Key Engineering Materials
ISSN: 1662-9795, Vol. 926, pp 2228-2237
doi:10.4028/p-6rou28

Submitted: 2021-12-12
Revised: 2022-01-28
Accepted: 2022-01-30
Online: 2022-07-22

On the Design of a Heterogeneous Mechanical Test Using a Nonlinear Topology Optimization Approach

Mafalda Gonçalves[1,a*], António Andrade-Campos[1,b], and Sandrine Thuillier[2,c]

[1]Department of Mechanical Engineering, Centre for Mechanical Technology and Automation, University of Aveiro, Portugal

[2]Univ. Bretagne Sud, UMR CNRS 6027, IRDL, F-56100 Lorient, France
[a]mafalda.goncalves@ua.pt, [b]gilac@ua.pt, [c]sandrine.thuillier@univ-ubs.fr

Keywords: Heterogeneous Test, Topology Optimization, Compliant Mechanisms, Nonlinear FEA.

Abstract. Numerical simulation is nowadays increasingly used to avoid the costs and time associated with the development and optimization of metal forming processes. However, the accuracy of the numerical results is still an issue. Material behavior and characteristics are required by simulation software, and these are usually obtained by performing a considerable number of classical mechanical tests. To improve this procedure, heterogeneous tests have been used instead. More and richer information can be obtained with a single test due to the heterogeneous displacement and strain fields that are induced. This work aims at designing a heterogeneous mechanical test using a topology-based optimization methodology. Highly heterogeneous displacement fields are induced on the sheet specimen by applying an extended version of the theory of compliant mechanisms. To account for large deformations, a geometrically nonlinear finite element analysis is proposed together with a consistent topology optimization approach. The material behavior is considered linear elastic. The performance of the obtained solutions is evaluated considering the heterogeneity of stress states using a mechanical indicator. Validation of the developed methodology is performed and an optimal mechanical test is obtained presenting a high diversity of stress states.

Introduction

Sheet metal forming processes are present in the majority of the engineering applications. Their virtualization through numerical simulation is of upmost importance resulting in reduced costs and delays that are usually associated with the development and optimization of metal forming processes. The correct reproduction of the material behavior is a key factor for its success. For that purpose, material parameters present in constitutive models need to be identified since these are required to model the material behavior. The classical characterization process establishes the use of a whole range of standard mechanical tests composed of tensile or shear tests, for example. However, several attempts have been made to improve this time-consuming and expensive process. One of the approaches consists in replacing multiple classical mechanical tests by a nonstandard one. The existence of heterogeneous displacement and strain fields in the sheet specimen was proven to provide richer information about the material behavior when compared to the homogeneous fields induced by classical mechanical tests [1].

Several authors have already addressed the design of a single nonstandard mechanical test to enhance the material characterization process [2]. Although considerable steps have already been taken, the goal of designing a single test configuration to identify all the material parameters of a constitutive model has not been achieved yet. Different approaches of increasingly complexity have been proposed over the years. A more straightforward approach consists in using the intuition and knowledge of the authors to design new nonstandard specimen geometries with strain heterogeneities. This approach can be seen, for instance, in [3–6]. Several works tried to induce in the specimen the maximum diversity of stress states using an optimization approach [7–10]. A step further makes use of the identification quality as a criterion for the test design process [11–15]. Full-field experimental techniques have been used for measuring the heterogeneous strain fields and their metrological parameters can also be introduced in the design procedure [16–19]. Most of the works does not propose an optimization approach for the design of the test and, in these cases, it is not certain that an optimal solution can be obtained. Moreover, the obtained solutions are usually dependent on the proposed initial geometry or on the set of boundary conditions. Therefore, this work aims at tackling these issues and making a considerable progress towards the improvement of the material characterization process.

The design of a heterogeneous mechanical test is addressed in this study. A topology-based optimization approach is proposed to find the best material layout in the sheet specimen, as already addressed by [10, 15, 20]. The theory of compliant mechanisms [21] along with the potential of the design by Topology Optimization (TO) in obtaining complex specimen geometries can be responsible for inducing highly heterogeneous displacement fields. In order to obtain results closer to reality, a geometrically nonlinear finite element analysis together with a topology optimization procedure is proposed to account for the large deformations. The developed methodology is validated using a numerical example of an inverter mechanism and then the methodology is applied to obtain an optimal mechanical test that presents a high diversity of stress states.

Methodology

The design of a heterogeneous mechanical test has been the focus of several works due to the benefits that its wide use can bring to the material characterization process. The heterogeneous strain fields that are generated on the specimen provide an important amount of information about the material behavior. This information is also of significant importance to enable an efficient material characterization procedure. Inducing directly strain fields in the specimen is a complex task so this works aims at introducing heterogeneity through the displacement field. It is proposed to use an extended version of the theory of the compliant mechanisms [21] to control the displacements in different locations of the specimen. Therefore, specific mechanical states could be induced depending on the applied displacements. A Compliant Mechanism (CM) is a mechanism known for transforming the displacement at least partly through the deformation of its flexible members [22]. Based on this idea, Figure 1 shows how two displacements applied in two different locations of the specimen can induce a specific mechanical state, i.e., shear, tension and compression.

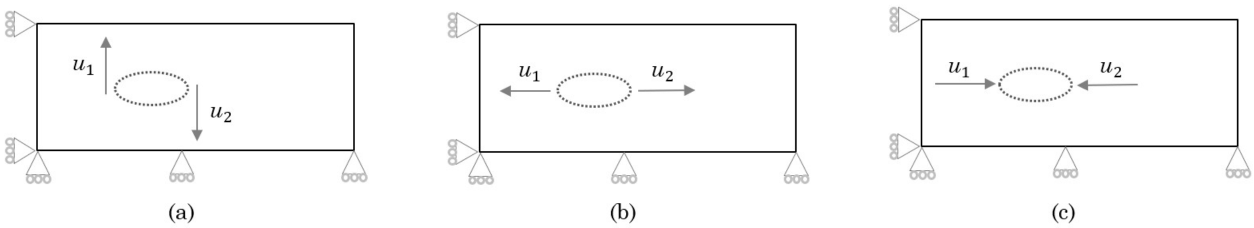


Fig. 1: Schematic representation of the application of the theory of compliant mechanisms to control the displacement field and, consequently, to induce a specific mechanical state: (a) shear, (b) tension, and (c) compression. The material points subjected to the imposed mechanical state are surrounded by an ellipse for easier understanding.

The methodology proposed in this work aims at coupling the presented approach with the design by topology optimization and its framework is depicted in Figure 2. Nonstandard specimen geometries can be obtained with the topology design method applying the theory of compliant mechanisms. The heterogeneity of the displacement field is therefore improved and specific strain/stress states can be induced depending on the applied displacements, leading to the enrichment of the strain field.

Topology optimization aims at finding the best material layout of the specimen considering a predefined design domain with established boundary conditions. The topology design domain is established as a starting point for the optimization process and its geometry is chosen in order to not restrict the design update and, consequently, the obtained solutions. Therefore, it is proposed the initial design domain represented in Figure 3. The size of the specimen is irrelevant due to the normalization

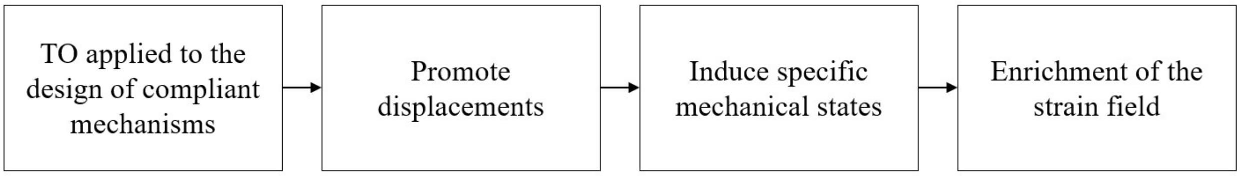


Fig. 2: Framework for the design methodology developed in this work.

of the analysis, being only the length/height ratio considered. Only one quarter of the specimen is represented and plane stress conditions are considered for the analysis. It is applied a uniaxial tensile loading, $\mathbf{F}_{\text{in}}$, that represents the load applied by the grips of the Universal Testing Machine (UTM) during a uniaxial tensile loading test. The height of the specimen is more than twice the height of half of the grips, supposing that this height is enough to achieve optimal solutions. Two displacements, u_{in} and u_{out}, are applied in specific locations of the specimen, input and output, respectively, that are chosen empirically by the authors. While the first one is applied in the lower right node, corresponding to the grips displacement, the output location is chosen to be far from the specimen boundary not to restrict the material layout. K_{in} and K_{out} stand for artificial stiffnesses that are introduced in input and output locations, respectively, to prevent the appearance of numerical issues.

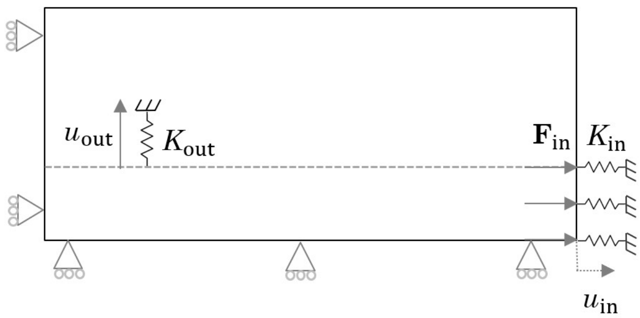


Fig. 3: Schematic representation of the topology design domain subjected to a uniaxial tensile loading.

Problem Formulation. The optimization procedure aims at finding an optimal specimen geometry. Considering the design by topology optimization, the specimen geometry can be defined by a discretized material distribution composed of solid and void elements. These are usually the same as in the finite element analysis and each one is described by a design variable, its relative density, X_{e}. Considering a void or material region, its value can be equal to 0 or 1, respectively. Along the optimization process, the material layout, represented by $\mathbf{X}$, is updated at each iteration, being material added or removed from each element with the goal of maximizing the objective function. Based on the extended version of the theory of complaint mechanisms, it is proposed to maximize the ratio between the displacements in the output and input locations in order to increase the heterogeneity of the displacement field. The optimization problem in the nonlinear framework can be defined as follows

$$\begin{aligned} \underset{\mathbf{X}}{\text{maximize}} \quad & T(\mathbf{X}) = \frac{u_{\text{out}}(\mathbf{X})}{u_{\text{in}}(\mathbf{X})}, \\ \text{subject to} \quad & \mathbf{R} = \mathbf{0}, \\ & \frac{\sum_{e=1}^{M} X_e V_e}{\sum_{e=1}^{M} V_e} - V^* \leq 0, \\ & 0 \leq \rho_{\text{min}} \leq X_e \leq 1,\ e = 1, 2, .., M, \end{aligned} \tag{1}$$

where X_e and V_e stand for the density and the volume of each element, respectively, and M for the total number of elements. The volume constraint is computed in each iteration to make sure the required volume fraction for the specimen, V^*, is respected. A minimum value for the element density, ρ_{min}, is established as 0.001 in order to avoid numerical issues in the optimization process. The mechanical equilibrium of the system is achieved when a balance between the internal and external loads is obtained and, consequently, the residual $\mathbf{R}$ is equal to the zero vector.

Nonlinear finite element analysis. A nonlinear finite element analysis is proposed to reproduce accurately the behavior of the specimen when submitted to a uniaxial tensile loading test. The material behavior is considered linear elastic, but geometric nonlinearity is introduced in the analysis as has already been addressed by Han *et al.* [23]. Although large displacements are considered in this work, the strains remain small. For that purpose, the Green-Lagrange strain tensor is used and can be written as linear and nonlinear parts as follows:

$$\mathbf{E} = \mathbf{E}_{\mathrm{L}} + \mathbf{E}_{\mathrm{N}} \; = \; \mathbf{B}_{\mathrm{L}}\mathbf{U} + \mathbf{B}_{\mathrm{N}}\mathbf{U}, \tag{2}$$

where $\mathbf{B}_{\mathrm{L}}$ and $\mathbf{B}_{\mathrm{N}}$ correspond to the linear and nonlinear transformation matrices between nodal displacements, $\mathbf{U}$, and element strains, $\mathbf{E}$, respectively. Along with the strain measure, the second Piola-Kirchoff stress tensor, $\mathbf{S}$, is also employed in the system analysis.

The equilibrium of the system is one of the constraints of the optimization procedure and can be defined as

$$\mathbf{R} \; = \int \mathbf{B}^{\mathrm{T}}\mathbf{S}\, dV - \; \mathbf{F}^{\mathrm{ext}} = \; \mathbf{F}^{\mathrm{int}} \; - \; \mathbf{F}^{\mathrm{ext}}, \tag{3}$$

where $\mathbf{F}^{\mathrm{int}}$ stands for the internal load of the system. $\mathbf{B}$ is composed of both the linear and nonlinear matrices, $\mathbf{B}_{\mathrm{L}}$ and $\mathbf{B}_{\mathrm{N}}$, respectively. The residual $\mathbf{R}$ has to be equal to zero when the equilibrium is achieved. The finite element procedure is solved iteratively using the Newton-Raphson algorithm and requires the determination of the tangent stiffness matrix defined as

$$\mathbf{K}_{\mathrm{T}} = -\frac{\partial \mathbf{R}}{\partial \mathbf{U}}, \tag{4}$$

that can be written as follows

$$\begin{aligned} \mathbf{K}_{\mathrm{T}} \;&=\; \mathbf{K}_{\mathrm{L}} + \mathbf{K}_{\mathrm{N}} + \mathbf{K}_{\mathrm{S}} \\ &=\; \int \mathbf{B}_{\mathrm{L}}^{\mathrm{T}}\mathbf{D}\mathbf{B}_{\mathrm{L}} dV + \int \mathbf{B}_{\mathrm{L}}^{\mathrm{T}}\mathbf{D}\mathbf{B}_{\mathrm{N}} dV + \int \mathbf{B}_{\mathrm{N}}^{\mathrm{T}}\mathbf{D}\mathbf{B}_{\mathrm{L}} dV + \int \mathbf{B}_{\mathrm{N}}^{\mathrm{T}}\mathbf{D}\mathbf{B}_{\mathrm{N}} dV + \int \mathbf{G}^{\mathrm{T}}\mathbf{M}\mathbf{G} dV, \end{aligned} \tag{5}$$

where, in this work, $\mathbf{D}$ stands for the elasticity tensor for a linear isotropic material in plane stress conditions. The first two terms, $\mathbf{K}_{\mathrm{L}}$ and $\mathbf{K}_{\mathrm{N}}$, are related to the constitutive tensor, one under the assumption of small displacement and the other caused by large ones, respectively. The latter term, $\mathbf{K}_{\mathrm{S}}$, is related to the initial stress state, in which $\mathbf{M}$ is composed of elements of the stress tensor $\mathbf{S}$ and $\mathbf{G}$ stands for a derivative matrix of shape functions with respect to coordinates.

The finite element analysis is carried out with the assumption of large displacements using the Green-Lagrange strain measure for the evaluation of the system. As a consequence, the tangent stiffness matrix may become singular and lead to the nonconvergence of the equilibrium equations. This issue is raised by elements with the minimum density that are subjected to large deformations. According to Buhl *et al.* [24], this is an artificial problem since these elements should be void and their behavior should not influence the structural response. Several approaches have already been proposed to overcome these issues from using a hyper-elastic material law to enhance the stiffness of these low-density elements [25] to only use the linear part of the stiffness matrix to characterize the stiffness in these same elements [26]. However, it was also mentioned by Buhl *et al.* [24] the noneffectiveness of these approaches, being suggested the relaxation of the convergence criterion for the equilibrium iterations. The convergence criterion is based on a limit value to the changes in nodal displacements. These keep oscillating in some nodes, specially, for nodes surrounded by void elements. Consequently, this study proposes to eliminate these nodes from the convergence criterion.

Sensitivity analysis. The objective-function of the topology optimization problem evaluates the ratio between the displacements in the output and input locations. Therefore, the sensitivity of the objective-function can be defined as

$$\frac{dT}{d\mathbf{X}} = \frac{\frac{du_{\text{out}}}{d\mathbf{X}} u_{\text{in}} - \frac{du_{\text{in}}}{d\mathbf{X}} u_{\text{out}}}{u_{\text{in}}^2}, \tag{6}$$

where the derivative of the output and input displacements can be derived using the adjoint variable method in the following manner,

$$u_{\text{in}} = \mathbf{L}_{\text{in}}^{\text{T}} \mathbf{U} + \boldsymbol{\gamma}^{\text{T}} \mathbf{R}, \tag{7}$$

$$u_{\text{out}} = \mathbf{L}_{\text{out}}^{\text{T}} \mathbf{U} + \boldsymbol{\lambda}^{\text{T}} \mathbf{R}, \tag{8}$$

where $\mathbf{L}_{\text{in}}$ and $\mathbf{L}_{\text{out}}$ stand for two vectors with the value of one at the input and output locations and zeros in the remaining ones, respectively. Assuming that the equilibrium has been found, the terms $\boldsymbol{\gamma}^{\text{T}}\mathbf{R}$ and $\boldsymbol{\lambda}^{\text{T}}\mathbf{R}$ are equal to zero and, therefore can be added to the displacements. The derivative of the input and output displacements can be written as

$$\frac{\partial u_{\text{in}}}{\partial X_e} = -p \left[(1 - X_e)\, \rho_{\text{min}}^{p-1} + X_e \right] \boldsymbol{\gamma}_e^{\text{T}} \mathbf{F}_{\text{int}}^e, \tag{9}$$

and

$$\frac{\partial u_{\text{out}}}{\partial X_e} = -p \left[(1 - X_e)\, \rho_{\text{min}}^{p-1} + X_e \right] \boldsymbol{\lambda}_e^{\text{T}} \mathbf{F}_{\text{int}}^e, \tag{10}$$

respectively. The adjoint vectors $\boldsymbol{\lambda}$ and $\boldsymbol{\gamma}$ can be easily obtained as the solutions to the linear adjoint equations $\mathbf{K}_{\text{T}}\boldsymbol{\lambda} = \mathbf{L}_{\text{out}}$ and $\mathbf{K}_{\text{T}}\boldsymbol{\gamma} = \mathbf{L}_{\text{in}}$. The tangent stiffness matrix has already been found during the equilibrium equations. The material properties are determined using the Solid Isotropic Material with Penalization (SIMP) method [27], in which the element densities are penalized using a penalization factor, p. Also, a linear interpolation between the two phases, solid and void, is required due to the nonhomogenized combination of solid and void in intermediate elements that is established by the adopted methodology [28].

Solution evaluation. An optimal specimen geometry with a heterogeneous displacement field is obtained from the design methodology proposed in this work. Based on its strain/stress field, its performance is then evaluated using a mechanical indicator [10]. The mechanical indicator analyzes the heterogeneity of the induced stress states and can be computed as

$$id = \prod_{s=1}^{3} \left[\frac{3}{\sum_{e=1}^{n} X_e} \sum_{e=1}^{n} \left({}^{s}\delta_e \, Z_e \, X_e \right) \right], \tag{11}$$

where the index s indicates the stress state (tension, compression, and shear) that the element e is subjected to, establishing the value of parameter ${}^{s}\delta_e$. Solutions with stress concentrations or unstressed material are penalized based on the von Mises stress using the parameter Z_e. The ideal solution would present the same amount of material in the three stress states (tension, compression, and shear) without stress concentrations or unstressed material.

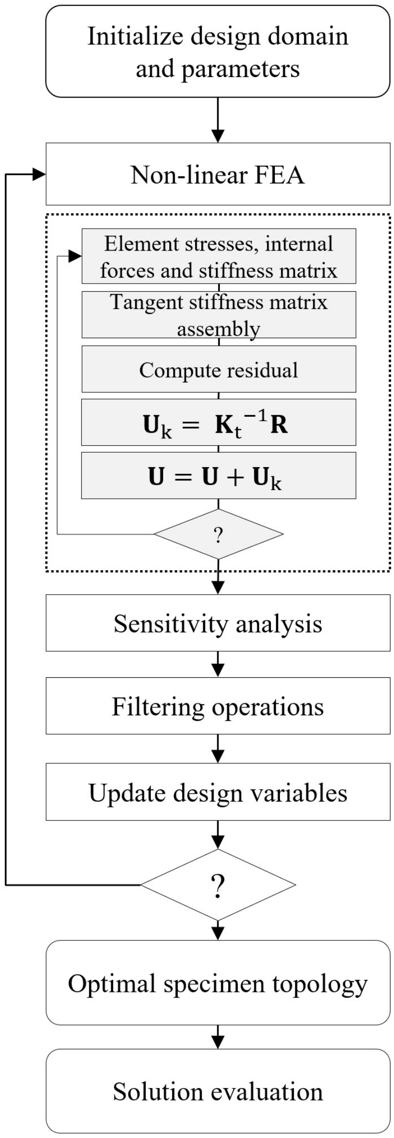


Fig. 4: Flow diagram of the nonlinear topology optimization methodology.

Implementation

The design procedure is based on a topology-based optimization methodology from which an optimal specimen is obtained. The structure of the developed methodology is depicted in Figure 4. The boundary conditions, the volume fraction and the filtering parameters [28] are initially defined. The initial design domain is submitted to a nonlinear finite element analysis, from which the displacements and internal forces are obtained. The objective function is computed along with the sensitivity matrix. After submitting the sensitivities to a filtering operation, the design variables are updated by the the Method of Moving Asymptotes (MMA) [29]. If the convergence criterion is reached, the optimal solution is obtained. Otherwise, a new iteration starts with an updated topology.

The finite element analysis procedure accounts for the large displacements that the specimen is submitted to. First, for the current topology, the stress, internal force and stiffness matrix are computed for each element. The tangent stiffness matrix is then assembled and the residual is computed. The equilibrium equation is evaluated and the obtained displacements for the analyzed iteration are added to the global ones. When the changes in the nodal displacements are below a certain value, the convergence is achieved. Therefore, the procedure is similar to a standard Finite Element Analysis (FEA).

Results and Discussion

For validation purposes, the proposed design optimization methodology is applied to the design of an inverter mechanism. This example is chosen due to its wide use in the literature [30, 31] to validate design methodologies of compliant mechanisms. The design domain and the boundary conditions are represented in Figure 5. The volume constraint is set to 25% of the total volume of the design domain.

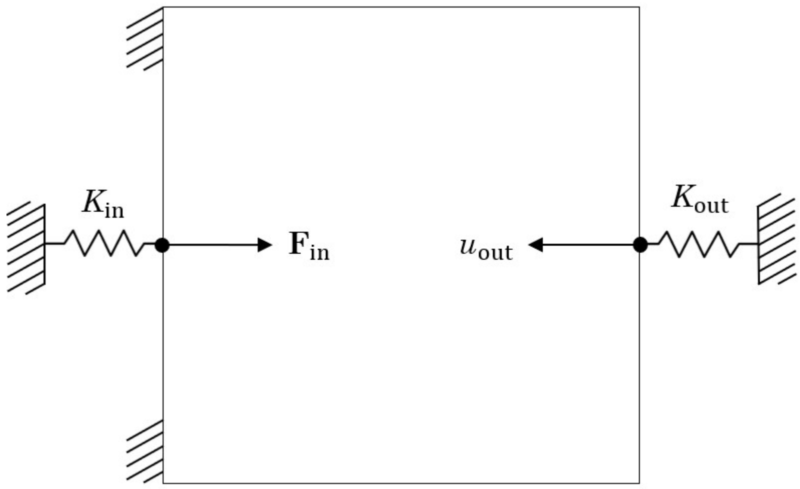
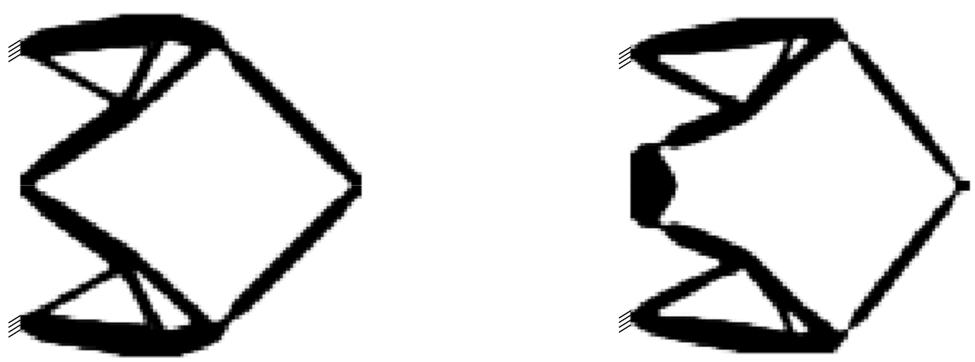


Fig. 5: Design domain of the displacement inverter mechanism. Adapted from [30].

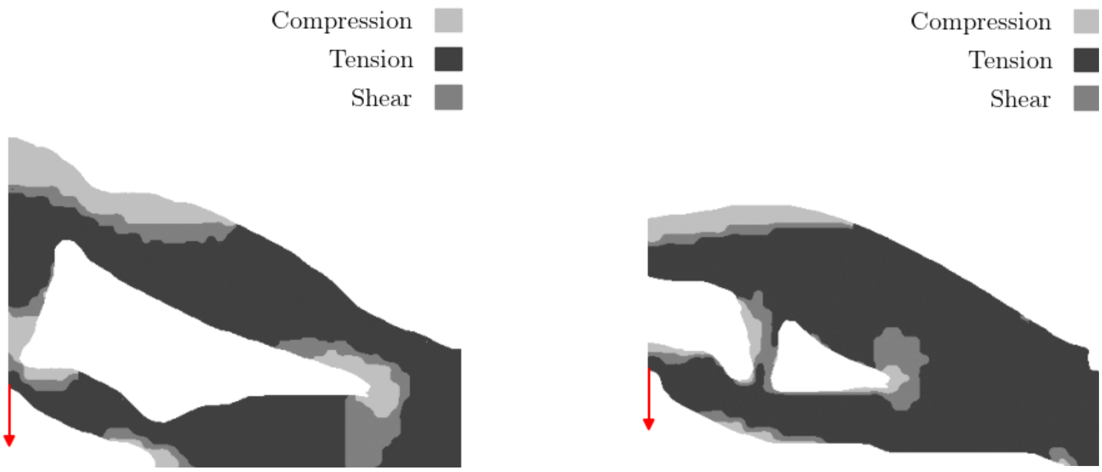
The optimized topologies obtained using a linear and nonlinear finite element analysis are shown in Figures 6(a) and (b), respectively. Similar analyses were addressed in the literature, in which both solutions were compared [31].The obtained topology using a nonlinear approach is quite similar to the one obtained in [30], validating the design methodology developed in this work. There are some differences between the topologies obtained with linear and nonlinear analysis, which proves the necessity of using a nonlinear finite element analysis to capture the proper behavior of the mechanism.



(a) (b)

Fig. 6: Obtained topologies for the displacement inverter mechanism using a: (a) linear and (b) nonlinear finite element analysis.

The developed methodology is now applied to design an optimal specimen from the initial design domain represented in Figure 3. A comparison is made between the topologies obtained with a linear and a nonlinear finite element analysis presented in Figures 7(a) and (b), respectively. The volume required for the specimen is 35% of the total initial volume. This volume was found to be the one that leads to the best results [20]. For the same reason, the output displacement is applied in the left symmetry boundary of the specimen and represented by the red arrow. It is used a mesh of 100×100 elements. The induced stress states (tension, compression and shear) are also represented in Figure 7. The topologies have some similarities related to the material placed around the symmetry boundary conditions and the locations of the applied load and displacements. However, several differences can be noticed due to the large-displacements approach, proving the necessity of using this strategy to reproduce accurately the specimen behavior.



(a) (b)

Fig. 7: Obtained specimen topologies using: (a) a linear and (b) a nonlinear finite element analysis. The stress states induced on the specimen are represented. The values of the mechanical indicator for the linear and nonlinear solutions are 0.0367 and 0.0120, respectively.

Concluding Remarks

In this work, a nonlinear topology-based optimization approach is proposed for the design of a heterogeneous mechanical test. Geometric nonlinearity is introduced in the finite element analysis in order to account for the large deformations. The proposed methodology is applied to the design of a displacement inverter mechanism for validation purposes. The obtained topology is compared to the ones from the literature, validating the developed methodology. The topologies obtained using a linear and nonlinear analysis are compared and it is concluded that a nonlinear analysis is required to reproduce in a correct way the mechanism behavior. Additionally, it is obtained an optimal specimen design, which presents an interesting diversity of stress states. Numerical issues appear due to the minimum stiffness of low-density elements leading to the nonconvergence of the finite element procedure. The relaxation of the convergence criterion is proposed to overcome this problem.

Acknowledgements

This project has received funding from the Research Fund for Coal and Steel under grant agreement No 888153. The authors also gratefully acknowledge the financial support of the Portuguese Foundation for Science and Technology (FCT) under the projects CENTRO-01-0145-FEDER-029713, POCI-01-0145-FEDER-031243 and POCI-01-0145-FEDER-030592 by UE/FEDER through the programs CENTRO 2020 and COMPETE 2020, and UIDB/00481/2020 and UIDP/00481/2020-FCT under CENTRO-01-0145-FEDER-022083. M. Gonçalves is grateful to the FCT for the Ph.D. grant Ref. UI/BD/151257/2021.

References

[1] M.G. Oliveira, S. Thuillier, and A. Andrade-Campos, Analysis of heterogeneous tests for sheet metal mechanical behavior, Procedia Manuf., 47 (2020) 831–838.

[2] F. Pierron, M. Grédiac, Towards Material Testing 2.0. A review of test design for identification of constitutive parameters from full-field measurements, Strain, 57 (2021) 1–22.

[3] D. Lecompte, A. Smits, H. Sol, J. Vantomme , D. Van Hemelrijck, Mixed numerical-experimental technique for orthotropic parameter identification using biaxial tensile tests on cruciform specimens, Int. J. Solids Struct., 44 (2007) 1643–1656.

[4] H. Haddadi and S. Belhabib, Improving the characterization of a hardening law using digital image correlation over an enhanced heterogeneous tensile test, Int. J. Mech. Sci., 62 (2012) 47–56.

[5] S. Belhabib, H. Haddadi, M. Gaspérini, P. Vacher, Heterogeneous tensile test on elastoplastic metallic sheets: Comparison between FEM simulations and full-field strain measurements, Int. J. Mech. Sci., 50 (2008) 14–21.

[6] T. Pottier, P. Vacher, F. Toussaint, H. Louche, T. Coudert, Out-of-plane Testing Procedure for Inverse Identification Purpose: Application in Sheet Metal Plasticity, Exp. Mech., 52 (2012) 951–963.

[7] N. Souto, A. Andrade-Campos, S. Thuillier, A numerical methodology to design heterogeneous mechanical tests, Int. J. Mech. Sci., 107 (2016) 264–276.

[8] A. Andrade-Campos, J. Aquino, J.M. Martins, B. Coelho, On the design of innovative heterogeneous sheet metal tests using a shape optimization approach, Metals 9 (2019).

[9] J. Aquino, A. Andrade-Campos, J.M. Martins, S. Thuillier, Design of heterogeneous mechanical tests: Numerical methodology and experimental validation, Strain 55 (2019) 1–18.

[10] B. Barroqueiro, A. Andrade-Campos, J. Dias-de-Oliveira, R.A.F. Valente, Design of mechanical heterogeneous specimens using topology optimization, Int. J. Mech. Sci., 181 (2020) 105764.

[11] F. Pierron, G. Vert, R. Burguete, S. Avril, R. Rotinat, M.R. Wisnom, Identification of the orthotropic elastic stiffnesses of composites with the virtual fields method: Sensitivity study and experimental validation, Strain 43 (2007) 250–259.

[12] P. Wang, F. Pierron, O.T. Thomsen, Identification of Material Parameters of PVC Foams using Digital Image Correlation and the Virtual Fields Method, Exp. Mech. 53 (2013) 1001–1015.

[13] M.B. Bertin, F. Hild, S. Roux, Optimization of a Cruciform Specimen Geometry for the Identification of Constitutive Parameters Based Upon Full-Field Measurements, Strain 52 (2016) 307–323.

[14] S. Zhang, T. Xing, H. Zhu, X. Chen, Experimental identification of statistical correlation between mechanical properties of FRP composite. Materials 13 (2020) 1–12.

[15] L. Chamoin, C. Jailin, M. Diaz, L. Quesada, Coupling between topology optimization and digital image correlation for the design of specimen dedicated to selected material parameters identification, Int. J. Solids Struct. 193-194 (2020) 270–286.

[16] M. Rossi, F. Pierron, On the use of simulated experiments in designing tests for material characterization from full-field measurements, Int. J. Mech. Sci., 49(2012) 420–435.

[17] M. Rossi, M. Badaloni, P. Lava, D. Debruyne, F. Pierron, A procedure for specimen optimization applied to material testing in plasticity with the virtual fields method, AIP Conference Proceedings 1769 (October 2016).

[18] X. Gu, F. Pierron, Towards the design of a new standard for composite stiffness identification, Compos. - A: Appl. Sci. Manuf., 91 (2016) 448–460.

[19] P. Wang, F. Pierron, M. Rossi, P. Lava, O.T. Thomsen, Optimised experimental characterisation of polymeric foam material using DIC and the virtual fields method, Strain 52 (2016) 59–79.

[20] M. Gonçalves, A. Andrade-Campos, J.A. Dias-de-Oliveira, On the design of mechanical heterogeneous specimens using both parametric and topology optimization, In XVI International Conference on Computational Plasticity. Fundamentals and Applications., 7-9 September 2021, Barcelona, Spain.

[21] O. Sigmund, On the design of compliant mechanisms using topology optimization, Mech. Struct. Mach., 25 (1997), 493–524.

[22] B. Zhu, X. Zhang, H. Zhang, J. Liang, H. Zang, H. Li, R. Wang, Design of compliant mechanisms using continuum topology optimization: A review, Mech. Mach. Theory, 143 (2020) 103622.

[23] Y. Han, B. Xu, Y. Liu, An efficient 137-line MATLAB code for geometrically nonlinear topology optimization using bi-directional evolutionary structural optimization method, Struct. Multidiscip. Optim., 63 (2021) 2571–88.

[24] T. Buhl, C.B.W. Pedersen, and O. Sigmund, Stiffness design of geometrically nonlinear structures using topology optimization, Struct. Multidiscipl. Optim. 19 (2000) 93–104.

[25] T.E. Bruns and D.A. Tortorelli, Topology optimization of nonlinear elastic structures and compliant mechanisms, 7th AIAA/USAF/NASA/ISSMO Symp. Multidiscip. Anal. Optim., 190 (1998) 1874–1882.

[26] F. Wang, B.S. Lazarov, O. Sigmund, J.S. Jensen, Interpolation scheme for fictitious domain techniques and topology optimization of finite strain elastic problems, Comput. Methods Appl. Mech., 276 (2014) 453-472.

[27] G.I.N. Rozvany, M. Zhou, T. Birker, Generalized shape optimization without homogenization, Struct. Optim., 4 (1992) 250–252.

[28] Y.F. Fu, B. Rolfe, L.N.S Chiu, Y. Wang, X. Huang, K. Ghabraie, SEMDOT: Smooth-edged material distribution for optimizing topology algorithm, Adv. Eng. Softw., 150 (2020) 102921.

[29] K. Svanberg, The method of moving asymptotes—a new method for structural optimization, Int. J. Numer. Methods Eng., 24 (1987), 359–73.

[30] L. Liu, J. Xing, Q. Yang, and Y. Luo, Design of Large-Displacement Compliant Mechanisms by Topology Optimization Incorporating Modified Additive Hyperelasticity Technique, Math. Probl. Eng., 2017.

[31] C.B.W. Pedersen, T. Buhl and O.Sigmund, Topology synthesis of large-displacements compliant mechanisms, Int. J. Numer. Methods Eng. 50 (2001) 2683-2705.