Applications of topology optimization considering physical nonlinearity

Matheus B. M. Cedrim[1], Eduardo N. Lages[2], Aline da S. R. Barboza[2]

[1]*Dept. of Civil Engineering, University Center CESMAC*
R. da Harmonia, 57081-350, Maceió - Alagoas, Brazil
matheuscedrim@hotmail.com
[2]*Laboratory of Scientific Computing and Visualization, Center of Technology, Federal University of Alagoas*
Av. Lourival Melo Mota, 57072-900, Maceió – Alagoas, Brazil
enl@ctec.ufal.br, aline@lccv.ufal.br

Abstract. Topology optimization is a powerful computational tool that assists designers in determining efficient structural configurations. Most applications are still limited to the field of theoretical/computational analysis. This limitation restricts the scope and applications of topology optimization concepts for practical problems. Extensive research has focused on topology optimization using isotropic material with linear elastic behavior. However, there is a paucity of studies comparing linear and nonlinear material behaviors, highlighting a research gap in result analysis and structural application fields. For concrete structures, in particular, there is generally a deeply nonlinear behavior, including effects such as cracking, creep, and shrinkage. Therefore, it becomes a challenge to realistically optimize this material. This study investigates the applications of topology optimization for evaluating the structural behavior of beams considering the physical nonlinearity of the material. Using the ABAQUS® software and the SIMP method, a finite element numerical analysis is conducted to determine the density distribution in a design domain. The obtained results of the optimized models are compared, and it is observed that the presence of plastic deformations influences relevant aspects of structural behavior. This study contributes to the use of nonlinear constitutive models, highlighting the potential of the concept of optimization-assisted design.

Keywords: ABAQUS®, Drucker-Prager, Plasticity.

1 Introduction

Modeling and structural analysis through numerical methods, such as the Finite Element Method (FEM), is the subject of study by numerous researchers. Numerical simulations aid in understanding and designing more efficient elements, capable of predicting events and outcomes.

For the calibration of numerical models, it is necessary to measure the variability of output parameters and correctly determine input parameters, thereby exploring the associated variable space for the problem at hand. With the increase in computational processing power, it has become possible to simulate various complex problems.

As a result, structural optimization has emerged as an important tool in the development of more daring projects. The use of numerical methods has enabled the inclusion of new design methodologies that allow for the reduction of total weight without compromising load-bearing capacity.

Recently, optimization techniques employing concepts of physical nonlinearity, as well as consideration of multiple loading cases, have been observed in the literature (Schwarz, Maute, and Ramm [1]; Jung and Gea [2]; Lotfi [3]; Xia, Fritzen, and Breitkopf [4]; Zhao [5]). The recent development enables greater possibilities for simulating static and dynamic behaviors, allowing for the incorporation of different constitutive models for various optimization problems, with objective functions and constraints defined according to the proposed simulation.

In recent years, Da [6] reports an increase in the use of high-performance heterogeneous materials, such as fiber-reinforced composites, cementitious materials, and materials aided by 3D printing (Figure 1, Vantyghem *et al.* [7]). The physical and mechanical characteristics of complex heterogeneous materials can be determined by the composition of their constituents; however, the volume fraction of inclusions, their geometric shape, and the presence of interfaces can drastically change the composite properties.

Figure 1. Concrete beam produced with the aid of 3D printing (Vantyghem *et al*. [7])

Gaganelis, Mark, and Forman [8] introduce the concept of Optimization-Aided Design (OAD). That is, the design and sizing of structures and structural elements through optimization methods. This line of research covers a generalist methodology that can be applied to different areas of application. This resource can be used for any type of concrete (from normal strength to ultra-high strength), as well as arbitrary types of reinforcement (steel, carbon, fibers), and can be extended to numerous fields of structural engineering, such as steel structures, wood, and foundations.

2 Methodology

Wen *et al.* [9] present a review of the use of topology optimization methods for nonlinear problems. In engineering practice, several problems can be considered nonlinear; however, it is common to treat them as linear. This simplification can lead to significant errors and even incorrect results, which can compromise structural integrity. From a computational standpoint, introducing nonlinearity brings challenges in solving the study equations due to local and global instability, as well as excessive distortion of low-density elements.

Kim [10] relates the sources of nonlinearity in a structural system (Figure 2). These are defined as nonlinearity in the displacement-strain relationship (geometric nonlinearity), nonlinearity in the stress-strain relationship (physical nonlinearity), and nonlinearity in boundary conditions (for example, present in contact problems).

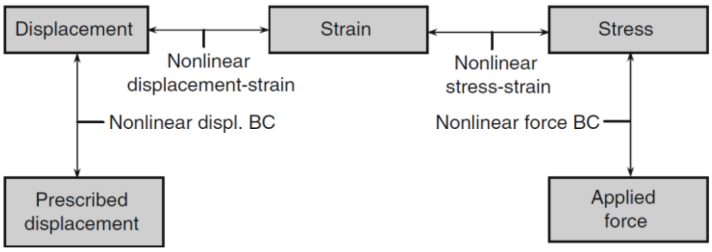

Figure 2. Sources of nonlinearity (Kim [10])

The most well-known finite element-based numerical optimization method is the SIMP (Solid Isotropic Material with Penalization), which was developed in the 1980s by Bendsøe and Kikuchi [11]. Sometimes the method is also referred to as material interpolation, artificial material, power law, or density method, but SIMP is the most used terminology worldwide [12].

In the discrete formulation of the problem, minimizing the strain energy, we have

$$
\begin{gathered}
\min c(x) = \boldsymbol{U}^{\boldsymbol{T}}\boldsymbol{K}\boldsymbol{U} = \sum_{e=1}^{N}(x_e)^p\, \boldsymbol{u}_e^T\, \boldsymbol{k}_e\, \boldsymbol{u}_e \\
s.t.\frac{V(\boldsymbol{x})}{V_0} = f \\
\boldsymbol{K}\boldsymbol{U} = \boldsymbol{F} \\
0 < x_{min} \leq x_e \leq 1
\end{gathered} \tag{1}
$$

where $\boldsymbol{U}$ and $\boldsymbol{F}$ are the global displacement and force vectors, respectively, $\boldsymbol{K}$ is the global stiffness matrix, $\boldsymbol{u}_e$ and $\boldsymbol{k}_e$ are the element displacement vector and stiffness matrix, respectively, $\boldsymbol{x}$ is the vector of design variables (relative densities of elements), x_{min} is a vector of minimum relative densities, N is the number of elements used to discretize the design domain, p is the penalty factor, $V(\boldsymbol{x})$ and V_0 are the material volume and design domain volume, respectively, and f is the prescribed volume fraction.

With the formulation of the method, the design variables defined in the problem are the relative densities associated with the finite elements or nodes of the mesh. Thus, the objective of minimizing compliance or maximizing stiffness can then be understood as an iterative process that seeks a better distribution of the design variables throughout the defined mesh.

The SIMP method, despite considering a homogeneous and isotropic material in its formulation, can be interpreted as a multiscale approximation, based on explanations by Wu, Sigmund, and Groen [13]. Based on this, a distribution of heterogeneous material within the element domain can be found for an intermediate density.

With a penalty factor parameter (p) greater than 3, it is possible to converge to 0-1 type solutions, which represent mono-scale structures. However, in a multiscale approach, the smaller the p, the greater the number of solutions containing a portion of intermediate densities, which provides a basis for porous microstructures (Figure 3 [13]).

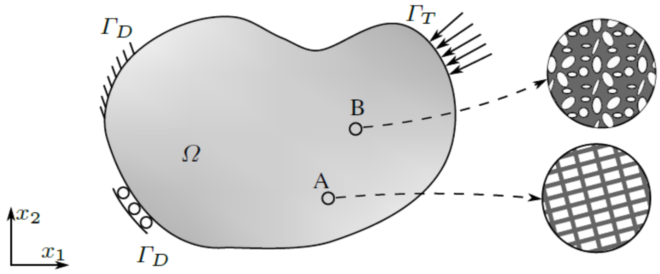

Figure 3. Multiscale structure representation (Wu, Sigmund and Groen [13])

Schwarz, Maute, and Ramm [1] note that in problems involving physical nonlinearity, where the material response depends on the intensity of the applied load, an incremental process is necessary. Consequently, the sensitivities involved in the optimization process need to be calculated after each incremental step. Analogously to the state variables, the derivatives are updated through increments added to previous steps.

For the computational simulation of the examples in this work, the software ABAQUS® 2019 is used for integrated finite element analysis and optimization processes. This software is recognized and highly regarded in the academic community due to its customization and modeling capabilities, making it applicable for the analysis of various engineering problems.

For nonlinear analysis, SIMULIA [14] demonstrates the solution process through Newton's method. The nonlinear response of a structure to a small increment of loading, ΔP, according to SIMULIA [14], for the Newton's method, utilizes the tangent stiffness K_0 calculated at configuration u_0, and ΔP is calculated for a displacement correction c_a. Using c_a, the updated structure configuration is obtained as u_a (Figure 4).

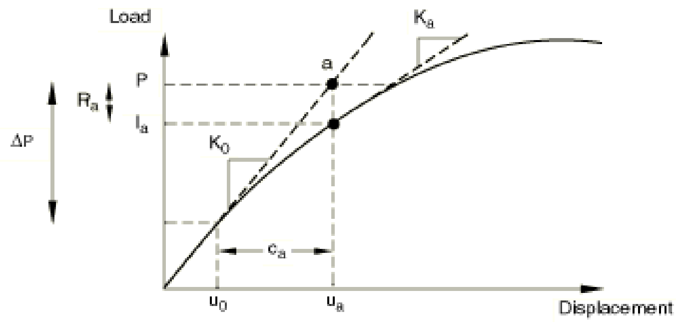

Figure 4. Incremental analysis for first iteration (SIMULIA [14])

Calculating the internal forces for the updated configuration, we have $R_a = P - I_a$, where R_a is the residual force for the iteration, P is the external force, and I_a is the internal force. If R_a is zero for each degree of freedom in the model, point a would coincide with the curve, and the structure would be in equilibrium. For typically nonlinear problems, R_a will never be exactly zero, requiring the introduction of convergence tolerance values. By default, a value of 0.5% is used as an average of the residual force in the structure, averaged over the loading time.

If R_a is less than the current tolerance value, P and I_a are considered in balance, and thus u_a is a valid equilibrium configuration for the structure with the applied loading. However, before ABAQUS/CAE® considers the solution accepted, it must be verified whether the last displacement correction c_a is small relative to the total displacement increment $\Delta u_a = u_a - u_0$. If c_a is greater than a fraction of the displacement increment (1% by default), the processing goes to another iteration. Both convergence criteria (force and displacement) must be met before the solution is considered convergent for that time increment.

If the iteration solution is not convergent, ABAQUS/CAE® performs a new iteration to achieve the balance of internal and external forces. Firstly, a new stiffness for the structure, K_a, is considered based on the updated configuration, u_a. The stiffness, along with the residual force R_a, determines a new displacement correction c_b, bringing the system close to equilibrium.

With the new residual force R_b determined in configuration u_b, the largest residual force in any degree of freedom is checked against the residual force tolerance. The displacement correction c_b for the second iteration is compared with the displacement increment Δu_b. If necessary, to meet the convergence criteria, processing goes to more iterations (Figure 5 [14]).

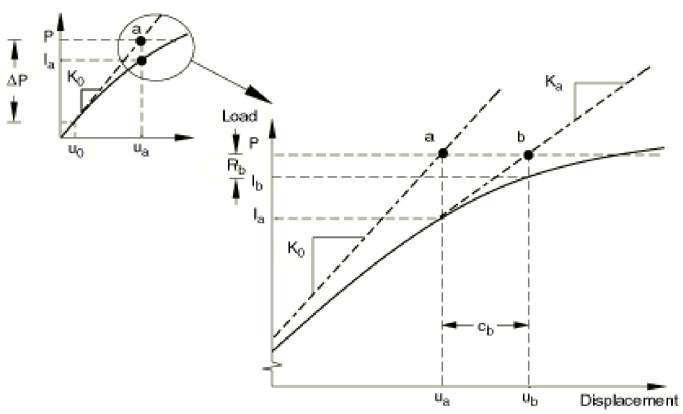

Figure 5. Incremental analysis for the second iteration (SIMULIA [14])

For each iteration in nonlinear analyses, the standard processor of ABAQUS/CAE® assembles the stiffness matrix of the model and solves the system of equations. The computational cost of each iteration is close to the cost of a complete linear analysis, which means that the computational demand of a nonlinear analysis can potentially be higher than the cost of a linear analysis.

3 Results and discussion

To obtain the results, a notebook with an Intel Core i7-12700H processor, 16 GB of RAM, and a Windows 11 Home 64-bit operating system was used.

The load application region is not considered in the optimization process. The density update strategy is normal. The solution algorithm is general, and the sensitivity filter is automatically defined according to the characteristic mesh dimension. The material interpolation technique used is SIMP, where a constant penalty factor is defined throughout the procedure. The other details are listed in Table 1.

Table 1: Topology optimization parameters

Parameters of the SIMP method	Value
Minimum density (ρ_{min})	0.001
Maximum density (ρ_{max})	1
Maximum change of the density per cycle of the project	0.25
Penalty factor (p)	3
Maximum tolerance for the objective function	0.001
Maximum tolerance for the element density	0.005

The following example illustrates the application of a clamped beam (Figure 6), proposed by Zhao *et al.* [15], subjected to a distributed loading in the bottom region. For modeling in ABAQUS®, a finite element mesh with an approximate size of 10 mm is considered for plane strain analysis using CPE4 elements (4-node bilinear quadrilateral elements), with 10,050 elements and 10,032 nodes.

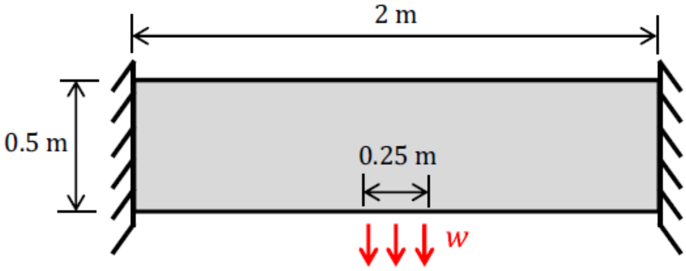

Figure 6. Clamped beam model (Zhao *et al.* [15])

Considering linear elastic behavior, the material properties for the analyzed beam are $E = 1.8 \cdot 10^5\ MPa$ and $\nu = 0.3$. For this model, with defined tensile strength $\sigma_t = 144\ MPa$ and compressive strength $\sigma_c = 1440\ MPa$, the linear Drucker-Prager model parameters are adopted as $\psi = 67.83^o$ and $c = 261.82\ MPa$. The applied distributed loading is $w = 80$ MN/m.

For the linear elastic model, topology does not vary with the intensity of the applied loading (Figure 7). The structural element does not have plastic deformations, indicating that the density distribution can be interpreted as a homogeneous domain of a material that possesses equivalent tensile and compressive strengths.

Figure 7. Optimized distribution of linear elastic material ($f = 25\%$)

For the elastoplastic model, since the material used has a higher compressive strength than tensile strength, there is a tendency for the density distribution to concentrate in the compressed regions (Figure 8).

Figure 8. Optimized distribution of elastoplastic material (Drucker-Prager) ($f = 25\%$)

Several parameters can influence the results presented in this model, from the characteristic dimension of the mesh used to convergence criteria and the filtering technique employed. This example seeks to visualize the possibilities of topological optimization for materials with nonlinear behavior and analyze its influence on the density distribution, reflecting a search for material reinforcement in regions with lower resistance.

With the surfaces extracted in the post-processing step, considering density values above $\rho > 0.50$ and using a moderate filtering technique, the structural behavior of the optimal topologies was analyzed. For both models, elastoplastic parameters are adopted to verify the structural capacity of the elements.

It can be observed in Figure 9 that, starting from a load of approximately 7,650.67 kN, the linear elastic model begins to enter the plastic regime, with a continuous increase in displacements until the ultimate load of 8,193.17 kN, where even a slight increase in loading results in up to a 10x increase in displacements. This is justified by the presence of large plastic deformations in the model, leading to a redistribution of forces in the structural element.

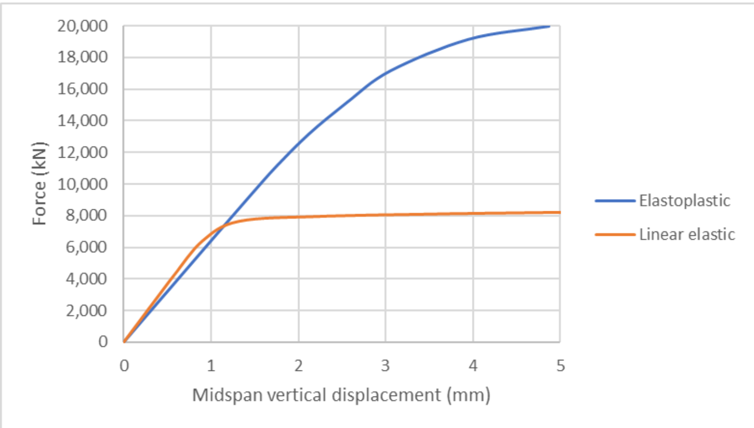

Figure 9. Load-bearing capacity for optimized beams

Yoon and Kim [16] mention that the stiffness, represented by the load-displacement curve dP/du, tends to be lower for the topology obtained with elastoplastic behavior in the elastic regime. However, it is higher in the plastic regime when compared to the model obtained by linear elastic topology.

Zhao [5] discusses that it is expected that the solution of the topology obtained by linear elastic analysis is stiffer than the topologies obtained by elastoplastic analysis in the elastic state. As soon as the structure initiates nonlinear deformations, structures based on plastic solutions tend to achieve higher load capacities.

4 Conclusions

In the field of topology optimization of nonlinear materials using ABAQUS®, efforts were made to clarify the key points in theoretical and numerical aspects for the density distribution procedure in a design domain. With the assistance of this tool, the behavior of optimized models was investigated in comparison to the elastoplastic Drucker-Prager model, contributing to the dissemination of the use of nonlinear models that provide more realistic responses for various types of materials.

The numerical responses obtained through the presented approach can assist designers in the decision-making process for optimized solutions, as the material distribution is determined by a rigorous mathematical criterion, enabling a reduction in time compared to physical tests. These responses, in general, are less resistant and less ductile than real ones, which may result in an additional safety factor for computationally performed investigations.

Acknowledgements. The authors acknowledge the Laboratory of Scientific Computing and Visualization (LCCV), from the Center of Technology (CTEC) at the Federal University of Alagoas (UFAL) for the support provided in this research. The second author acknowledges the support from the Brazilian National Council for Scientific and Technological Development (CNPq) to develop this work.

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] S. Schwarz, K. Maute, E. Ramm. "Topology and shape optimization for elastoplastic structural response". *Comput. Methods Appl. Mech. Eng.*, vol. 190, n. 15-17, p. 2135-2155, 2001.
[2] D. Jung and H. C. Gea. "Topology optimization of nonlinear structures". *Finite Elem. Anal. Des.*, vol. 40, n. 11, p. 1417-1427, 2004.
[3] R. Lotfi. *Topology optimization under nonlinear mechanics*. Dissertation. John Hopkins University, 2013.
[4] L. Xia., F. Fritzen, P. Breitkopf, P. "Evolutionary topology optimization of elastoplastic structures". *Struct. Multidiscip. Optim.*, vol. 55, p. 569-581, 2017.
[5] T. Zhao. *Nonlinear topology optimization: theory and applications*. Dissertation. Georgia Institute of Technology, 2021.
[6] D. Da. *Topology Optimization Design of Heterogeneous Materials and Structures*. Wiley. 2019.
[7] G. Vantyghem, V. Boel, W. De Corte, M. Steeman. "Compliance, stress-based and multi-physics topology optimization for 3D-printed concrete structures". *RILEM International Conference on Concrete and Digital Fabrication*. Springer, p. 323-332, 2018.
[8] G. Gaganelis, P. Mark, P. Forman. *Optimization Aided Design: Reinforced Concrete*. John Wiley & Sons, 2022.
[9] G. Wen, J. Liu, Z. Chen, P. Wei, K. Long, H. Wang, J. Rong, Y. Xie. "A survey of nonlinear continuum topology optimization methods". *Chin. J. Theor. Appl. Mech.*, vol. 54, n. 9, p. 1-17, 2022.
[10] N. H. Kim. *Introduction to Nonlinear Finite Element Analysis*. Springer, 2015.
[11] M. P. Bendsøe and N. Kikuchi. "Generating optimal topologies in structural design using a homogenization method". *Comput. Methods Appl. Mech. Eng.*, vol. 71, n. 2, p. 197-224, 1988.
[12] G. I. N. Rozvany. "A critical review of established methods of structural topology optimization". *Struct. Multidiscip. Optim.*, vol. 37, n. 3, p. 217-237, 2009.
[13] J. Wu, O. Sigmund, J. P. Groen. "Topology optimization of multi-scale structures: a review". *Struct. Multidiscip. Optim.*, vol. 63, p. 1455-1480, 2021.
[14] SIMULIA. *Abaqus User's Manual volume 2*. 2014.
[15] T. Zhao, E. N. Lages, A. S. Ramos Jr., G. H. Paulino. "Topology optimization considering the Drucker–Prager criterion with a surrogate nonlinear elastic constitutive model". *Struct. Multidiscip. Optim.*, vol. 62, n. 6, p. 3205-3227, 2020.
[16] G. H. Yoon and Y. Y. Kim. "Topology optimization of material-nonlinear continuum structures by the element connectivity parameterization". *Int. J. Numer. Methods Eng.*, vol. 69, n. 10, p. 2196-2218, 2006.