# Topological Optimization with SIMP

Nathaniel Jones

December 17, 2022

## 1 Introduction

Modern engineering requires designing structures that can meet strict criteria and withstand harsh environments. As requirements grow more and more complex, it is increasingly labor-intensive and difficult to design such structures manually. Topology optimization is a field dedicated to finding numerical models and procedures to create such structures algorithmically. The field is applicable to many different areas of engineering and manufacturing, from the design of household applicances to aerospace engineering; indeed, the latter has taken a keen interest in topology optimization due to the necessity of reducing weight while withstanding high G-forces. While topology optimization as a field has existed for several decades, it is also increasingly relevant due to the rise of additive manufacturing, also known as 3D printing. Through additive manufacturing techniques, finely-detailed objects cost no more to produce than more coarsely-detailed ones, making it feasible to manufacture optimal designs that exhibit small-scale structure. Furthermore, it is easier than ever to actually create optimal designs thanks to the inclusion of topology optimization models in CAD software. This paper focuses on one such model, the Solid Isotropic Material with Penalization (SIMP) model.

SIMP was first proposed by Martin Bendsøe and Noboru Kikuchi in 1988. Prior to their work, finite-element methods of solving topology optimization problems framed these problems as integer programming problems. A reference domain on which forces act would be divided up into elements, and each element would either have material or no material. While this problem is possible to solve, integer optimization is notorious for its difficulty. To combat this, Bendoe and Kikuchi proposed assigning each element a density in $[0, 1]$, with intermediate values being punished. This transforms the optimization problem into a continuous one, which is more computationally tractable. While in a real-world scenario SIMP would be applied on a 3D domain, for simplicity's sake and ease of illustration we will consider the model in two dimensions.

### 1.1 Review of Linear Elasticity

In order to understand the SIMP model, it is necessary to understand the basics of linear elasticity. Essentially, objects in linear elasticity are not treated as rigid point masses, but instead are treated as deformable volumes. Displacement is not a single value as it is in elementary physics; rather, displacement is a vector field describing how every point in the

object is individually displaced from its initial state. The displacement (or more accurately, its spatial gradient) follows a generalization of the familiar Hooke's Law. Each point of an object can be displaced when acted upon by force, and the point increasingly resists further displacement. Written symbolically,

$$\sigma = K\epsilon$$

Here, $\sigma$ refers to the *stress matrix*. Stress is measured in pascals (Pa), and is essentially a description of the forces acting on an object to stretch and compress it at an infinitesimal scale. Unlike force, however, stress acts in multiple directions at once; these directions are in fact the eigenvectors of the stress matrix. $K$ refers to the *stiffness tensor*, which is an innate property of the material in question. It is analogous to the spring constant in the ordinary Hooke's Law. $\epsilon$ is the *strain matrix*. If stress refers to the forces that stretch an object, strain describes the actual stretch. Similarly, its eigenvectors describe the priniciple directions of deformation at a point. Crucially, the elements of strain are computed as

$$\epsilon_{ij} = \frac{1}{2}(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})$$

That is, strain is the *change* in displacement $u$ with respect to space. This is somewhat intuitive; an object undergoes deformation when some parts of it have been displaced more than others.

One of the most common problems in topology optimization is the minimum compliance problem, which can also be thought of as the maximum rigidity problem. The goal is to distribute material throughout a reference domain $\Omega$ such that the work done on on the structure is minimized. Written symbolically:

$$\text{Minimize} \int_\Omega f \cdot u \mathrm{d}x + \int_{\partial\Omega} t \cdot u \mathrm{d}s$$

In the first term, we are integrating throughout the volume of the reference domain $\Omega$. $f$ refers to the vector field of *body forces* acting on the object at any point, which are measured in newtons (N). Body forces act within the internal volume of the object. For example, gravity is a body force because it pulls on all points of an object simultaneously (but not necessarily at equal strength or direction). Recall that work is force applied over a distance and is measured in Joules (J). Thus, in the first integral we are summing the work done by the body forces along each coordinate axis at every point to get the total work done throughout on the structure by these forces.

In the second term, we are integrating along the surface $\partial\Omega$ of the reference domain. $t$ refers to the *traction* along this boundary, which is measured in pascals (Pa). If body forces are forces internal to the object, then traction can be thought of as forces external to the object. For example, a book on a bookshelf is applying traction to the bookshelf, since the weight of the book is pushing on the bookshelf's surface. Similar to the first integral, the second integral is summing the work done by traction (i.e. external forces) along each coordinate at each point of the surface of the domain to get the total work done by these forces. The total work applied to the object, then, is the sum of the total work done by body forces and the total work done by boundary tractions. To achieve minimum compliance, we want to control the displacement field so as to minimize this sum.

However, we cannot arbitrarily create the displacement field, but rather it must be derived from the design variables (namely, the distribution of material) and is subject to a constraint. Specifically, the displacement field must result in the object being in elastic equilibrium. This is because we want to analyze the final displacement of the object when computing the work done, which will necessarily be at equilibrium with the body forces and boundary tractions. To describe the equilibrium condition symbolically, we first start with the linearly elastic version of the classic $F = ma$ equation:

$$\int_\Omega f \mathrm{d}x + \int_{\partial\Omega} t \mathrm{d}s = \int_{Omega} \rho \frac{\partial^2 u}{\partial t^2}$$

In equilibrium, the object is not accelerating, so the right hand side becomes zero.

$$\int_\Omega f \mathrm{d}x + \int_{\partial\Omega} t \mathrm{d}s = 0$$

Noting that the boundary tractions are equal to $\sigma \cdot \overrightarrow{n}$, where $\overrightarrow{n}$ is the surface normal of the domain, we write

$$\int_\Omega f \mathrm{d}x + \int_{\partial\Omega} \sigma \overrightarrow{n} \mathrm{d}s = 0$$

And by the divergence theorem, we have

$$\int_\Omega f \mathrm{d}x + \int_\Omega \nabla \cdot \sigma \mathrm{d}x = 0$$

Rearranging, we get

$$-\nabla \cdot \sigma = f$$

This is known as the strong form of the linearly elastic equilibrium equation. This formulation requires that displacement be twice-differentiable with respect to space. While $u$ would ideally meet this constraint, in practice it is difficult to compute such a displacement field via finite element methods such as those used with SIMP. Therefore, through the equations relating stress, strain, and with a bit of real analysis (omitted for brevity), this constraint is instead written in the weak form:

$$\forall \mathbf{v} \in U, \int_\Omega \epsilon(u)^T K \epsilon(v) \mathrm{d}x = \int_\Omega \mathbf{f} \cdot v \mathrm{d}x + \int_{\partial\Omega} \mathbf{t} \cdot v \mathrm{d}s$$

Note that in the weak form, $u$ is only required to be once-differentiable, which is a great simplification.

# 2 The SIMP Model

## 2.1 Assumptions

SIMP imposes a few assumptions on the problem. First, as the name suggests, the structure is assumed to be made of a single material and the material must be isotropic. In other words, it must have the same resistance to stretching in all directions. This is a reasonable

assumption; many materials used in manufacturing are isotropic, such as metals and most plastics. Second, it is assumed that the only stresses on the object are mechanical, so for example there is no concern of deformation due to high temperature. This is also reasonable for many materials and environments. The loads are also assumed to not depend on time. Finally, in this project, I assumed that the effect of body forces on displacement will be negligible compared to the loads (i.e. the object will not deform under its own weight) for simplicity's sake.

## 2.2 Methodology

As a finite element method, SIMP first discretizes the reference domain $\Omega$ into a mesh of elements. While there are many possible choices of elements to be made, in this report we will simply use a grid of squares. Each element is assigned a density $x \in [0, 1]$. Then, we write

$$K = x^p \bar{K}$$

,

where $\mathbf{K}$ is the stiffness matrix for the individual element, $p \geq 3$ (to ensure the optimization problem is well-posed), and $\bar{\mathbf{K}}$ is the constant stiffness matrix for the material itself. When the volume constraint $\int_\Omega x \mathrm{d}x = V$ is added, this formulation has the effect of punishing intermediate values of $x$ by increasing the volume without adding much stiffness. The overall problem formulation is as follows:

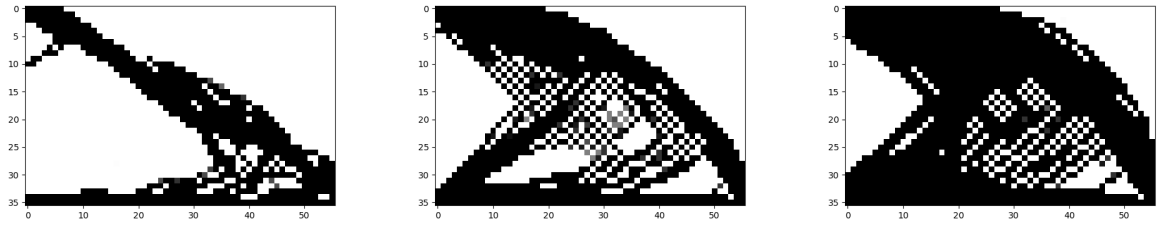
$$\text{Minimize;} \mathbf{U}^T \mathbf{K} \mathbf{U}$$

$$s.t.$$

$$\sum_{i=1}^{N} = VN$$

$$\mathbf{KU} = \mathbf{F}$$

Here, $\mathbf{U}$ is a large column vector that encodes the displacement of each vertex of all the elements of the mesh. $\mathbf{F}$ is a large column vector of the same size as $\mathbf{U}$ that contains the load placed on each vertex. We will be examining a single-load cantilever beam, so only one node has a load placed on it. $\mathbf{K}$ is a large matrix containing submatrices that describe the stiffness of each node, which depends on the density values $x_i$ in the surrounding elements. $N$ is the number of elements, and $V$ is a dimensionless parameter describing the fraction of the volume that we wish to fill with material. While the objective function does not initially



(a) $p = 3.0, V = 0.3$ (b) $p = 3.0, V = 0.5$ (c) $p = 3.0, V = 0.7$

Figure 1: Cantilever beams at different volumes

appear to be compliance, it in fact is. To compute compliance, that is, work performed on the structure, we want to compute $\mathbf{U}^T\mathbf{F}$, which is the work performed on the nodes. However, $\mathbf{KU} = \mathbf{F}$, so $\mathbf{U}^T\mathbf{KU} = \mathbf{U}^T\mathbf{F}$. We write the objective function in this way so that its value can be directly controlled by adjusting the densities, which in turn adjusts the stiffness.

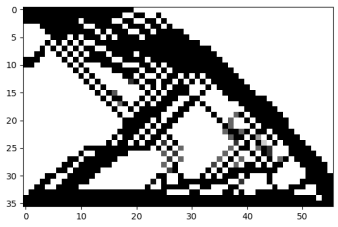
In order to perform the optimization, we first compute the nodal displacements $\mathbf{U}$ by solving the linear system above. Using the formulation by Ole Sigmund [**?**], $\mathbf{K}$ is a positive definite symmetric matrix, so the system can easily be solved via Cholesky decomposition or other means. Then, once displacement has been computed, we linearize the objective function by a first-order Taylor approximation, taking its partial derivatives with respect to the element densities. This linearized function is simply $\sum_{i=1}^{N} px_i^p \mathbf{u}_i^T \mathbf{k}_0 \mathbf{u}_i$, where $\mathbf{u_i}$ is a column vector containing the nodal displacements of element $i$, and $\mathbf{k}_0$ is the innate stiffness matrix of the material. In each iteration, we want to *maximize* this linearized objective function. While this is counterintuitive, the goal is to make it as difficult as possible to displace the object. Therefore, we want to maximize the work needed to displace the object by the computed amount. The optimization continues in this way until convergence is reached.

# 3 Numerical Experiments and Analysis

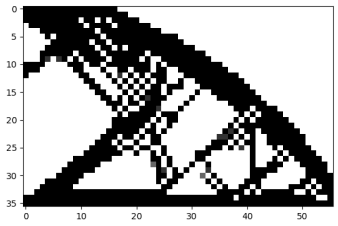
In this section, we will examine a cantilever beam clamped to the left side of the domain with a small load pulling down on the bottom-right corner. The following figures were generated with a mesh size of 56 by 36 elements. Each figure was created with an elastic modulus of 1.0 GPa and a load of 1 N on the bottom rightmost node.

Figure 1 shows the results of optimization with volume fractions of 0.3, 0.5, and 0.7. As is clearly shown, increasing the volume has a dramatic effect on the character of the structure. At higher volume fractions, the model tries to create a supporting beam for the top beam, which becomes increasingly thick as the volume becomes larger. At lower volume fractions, however, this support beam disappears, as there is not enough material for it to be worth creating.

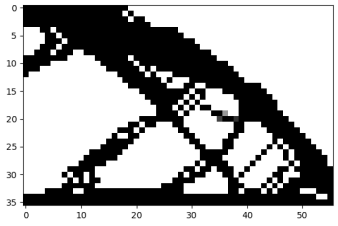
Increasing the penalty term has a more subtle effect. Figure 2 shows the results of optimization with penalties of 3.0, 3.5, and 4.0. The overall character of the structure remains the same, but the model makes an effort to avoid mesh or chainmail-like structures. This




(a) $p = 3.0, V = 0.4$

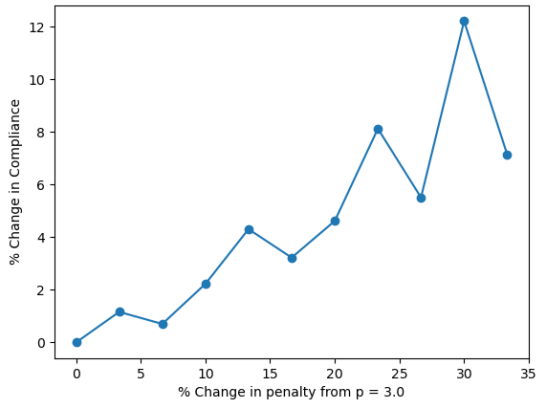



(b) $p = 3.5, V = 0.4$

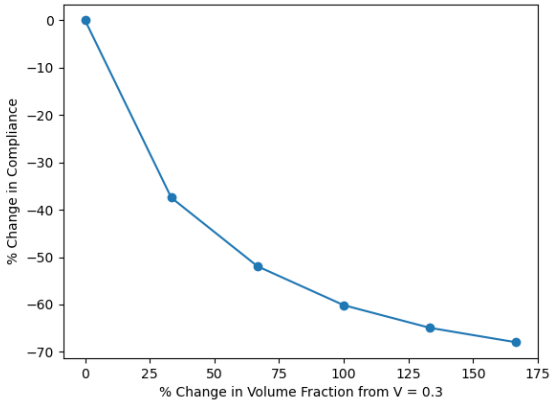



(c) $p = 4.0, V = 0.4$

Figure 2: Cantilever beams at different penalties




(a) Sensitivity of compliance to $p$




(b) Sensitivity of compliance to $V$

Figure 3: Sensitivity graphs

makes sense, as an increased penalty will reduce the quantity of elements with intermediate densities, which these mesh-like structures are partially made from.

Figure 3 shows graphs of the sensitivity of the compliance to the volume fraction and the penalty term. As can be seen, increasing the penalty tends to increase compliance, whereas increasing volume tends to decrease compliance. On average, a 1% increase in the penalty factor will increase the final compliance by 0.26%, whereas a 1% increase in the volume fraction will decrease the final compliance by 0.68%. This indicates that compliance is somewhat robust to changes in the penalty factor but fairly sensitive to the amount of volume used. These sensitivities are in line with what we see in the qualitative analysis of the structure itself, where the structure only changes slightly with increased penalty but drastically with increased volume.

# 4 Conclusion

In this project, I implemented and analyzed the SIMP model for topology optimization. One question that remains to be answered satisfactorily is how to interpret the intermediate densities as a practical matter. Bendsøe [**?**] describes in a followup paper how SIMP can be formulated as a sizing problem in which the objective is to determine the size of pores in an element. Elements with no material simply have pores as large as the element, elements with full density have no pores, and intermediate densities have various pore sizes. While this formulation is theoretically appealing, I personally still find it unsatisfying, as manufacturing infinitesimally porous material as Bendsøe describes is not feasible. Still, with advances in 3D printed lattice structures, we may be able to approximate such porous materials in due time. Until then, it may be best to handle these intermediate densities by removing them with a post-process, such as a morphological filter.

# References


[1] Martin P Bendsøe. Optimal shape design as a material distribution problem. *Structural optimization*, 1(4):193–202, 1989.

[2] Martin Philip Bendsøe and Noboru Kikuchi. Generating optimal topologies in structural design using a homogenization method. *Computer methods in applied mechanics and engineering*, 71(2):197–224, 1988.

[3] Phillip L Gould and Yuan Feng. *Introduction to Linear Elasticity*. Springer, 2018.

[4] Ole Sigmund. A 99 line topology optimization code written in matlab. *Structural and multidisciplinary optimization*, 21(2):120–127, 2001.