USE OF TOPOLOGY OPTIMIZATION IN CONCRETE REINFORCEMENT DESIGN



Michael Konečný **Jaromír Kabeláč** **Jaroslav Navrátil**

Abstract

Topology optimization may be used to facilitate reinforcement design of concrete structures. A brief overview of the method is given and ways it can be used for reinforcement design are described, along with several test case results. Three practical examples of design of typical concrete structures based on results generated by the method are shown.

Keywords: reinforced concrete, topology optimization, reinforcement design, discontinuity regions

1 Introduction

Current practice of the design and assessment of so called discontinuity regions of concrete structures is based on either the strut and tie method or on the use of scientifically oriented programs. Both of these methods suffer from different disadvantages, but none of them can be used for a concept-decision analysis of the structure or its detail as they require accurate dimensions, location directions and amount of reinforcement in advance. The author's experience is that even professionals have inadequate knowledge when it comes to the problem of the determination of positions and directions of the reinforcement in cases of atypical details of concrete structures.

2 Topology optimization method

The use of the topology optimization method may be of great help in reinforcement design. It can be used to generate a geometry by using only a certain percentage of the original material volume and rearranging it in a way that is "most effective" for the given set of loads, according to some criteria. This may result in a structure with a number of empty spaces where there is no material, which is of course impractical to manufacture by conventional methods other than 3D printing, but this geometry may be used as quite an accurate guide tool to identify the areas of tension and compression of the original concrete structure. This process is not much unlike the commonly used strut-and-tie method, but using this method, it can be done automatically, with much less need for human reflection and trial and error.

2.1 Objective function

In order to compute the optimal structure, it is necessary to define an objective of the optimization. Different options exist, but a good way is to try to maximize the global stiffness of the structure for a given load set. This is equivalent to minimizing the strain energy (also called compliance) which is equal to the work done by external loads. The suitability of this objective can easily be demonstrated on a simple model, see figure 2

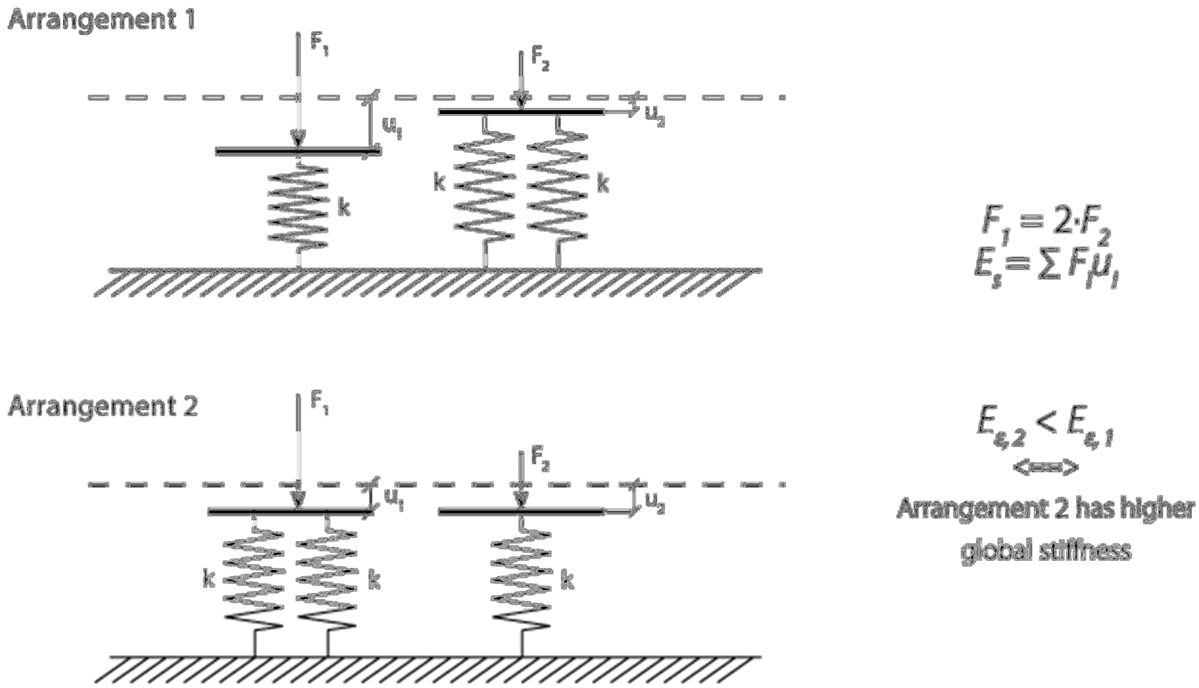


Figure 2 Strain energy in two systems with the same amount of material

Let's imagine that the springs in this model can be moved around voluntarily with the goal of achieving the best arrangement. One can intuitively see that Arrangement nr. 2 is better as the model is stiffer where higher load is applied and more compliant where less load is applied. Arrangement 2 is also the one with the lowest strain energy, so the optimization objective seems be set properly.

An analogy can be made with a 2D finite element model, where the number of springs per plate in our simple model is analogical to the density of material in a finite element and the strain energy can be calculated as the sum of external forces multiplied by displacement in nodes. The optimization problem can be then stated as:

$$\min(E_\varepsilon) <=> \min(\sum F_{e,i} u_i)$$

2.2 Optimization algorithm

The optimized structure can be calculated using an iterative algorithm, by starting with a geometry with a homogeneous material "density" distribution and changing the density of each element in each iteration in a way that leads to decreasing the total strain energy.

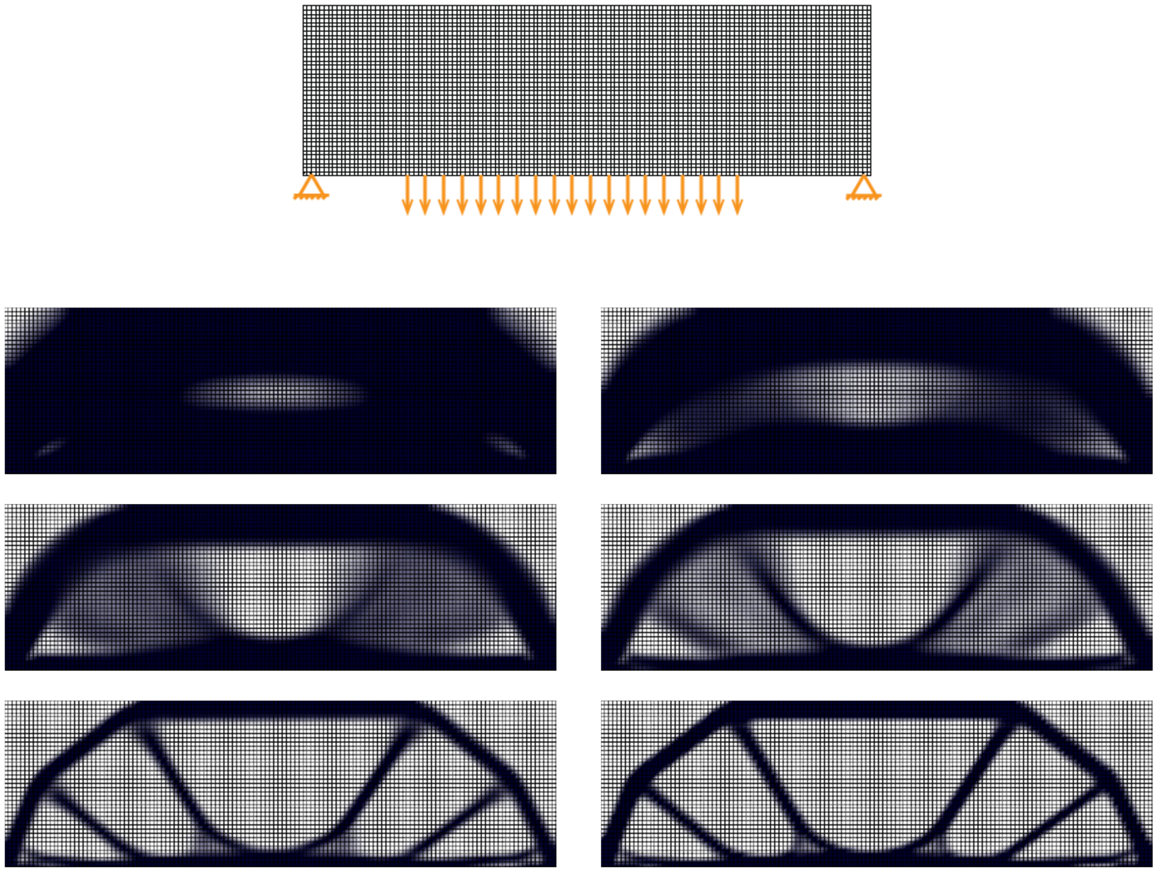


Figure 3 Gradual convergence of a vertically loaded structure with two point supports, iterations 1, 3, 7, 10, 15 and 19

It is to be noted, that in this model, the material "density" does not represent any real physical property; it is merely used to allow the optimization method to function. In the computation model, an element's stiffness is proportional to its density – an element with 100% density has its full original stiffness, whereas an element with 0% density has zero stiffness. However, when the optimization reaches the optimum, it results solely in elements with either 100% or 0 % density, which can be interpreted as places where material either is or isn't present.

In each iteration, a linear FEM analysis is performed. This is used to calculate the total strain energy and also to calculate the derivative of the total strain energy with respect to the element density ($\frac{\partial E_\varepsilon}{\partial \rho_i}$) in each element. This derivative is then used to adjust the density of the element in a way to decrease the strain energy. The changes in all elements' densities are calculated so that the total material volume converges towards a pre-set value, called the *effective volume* (typically 20 – 80 % of the original structure's volume). This

results in new element densities. A next iteration is then performed.
The algorithm is stopped once the change in total strain energy is reasonably small.

2.3 Effect of different effective volumes

The amount of material volume used to generate the optimized structure can have a significant effect on the character of the generated shape. In Figure 4, one can see that although for some geometries the generated shape always has the same character, there are other for which a change in used volume yields very different shapes.
It is therefore beneficial to show to the user results generated using several different values of effective volume, in order for them to be able to well understand the character of the structure.

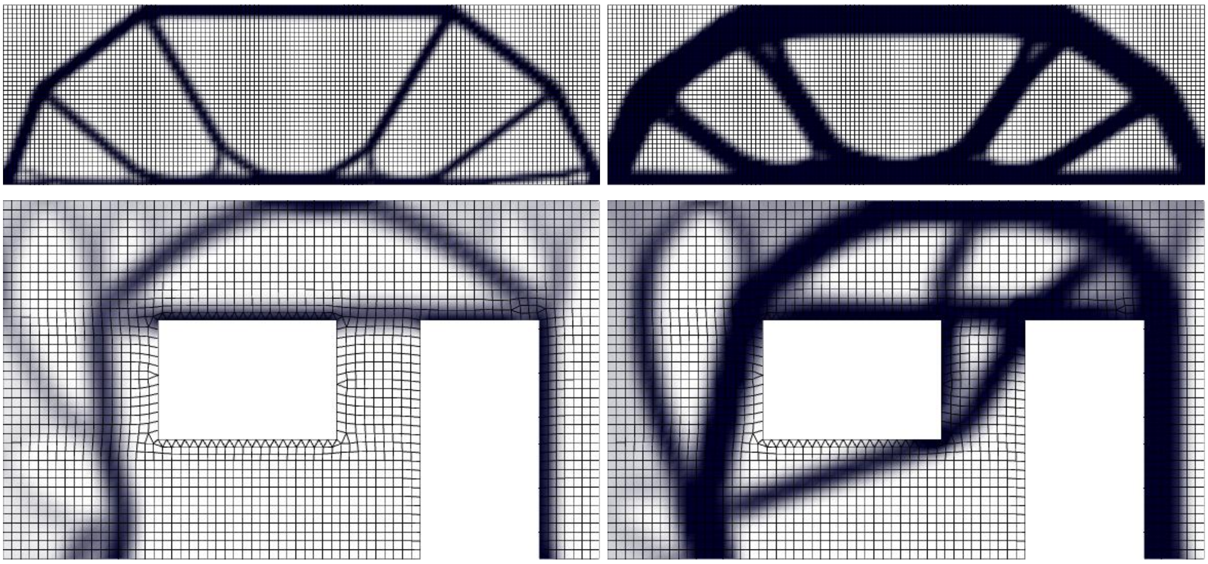


Figure 4 Low and significant effects of effective volume on generated geometry
Effective volumes used: Vf = 0.2 and 0.466

2.4 Examples of generated structures

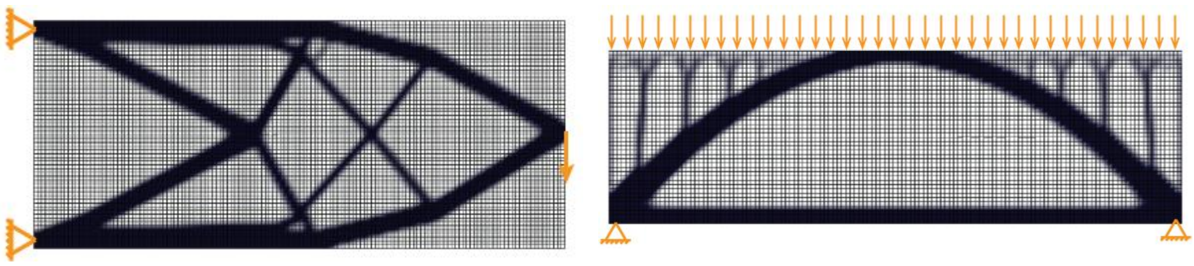
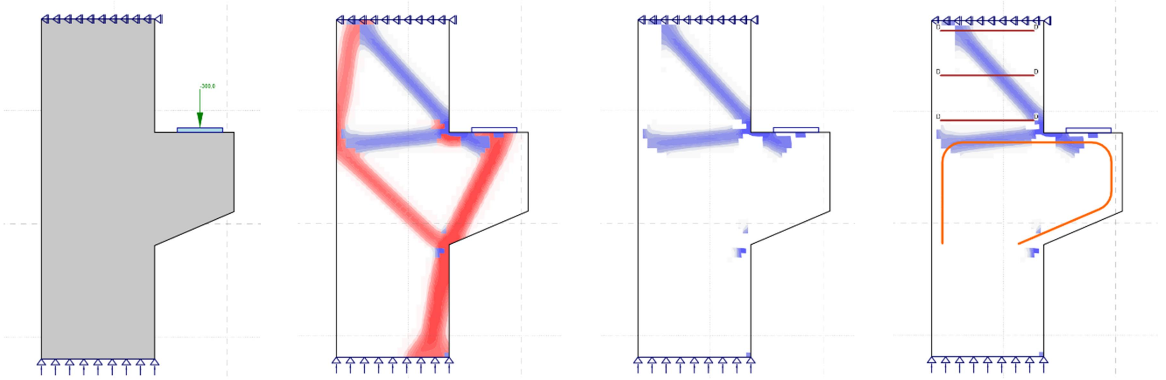


Figure 5 Load-carrying structure of cantilever and beam

2.5 Optimizing for multiple load cases

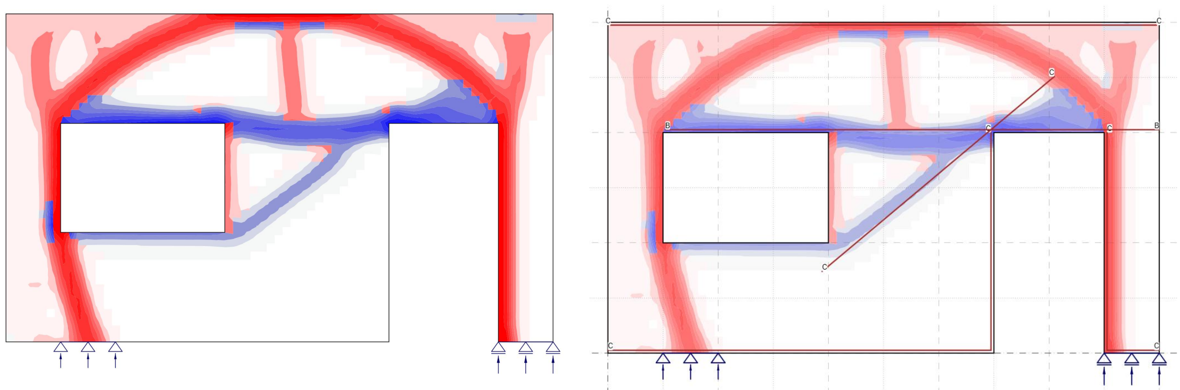
Structures usually need to withstand not just one, but multiple types of loads. Therefore, it is necessary for a reinforcement design tool to be able to take multiple load cases into account. For topology optimization, this can be done by slightly modifying the standard topology optimization algorithm. For each load case, a FEM analysis needs to be performed separately. This yields the desired changes in density for each of the load cases. These values are then averaged and the averages are used to update the densities of the structure.

3 Results for typical concrete structures

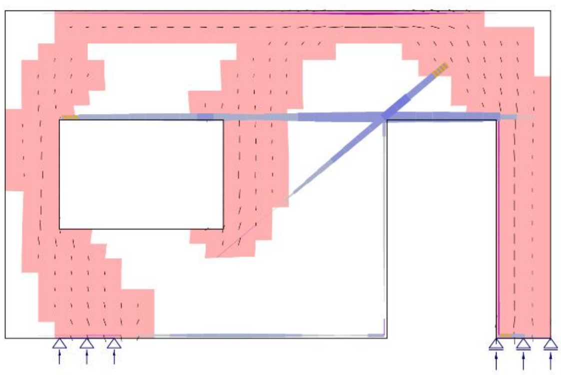


Model with constraints and load | Topology optimization | Compression parts filtered out | model with added reinforcement

Figure 6 Reinforcement of a bracket



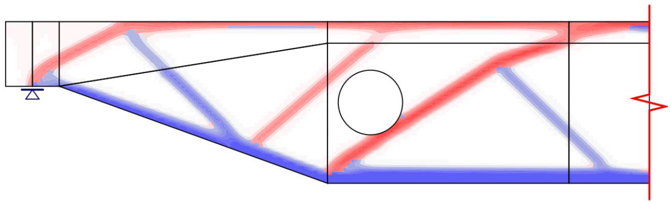
Topology optimization result | Model with added reinforcement



Result of non-linear analysis, red – stress in compressed concrete, blue – forces in reinforcement bars

Figure 7 Wall with door and window

Topology optimization has been implemented in a new user-friendly software IDEA StatiCa Detail. Results for typical concrete structures are shown in figures 6 to 8.



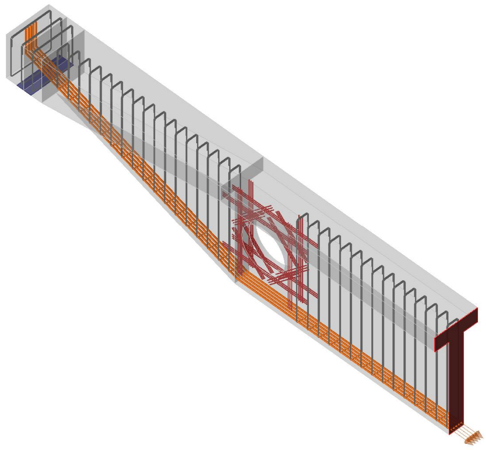


Figure 8 Dapped end with haunch and opening
Results of topology optimization
3D view of reinforced member

4 Summary

Although the results yielded by the topology optimization method still require some level of an engineer's reflection and interpretation, we believe it represents a fast and easy to use tool that can facilitate and speed up the task of reinforcement design significantly. Especially in cases of non-typical structures and/or multiple load cases, it can lead to results that would not otherwise be obvious, if conventional methods were used.
This can result in savings not only in engineering time, but also in the amount of reinforcement steel that needs to be used.

5 Acknowledgments

This project has received funding from the Eurostars-2 joint program, project ID 10 571, code 7D16010 with co-funding from the European Union Horizon 2020 research and innovation program. Related software IDEA StatiCa Detail was developed by IDEA RS Ltd in collaboration with IBK ETH Zürich

Ing. Michael Konečný
✉ IDEA RS s.r.o.
U Vodárny 2, 616 00 Brno
☎ +420 511 205 263
☺ konecny@ideastatica.com
URL www.ideastatica.com

Ing. Jaromír Kabeláč, Ph.D.
✉ Hypatia Solutions s.r.o.
třída Kpt. Jaroše 3, 602 00 Brno
Czech Republic
☺ kabelac@ideastatica.com
URL www.ideastatica.com

Doc. Ing. Jaroslav Navrátil, CSc.
✉ IDEA RS s.r.o., U Vodárny 2, 616 00 Brno, Czech Republic
☎ +420 511 205 263, ☺ navratil@ideastatica.com
URL **www.ideastatica.com**