MECSOL 2022 - Proceedings of the 8th International Symposium on Solid Mechanics
M.L. Bittencourt, J. Labaki, L.C.M. Vieira Jr. and E. Mesquita (Editors), Campinas SP, Brazil, October 17th to 19th, 2022

STRUCTURAL TOPOLOGY OPTIMIZATION IN 3D MULTI-COMPONENT ELEMENTS USING FENICS

Rafael Marin Ferro [1,2], **Renato Pavanello** [2]

[1] Coordination of Mechanical Engineering, Federal Institute of Science and Technology of ES – IFES

[2] Department of Computational Mechanics, Faculty of Mechanical Engineering – Unicamp

Abstract: This work analyzes the implementation of a discrete method of structural topology optimization in 3D or Three-Dimensional multi-component elements using the open-source FEniCS tools and other complementary tools such as Dolfin Adjoint, Gmsh, meshio, Ipopt and Paraview. Structural topological optimization is widely used by several researches and projects in the process of improving structural elements. Research and techniques that use structural topological optimization, mostly, deal with continuous structural elements with a single domain, or single component, and it is observed that the real structures are complex and multi-component. Currently, there are few approaches to the analysis of multi-component structures and they are still made using commercial software. The implementation of structural topological optimization in multi-component elements in this work involves four main components: multi-component meshing, numerical analysis using the Finite Element Method (FEM), sensitivity analysis by an adjoint method and an optimization solver. In order to allow the automated numerical solution of Partial Differential Equations (PDEs) and to perform a sensitivity analysis, the FEniCS and Dolfin Adjoint software are used, which are open-source code. For the optimization process, Ipopt is used, which is a large-scale nonlinear optimization software designed to find solutions of mathematical optimization problems. To generate the meshes, the Gmsh software is used, which is an open-source finite element mesh generator with an integrated CAD engine and post-processor. For the conversion/reading of the generated models, the Meshio package is used. Paraview software is used to visualize the results, which is an open-source cross-platform data visualization and analysis application. The topological optimization method used is based on SIMP interpolation – Solid Isotropic Material with Penalisation. The problem considered is compliance minimization / stiffness maximization, considering the examples of multi-component structures recurrent in the literature. A density filtering algorithm based on the Helmholtz formulation is used.

Keywords: 3D multi-component, structural topology optimization, Finite Element Method, FEniCS

INTRODUCTION

In recent decades, structural topology optimization has been studied for systems a single component (Bendsøe, 1995; Hassani and Hinton, 1999; Rozvany et al. (1995); Xie and Steven, 1997), however, in real projects, the great the most engineered structures are composed of more than one structural component, called multi-components. According to Fig. 01, various trusses, structures automotive, machine tool systems, aircraft components, among others, are real examples of situations with applications of multi-component structures.

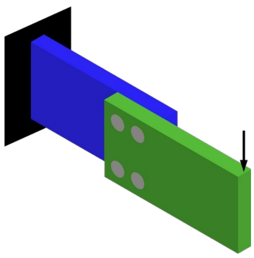

Figure 01 – multi-component structure with two distinct domains

The standard process of analyzing multi-component systems is done through the individual components for the purpose of simplifying the modeling and design process of the project, Li et al. (2001). In the individual analyzes of each component, the consideration of the connections between the elements, are made by being dimensioned as an appropriate load transfer system from one component to another, including restrictions miscellaneous kinematics that may not fit the actual conditions.

As this load transfer system is imposed on connection locations among the components that are dimensioned and pre-defined, each component needs to be dimensioned considering the impositions and restrictions of the set of components, evaluating the loading and boundary conditions considering all multi-components, and this may not be clear or sufficient for the effective development of multi-components. The layout of the multiple connected elements and their components must be incorporated with topology optimization to ensure desirable performance of multi-components systems.

Over the years, some works considering multi-component optimization have made extensive use of commercial software, which is very good, but has a high acquisition and maintenance cost, Li et al. (2001). So, in order to amplify the structural topological optimization approach in elements multi-components, a new approach is made considering the use of software with open-source language.

With the advancement of numerical methods and the power of technological dissemination available openly, some open-source software has been widely improved and disseminated. In this context, FEniCS, Alnaes et al. (2015); Logg et al. (2011), is a software that has good characteristics for the application of numerical methods or even the finite element method for the calculation of partial differential equations and being used in this approach. The Dolfin-Adjoint software, Mitusch et al. (2019) is a complementary package to FEniCS that calculates the sensitivities of models. Dolfin-Adjoint takes objective function derivatives and constraint derivatives of volume. For optimization, Ipopt (Interior Point OPTimizer) (Wächter and Biegler, 2006) which is a large-scale nonlinear optimization package designed to find (local) solutions. The Gmsh software is used to generate the meshes. (Remacle and Geuzaine, 2007) which is a finite element mesh generator in Three Open source (3D) dimensions with an integrated CAD engine and post processor. For the conversion/reading of the generated models, the Meshio Package is used, Schlömer et al. (2018) which transforms Gmsh files into .xdmf format which are optimized files in size, reading speed and used memory space. For viewing the results, the software Paraview, Moreland et al. (2016) is used, which is an open-source cross-platform data visualization and analysis.

Multi-component Structures

Historically, the advancement of design over structural projects has demanded many techniques and assumptions about the methodologies to be used in the sizing of their elements (Chickermane and Gea, 1997). However, what can be observed is that many projects, if not most, are formed by multi-component elements, and that even with all available technology, are still treated as a single component, and yet, for the development of the individual elements, a decomposition of the loads and displacements in order to perform an individual analysis of each component.

According to the purpose of the work, the real structures are composed by the "assembly" of several individual components or multi-component structures, where the need for division of treated structures into their individual components is an improvement for the manufacturing, assembly and transport processes. The problem of optimizing the topology of multi-component elements, is done by (Chickermane and Gea, 1997) in a direct, that is, as potentially force-transferring joints such as a grid of spring elements, acting between the nodes of two parts, and in this case, this direct approach, may not generate the idea transfers between the elements. According to Thomas et al. (2020) in their recent work for periodic multi-component elements, several works have studied this multi-component approach, but using single component optimization with element decomposition. Other works have same approach of direct decomposition of elements, as done by (Menassa and Devries, 1991) which deals with the insertion of fixed positions between elements, or also on the fixed location of spot welds, seen in (Chirehdast and Jiang, 1996). Still considering fixed elements, the work done by (Jiang and Chirehdast, 1997) considers adhesive bonding elements generating a high degree of dynamic assumption. In Li et al. (2001) also configure discrete joint fixed elements, where even considering 3D elements, load transfers are still performed by applications of assumptions. In the work of Zhu et al. (2017) it is proposed to incorporate non-design within a macro-design domain, allowing the structural integration of small components within a larger assembly, which exhibits significant advantages in design of complex structures, a technique similar to the one used in the present work. Zhang et al. (2011) adopted the finite circle method to impose restrictions of no overlap between components to approximate a complex component through a set of circles of varying size and placement. (Liu and Kang, 2018) explored the allocation field to solve multi-component problems modeling material interfaces between components, providing more realistic modeling of mounting conditions. It is observed that the allocation of interface material, as screws, welds or rivets, can represent a special case of overlap problems, where the limitation of the approach is to deal with some specific characteristics of the method used (Qian and Ananthasuresh, 2004). More recent work on optimizing several components subdivided the structure into a set of multi-bar components Wang et al. (2018), which are mobile and morphable, allowing conform approximately to a similar project achieved through a single classic optimization component. Thus, it is observed that these works are largely based on topology optimization techniques, but developed for ingle component.

Other methods for optimization are those that consider the simultaneous combination of topology and layout and that directly use component location variables as part of the design variables, thus allowing a flexible positioning of the connecting elements. This approach to component layout optimization incorporated into a framework for the optimization of the final topology was made by Zhu et al. (2017). The method was further refined by (Ambrozkiewicz and Kriegesmann, 2021) with the inclusion of predefined fixed shape connector elements to model the transfer of force between the components and the supporting structure. These elements fasteners are connected to the structure using coupling equations, bypassing the need to re-mesh when components move. Still recent, Rakotondrainibe et al. (2020) modeled together as supports rigid using movable boundary condition zones, whose positions were part of the optimization in a level set topology optimization framework.

A fact seen in most of the works and their approaches is that they follow good standards of analysis in multi-component elements, and in practically all that explain the methodologies, make current use of commercial numerical support that, due to the purpose of their commercialization have a certain ease in handling and applicability. So, an approach with unconventional programs, or even open-source programs such as FEniCS, arise for an even didactic approach on the optimization of the multi-component structures.

The present work demonstrates a method of structural topology optimization using the SIMP – Solid Isotropic Material with Penalisation (Sigmund, 2001) considering the elements multi-components with individual but simultaneous structural optimization, thus optimizing the material distribution of each component. The approach considers a distribution material optimization by analyzing each component considering the loading to the set as a whole without any simplification.

For the models considered in this work, the position of the connection elements between the elements that make up the structure is pre-established. According to Li et al. (2001) in their analyses, in the topological optimization of the region between the elements, the final results for the optimal position of the connection elements are very close to the common positions of structural connections projects commonly seen in real projects, which shows the validation of its approach, and with that, the optimization projects of the connection regions have a strong influence on the final result, but it has a periodicity independent of the method, thus, the present work will use these results to pre-fix the connection ideas layouts.

Finite element method using FEniCS

The implementation of the finite element method was done using the platform FEniCS. The problem must be formulated using functional analysis (Langtangen and Mardal, 2019). FEniCS is a tool capable of solving PDEs by the method of finite elements, and was designed to make implementations more compact, which is attractive because the abstract formulation of the method is used (Langtangen and Mardal, 2019). Over the last few decades, much work has been done on the finite element method and generally speaking, it is considered that among most of these works there is a format division of the method approach into two categories, the first is version abstract mathematics of the method and the second is the formulation of “structural analysis” of engineering (Langtangen and Logg, 2016). Thus, the FEniCS software is based on applying the concepts of the first approach, the abstract mathematics of the finite elements. Other works that follow this same approach in the development of the finite element method are made by (Gockenbach, 2007; Larson and Bengzon, 2013). The formulation can be written using the weighted residual method of the type GALERKIN ("CG" or Continuous Galerkin), where the solution of an PDEs can be obtained considering a nodal polynomial approximation by subdomains. For this, they are defined the test functions that are used in FEniCS programs. The test functions belong to certain function spaces that specify the properties of approximations numerical values adopted (Langtangen and Logg, 2016). In order to reduce the number of degrees of freedom of the models, only Lagrange CG_1 tetrahedron elements are used, both for the analysis of displacements and for the filtering system. See Fig. 02 for an example of the type of CG_1 Lagrange element.

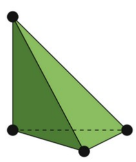

Figure 02 – The linear Lagrange tetrahedron

MECSOL 2022 - Proceedings of the 8th International Symposium on Solid Mechanics
M.L. Bittencourt, J. Labaki, L.C.M. Vieira Jr. and E. Mesquita (Editors), Campinas SP, Brazil, October 17th to 19th, 2022

Mesh Generation using Gmsh Software

Gmsh Software is a three-dimensional finite element mesh generator with an integrated CAD engine and post processor. Its design objective is to provide a fast, lightweight and easy-to-use mesh tool with parametric input and advanced visualization. Gmsh is built around four modules: geometry, mesh, solver and post-processing. All geometric, mesh, solver, and post-processing are interactively prescribed using the graphical user interface (GUI) or in text files using Gmsh's own scripting language. The interactive actions generate language bits in the input files and vice versa. A programming API too is available, to integrate Gmsh into your own C++, C, Python or Julia code. In this work will be used only the generation of three-dimensional mesh and later converted to read in FEniCS.

A model in Gmsh is defined using its Boundary Representation (BRep): a volume is bounded by a set of surfaces, a surface is bounded by a series of curves, and a curve is bounded by two endpoints. Model entities are topological entities, that is, they only deal with adjacencies in the model and are implemented as a set of abstract topological classes. This BRep is extended by defining built-in, or internal, model entities: internal points, edges, and surfaces can be embedded in volumes; and internal points and curves can be embedded in surfaces. The great advantage of using Gmsh is that due to its structuring of geometries, it is possible to define each volume of each domain including the connection elements and convert to the reading of FEniCS exactly as follows the mathematical process of the topological optimization method.

Considering the example of Fig. 01, follows the process of creating the finite elements in Gmsh: Creation of geometry, application of boolean elements to generate independent domains and generation of faces and volumes, automatic mesh generation and transformation of the Gmsh model, .msh file, to .xdmf file.

As shown in Fig. 03, we have: Items 1 and 2 are the loading and fixing surfaces respectively. Items 3 and 4 are the domain volumes. Items 5, 6, 7 and 8 are the volumes of the connection elements. All geometric volume elements are independent, where the nodes that coincide with each other, have the same load transfer conditions and displacement.

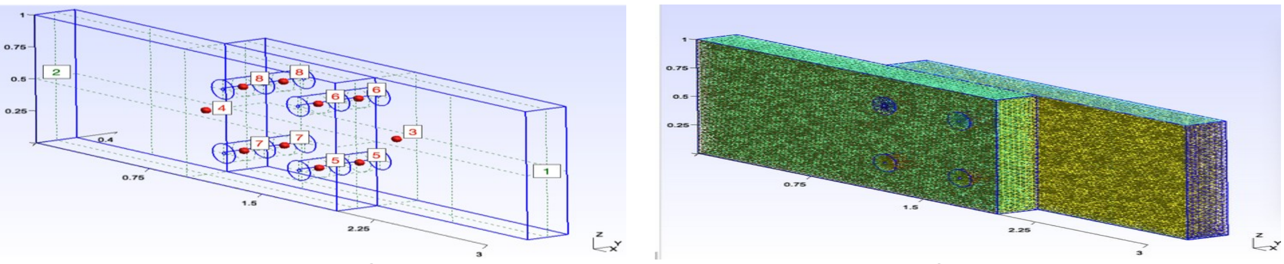

Figure 03 – Generation of Faces and Volumes and MEF Mesh Generation

FEniCS reads the models through .xdmf files. This process of file transformation is done using the Meshio toolkit which can read and write to various data formats that represent unstructured meshes, such as DOLFIN, Gmsh, H5M or VTK. This generates two files for the analysis of FEniCS. One with the meshes of faces where the boundary conditions for fixing the model in question will be inserted, or any other desired, and also where the loads will be applied. And the other is the volumetric meshes of the domains considered in the study. Each domain and each connection element is represented by a specific color showing that they are independent geometric elements. The final images of these models used in this work and later the final results of the optimization are visualized in the Paraview Software.

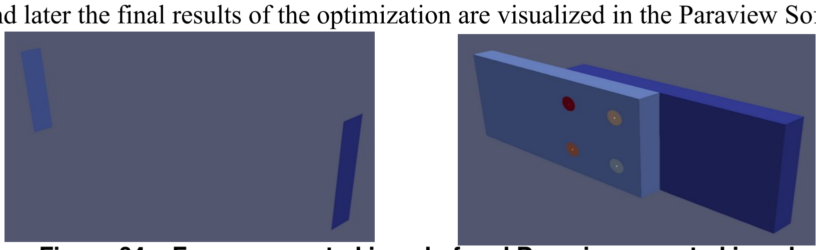

Figure 04 – Faces generated in .xdmf and Domain generated in .xdmf

Structural Topology Optimization in multi-component Elements

Thus, in order to perform the structural topological optimization in elements multi-component, the problem of minimizing compliance (mean sensitivity) structural through an objective function over the domains considered, according to the Eq. (1):

(1)

$$\mathbf{L}(\mathbf{v}) = \sum_{i=1}^{D} \left(\int_{\Omega_i} \mathbf{f_i} \cdot v \mathrm{d}\Omega_i \ + \int_{\partial\Gamma_{iN}} \mathbf{T_i} \cdot v \ \mathrm{d}\Gamma_i \right) \tag{1}$$

Where $\mathbf{L}(\mathbf{v})$ is the work of external forces of volume $\mathbf{f_i}$ and surface $\mathbf{T_i}$ applied to each domain, D represents the number of domains analyzed, i represents the subindex of each domain, Ω_i is the individual domains, v represents the test functions, and $\partial\Gamma_{iN}$ is the portion of the contour where the surface forces of each domain are applied.

Equilibrium conditions are considered as constraints of the problem and are given per:

$$-\nabla.\sigma \ = \ \mathbf{f} \ in \ \Omega \tag{2}$$

$$\sigma.n = \mathbf{T} \ on \ \Gamma_N \tag{3}$$

Where σ is the Cauchy stress tensor, n represents the external normal direction in Γ_N and $\nabla.\sigma$ is the stress divergence, as follows:

$$\frac{\partial\sigma_{xx}}{\partial x} + \frac{\partial\sigma_{xy}}{\partial y} + \frac{\partial\sigma_{xz}}{\partial z} = -f_x \tag{4}$$

$$\frac{\partial\sigma_{xy}}{\partial x} + \frac{\partial\sigma_{yy}}{\partial y} + \frac{\partial\sigma_{yz}}{\partial z} = -f_y \tag{5}$$

$$\frac{\partial\sigma_{xz}}{\partial x} + \frac{\partial\sigma_{yz}}{\partial y} + \frac{\partial\sigma_{zz}}{\partial z} = -f_z \tag{6}$$

$$\sigma_{xx}n_x + \sigma_{xy}n_y + \sigma_{xz}n_z = T_x \tag{7}$$

$$\sigma_{xy}n_x + \sigma_{yy}n_y + \sigma_{yz}n_z = T_y \tag{8}$$

$$\sigma_{xz}n_x + \sigma_{yz}n_y + \sigma_{zz}n_z = T_z \tag{9}$$

And the following additional restrictions:

$$\int_{\Omega} \rho(x)\mathrm{d}x \leqslant V \tag{10}$$

$$0 \leq \rho(x) \leqslant 1, \ \forall x \in \Omega$$

Since the parameter $\rho(x)$ is the design variable ($\rho(x) = 1$ means presence of material and $\rho(x) = 0$ means absence of material), V, in Eq. (10), is the desired final volume. The strain displacement relations considering the small elastic deformations of a multi-component Ω_i can be written as:

$$\epsilon_i\left(v\right) \ = \ \frac{1}{2}\left(\nabla v + (\nabla v)^T\right) \tag{11}$$

The constitutive equations can be written as:

$$\sigma_i(u_i) = \lambda tr(\epsilon_i)I + 2\mu\epsilon_i \tag{12}$$

And the linear integral form of the equilibrium conditions can be written as:

$$a(u,v) = \sum_{i=1}^{D} \left(\int_{\Omega_i} \sigma_i\left(u_i\right) : \epsilon_i(v)\mathrm{dx}_i \right) \tag{13}$$

Where u_i is the displacement field of each domain, $\epsilon_i(v)$ is the linear part of the Green strain stresses of each domain, tr is the trace and I is the identity tensor. The Lamé constants are λ and μ according to:

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}, \quad \mu = \frac{E}{2(1+\nu)} \tag{14}$$

Being in Eq. (14), E the Young's modulus and v a Poisson coefficient.

With this, the variational formulation is summarized on how to find u, total displacements considering the influence of all domains, so that $u \in V$ where:

$$a(u,v) = L(v) \quad \forall v \in \hat{V} \tag{15}$$

Being $\hat{V}$ the Hilbert subspace of v admissible functions.

Isotropic elastic conditions are assumed in the examples of this work. The update material distribution $\rho(x)$ is done by updating the material stiffness through compliance minimization (Bendsøe and Sigmund, 2003) which can be written as follows:

min. $L(v)$

s.a: $a(u,v) = L(v) \quad \forall v \in \hat{V}$ (16)

$\sum_{i=1}^{D}\left(\int_{\Omega_i}\rho(x)\mathrm{d}x_i \leqslant V\right)$ with $0 \leq \rho(x) \leqslant 1,\ \forall x \in \Omega_i$

Thus, a continuous SIMP relaxation is performed, proposed by, proposed by (Bendsøe and Sigmund, 2003) according to:

$$E = E_{min} + (E_{max} - E_{min})\rho(x)^p \tag{17}$$

Density filter

Due to the choice of a power law for material interpolation, as Eq. (17), problems of alternating solutions with $p = 0$ and $p = 1$ may arise with numerical problem of the “checker board” type (Díaz and Sigmund, 1995; Jog and Haber, 1996; Sigmund and Petersson, 1998). So, to alleviate this problem, a density filter, which can be implicitly represented by the solution of a partial differential equation of the Helmholtz type with homogeneous Neumann boundary conditions according to (Lazarov and Sigmund, 2011) and described in Eq. (18).

$$-R_{min}^2\nabla^2\tilde{\psi} + \tilde{\psi} = \psi \tag{18}$$

Where ψ is the continuous representation of the unfiltered design variable and $\tilde{\psi}$ is the filtered design variable. The R_{min} parameter plays a similar role to the r_{min} used in classical filtering approaches that use the SIMP method. In (Lazarov and Sigmund, 2011) an approximate relationship between the length scales for the classical filter and the Helmholtz approach is described as $R_{min} = r_{min}/2\sqrt{3}$.

The solution of Eq. (18) can be written in the form of an integral convolution which is equivalent to the classic filter. In terms of variational formulation, it is as follows:

$$-R_{min}^2\int_\Omega \nabla\widetilde{\rho(x)}.\nabla w d\Omega\ + \int_\Omega \widetilde{\rho(x)} w d\Omega\ =\ \int_\Omega \rho(x) w d\Omega \quad \forall w \in \hat{W} \tag{19}$$

Being $\hat{W}$ Hilbert subspace of w admissible functions.

Numerical implementation

The equilibrium equations, Eq. (15), are solved using the method of finite elements using the FEniCS framework, which converts the models described by variational forms in efficient finite element code. The optimization problem described in Eq. (16) is solved using the optimization routines from the Ipopt library, which is a large-scale nonlinear optimization software package that implements an "interior-point" and a "line-search". The analysis and optimization methods used are suitable for large problems with up to millions of variables and constraints. At sensitivities are calculated by the adjoint method using the Dolfin Adjoint package that derives automatically the adjoint and tangent discrete linear models of a direct model written in the Python interface for FEniCS. In Fig. 05 a general scheme of the implementation numeric is shown.

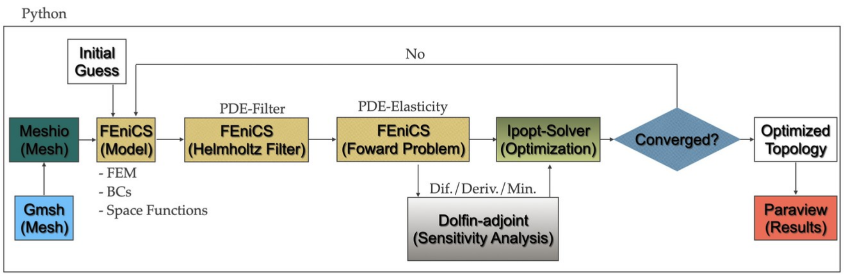

Figure 05 – Numerical implementation routine

Models analyzed

To demonstrate the application of the approach described, some recurrent models in the literature will be generated and analyzed. The first model, a Cantilever Beam with two domains, is the most commonly seen in structural topology optimization work on multi-component elements, and demonstrates the overlapping domains and their connecting elements. Models with overlapping multi-component, as shown in Fig. 01, considering a 3D analysis, have an implication with a lot of distortion in the final result, due to the eccentricity of the loading in relation to the components, which is not so well observed in 2D analysis and practically not being considered in the referenced works due to the majority being in 2D. To circumvent this problem and make the models more real, in other studies carried out in this work, an additional connecting element will be considered, such as a connecting plate between the models, and with that there is a concentric loading between the multi-components in the models. According to the numerical approach, it is possible to decide which domain(s) will be optimized, just describing the need in the volume constraint functions. Thus, examples optimizing only the domains will be considered and examples optimizing the connecting plate in addition to the domains. The considered connection elements are modeled as pins or screws and to make the simulation even more realistic, these connection elements are inserted hollow, this to reduce the rigidity of the connection making the connection more flexible and improving the applicability in the conditions of analysis in each individual element.

Cantilever with two connections

The first model made is the classic Cantilever with two connecting elements. A multicomponent was assembled with two overlapping beams and two fixed connection elements. The left domain is fixed to the left face and a load is applied to the central part of the right face of the right domain. Volume reduction to 30%.

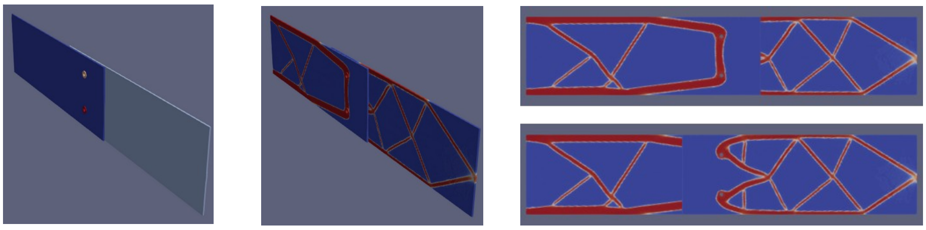

Figure 06 – Cantilever (Two Connections): Base and Optimized Geometry and Side Views

Cantilever with four connections

Now, still for the first model, the classic Cantilever, but with four connection elements. A multi-component with two overlapping beams and four fixed connection elements. The left domain has its fixation on the left face and a load is applied to the central part of the right face of the right domain. Volume Reduction to 30%.

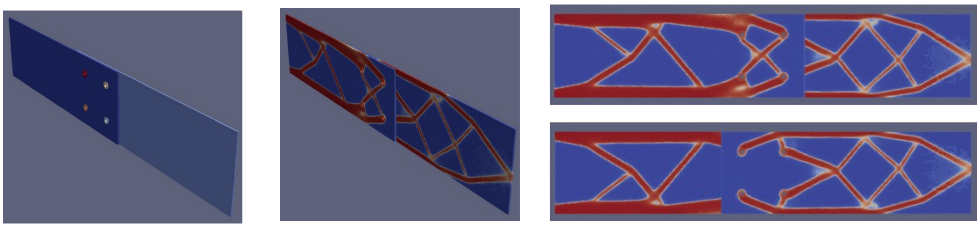

Figure 07 – Cantilever (Four Connections): Base and Optimized Geometry and Side Views

Cantilever with plate

Now the Cantilever model is made considering two non-overlapping aligned beams and a plate or an additional joining element between the beams. Each bam is connected by two connecting elements through the additional plate. The left domain is fixed to the left face and a load is applied to the central part of the right face of the right domain. Volume reduction to 30%.

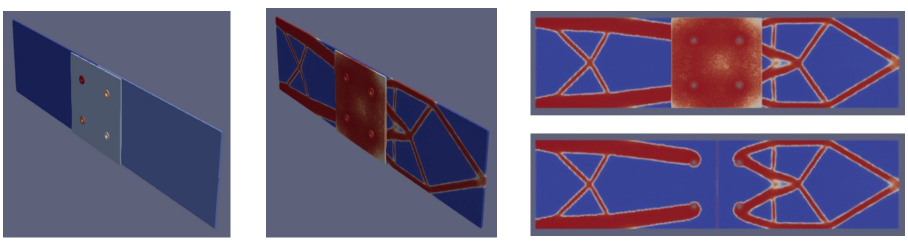

Figure 08 – Cantilever (With connecting plate): Base and Optimized Geometry and Side Views

Now follow the model results considering the connection plate optimization.

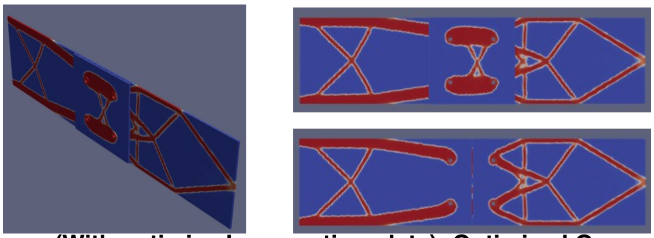

Figure 09 – Cantilever (With optimized connection plate): Optimized Geometry and Side Views

This result shows that the application of the connecting plate, removing the eccentricity loading, generates a more uniform result without distortion, very similar to results observed when considering individual optimization.

Bridge

The second model made is a representation of a bridge. It is an extension of the models above but considers the two external side faces fixed and the load applied on the board. Volume reduction to 50%.

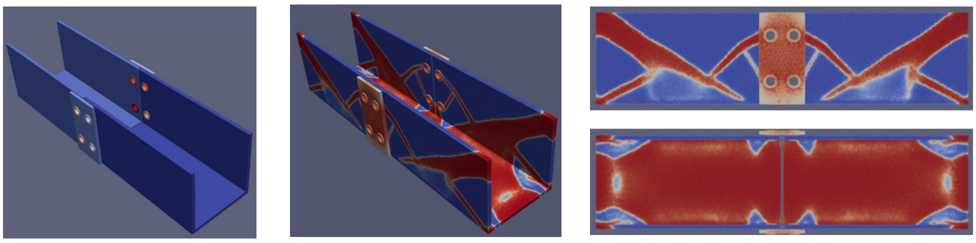

Figure 10 – Bridge: Base and Optimized Geometry and Side and Top Views

Now follow the model results considering the connection plate optimization.

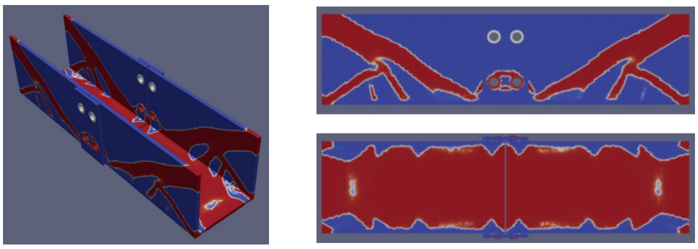

Figure 11 – Bridge (With optimized connection plate): Base and Optimized Geometry and Side and Top Views

This bridge result shows that the numerical approach actually makes the analysis considering all the domains, performing the optimization in the connection elements and in the domains, considering the whole set and showing that in the final process the connections superiors are not required after the final optimization process.

Conclusion

This work demonstrated the implementation of a discrete method of structural topology optimization in 3D or three-dimensional multicomponent elements using the open-source FEniCS, Dolfin Adjoint and other complementary tools such as Gmsh, meshio, Ipopt and Paraview. Thus, demonstrating the feasibility of implementation in a fully open-source environment. Classic examples of structural topology optimization were considered as multi-component elements and their results are shown. At first, the integration between the different platforms used may seem expensive, but the progress of learning in this approach has a smoothed curve along the works and models made. Understanding the mesh generation process by Gmsh and how to make it available, within the 3D multi-component elements approach, both for the finite element analysis performed by FEniCS and for the Dolfin-Adjoint sensitivity calculation, is a complex process and shown in this work in an integrated manner across all programs. An advantage in the numerical method developed in this work is the possibility of optimizing only the desired domain, with no need for any simplification, only describing in the volume constraint functions which domain(s) will be optimized. Thus, through the process of penalization by isotropic solid material, SIMP, it was possible to use the adjoint method to calculate the sensitivities considering the derivation of the functionals and later the optimization of the results through the maximization of the displacements of the multi-component, or minimization of the compliance, with using the Ipopt package. The results of the visualization of the models and the generation of images are made by the software paraview, and thus, closing an entire process developed with open tools.

When comparing the results made here with already renowned works such as those seen in Thomas et al. (2020) that use commercial packages or even more recent works such as (Ambrozkiewicz and Kriegesmann, 2021), we see that the results are very close, showing the viability of the codes generated here. And yet a particularity is that there is no assumption about loading on multi-components and also all models are applied directly to three-dimensional geometries. Finally, this work, albeit modestly, is the first version that performs structural topological optimization analysis on multicomponent elements using FEniCS.

The optimization of classic examples such as a multi-component cantilever beam with a connecting plate and four connecting elements between them, with 145.025 tetrahedral elements and 48.281 nodes, it is done in a few minutes and shows the capacity of high performance of open-source packages. Thus, the present work shows that it is possible to use open tools in multi-component structural topology optimization projects 3D.

ACKNOWLEDGMENTS

I thank IFES – Federal Institute of Science and Technology of ES for the encouragement and availability in provide the opportunity for Dinter – Inter-institutional Doctorate with Unicamp – Campinas State University. Thanks to CAPES – Improvement Coordination of Higher Education Personnel for the scholarships made available during part of the research period.

REFERENCES

Alnaes, M.S. et al, 2015, “The FEniCS Project Version 1.5”, Archive of Numerical Software 3.

Ambrozkiewicz, O.; Kriegesmann, B., 2021, “Simultaneous topology and fastener layout optimization of assemblies considering joint failure”, International Journal for Numerical Methods in Engineering, 2021.

Andreassen, E. et al., 2011, “Efficient topology optimization in MATLAB using 88 lines of code”. Structural and Multidisciplinary Optimization, v. 43, n. 1.

Bendsøe, M.P., 1995, “Optimization of Structural Topology, Shape, and Material”, Springer Berlin Heidelberg.

Bendsøe, M.P. and Sigmund, O., 2003, “Topology Optimization - Theory, Methods, and Applications”, Springer Verlag.

Chickermane, H. and Gea, H.C., 1997, “Design of multi-component structural systems for optimal layout topology and joint locations”, Engineering with Computers, v. 13, n. 4, p. 235–243.

Chirehdast, M. and Jiang, T., 1996, “Optimal design of spot-weld and adhesive bond pattern”, SAE Technical Papers.

Díaz, A. and Sigmund, O., 1995, “Checkerboard patterns in layout optimization. Structural Optimization”, v. 10, n. 1.

Farrell, P. E. et al., 2013, “Automated derivation of the adjoint of high-level transient finite element programs: , SIAM Journal on Scientific Computing, v. 35, n. 4.

Gockenbach, M., 2007, “Understanding and Implementing the Finite Element Method”, Scientific Programming, v. 15, p. 864256.

Hassani, B. and Hinton, E., 1999, “Homogenization and Structural Topology Optimization”, Springer.

Jiang, T. and Chirehdast, M., 1997, “A systems approach to structural topology optimization: Designing optimal connections”, Journal of Mechanical Design, Transactions of the ASME, v. 119, n. 1.

Jog, C.S. and Haber, R.B., 1996, “Stability of finite element models for distributed-parameter optimization and topology design”, Computer Methods in Applied Mechanics and Engineering.

Langtangen, H.P. and Logg, A., 2016, “Solving PDEs in Python”, Springer.

Langtangen, H.P. and Mardal, K.A., 2019, “Introduction to Numerical Methods for Variational Problems”.

Larson, M. and Bengzon, F., 2013, “The Finite Element Method: Theory, Implementation, and Applications”, Springer.

Lazarov, B. S. and Sigmund, O., 2011, “Filters in topology optimization based on Helmholtz-type differential equations”, International Journal for Numerical Methods in Engineering, v. 86, n. 6.

Li, Q., Steven, G.P., Xie, Y.M., 2001, “Evolutionary structural optimization for connection topology design of multi-component systems”, Engineering Computations (Swansea, Wales), v. 18, n. 3–4, p. 460–479.

Liu, P. and Kang, Z., 2018, “Integrated topology optimization of multi-component structures considering connecting interface behavior”, Computer Methods in Applied Mechanics and Engineering, v. 341.

Logg, A., Wells, G., Mardal, K.A., 2011, "Automated solution of differential equations by the finite element method", The FEniCS book. Springer. v. 84.

Menassa, R.J. and Devries, W.R., 1991, "Optimization Methods Applied to Selecting Support Positions in Fixture Design", Journal of Engineering for Industry, v. 113, n. 4, p. 412–418.

Mitusch, S., Funke, S., Dokken, J., 2019, "dolfin-adjoint 2018.1: automated adjoints for FEniCS and Firedrake", Journal of Open-Source Software, v. 4, n. 38.

Moreland, K. et al., 2016, "The ParaView Guide", Sandia National Laboratories.

Qian, Z. and Ananthasuresh, G.K., 2004, "Optimal embedding of rigid objects in the topology design of structures", Mechanics Based Design of Structures and Machines, v. 32, n. 2.

Rakotondrainibe, L., Allaire, G., Orval, P., 2020, "Topology optimization of connections in mechanical systems", Structural and Multidisciplinary Optimization, v. 61, n. 6.

Remacle, J. adn Geuzaine, C., 2007, "Gmsh : a three-dimensional finite element mesh generator with built-in pre- and post- processing facilities What is Gmsh ?", International Journal For Numerical Methods In Engineering, v. 0.

Rozvany, G.I.N., Bendsø, M.P., Kirsch, U., 1995, "Layout optimization of structures", Applied Mechanics Reviews, v. 48, n. 2.

Schlömer, N., Nilswagner; Li, T., Coutinho, C., Dalcin, L., Mcbain, G., Cervone, A., Langlois, T., Peak, S., Bussonnier, M., Vaillant, G. A., Croucher, A., 2018, "nschloe/meshio v1.11.7 - I/O for various mesh formats", https://doi.org/10.5281/zenodo.1173116.

Sigmund, O., 2001, "A 99 line topology optimization code written in matlab", Structural and Multidisciplinary Optimization, v. 21, n. 2.

Sigmund, O. and Petersson, J., 1998, "Numerical instabilities in topology optimization: A survey on procedures dealing with checkerboards, mesh-dependencies and local minima", Structural Optimization, v. 16, n. 1.

Thomas, S., Li, Q., Steven, G., 2020, "Topology optimization for periodic multi-component structures with stiffness and frequency criteria", Structural and Multidisciplinary Optimization, v. 61, n. 6, p. 2271–2289.

Wang, X. et al., 2018, "An explicit optimization model for integrated layout design of planar multi-component systems using moving morphable bars", Computer Methods in Applied Mechanics and Engineering, v. 342.

Wächter, A. and Biegler, L.T., 2006, "On the implementation of an interior-point filter line-search algorithm for large-scale nonlinear programming. Mathematical Programming, v. 106, n. 1.

Xie, Y.M. and Steven, G.P., 1997, "Basic Evolutionary Structural Optimization. In: Evolutionary Structural Optimization.", Springer.

Zhang, W. et al., 2011, "Some recent advances in the integrated layout design of multi-component systems", Journal of Mechanical Design, Transactions of the ASME, v. 133, n. 10.

Zhu, J. H. et al., 2017, "Integrated layout and topology optimization design of multi-frame and multi-component fuselage structure systems", Structural and Multidisciplinary Optimization, v. 56, n. 1.

RESPONSIBILITY NOTICE