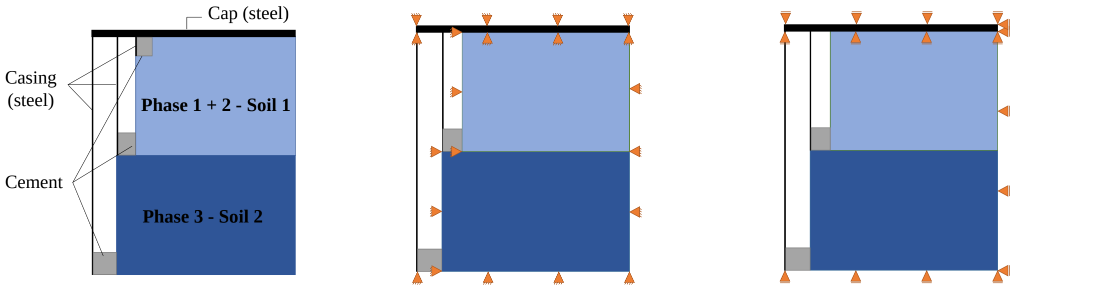
Given this, the present work aims to study and compare two existing numerical modeling strategies for reproducing the APB phenomenon. To address the challenge of analytically reproducing APB, several authors have turned to finite element modeling, using software like Abaqus [3]. In this context, to achieve the proposed objective, the adopted methodology is based on the following steps: a) Study and understand two APB modeling strategies in Abaqus; b) Verify these strategies in a simplified problem; and c) Conduct a case study of a well model. Two modeling strategies in Abaqus, both found in the literature, are evaluated, each utilizing fluid cavity interaction for fluid behavior in a 2D axisymmetric analysis. The main distinction between the strategies lies in how they address thermal expansion: while one directly applies temperature variation, the other uses an equivalent mass flow.

The study compares these strategies based on results and accuracy, aiming to identify the most efficient one. Despite methodological differences, both approaches yield similar results, with the latter offering flexibility in modeling fluids with different behaviors. This research contributes to improving APB modeling, aiding in the development of more robust strategies for predicting the phenomenon's effects.

2 Modeling APB in Abaqus

Calculating the APB phenomenon involves thermal, mechanical, and hydraulic analyses with various materials. Due to the cylindrical geometry and varying stress fields around the well, predicting pressure increases is complex and requires computational tools, as shown by Almeida [4]. This work uses Abaqus software, which employs the finite element method [5], to model APB under different geometries, material properties, and transient conditions. A planar axisymmetric model with static thermal expansion analysis is used to estimate APB.

The methodology proposed by [6] is adopted. The initial step is to create a single part for the entire model, avoiding the need to create interactions between regions and allowing joint deformation. The part is then partitioned into regions to assign different materials (steel, cement, formation) and to map the annular region as a fluid cavity. Figure 1-a shows the generalized regions and partitions of the well. The model is meshed using bilinear quadrilateral axisymmetric elements with reduced integration, referred to as CAX4 in Abaqus. The mesh is greater refined near the wellbore's inner wall and lower refined towards the outer edge, ensuring finer resolution where differential stress and displacement are highest. Element size is assumed constant in the vertical direction.



(a) Generalized Well Structure (b) Initial boundary conditions (c) Boundary conditions after well completion

Figure 1. Study scenario with two annuli and material assignment for each region of the model

Two simulation steps are created: the geostatic step, which simulates the rock's intact state with geostatic stresses, and the elastic step, where regions can deform and temperature variation occurs. Boundary conditions change before and after drilling. Initially, the model is under geostatic stress; after drilling, hydrostatic stress on the annuli walls is considered. Fluids in the annuli are modeled using the Fluid Cavity interaction, assuming uniform pressure and temperature. Before drilling, the entire model is undeformed with fixed boundary edges, as shown in Fig. 1-b. After drilling, the rock can move towards the well's closure, with displacement constraints applied vertically to the top and bottom and horizontally towards the outer edge, as shown in Fig. 1-c.

Due to space limitations, the description of the modeling strategy has been summarized; further details on the APB modeling strategy can be found in [6].

3 Modeling Thermal Expansion

In this session, the specific problem of thermal expansion will be discussed. Two approaches to modeling this problem are presented. The first approach, based on the native Abaqus equation, is detailed in Lins work [6], therefore this approach will lead to a replication of the entire methodology proposed by Lins [6]. The second approach, which focuses on injecting an equivalent mass flow for the APB calculation, is presented by Almeida [4], thus, this approach will result in a methodology analogous to that proposed by Lins [6], with the exception of the strategy employed for modeling the thermal expansion of the confined fluid.

To model the thermal expansion of fluids in the well annuli, the Fluid Cavity interaction is used. This interaction simulates a closed cavity filled with hydraulic or pneumatic fluid, assuming uniform pressure and temperature throughout the cavity over time. As the system volume is fixed, fluid expansion leads to increased internal pressure.

This tool models fluid-structure interactions, considering both thermal and pressure effects on the fluid. The approach involves defining the cavity, specifying fluid properties (density, viscosity, thermal expansion coefficient, and heat capacity), and setting initial pressure and temperature conditions. Boundary conditions are established for heat transfer and fluid flow.

3.1 Abaqus Native Equation

In the approach proposed by Lins [6], the mass, momentum, and energy conservation equations are solved under the assumptions of uniform pressure and temperature. Thermal expansion is simulated by applying temperature variations to the cavity, which alters the fluid pressure as the volume increases. This approach assumes a simplified model with uniform fluid pressure and temperature, considering isotropic thermal and isobaric expansion coefficients. However, it is important to highlight the limitations of this method. In very deep wells, the temperature tends to vary significantly from the top to the bottom, which is not adequately captured by the assumptions of uniform pressure and temperature. Additionally, there are more complex equations that offer a more accurate representation of the behavior of fluids used in wells, as their properties, such as density and viscosity, vary significantly with temperature. These more elaborate equations are essential for simulations seeking greater accuracy, especially under extreme conditions.

3.2 Mass Flow

This second strategy used is based on the study by Almeida [4], which also models the thermal expansion in Abaqus. Instead of calculating APB by applying a unique temperature variation in the annulus using the Fluid Cavity interaction, this approach calculates the pressure variation by applying a mass flow into the annulus corresponding to the desired thermal variation. According to Almeida [4], the mass variation corresponding to a given temperature change is given by:

$$\Delta m = \rho \gamma A \int \left[T_i(y) - T_j(y) \right] \, dy, \tag{1}$$

where ρ is the density of the fluid in the annulus; γ is the volumetric thermal expansion coefficient, equivalent to 3α (the thermal expansion coefficient); A is the cross-sectional area of the annulus, considered constant along its entire length at the beginning of heating; $T_i(y)$ and $T_j(y)$ represent two thermal profiles at different times during well production. The representative temperature of the profile is taken as the average value of the function defining the temperature profile of the annulus for the depth section containing the fluid. Thus, by considering the temperature profiles and the relationship between the volumetric thermal expansion coefficient (γ) and the linear thermal expansion coefficient (α), eq. (1) can be rewritten as: $m = \rho 3\alpha V_0 \Delta T$.

For each thermal variation, the corresponding mass change is calculated and used as the volume flow applied in the annulus. In addition to the Fluid Cavity interaction, this approach uses Fluid Exchange to model the mass/volume flow transfer between cavities. This requires creating a new closed cavity region with an independent Fluid Cavity interaction and the same fluid properties. Since the calculations are performed independently of Abaqus, other fluid equations can be considered, as well as different temperature profiles, making this strategy more suitable for modeling the problem under study. On the other hand, its application is somewhat more complex.

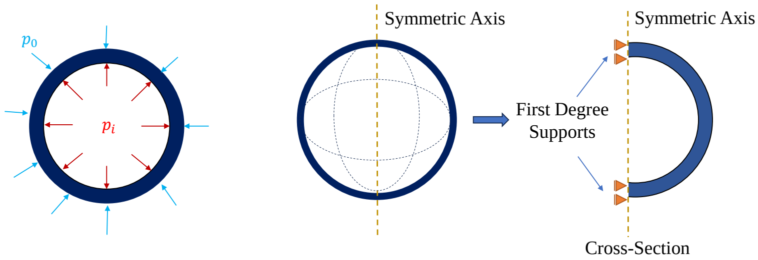
4 Verification of Strategies

To evaluate the methodologies presented for modeling thermal expansion in the Abaqus software, a problem involving a thick-walled spherical pressure vessel was studied. A pressure vessel is a container designed to store fluids, such as liquids or gases, under an internal pressure p_i significantly different from the external pressure p_0. Figure 2-a illustrates the loading applied to a spherical vessel, showing the internal and external pressures, while Fig. 2-b presents the axisymmetric representation of the model.

To validate the model strategies, a comparison with the analytical calculation presented by Lins [6] is performed. Furthermore, it was observed that it is necessary to determine the final density of the fluid to calculate the ideal amount of mass to be injected. This is due to the fact that the fluid density can change due to variations in pressure and temperature, reflecting real operational conditions and providing a more accurate estimate of the required mass for the system. Thus, the final density can be determined from eq. (2).

$$\rho_f = \frac{V_i \cdot \rho}{V_i - V} = \frac{\rho \cdot V_i + \rho_f \cdot \Delta V}{V_f}. \tag{2}$$

Considering that the annular volume will not undergo a significant variation, we can assume that the final



(a) Pressures applied to a spherical pressure vessel.

(b) Axisymmetric representation of the spherical model.

Figure 2. Representative model of a spherical pressure vessel

volume V_f is approximately equal to the initial volume V_i. Therefore, as $V_f \approx V_i$, eq. (2) can be rewritten as:

$$\rho_f = \frac{\rho \cdot V_i + \rho_f \cdot \Delta V}{V_i}. \tag{3}$$

Based on the presented example, a numerical analysis of the model is performed in the Abaqus software, first applying the thermal expansion strategy and then the mass flow strategy. The obtained results are shown in Table 1.

Table 1. Comparison of the results of APB and thermal expansion (ΔV) for the simplified model.

Strategy	Initial Volume [m^3]	Final Volume [m^3]	ΔP [MPa]
Analytical calculation	4.1888	4.1930	34.47
Abaqus Native Equation	4.1802	4.1879	33.28
Mass Flow - Initial Density	4.1888	4.1962	32.55
Mass Flow - Calculated Final Density	4.1888	4.1964	33.34

Both strategies yield satisfactory results. The mass flow approach using the final density showed a relative difference of -3.29% compared to the analytical calculation, while the thermal expansion strategy resulted in a difference of -3.45%. Notably, using the initial density increased the relative difference to -5.56%, emphasizing the importance of considering real operational conditions. This highlights the superiority of using the final density for a more accurate and consistent evaluation of the system.

5 Case Study

The case study presented by Almeida [4] and also discussed in Lins [6] is adopted to compare the implemented strategies. The model has two annuli, denoted as A and B, with B being the inner annulus and A the outer one. Both annuli are partially filled with cement and, for the most part, with fluid. The outer edge of the model has a radius of 10 meters, chosen to ensure that the radius would provide a response free from boundary interference. Detailed information about the model, including geometric parameters, is provided in detail in Lins [6].

The rock formations are assumed to be elastic and with constant properties along the depth of the well. The material used for the well casing columns is steel, and the same properties are applied to all three casings. The elastic properties of the regions in the scenario follow those described in the study by Lins [6].

For the Abaqus native equation method, the initial fluid temperatures are 37.96°C in annulus A and 19.74°C in annulus B. After 4096 hours, they rise to 54.47°C and 38.17°C, respectively. For the mass flow strategy, the temperature profile from Almeida [4] is used.

Based on the model, both methodologies are analyzed as shown in Table 2. The most suitable methodology for modeling APB in confined annuli is determined by comparing the relative percentage difference (RPD) to the results of Almeida [4].

Table 2. Comparison of volume variation and thermal expansion results.

Strategy	Annular A		Annular B	
	ΔV [m^3]	ΔP [MPa]	ΔV [m^3]	ΔP [MPa]
Almeida [4]	1.20	17.82	1.33	10.80
Abaqus Native Equation	0.79	16.31	0.85	10.93
Mass Flow - Calculated Final Density	0.88	18.86	0.85	11.25
DPR - Abaqus Native Equation	-33.83%	-8.48%	-36.17%	1.19%
DPR - Mass Flow	-26.58%	-5.85%	-36.17%	5.12%

6 Conclusions

The comparison of thermal expansion modeling methodologies in Abaqus shows that both approaches are accurate and efficient. The native Abaqus solution is praised for its simplicity and ease of implementation, while the equivalent mass flow approach, as described by Almeida [4], aligns more closely with reference results and better reflects real-world conditions. This method integrates the native Abaqus solution with a fluid cavity, complemented by Almeida's methodology. In the case study, annuli A and B showed specific responses due to the model's complexity and the interaction between the annuli, which complicated boundary conditions and material properties. Both methodologies produced satisfactory results, with minor variations based on input parameters and simplifications described in the reference work or assuming a constant temperature in the Fluid Cavity. Regarding computational time, strategy 1, using the native Abaqus equation, averaged 24.5 seconds, while strategy 2, employing the mass flow method, averaged 28 seconds. Mesh refinement also impacts execution time significantly; finer meshes, which enhance accuracy, can substantially increase computational time. Despite the second strategy's longer computational time, the difference is minimal. Notably, the second strategy allows for more complex fluid modeling, which may better capture thermal expansion behavior. Choosing between methodologies depends on the desired accuracy, available computational resources, and user familiarity. Both the native Abaqus solution and the equivalent mass flow approach are suitable for thermal expansion analysis. The decision should reflect the problem's specific characteristics and study objectives to ensure reliable results for the oil industry.

Acknowledgements. The first author acknowledges the financial support related to the development of this research provided by the Institutional Program for Scientific Initiation Scholarships – PIBIC/UFAL (2023-2024). 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).

References

[1] 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.
[2] U. Sathuvalli, M. Payne, P. Pattillo, S. Rahman, and P. Suryanarayana. Development of a screening system to identify deepwater wells at risk for annular pressure build-up. *SPE/IADC*, , n. 92594, 2005.
[3] M. Smith. *ABAQUS/Standard User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp, Providence, RI, 2009.
[4] L. F. M. Almeida. Modelagem termomecânica do crescimento de pressão em anulares confinados, frente a formações salinas. Dissertação (mestrado em engenharia mecânica), Pontifícia Universidade Católica do Rio de Janeiro, Rio de Janeiro, 2016.
[5] J. Fish and T. Belytschko. *Um Primeiro Curso em Elementos Finitos*. LTC, 2009.
[6] G. K. M. Lins. Modelagem computacional do apb para previsão de esforços em revestimentos de poços de petróleo, 2022.