

Lab11: Topologic optimization

05/12/2025

Abstract

This report explores topological optimization as a method for lightweight vehicle design through two case studies. The first study uses the MATLAB 99-line code to analyze mesh dependency and penalization factors in a simple beam. The second study applies Altair Inspire to a MacPherson suspension's lower control arm, reducing the material to 30% of the original under realistic loading conditions.

Results demonstrate that while optimization increases structural compliance, it successfully identifies the material distribution necessary to achieve significant weight reduction.

Contents

1. Introduction.....	3
1.1 Beam: problem definition	3
1.2 Lower arm: problem definition	3
2. MATLAB 99 lines optimization code	4
2.1 Mesh size dependence	5
2.2 Black & White	6
3. Topological optimization of lower control arm using Altair Inspire	8
3.1 Setup of the optimization	8
3.2 Results of the optimization	11
4. Conclusion.....	13

1. Introduction

Nowadays, lightweight is one of the key aspects in vehicle design. It helps, for example, to reduce emissions for ICE cars and improve range drivability for EV cars. Structural optimization can be divided into three categories: size, shape and topology optimization. The focus of this lab was the latter: it describes where the material is needed in a structure to support the loads and the stresses. The aim of this assignment was to explore topological optimization, first on a simple beam and afterwards on the lower arm of a MacPherson suspension.

1.1 Beam: problem definition

The structure of the beam had to be optimized using MATLAB and specifically the *99 lines optimization code*.

The aim of this first task was to investigate first the mesh dependency of the result and afterwards change the penalization factor: this last request is linked to the correct definition of a topological problem.

1.2 Lower arm: problem definition

This part of the lab was implemented on Altair Inspire software. The design domain was provided, and it was defined so to give extra material to the software to have freedom to change density where necessary. The steps required to run the optimization were:

- Partition the design domain so to freeze the regions where load and constraints are applied (The material should not be modified in those regions)
- Definition of the design domain
- Loads and constraints were applied (as shown in Figure 1). The static vertical load considered was 1000kg

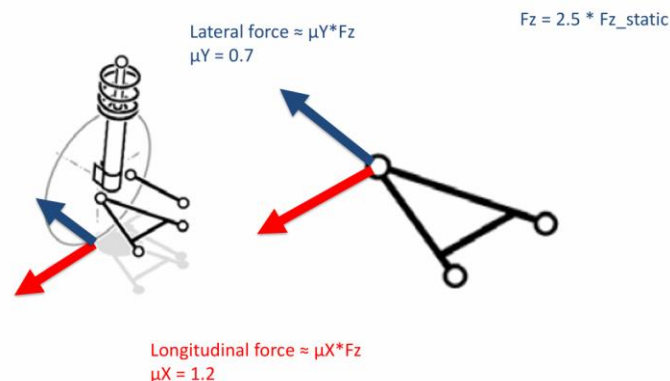


Figure 1

2. MATLAB 99 lines optimization code

The code implements the SIMP topology optimization algorithm. The main features of this algorithm are listed below:

- The material is selected a priori, its distribution within the domain is what needs to be optimized
- The structure is discretized into finite elements
- The objective function to be minimized is the internal deformation energy (\approx strain energy \approx compliance)
- The constraint is given as a ratio between the final/initial volume
- The (normalized) density of each finite element can assume continuous values between 0,1 and these are the variables that are going to be optimized. The fact that density assumes continuous values in a discretized domain is what ensures the well-posedness of the problem
- The relationships between the quantities involved in the minimization are the following:

$$\begin{cases} E_e = \rho_e^p E_e^* \\ M_e = \rho_e^q M_e^* \end{cases}$$

Equation 1

- $E_e \rightarrow$ stiffness of the finite element considered
- $\rho_e \rightarrow$ density of the finite element considered
- $p \rightarrow$ penalization power for E
- $E_e^* \rightarrow$ stiffness of the selected material when $\rho_e = 1$
- $M_e \rightarrow$ mass of the finite element considered
- $q \rightarrow$ penalization power for M
- $M_e^* \rightarrow$ mass of the selected material when $\rho_e = 1$

Penalty

Solving the problem with $p = 1$ (no penalty) leads to the computation of a solution with the lowest possible compliance but every finite element has an intermediate normalized density between 0,1. The physical meaning of this result is that each finite element is made of a material with intermediate properties between “nonmaterial” (void) and the selected material \Rightarrow **this solution is not feasible**.

The penalty for instance is needed to make the relationships written previously highly nonlinear and force the algorithm to choose between attributing each finite element a normalized density ≈ 0 (void element) or ≈ 1 (“full” element).

Note that increasing the penalty is like introducing an additional numerical constraint, hence, the result with increased penalty will be worse than the one with $p = 1$ in terms of compliance.

Filter

The final aspect that needs to be considered in the definition of the topological optimization is the introduction of a filter, that homogenizes the density of elements close to each other; this prevents the final solution from being a checkerboard.

All of the aspects previously described are taken into consideration in the function implemented with the MATLAB 99 line code

topology(nelx, nely, volfrac, penal, rmin)

- *nelx* → number of finite elements in horizontal direction
- *nely* → number of finite elements in vertical direction
- *volfrac* → ratio between final/initial volume. It is the constraint of the optimization problem
- *penal* → penalization of intermediate densities
- *rmin* → filter radius

2.1 Mesh size dependence

(50, 10, 0.5, 3, 0.575)

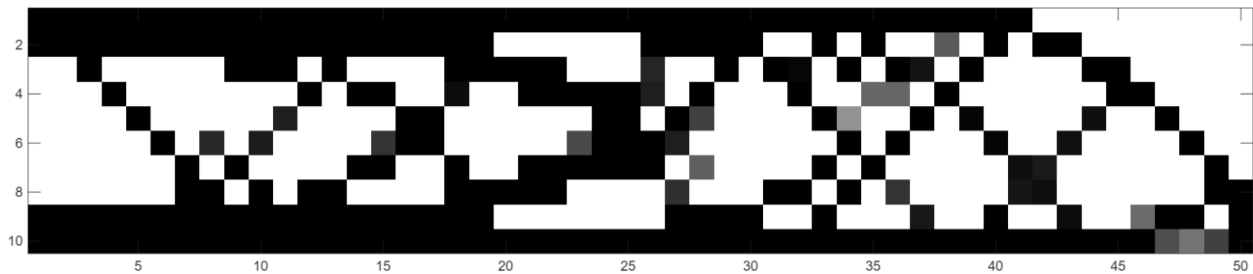


Figure 2

(100, 20, 0.5, 3, 1.15)

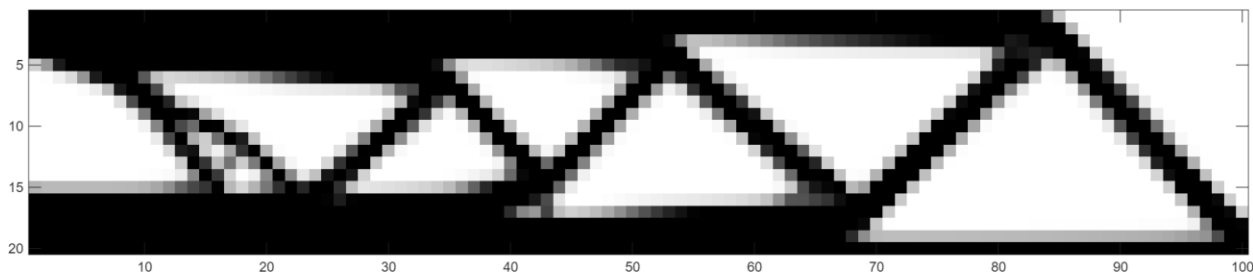


Figure 3

(200, 40, 0.5, 3, 2.5)

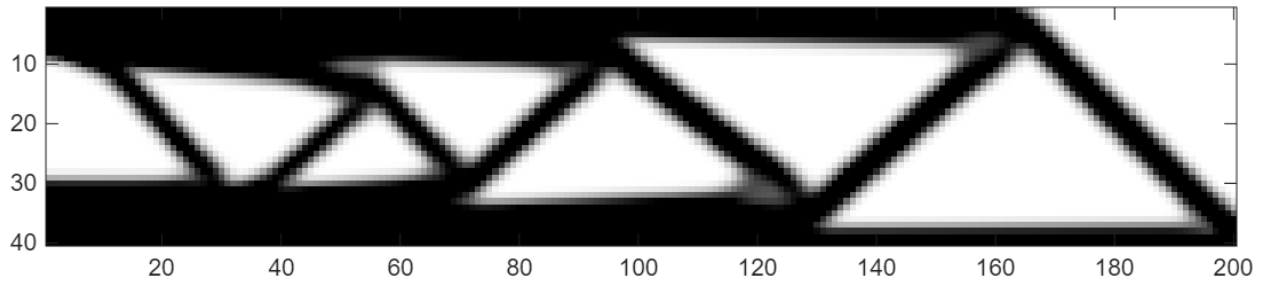


Figure 4

Increasing the filter radius proportionally to the discretization in vertical and horizontal direction, it is possible to evaluate that the design is mesh independent. Especially from Figure 3 to Figure 4, it is possible to see that the design, apart from the resolution remains almost equal.

2.2 Black & White

(100, 20, 0.5, 1, 1.15)

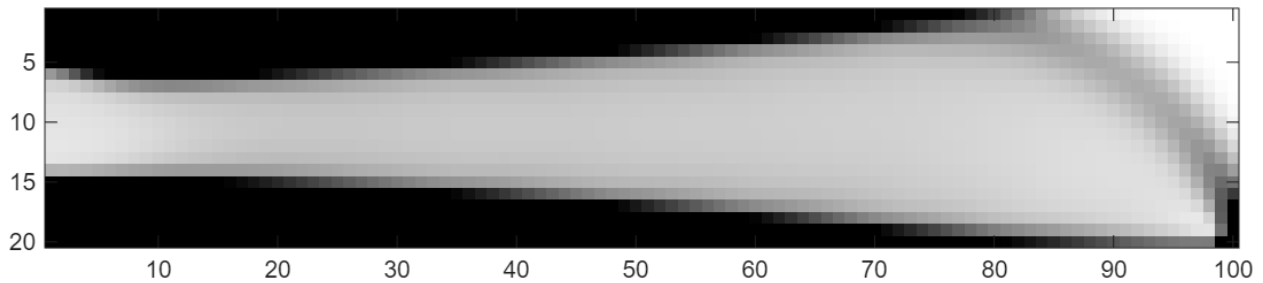


Figure 5

- **Compliance:** 631.6990

(100, 20, 0.5, 3, 1.15)

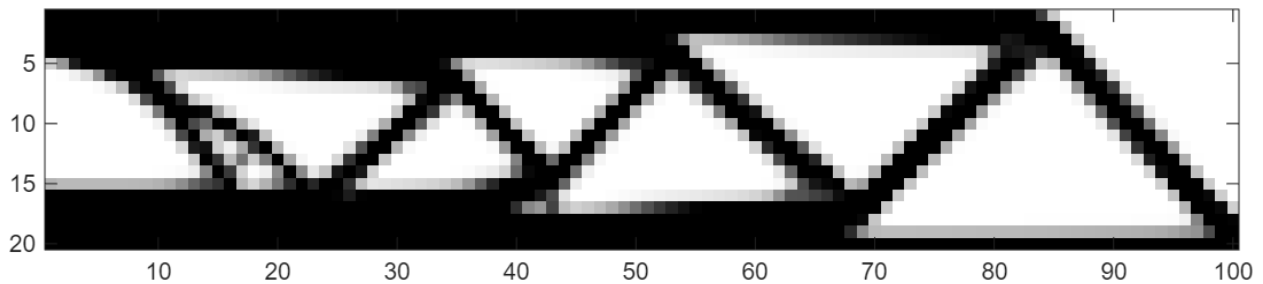


Figure 6

- **Compliance:** 725.0305

(100, 20, 0.5, 6, 1.15)

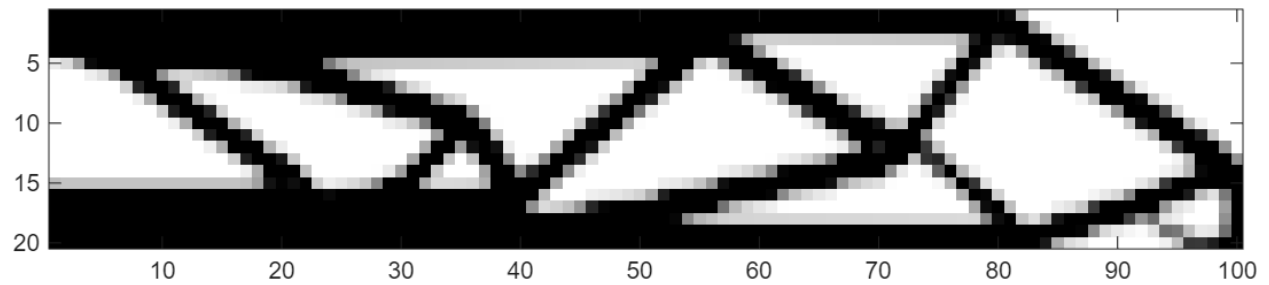


Figure 7

- **Compliance:** 763.2529
- In this case the solution is oscillating around a certain value, which is a typical problem for high value penalty. The higher penalty makes the curve extremely non-linear, preventing it to reach convergence (see **Error! Reference source not found.**)

3. Topological optimization of lower control arm using Altair Inspire

3.1 Setup of the optimization

The following steps have been implemented in the software:

1. Definition of the initial geometry

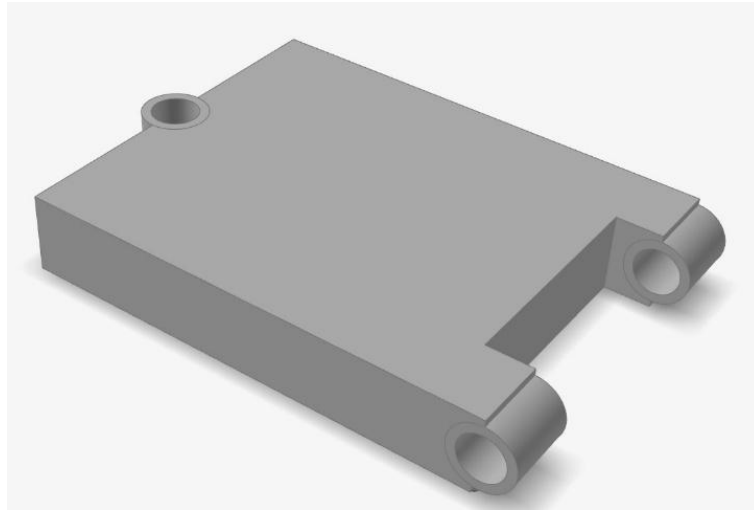


Figure 8

2. Definition of the material: Al6061-T6
3. Partition of the system the part to keep frozen the regions where the loads and the constraints are applied and definition of the design domain (Figure 9).
 - a. “frozen regions” are highlighted in gray
 - b. The design domain is highlighted in darker color

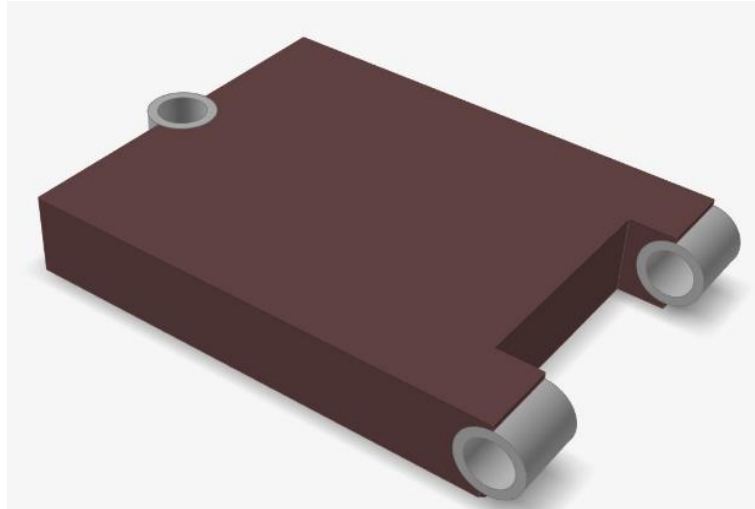


Figure 9

4. Definition of the applied loads

- Car selected : Fiat Panda ($m \approx 1000\text{kg} \rightarrow F_{z,tot} = 9810\text{ N}$)
- Vertical static load on the single wheel: $F_{z,stat} = 2452.5\text{ N}$
- Vertical load on one wheel $F_z = 2.5F_{z,stat} = 6131.25\text{ N}$
- $F_x = \mu_x F_z = 7357.5$ ($\mu_x = 1.2$)
- $F_y = \mu_y F_z = 4291.87$ ($\mu_y = 0.7$)
- Note that the vertical force along z is not considered since the constraints allow the rotation about the x axis

5. Definition of the constraints

- **Two pins** are placed in correspondence of the two bushings that connect the lower arm to the chassis. They block every movement except the rotation about the bushings' axis
- **Manufacturing constraint:** 2 dies stamping closing on x,y plane
- **Symmetry constraint** on the y,z plane (the example of optimized design provided by the assignment was taken as reference and that is symmetric)
- **Mass target:** 30% of the original material is left (Figure 10)
- **Minimum thickness constraint:** suggested by the software (Figure 10)

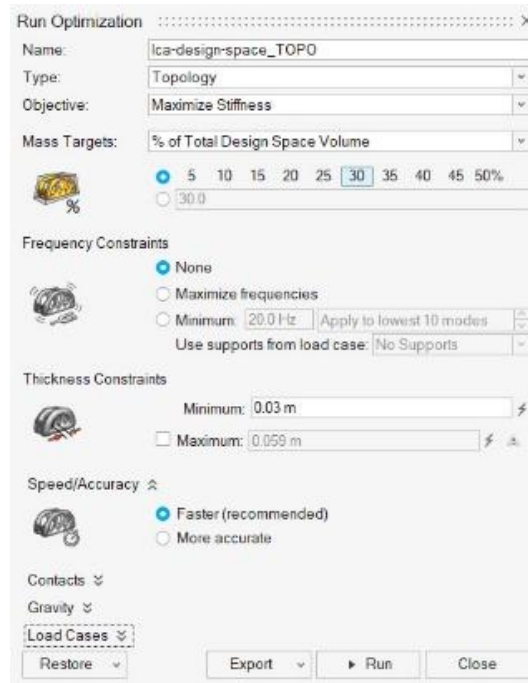


Figure 10

6. Static FEM analysis of the initial design (Figure 11)

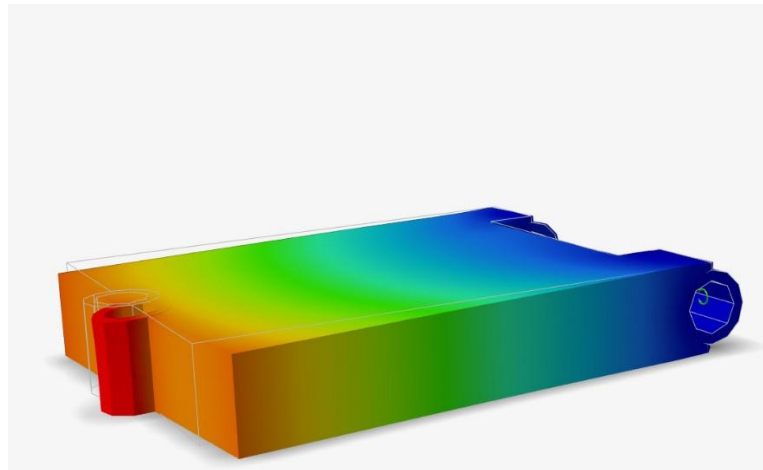


Figure 11

7. Topological optimization is run

3.2 Results of the optimization



Figure 12

The material is kept where it is more needed, so where the stresses are higher. It is important to notice that the geometry obtained is only an indication so to develop a definitive geometry to be verified again.

Fem of optimized design

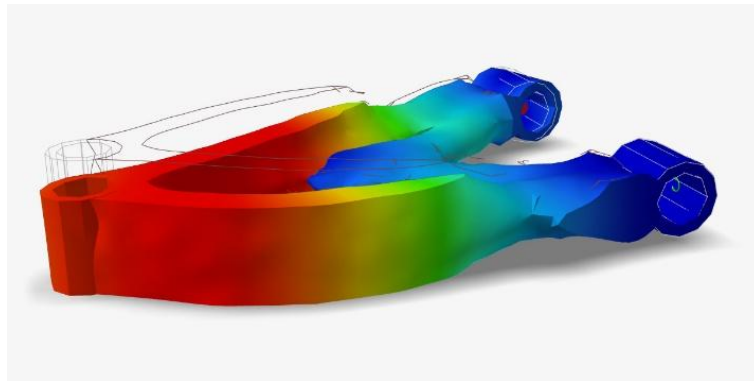


Figure 13

The computed compliance is higher than the one of the baseline case (as expected).

Fem of optimized simplified design

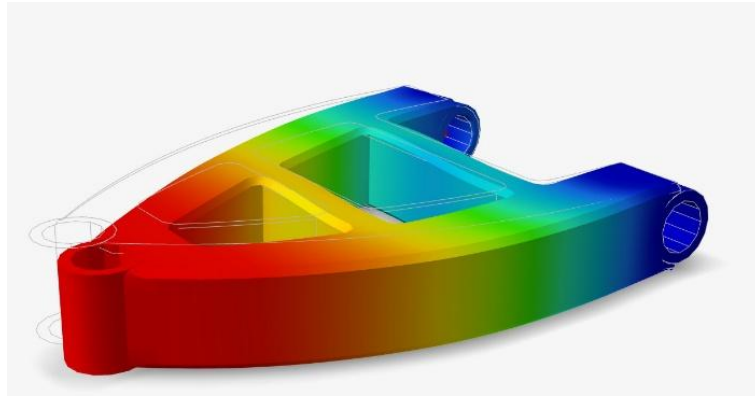


Figure 14

As already mentioned, the final solution is being verified again. The final compliance is increased compared to the initial block, but the piece geometry is optimized to have a lighter component.

4. Conclusion

The laboratory exercises confirmed that topological optimization is a powerful tool for balancing weight reduction with structural performance. The MATLAB simulations highlighted the necessity of **penalization factors** to eliminate intermediate densities and **filters** to prevent numerical checkerboard patterns. In the suspension arm application, the software successfully generated a design that retains material only in high-stress regions. It provides a crucial preliminary design that can be refined and verified for automotive production.