XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGENEERING

Integration of strain gradient notation finite elements into SIMP-based topology optimization

Leilson J. Araujo[1], João E. Abdalla Filho[1]

[1]*Graduate Program in Civil Engineering, Federal University of Technology – Paraná (UTFPR)*
Rua Deputado Heitor Alencar Furtado, 5000, Curitiba, 81280-340, Paraná, Brazil
leilsonjoaquim@gmail.com, joaofilho@utfpr.edu.br

Abstract. This work investigates the integration of the Strain Gradient Notation Finite Element Method (SGN-FEM) into a Solid Isotropic Material with Penalization (SIMP) topology optimization framework. Unlike conventional isoparametric quadrilateral elements, SGN-FEM eliminates parasitic shear terms directly at the formulation stage, avoiding shear locking effects without the need for numerical corrections such as reduced integration. This straightforward formulation allows its direct implementation into existing optimization codes with minimal computational overhead. Numerical experiments were conducted on classical benchmark problems, including the MBB beam and a cantilever beam with a passive hole. The results showed that SGN-FEM achieved solutions with topologies equivalent to conventional formulations, maintaining compliance and volume constraints while exhibiting comparable or improved convergence behavior. These findings highlight SGN-FEM as a robust and efficient alternative for topology optimization, providing a stable basis for future applications in multiscale optimization.

Keywords: Topology optimization, SGN-FEM, SIMP method, Shear locking, Finite element method

1 Introduction

Topology optimization is a well-established method for designing lightweight and high-performance structures, enabling the optimal distribution of material within a fixed domain under load, boundary, and volume constraints. Since the pioneering work of Bendsøe [1], topology optimization has become a key tool in structural design, with increasing demand for more efficient structural solutions. In this context, multiscale optimization strategies have gained relevance, allowing microstructural information to be directly integrated into the macrostructural design process [2–6]. Ensuring the accuracy and robustness of such methods depends largely on the finite element formulation adopted at each scale [7].

The Solid Isotropic Material with Penalization (SIMP) method is one of the most applied techniques in multiscale topology optimization problems. Its continuous formulation, based on a scalar density field, enables the computation of equivalent constitutive properties at the microscale through homogenization procedures, which facilitates efficient coupling between macro and micro levels [1]. Due to its simplicity and compatibility with consolidated optimization frameworks, SIMP has become a standard choice in multiscale design.

However, despite its widespread adoption, SIMP-based results are sensitive to the quality of discretization at the macroscopic level. Conventional finite elements, such as the bilinear quadrilateral (Q4), are prone to numerical issues like shear locking, especially in bending-dominated problems or in meshes with low refinement level [8]. These limitations can significantly affect the accuracy of the optimized topology, particularly in cases where fine-scale features are critical.

To overcome these drawbacks, this work applies the Strain Gradient Notation Finite Element Method (SGN-FEM) [8] within a SIMP-based topology optimization framework. The SIMP implementation developed by Andreassen et al. [9] is used as the baseline code, in which the SGN finite element formulation is integrated for structural analysis at the macro scale. As a preliminary study, numerical results for classical benchmark problems are presented to validate the SGN-FEM implementation and establish a foundation for its future application in multiscale optimization frameworks.

2 Theoretical Background

2.1 Topology Optimization using SIMP

Topology optimization aims to distribute material within a fixed domain to achieve optimal structural performance under loading and volume constraints. Among the available methods, the Solid Isotropic Material with Penalization (SIMP) technique is widely used due to its simplicity and efficiency [1, 9].

In SIMP, a continuous density field ρ is associated with each finite element, interpolating its stiffness as $E(\rho) = \rho^p E_0$, where p is the penalization factor and E_0 is the material's Young's modulus. The optimization problem is formulated as:

$$\begin{aligned} \text{Minimize} \quad & c(\rho) = \mathbf{f}^T \mathbf{u} \\ \text{Subject to} \quad & \mathbf{K}(\rho)\mathbf{u} = \mathbf{f}, \\ & \frac{v(\rho)}{V_0} \leq F, \\ & 0 < \rho_{\text{min}} \leq \rho_j \leq 1, \quad j = 1, \ldots, n. \end{aligned} \tag{1}$$

Here, $c(\rho)$ is the structural compliance, $\mathbf{f}$ is the global load vector, $\mathbf{u}$ is the global displacement vector, and $\mathbf{K}(\rho)$ is the global stiffness matrix assembled from the element stiffness matrices dependent on ρ. The term $v(\rho)$ denotes the material volume, V_0 is the design domain volume, and F is the prescribed volume fraction. The lower bound ρ_{min} avoids singular stiffness matrices, ρ_j is the density variable of the j-th element, and n is the total number of elements in the mesh.

The SIMP implementation adopted in this work is based on the 88-line MATLAB code by Andreassen et al. [9], serving as the baseline for incorporating the SGN-FEM.

2.2 Strain Gradient Notation Finite Element Method (SGN-FEM)

Isoparametric quadrilateral (Q4) finite elements are widely employed in topology optimization due to their simplicity and straightforward implementation. However, these elements are susceptible to *shear locking*, particularly in bending-dominated problems or in meshes with low refinement. This numerical pathology leads to an artificially stiff structural response, compromising the accuracy of the solution.

The Strain Gradient Notation Finite Element Method (SGN-FEM), introduced by Dow [8], reformulates the element kinematics by expressing the displacement field in terms of strain-gradient coefficients. For a Q4 element, the displacement components in the x and y directions, denoted by $u(x, y)$ and $v(x, y)$, respectively, are approximated by polynomial expansions including first-order derivatives of the strain field as:

$$u(x, y) = (u_{rb})_0 + (\varepsilon_x)_0\, x + \left(\frac{\gamma_{xy}}{2} - r_{rb}\right)_0 y + (\varepsilon_{x,y})_0\, xy, \tag{2}$$

$$v(x, y) = (v_{rb})_0 + \left(\frac{\gamma_{xy}}{2} + r_{rb}\right)_0 x + (\varepsilon_y)_0\, y + (\varepsilon_{y,x})_0\, xy, \tag{3}$$

where:

- $(u_{rb})_0$ and $(v_{rb})_0$ are rigid body translations in the x and y directions, respectively;
- $(\varepsilon_x)_0$ and $(\varepsilon_y)_0$ are constant normal strain components in the x and y directions;
- γ_{xy} is the shear strain;
- r_{rb} is the rigid body rotation about the z axis;
- $(\varepsilon_{x,y})_0$ and $(\varepsilon_{y,x})_0$ are the derivatives of the normal strain components, representing strain gradients in the y and x directions, respectively.

These strain-gradient coefficients possess a direct physical interpretation. For instance, $(\varepsilon_{x,y})_0$ represents the linear variation of the constant term $(\varepsilon_x)_0$ along the y direction, corresponding to the bending strain distribution produced by a constant bending moment.

The strain–displacement relationships are obtained by differentiating the displacement field with respect to the spatial coordinates:

$$\frac{\partial u}{\partial x} = \varepsilon_x = (\varepsilon_x)_0 + (\varepsilon_{x,y})_0\, y, \tag{4}$$

$$\frac{\partial v}{\partial y} = \varepsilon_y = (\varepsilon_y)_0 + (\varepsilon_{y,x})_0\, x, \tag{5}$$

$$\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \gamma_{xy} = (\gamma_{xy})_0 + (\varepsilon_{x,y})_0\, x + (\varepsilon_{y,x})_0\, y. \tag{6}$$

Equations (4) and (5) correctly describe the linear variation of the normal strain components ε_x and ε_y within the element. However, the shear strain component γ_{xy} in eq. (6) contains additional terms proportional to $(\varepsilon_{x,y})_0$ and $(\varepsilon_{y,x})_0$, which correspond to derivatives of the normal strains. These parasitic shear terms introduce coupling between normal and shear strains.

In SGN-FEM, these spurious terms are removed directly at the formulation stage, yielding the corrected shear strain expression:

$$\gamma_{xy} = (\gamma_{xy})_0, \tag{7}$$

thereby ensuring that shear deformation is represented independently of normal strain variations. This *a priori* correction eliminates shear locking without requiring numerical remedies such as reduced or selective integration.

The element stiffness matrix in SGN-FEM is expressed as:

$$[K_e] = [\Phi]^{-T}\, \bar{U}\, [\Phi]^{-1}, \tag{8}$$

where:

- $[K_e]$ is the element stiffness matrix;
- $[\Phi]$ is the transformation matrix that relates nodal displacements to strain-gradient degrees of freedom;
- $\bar{U}$ is the strain energy matrix, defined in terms of the corrected strain measures.

A detailed derivation of the Q4 SGN-FEM formulation can be found in Dow [8].

This formulation ensures a physically consistent representation of the element behavior and provides a locking-free alternative to conventional isoparametric Q4 elements, while maintaining compatibility with standard finite element assembly procedures.

3 Methodology

This study integrates the Strain Gradient Notation Finite Element Method (SGN-FEM) into a SIMP-based topology optimization framework. The original 88-line MATLAB code by Andreassen et al. [9] was adapted to replace the isoparametric bilinear quadrilateral (Q4) element with the SGN-FEM formulation presented in Section 2, without altering the optimization algorithm.

Two finite element models were considered: the **Conventional Element (CE)**, using the isoparametric Q4 formulation, and the **Strain Gradient (SG)** element, with parasitic shear terms eliminated at the formulation stage.

Numerical experiments were conducted on two classical benchmark problems. The first is the MBB beam, with pinned supports on the left edge, a roller at the bottom right, and a vertical load applied at the upper left corner. The second is a cantilever beam with a circular hole, fully clamped on the left edge and subjected to a vertical tip load. Mesh discretizations of 60×20, 150×50, and 300×100 elements were used for the MBB beam, while the cantilever beam with hole was discretized with 120×60 elements. The boundary conditions and load configurations are shown in Fig.s 1 and 2.

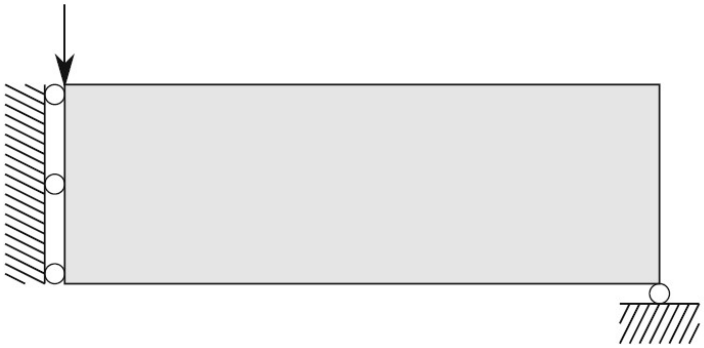


Figure 1. Design domain, boundary conditions, and external load for the MBB beam.

Both models were evaluated under a volume constraint of 50%, using a penalization factor $p = 3$ and a sensitivity filter radius of 1.5 element widths. The optimization process was considered converged when compliance changes between successive iterations became negligible, while maintaining the volume constraint.

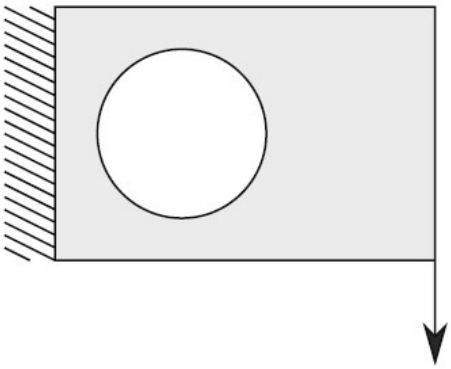


Figure 2. Design domain, boundary conditions, and external load for the cantilever beam with hole.

The number of iterations and computational time were recorded to compare the performance of SGN-FEM against the conventional formulation, with particular attention to convergence behavior and numerical robustness in bending-dominated scenarios.

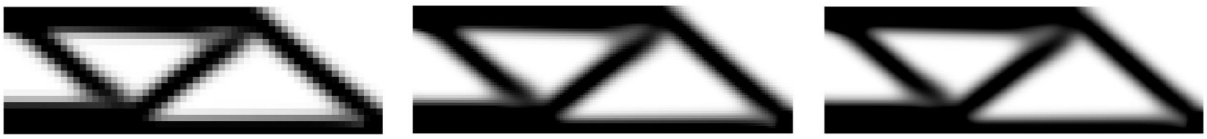
4 Results

Numerical experiments were conducted to evaluate the performance of the SGN-FEM formulation in compliance minimization problems. Two benchmark cases were analyzed: the MBB beam and a cantilever beam with a hole. The results obtained using the SGN-FEM (**SG**) were compared against the conventional quadrilateral element (**CE**), considering iteration count, compliance, volume constraint, and computational time.

Tab. 1 presents the numerical results for the MBB beam problem. Both formulations achieved similar compliance values and volume constraints, with the SG model exhibiting a slight increase in compliance (approximately 0.3% to 0.4%) due to the elimination of parasitic shear terms. The SG model also demonstrated a marginal reduction in iteration count for the coarser mesh (60x20) and slightly lower computational times across all mesh configurations. The optimized topologies obtained with SGN-FEM are illustrated in Fig. 3, showing well-defined and physically consistent structures.

Table 1. Numerical results for the MBB beam using the SIMP method.

Mesh	Method	Iterations	Compliance	Volume	Time (s)
60x20	CE	106	216.8137	0.5000	3.5509
60x20	SG	104	217.4516	0.5000	3.5334
150x50	CE	95	219.5199	0.5000	7.5270
150x50	SG	96	220.2974	0.5000	7.4812
300x100	CE	93	222.2857	0.5000	28.1940
300x100	SG	93	222.8015	0.5000	27.8989



(a) Mesh 60x20 (b) Mesh 150x50 (c) Mesh 300x100

Figure 3. Optimized topologies for the MBB beam problem using SGN-FEM.

The second numerical experiment addressed a cantilever beam with a passive circular hole. Tab. 2 summarizes the results. Both CE and SG configurations reached convergence after 36 iterations, with SG achieving a slightly higher compliance value (0.93% increase) but requiring a lower computational time. The optimized topology

obtained with SGN-FEM is presented in Fig. 4, highlighting its capability to generate well-defined boundaries even in the presence of passive regions.

Table 2. Numerical results for the cantilever beam with a passive hole using the SIMP method.

Mesh	Method	Iterations	Compliance	Volume	Time (s)
150x100	CE	36	60.9485	0.5000	5.7589
150x100	SG	36	61.5160	0.5000	5.0995



Figure 4. Optimized topology for the cantilever beam with a passive hole using SGN-FEM.

Overall, the SGN-FEM formulation provided solutions free from locking effects while maintaining similar compliance and volume results to conventional elements. Additionally, SGN-FEM demonstrated slight improvements in computational efficiency, with reduced iteration counts and execution times, especially in cases involving bending-dominated behavior and geometric complexities.

5 Conclusions

This work investigated the integration of the Strain Gradient Notation Finite Element Method (SGN-FEM) into a SIMP-based topology optimization framework. Numerical experiments were performed on two benchmark problems (MBB beam and cantilever beam with a passive hole), comparing SGN-FEM to conventional quadrilateral elements (Q4).

The SGN-FEM formulation effectively eliminates locking phenomena through an *a priori* correction of parasitic shear terms, resulting in a slightly more flexible structural response compared to conventional elements. This characteristic led to marginally higher compliance values (less than 1%), indicating a more physically accurate representation of the structural behavior. Despite this minor difference, the optimized topologies remained consistent, and the volume constraints were fully satisfied.

Additionally, SGN-FEM required an equal or smaller number of iterations and achieved competitive computational times. These results demonstrate that SGN-FEM is a robust and efficient alternative to isoparametric finite elements in topology optimization. Future work will explore its application to multiscale optimization problems.

Acknowledgements. The authors acknowledge the Graduate Program in Civil Engineering (PPGEC) at the Federal University of Technology – Paraná (UTFPR).

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] M. P. Bendsøe. Optimal shape design as a material distribution problem. *Structural Optimization*, vol. 1, n. 4, pp. 193–202, 1989.

[2] J. Wu, O. Sigmund, and J. P. Groen. Topology optimization of multi-scale structures: a review. *Structural and Multidisciplinary Optimization*, vol. 63, pp. 1455–1480, 2021.
[3] F. Ferrari and O. Sigmund. A new generation 99 line matlab code for compliance topology optimization and its extension to 3d. *Structural and Multidisciplinary Optimization*. Accepted paper, arXiv preprint arXiv:2005.05436, 2020.
[4] V.-N. Hoang, P. Tran, V.-T. Vu, and H. Nguyen-Xuan. Design of lattice structures with direct multiscale topology optimization. *Composite Structures*, vol. 252, pp. 112718, 2020.
[5] G. Kazakis and N. D. Lagaros. A simple matlab code for material design optimization using reduced order models. *Materials*, vol. 15, n. 14, pp. 4972, 2022.
[6] H. Liu, Y. Wang, H. Zong, and M. Y. Wang. Efficient structure topology optimization by using the multiscale finite element method. *Structural and Multidisciplinary Optimization*, vol. 58, pp. 633–653, 2018.
[7] T. H. Nguyen, C. H. Le, and J. F. Hajjar. Topology optimization using the p-version of the finite element method. *Structural and Multidisciplinary Optimization*, vol. 56, n. 2, pp. 335–348, 2017.
[8] J. O. Dow. *A Unified Approach to the Finite Element Method and Error Analysis Procedures*. Academic Press, San Diego, USA, 1999.
[9] E. Andreassen, A. Clausen, M. Schevenels, B. S. Lazarov, and O. Sigmund. Efficient topology optimization in matlab using 88 lines of code. *Structural and Multidisciplinary Optimization*, vol. 43, n. 1, pp. 1–16, 2011.