Faculdade de Engenharia Mecânica
r. Mendeleyev, 200
13081-970 Campinas, SP
mail:pava@fem.unicamp.br

Eco-Design and Topological Optimization

INSA de Lyon - Unicamp

Renato Pavanello and Jarir Mahfoud

October 2023

Suported by Région Auverne-Rhnes-Alpes, INSA de Lyon and FAPESP

1 Introduction

Numerous structural optimization methods have emerged over the years. The fundamental challenge of creating lighter structures that maintain performance standards inherently connects with sustainability and cost considerations. However, environmental demands have grown in relevance due to the escalating climate and environmental crises [1]. Consequently, new approaches for evaluating environmental impact and cost must be integrated in the design, optimization and analysis procedures of mechanical structures. In this course, we explore the possibilities to combine ECO-Design rules with Topological Optimization methods applied to design of mechanical structures [2].

2 Eco-Design of multi-material Mechanical structures

The principal goal of the course is integrated the concepts of Eco-Design on to classical structural designs methodologies based on the numerical simulations.

Today, the "Simulation Based Engineering (SBE)" is a common methodology used in practice of Mechanical and structural Design. In Figure 2 a global view of SBE methodology is illustrated, where the Experimental, Analytical and Numerical techniques must are used in a integrated manner.

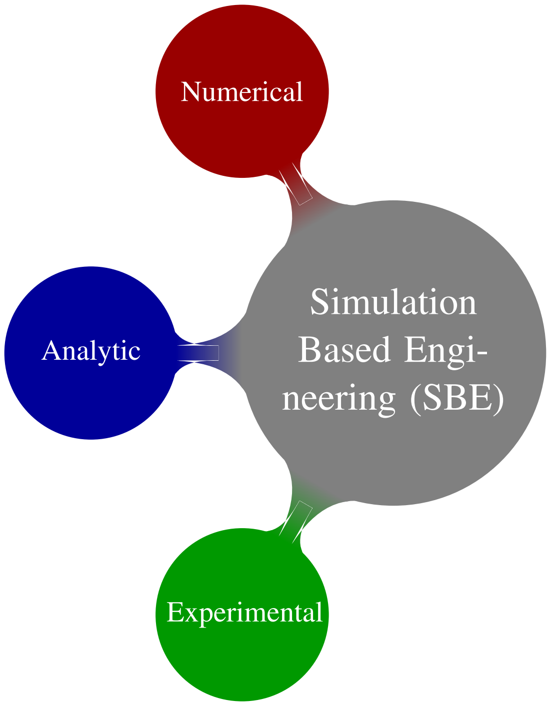


Figure 1: Simulation Based Engineering

Due to the exceptional advance in computational processing capacities, the use of computer simulation advanced rapidly at the beginning of the 21st century, being applied extensively the SBE methodology to problems of synthesis and optimization of structures and materials, with great emphasis on multi-physics and multi-scale problems. In this context, areas such as Computational Materials Engineering and Evolutionary Computation have grown rapidly, seeking not only to simulate and optimize structures and materials but also to predict how systems will evolve.

In the present course, we will introduce the use of Eco-Design methods in the context of the SBE, as illustrated in Figure 9. The chosen applications are the truss structures, very common in civil and mechanical engineering applications such as: bridges, cranes, towers, satellites, etc. (see Figure 3)

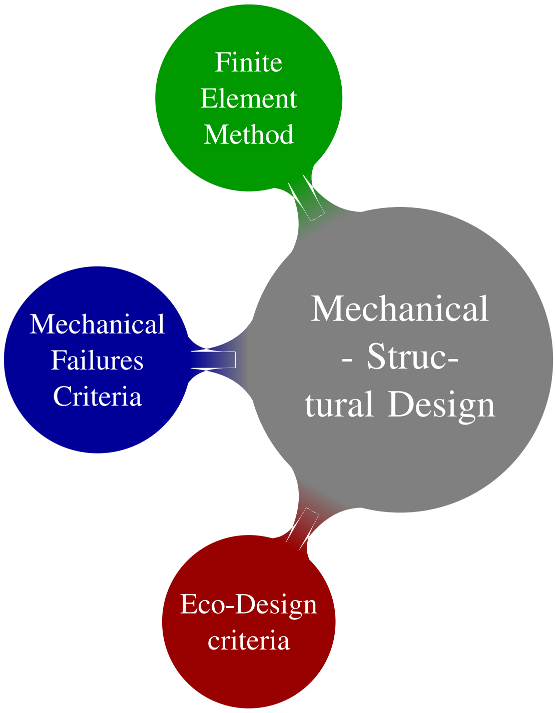


Figure 2: Eco-Design in SBE

To introduce this method, we consider the case of multi-material structures, for example a Truss build with steel mixed with parts processed using wood, as shown in figures 3,4,5 and 6. Timber is wood that has been processed into uniform and useful sizes, including beams or planks. Glulam, acronym from Glued Laminated Timber, is the more complex parts or components of structures manufactured using Timber.

Figure 3: Truss Applications: Ponts

Figure 4: Metal Timber Structures

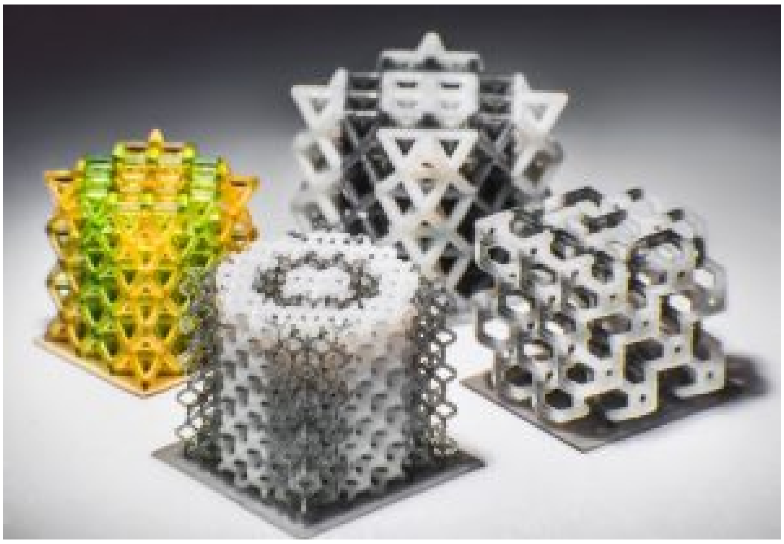


Figure 5: Multi-material micro-lattice polymeric structures fabricated using 3D printing

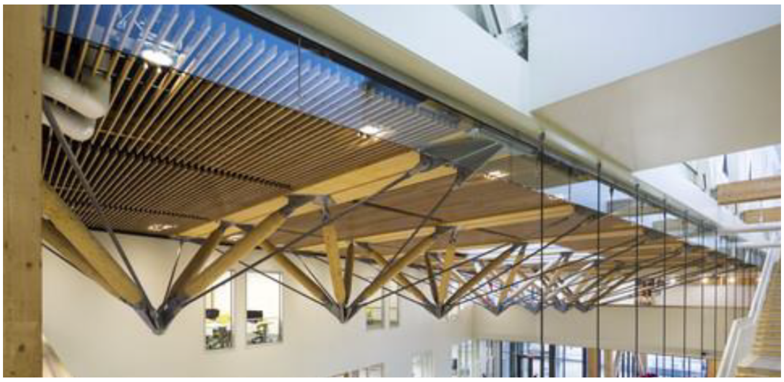


Figure 6: Timber-steel roof at UMass Amherst Design Building

3 Course ECO-Topological design

The present version of this course, is a practical computational implementation course, focused in introduce the ECO design criteria in the context of structural design of truss structures also named in this context of ground structures.

3.1 Truss Analysis

The first part of this course, is devoted to remember the principal concepts of bar modeling and solution using finite element method. The basic problem can be stated as:

> Formulation of static equilibrium equation of 2D truss using the finite element method. The state variables for each element can be calculate: volume, displacements, forces, stress.

The principal steps that will be remember are:

- Equilibrium equation , Strong and weak form of the problem;
- Discretization and approximation of the solution using Finite Element Method;
- Assemblage of the global system and Local to global transformation.
- Solve the discrete equilibrium equation to find nodal displacements.
- Post processing to evaluate the stresses.

3.2 Equilibrium Differential equation for bar problems

In the case of the straight bars, the formulation of the static equilibrium problem can be illustrated as shown in Figure 7.

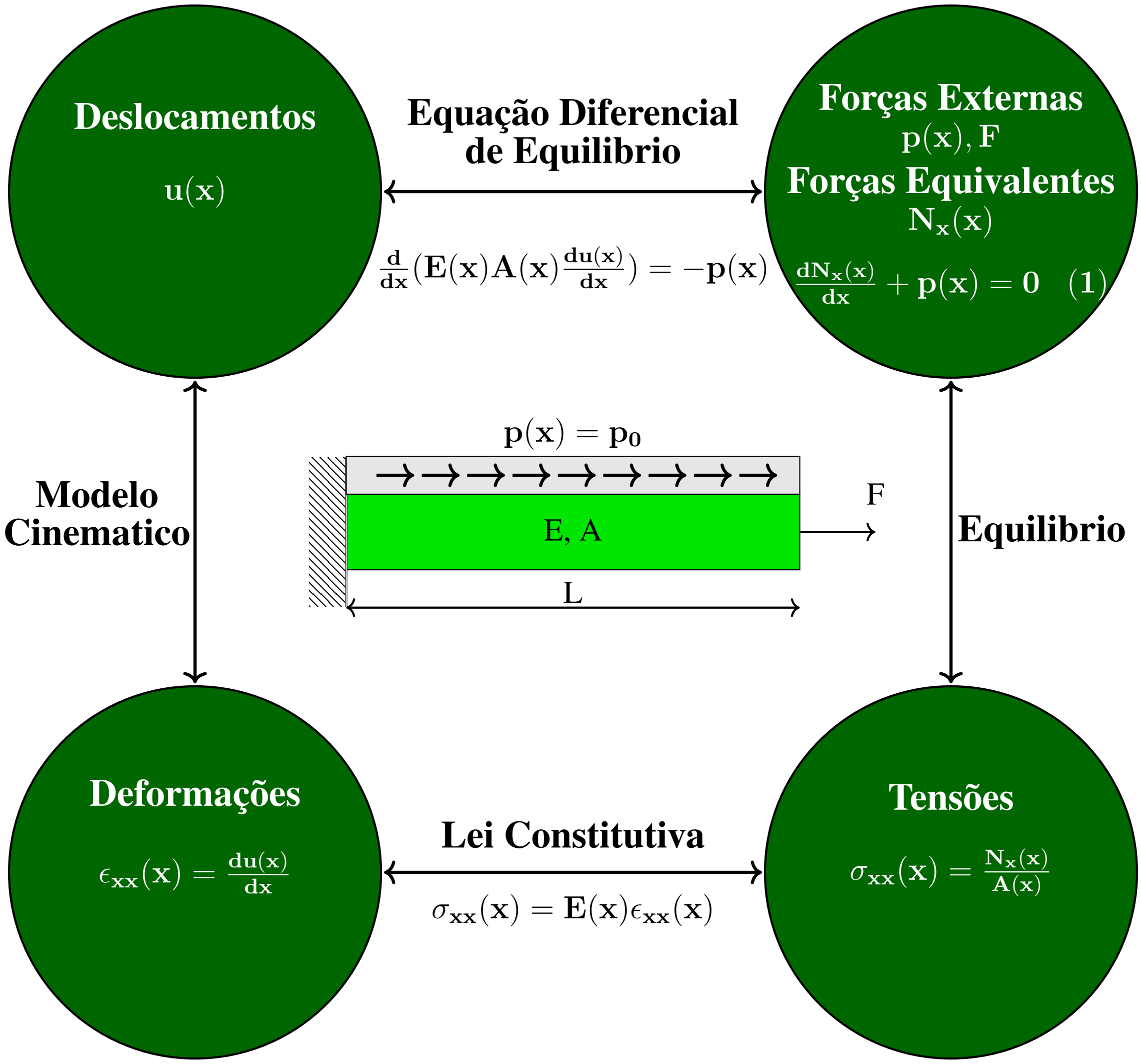


Figure 7: Bar formulation.

3.2.1 Kinematic model

Consider the bar represented in Figure 8, subject to a external Load F, and length L, with a are section A.

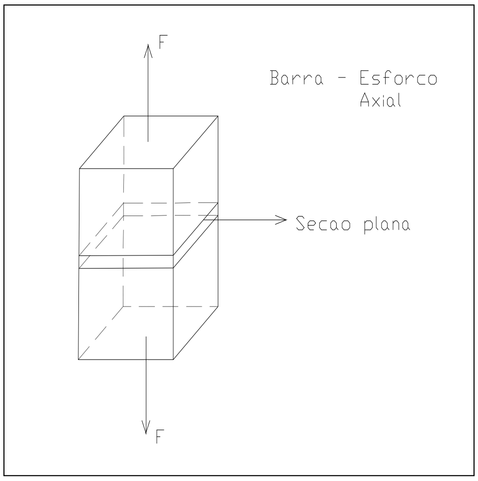


Figure 8: Bar in traction/compression

The basic kinematic assumption is that after the load is applied, all the sections that were initially straight remain straight. Thus the deformation in the direction of the longitudinal axis of the bar is given by:

- Differential element of the non deformed bar is Δx.
- After the load application, the length is :

$$\Delta x + (\frac{\partial u}{\partial x}\Delta x) \tag{1}$$

- and the expression of the strain is:

$$\varepsilon_{xx} = \frac{[\Delta x + (\frac{\partial u}{\partial x})\Delta x] - \Delta x]}{\Delta x} \tag{2}$$

or:

$$\varepsilon_{xx} = \frac{\partial u}{\partial x} \tag{3}$$

3.2.2 Constitutive model

We consider that the bars are made with material homogeneous, isotropic with a linear elastic behaviour. In this conditions, we can adopted the Hooke's Law:

$$\sigma_{xx} = E\varepsilon_{xx} \tag{4}$$

where,

- σ_{xx} is the component of the stress tensor;
- E is the Young modulus;
- ε_{xx} is the component of the strain tensor.

3.2.3 Equilibrium conditions

Using the Newton's Law, for a bar subjected to a traction load F, see Figure 9, where due top the kinematic model the internal forces and axial stress σ_{xx} can be considered constant, then:

$$F = \int_A \sigma_{xx} dA = \sigma_{xx} \int_A dA = \sigma_{xx} A \tag{5}$$

where σ_{xx} is the noram stress and A is the area cross section of the bar.

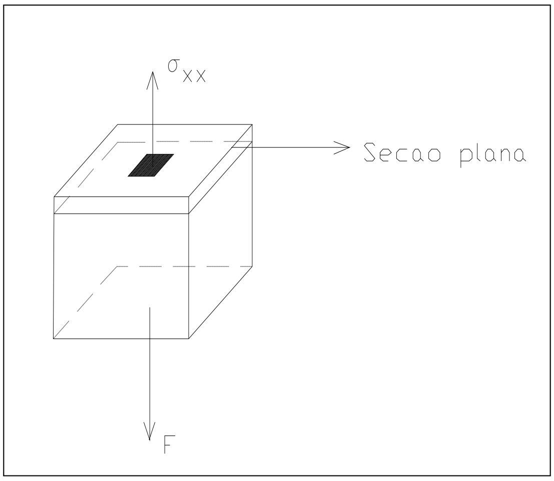


Figure 9: Internal forces of the bar

Using equations (35) and (36) in equation (37) we can have the first equilibrium equation:

$$F = \sigma_{xx} A = E \varepsilon_{xx} A = EA \frac{\partial u}{\partial x} \tag{6}$$

The second equilibrium equation consider a distributed load $p(x)$, as shown in Figure 10.

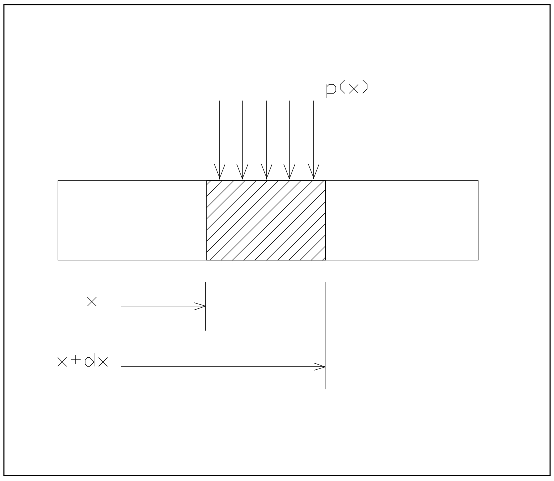


Figure 10: Bar subjected to a distributed load

In this condition, the equilibrium in the x direction is:

$$-F_x + p(x)dx + F_x + dF_x = 0 \tag{7}$$

and:

$$p(x) = -\frac{dF_x}{dx} \tag{8}$$

Using equation (39) in equation Eq. (8) we can written:

$$p(x) = -\frac{d}{dx}(EA\frac{du}{dx}) \tag{9}$$

considering the area and the Young's modulus constant along the bar:

$$EA\frac{d^2u}{dx^2} + p(x) = 0 \tag{10}$$

The equation (10) govern the static equilibrium problem for a bar.

3.2.4 Boundary conditions

To solve the problems governed by the equation (10) it is necessary two boundary conditions.

- the essential boundary condition is:

$$u = \bar{u} \qquad\qquad \text{em } \Gamma_1$$

- and the natural boundary condition:

$$\frac{\partial u}{\partial x} = \frac{\bar{F}}{EA} \qquad \text{em } \Gamma_2$$

where $F = 0$ for free boundaries or $F = \bar{F}$ in the cases of externals loads is applied. The distributed loads $p(\bar{x})$ are the body forces included directly in the equilibrium equation (10).

3.2.5 Summary - bar problem

Find u such that:

$$\begin{cases} EA\frac{d^2u}{dx^2} + p(x) = 0 & \text{no domínio } \Omega \\ \text{sujeito a:} & \\ u = \bar{u} & \text{em } \Gamma_1 \\ \frac{\partial u}{\partial x} = \frac{\bar{F}}{EA} & \text{em } \Gamma_2 \end{cases}$$

3.3 Finite Element approximation - Bar

Using the Weighted Residual method we can define the residue $R(x)$ as:

$$R(x) = L(u(x)) - f(x)$$

where

- $L(u(x))$ is a linear differential operator
- $f(x)$ is independent of $u(x)$
- $u(x)$ is the displacement field

To minimize the residue, we can use the Galerkin method, where:

$$\int_\Omega W_l R(x) dx = 0 \qquad l = 1, 2, 3, ..., M \tag{11}$$

where $W_l(l = 1, 2, 3, ..., M)$ are the weight or test functions and Ω is the spacial domain of the problem. For a bar with length L:

$$\int_0^L \left(\frac{d}{dx} \left(EA \frac{du}{dx} \right) + p(x) \right) W_l \, dx = 0 \tag{12}$$

Using the Green theorem:

$$\int_0^L EA\left(\frac{dW_l}{dx}\frac{du}{dx}\right)dx - \int_0^L W_l p(x)dx - \int_{\Gamma_1} EA\frac{du}{dx}W_l d\Gamma - \int_{\Gamma_2} EA\frac{du}{dx}W_l d\Gamma = 0$$

and the Weak Form of the problem is:

$$\int_0^L EA\left(\frac{dW_l}{dx}\frac{du}{dx}\right)dx - \int_0^L W_l p(x)dx - \left[EA\frac{du}{dx}W_l\right]_{\Gamma_1} - \left[EA\frac{du}{dx}W_l\right]_{\Gamma_2} = 0 \quad (13)$$

3.3.1 Weight Functions - Galerkin method

Using the Galerkin method, the shape functions $N_l(l = 1, 2, 3, ..., M)$, used in the Finite Element approximation is used as the weight functions in the Weak Formulation.

$$W_l = N_l \qquad\qquad l = 1, 2, 3, ..., M$$

and the Weak form can be re-written as:

$$\int_0^L EA\frac{dN_l}{dx}\frac{du}{dx}dx = \int_0^L N_l p(x)dx + \left[EA\frac{du}{dx}N_l\right]_{\Gamma_1} + \left[EA\frac{du}{dx}N_l\right]_{\Gamma_2} \quad (14)$$

3.3.2 Application of the Finite Element Method

To apply the FEM method, we discretize the domain Ω in a certain number of subdomains Ω^e.

$$\int_\Omega W_l R_\Omega d\Omega = \sum_{e=1}^{nel} \int_{\Omega^e} W_l R_{\Omega^e} d\Omega^e$$

where

- nel is the numbers of subdomains or elements.

To find an approximation for the displacements $\tilde{u}$, for each subdomains in $\Omega = 0 \leq \bar{x} \leq L$, we have:

$$u \cong \tilde{u} = \sum_{m=1}^{M} u_m N_m$$

where,

- $u_m(m = 1, 2, 3, ..., M)$ are the nodal parameters, or nodal displacements

- $N_m(m = 1, 2, 3, ..., M)$ are the shape functions, defined for each element

The nodal parameters u_m and the shape functions enforced that when $M \to \infty$ then $\tilde{u} \to u$. For a finite value of M the equation 14 can be represented in the matrix form:

$$\tilde{u} = [N]\{u\}$$

Considering a bar element with two nodes, as shown in Figure 11, the shape function are defined by:

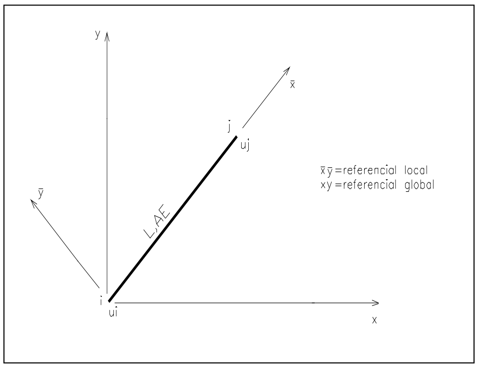


Figure 11: Bar element

$$\tilde{u}(\bar{x}) = \left\{ \begin{array}{cc} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{array} \right\} \left\{ \begin{array}{c} u_i \\ u_j \end{array} \right\}$$

where:

- $N_i = 1 - \frac{\bar{x}}{L}$ shape function
- $N_j = \frac{\bar{x}}{L}$ shape function
- $\{u_i u_j\}$ nodal displacements on node i and j

3.3.3 Stiffness Matrix and Equivalent load vector

Using the approximation $\tilde{u}$, and considering the local referential, (see Figure 11), we can write:

$$EA \sum_{m,l=1}^{2} \int_0^L \frac{dN_m}{d\bar{x}} \frac{dN_l}{d\bar{x}} d\bar{x} u_m = \int_0^L N_l p(\bar{x}) d\bar{x} + \left[EA \frac{d\tilde{u}}{d\bar{x}} N_l \right]_{\bar{x}=0} + \left[EA \frac{d\tilde{u}}{d\bar{x}} N_l \right]_{\bar{x}=L} \tag{15}$$

In matrix form,

$$EA\int_0^L \left[\begin{array}{cc} \frac{dN_1}{d\bar{x}}\frac{dN_1}{d\bar{x}} & \frac{dN_2}{d\bar{x}}\frac{dN_1}{d\bar{x}} \\ \frac{dN_1}{d\bar{x}}\frac{dN_2}{d\bar{x}} & \frac{dN_2}{d\bar{x}}\frac{dN_2}{d\bar{x}} \end{array} \right] d\bar{x} \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\} =$$
$$\int_0^L \left\{ \begin{array}{c} p(\bar{x})N_1 \\ p(\bar{x})N_2 \end{array} \right\} d\bar{x} + EA \left\{ \begin{array}{c} \frac{d\tilde{u}}{d\bar{x}} \mid_{\bar{x}=0} N_1 \\ \frac{d\tilde{u}}{d\bar{x}} \mid_{\bar{x}=0} N_2 \end{array} \right\} + EA \left\{ \begin{array}{c} \frac{d\tilde{u}}{d\bar{x}} \mid_{\bar{x}=L} N_1 \\ \frac{d\tilde{u}}{d\bar{x}} \mid_{\bar{x}=L} N_2 \end{array} \right\}$$

or, in a compact matrix notation:

$$[K]\{u\} = \{R_T\} + \{R_B\} \tag{16}$$

where $[K]$ is the stiffness matrix of one element, u are the nodal displacements of the element and $R = \{R_T\} + \{R_B\}$ represent the vector load due to the boundary conditions. Integrating the terms of stiffness matrix in a analytic manner, we can write:

$$K = \frac{AE}{L} \left[\begin{array}{rr} 1 & -1 \\ -1 & 1 \end{array} \right]$$

For the equivalent load vector, considering $p(\bar{x}) = p_o$ we can calculate:

$$R_T = \left\{ \begin{array}{c} \frac{p_o L}{2} \\ \frac{p_o L}{2} \end{array} \right\}$$

3.3.4 Global stiffness matrix for truss structures

Starting with a 2D truss. The local and global coordinate system are $(\bar{x}, \bar{y})$ and (x, y) respectively, as shown in Figure (12).

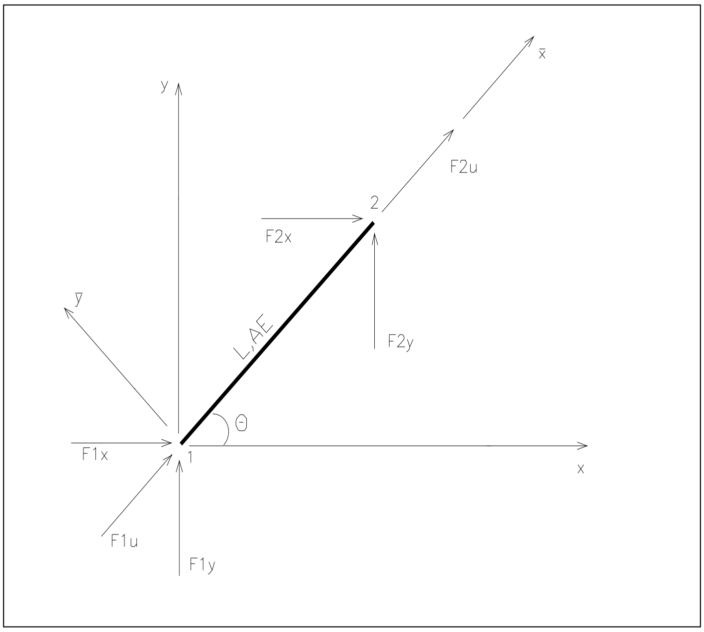


Figure 12: Global and Local coordinate systems

Using the Figure (12) it is possible to write the matrix T relating the global and local vectors quantities,

$$[T] = \begin{bmatrix} cos\theta & sen\theta \\ -sen\theta & cos\theta \end{bmatrix} \tag{17}$$

Applying the referential transformation to the elemental Stiffness matrix, and for the load vector, we can write:

$$[K] = [T]^{'}[K][T] \tag{18}$$

and using $cos\theta = \lambda$ e $sen\theta = \mu$, the elemental matrix can be written as:

$$[K] = \frac{AE}{L} \begin{bmatrix} \lambda^2 & & & \text{Sim.} \\ \lambda\mu & \mu^2 & & \\ -\lambda^2 & -\lambda\mu & \lambda^2 & \\ -\lambda\mu & -\mu^2 & \lambda\mu & \mu^2 \end{bmatrix} \tag{19}$$

and the force vector for one element:

$$F = \begin{Bmatrix} F_{ix} \\ F_{iy} \\ F_{jx} \\ F_{jy} \end{Bmatrix}$$

with all forces in the global (x, y) . The displacement vector u is:

$$u = \left\{ \begin{array}{c} u_1 \\ v_1 \\ u_2 \\ v_2 \end{array} \right\}$$

After Assemblage procedure, we can find the stiffness matrix and load vector for truss as shown bellow:

$$[Kg]\{ug\} = \{Fg\} \tag{20}$$

where $[Kg]$ is the global stiffness matrix, and $\{Fg\}$ is the global load vector, calculating using:

$$[Kg] = \sum_{1}^{nel}[K_e] \ \ ; \ \ \{Fg\} = \sum_{1}^{nel}\{F_e\} \tag{21}$$

where $[K_e]$ is the stiffness matrix of the element e and $\{F_e\}$ is the load vector of the element e.

For a 3D case, as shown in Figure (13) , the geometric transformation for the coordinate system can be express by:

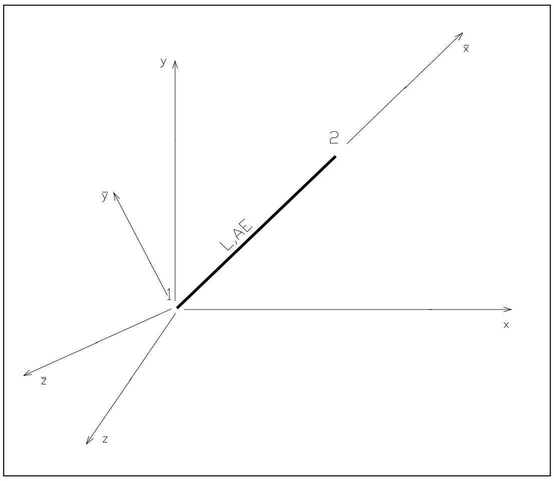


Figure 13: Representação Genérica de Elemento de Treliça Espacial

$$\begin{aligned}
\bar{x} &= cos\theta_{x\bar{x}}\mathbf{i} + cos\theta_{y\bar{x}}\mathbf{j} + cos\theta_{z\bar{x}}\mathbf{k} \\
\bar{y} &= cos\theta_{x\bar{y}}\mathbf{i} + cos\theta_{y\bar{y}}\mathbf{j} + cos\theta_{z\bar{y}}\mathbf{k} \\
\bar{z} &= cos\theta_{x\bar{z}}\mathbf{i} + cos\theta_{y\bar{z}}\mathbf{j} + cos\theta_{z\bar{z}}\mathbf{k}
\end{aligned}$$

and using the compact notation:

$$\begin{array}{lll} \lambda_u = cos\theta_{x\bar{x}} & \lambda_v = cos\theta_{x\bar{y}} & \lambda_w = cos\theta_{x\bar{z}} \\ \mu_u = cos\theta_{y\bar{x}} & \mu_v = cos\theta_{y\bar{y}} & \mu_w = cos\theta_{y\bar{z}} \\ \nu_u = cos\theta_{z\bar{x}} & \nu_v = cos\theta_{z\bar{y}} & \nu_w = cos\theta_{z\bar{z}} \end{array}$$

The matrix of coordinate systems can be written as:

$$T = \begin{bmatrix} \lambda_u & \mu_u & \nu_u \\ \lambda_v & \mu_v & \nu_v \\ \lambda_w & \mu_w & \nu_w \end{bmatrix} \tag{22}$$

and the stiffness matrix of one element is:

$$[K] = \frac{AE}{L} \begin{bmatrix} \lambda^2 & & & & & \text{Sim.} \\ \lambda\mu & \mu^2 & & & & \\ \lambda\nu & \mu\nu & \nu^2 & & & \\ -\lambda^2 & -\lambda\mu & -\lambda\nu & \lambda^2 & & \\ -\lambda\mu & -\mu^2 & -\mu\nu & \lambda\mu & \mu^2 & \\ -\lambda\nu & -\mu\nu & -\nu^2 & \lambda\nu & \mu\nu & \nu^2 \end{bmatrix} \tag{23}$$

where, only the geometric transformation terms relative to $\bar{x}$ to x are involved, then:

$$\begin{array}{l} \lambda = \lambda_u \\ \mu = \mu_u \\ \nu = \nu_u \end{array} \tag{24}$$

Similar than the 2D case, after Assemblage procedure, we can find the stiffness matrix and load vector for truss as shown bellow:

$$[Kg]\{ug\} = \{Fg\} \tag{25}$$

where $[Kg]$ is the global stiffness matrix, and $\{Fg\}$ is the global load vector, calculating using:

$$[Kg] = \sum_{1}^{nel} [K_e] \;\; ; \;\; \{Fg\} = \sum_{1}^{nel} \{F_e\} \tag{26}$$

where $[K_e]$ is the stiffness matrix of the element e and $\{F_e\}$ is the load vector of the element e.

3.4 Matlab script

Implementation and testes of a simple truss using a Matlab script. Code Directory» Truss Analysis

The matlab implementation can be divided in the following tasks:

- Data base: node coordinates, elemental incidences, geometric and material properties
- Boundary conditions and load cases.
- Elemental properties: stiffness matrices and load vectors.
- Assemblage, boundary conditions ans solve.
- Post processing: stress.

One script has been developed, to simulated the static equilibrium conditions, considering a cantilever boundary conditions, as represented in Figure (14). Two mesh generators have been developed. The first one for a X reinforced truss and, the second mesh generator all bar are connected with all nodes.

3.5 Multi material Truss - Mechanical and Ecological variables

Implementation of a typical truss 2D case, and evaluate local/global indicator for strain energies/compliance, Embodied carbon Coefficients of the structure and evaluate the stress for each bars. Compare the results of a steel truss with a timber/steel truss and timber trusses.

The second part of this implementation are related with the pos-processing of variables. In figure (15) we can observe one example with two materials and in figure (16) we can see the displacements and bar stress for a simple supported beam case.

The numerical results obtained using the script Matlab named Truss-Analysis are:

- $\mathbf{u_g}$ the nodal displacements;
- σ_{xx} the elemental stress;
- $C = \mathbf{F_g^T u_g}$ the structure Compliance ;
- $C_e = \mathbf{u_e}^T \mathbf{K_e u_e}$ the elemental compliance;
- $\frac{dC}{dA_e}$ the sensitivity number;
- W the weight of the structure;
- $W_e = \rho_e A_e L_e$ the elemental weight;

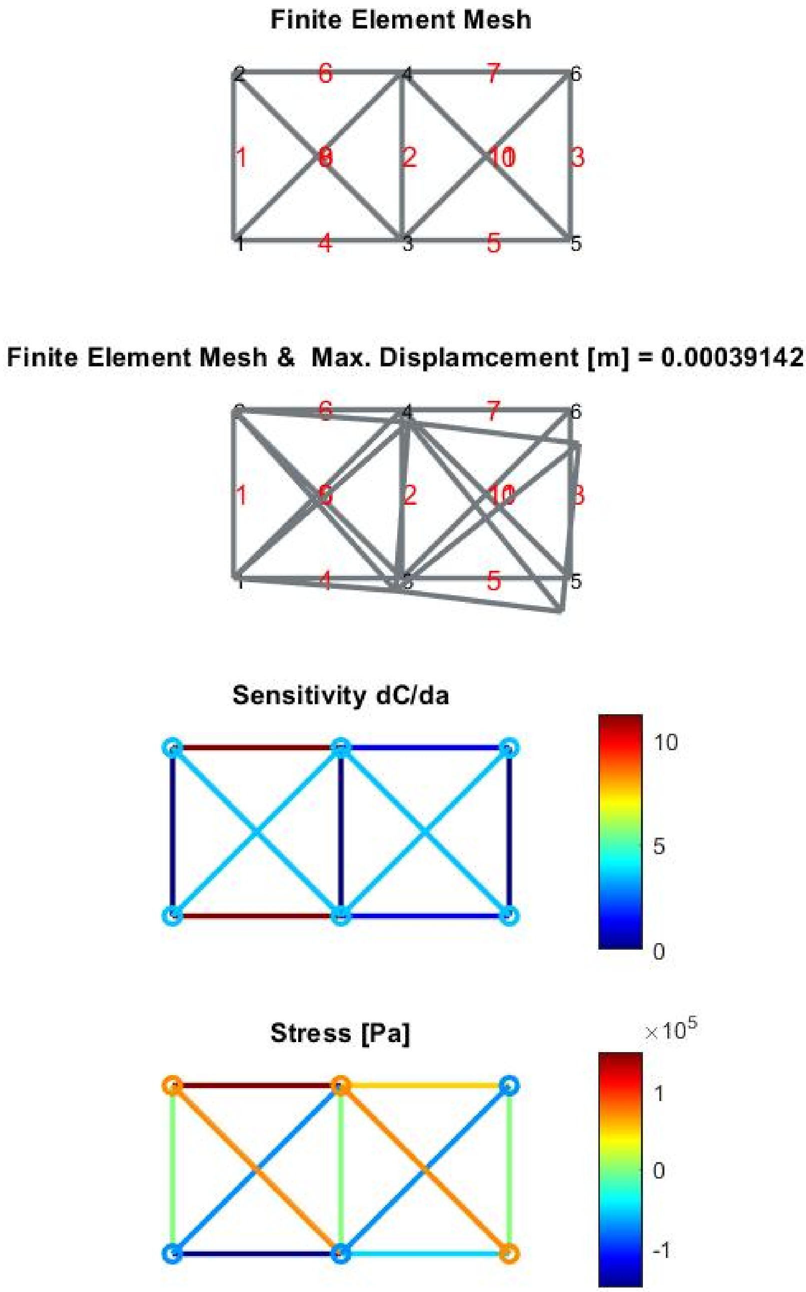


Figure 14: Mesh, and responses of the truss structure

- $ECC = \rho_e A_e L_e ECC_e$ the Embodied Carbon;
- ECC_e the elemental Embodied Carbon Coefficient.

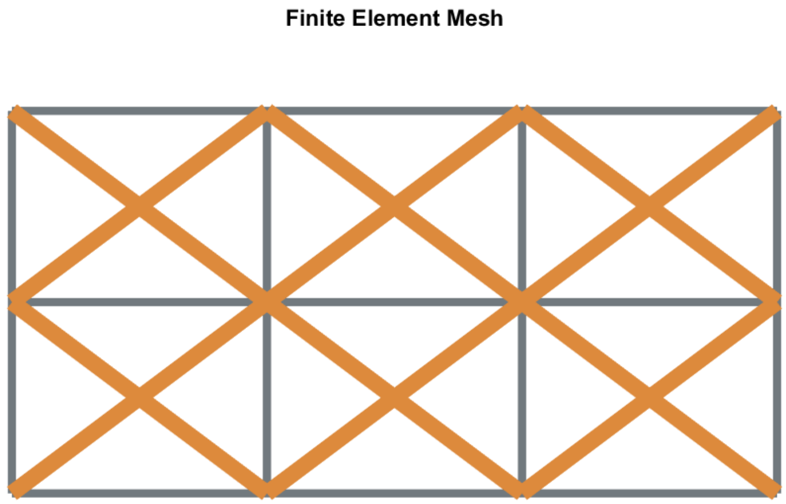


Figure 15: Steel(gray)/Timber(brown) truss

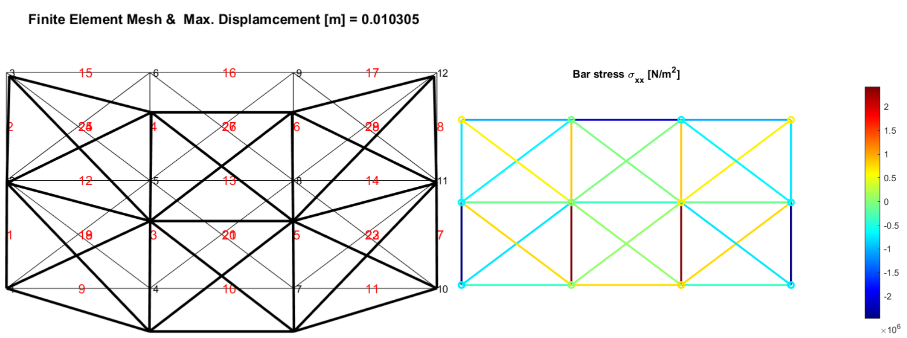


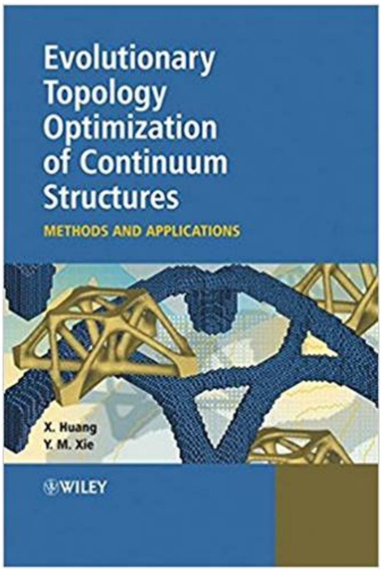
Figure 16: Displacements and Stress of Steel/Timber Truss

3.6 Evolutionary Compliance Optimization: An heuristic Method

in this section we perform a single-material truss topology optimization for minimum compliance with weight and box constraints. The limits A_{min} and A_{max} cross section area variables are defined by the user. Formulation and sensitivity analysis, algorithm, implementation and tests are performed.

To start, we use a Evolutionary Structural Optimization method, with following characteristics:

- Method was inspired in the nature evolution of the plants and natural structures: Bones, plants, etc
- It can be applied for different Criteria: Stress, Compliance, Natural frequencies, etc. We try to expanded to use this method for Eco-design functions.
- This version of the Evolutionary method have been proposed by Xie, Stevens and Huang [3], [4].
- The method is intuitive and simple to implement.
- The version allow add and remove Material, changing the cross section Area of the bars.
- The material must be gradually removed.
- The variations are made by constant discrete values of the cross section Area A_i.
- The rejection criteria is based in the sensitivity analysis.



In the discrete form, the problem of compliance C minimization with weight constraint g_w, can be written as:

$$\begin{array}{ll} Minimize & C = \mathbf{F_g}^T \mathbf{u_g} \\ Subjectto & \mathbf{K_g}(A_e)\mathbf{u_g} = \mathbf{F_g} \\ & g_w = \sum_{e=1}^{nel} A_e L_e \rho_e \leq W_{lim} \\ & A_{min} \leq A_e \leq A_{max} \end{array} \tag{27}$$

where,

- C is the Compliance (objective function);
- $\mathbf{F_g}$ is the nodal force vector;
- $\mathbf{u_g}$ is the nodal displacement vector;
- $\mathbf{K_g}(A_e)$ is the global stiffness matrix ;
- A_e is the elemental cross section area;
- L_e is the elemental length;
- ρ_e is the material density of the element e;
- W_{lim} is the limit value of the total weight of the truss;
- A_{min} and A_{max} are the box constraints, lower and upper limits.

The Compliance minimization is equivalent to Stiffness maximization problem. The objective function of this problem is:

$$C = \frac{1}{2}\mathbf{F_g}^T\mathbf{u_g} \tag{28}$$

where C is the structure Compliance, $\mathbf{F_g}$ is the constant external loads and $\mathbf{u_g}$ is the structural displacement. We consider that the structure is modeled by Finite Element Method, and the static equilibrium equation can be written by:

$$\mathbf{K_g}\mathbf{u_g} = \mathbf{F_g} \tag{29}$$

Introducing the design variable A_e, as area cross section for the element e, the strain energy can be re-written, for the cases where the stiffness matrix is symmetric, by:

$$C = \frac{1}{2}\mathbf{F_g}^T\mathbf{u_g}(A_e) = \mathbf{u_g}(A_e)^T\mathbf{K_g}(A_e)\mathbf{u_g}(A_e) \tag{30}$$

where A_e is the vector of bars cross section area, e indicate a specific element, nel is total number of elements. To simplify the notation we use $\mathbf{K_g}(A_e) = \mathbf{K_g}$ and $\mathbf{u_g}(A_e) = \mathbf{u_g}$. But, since $\mathbf{K_g}\mathbf{u_g} = \mathbf{F_g}$ if $\mathbf{F_g}$ is constant, than, the first derivative of the equilibrium equation with respect to the design variables A_e is:

$$\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} + \mathbf{K_g}\frac{d\mathbf{u_g}}{dA_e} = 0 \tag{31}$$

than:

$$\mathbf{K_g}\frac{d\mathbf{u_g}}{dA_e} = -\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} \tag{32}$$

and,

$$\frac{d\mathbf{u_g}}{dA_e} = -\mathbf{K_g}^{-1}\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} \tag{33}$$

And calculating the derivative of C we have:

$$\frac{dC}{dA_e} = \frac{1}{2}(2 * \mathbf{u_g}^T\mathbf{K_g}\frac{d\mathbf{u_g}}{dA_e} + \mathbf{u_g}^T\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g}) \tag{34}$$

Replacing the equation (33) in (34) we have:

$$\frac{dC}{dA_e} = \frac{1}{2}(-2 * \mathbf{u_g}^T\mathbf{K_g}\mathbf{K_g}^{-1}\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} + \mathbf{u_g}^T\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g}) \tag{35}$$

and we arrive to:

$$\frac{dC}{dA_e} = -\frac{1}{2}\mathbf{u_g}^T\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} \tag{36}$$

The stiffness Matrix variation with respect to the element surface A_e can be done by:

$$\frac{d\mathbf{K_g}}{dA_e} = \frac{1}{A_e}\mathbf{K_e} \tag{37}$$

or, in an displayed form,

$$\frac{d\mathbf{K_g}}{dA_e} = \frac{E}{L}\begin{bmatrix} \lambda^2 & & & \text{Sim.} \\ \lambda\mu & \mu^2 & & \\ -\lambda^2 & -\lambda\mu & \lambda^2 & \\ -\lambda\mu & -\mu^2 & \lambda\mu & \mu^2 \end{bmatrix} \tag{38}$$

where $\mathbf{K_e}$ is the stiffness matrix of an element e. The variation of the nodal displacements is $\frac{d\mathbf{u_g}}{dA_e}$, written by:

$$\frac{d\mathbf{u_g}}{dA_e} = \mathbf{K_g}^{-1}\frac{d\mathbf{K_g}}{dA_e}\mathbf{u_g} \tag{39}$$

Using equation (39), we can use the quadratic form, as a positive number that represent the variations of the objective function C with respect to A_e:

$$\frac{dC}{dA_e} = \frac{1}{2}\mathbf{u_e}^T\mathbf{K_e}\mathbf{u_e} \tag{40}$$

where $\mathbf{u_e}$ is the displacement vector of one element e. In this contest, to minimize the compliance, we need decreased the A_e of the elements with the lowest values of $\frac{dC}{dA_e}$ and increase the area cross section of the elements with the highest value of the sensitivity function.

The algorithm for update the design variables, in this case the cross section elemental area A_e is:

3.6.1 Compliance - Evolutionary Algorithm

The Evolutionary Algorithm can be stated as:

1. Discretize the design domain using a finite element method using bars elements. Impose boundary conditions and loads;
2. Solve the static structural analysis, finding the nodal displacements;
3. Calculate the sensitivity number $\frac{dC}{dA_e}$, using the equation 40,
4. Reduce the cross section area A_e by a constant value $A_e - \delta A_e$ for a fixed number of elements with smaller $\frac{dC}{dA_e}$ values,
5. Increase the cross section area A_e by a constant value $A_e + \delta A_e$ for a fixed number of elements with higher $\frac{dC}{dA_e}$ values,
6. Repeat pass 2 and 5 until reach the final volume or an limit value for the compliance.

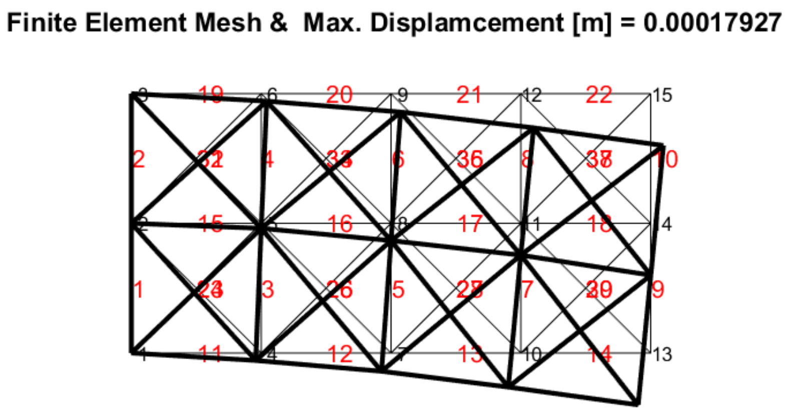


Figure 17:

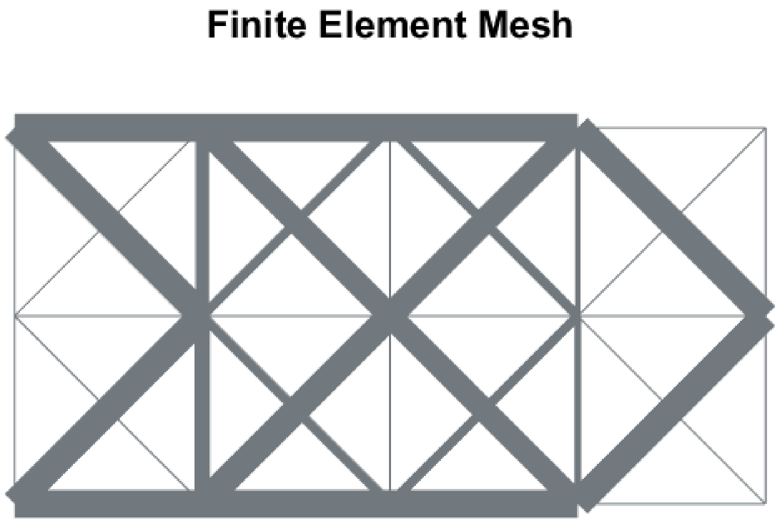


Figure 18:

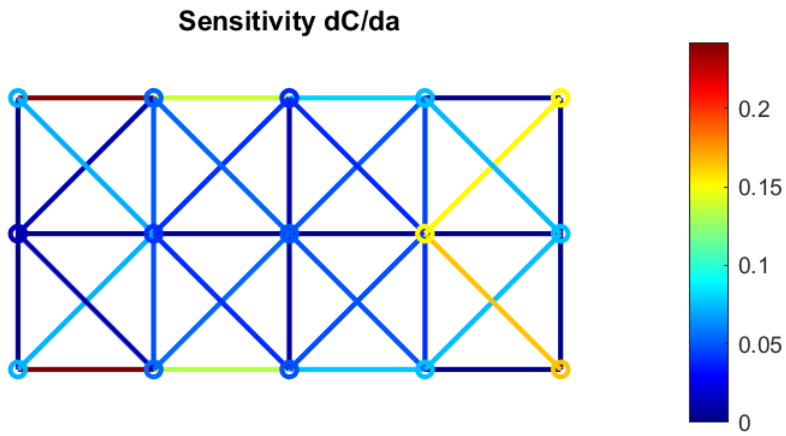


Figure 19:

3.7 Two-material truss topology optimization for minimum compliance with weight and/or Embodied Carbon constraints. Formulation and sensitivity analysis. Algorithm, implementation and testes.

To implement a two materials models, we need to use a pseudo density variable define as x_e one value per element. We consider that, if $x_e = 0$ means that the element is of timber and $x_e = 1$ means that the element is of steel.

$$
\begin{array}{ll}
Minimize & C = \mathbf{F_g}^T \mathbf{u_g} \\
Subjectto & \mathbf{K_g}(A_e, x_e)\ \mathbf{u_g} = \mathbf{F_g} \\
& g \leq g_{lim} \\
& A_{min} \leq A_e \leq A_{max} \\
& 0 \leq x_e \leq 1
\end{array}
\tag{41}
$$

where A_e and x_e are the design variables. The equation (41) can be solve with two different constraints, $g = g_W$ the weight constraint and $g = g_{EC}$ the global warning potential. The g_{EC} is calculated by:

$$
g_{EC} = \sum_{e=1}^{nel} = A_e l_e (\rho_e ECC_e) \tag{42}
$$

and the limit values are W_lim and ECC_lim. To simulate the material variations we use a material interpolation model. Here, the we use the SIMP model to interpolate the material. The SIMP model has been proposed by Sigmund [5].

$$
\begin{array}{l}
E_e = (x_e)^p \Delta E + E_{timber} \\
\rho_e = (x_e)^p \Delta \rho + \rho_{timber} \\
(\rho_e ECC_e) = (x_e)^p \Delta(\rho ECC) + (\rho ECC)_{timber}
\end{array}
\tag{43}
$$

where:

$$
\begin{array}{l}
\Delta E = E_{steel} - E_{timber} \\
\Delta \rho = \rho_{steel} - \rho_{timber} \\
\Delta(\rho ECC) = (\rho ECC)_{steel} - (\rho ECC)_{timber}
\end{array}
\tag{44}
$$

3.8 Two material trusses with stress constraints. Minimization of the global Embodied Carbon subject to stress constraints. Formulation and sensitivity analysis. Algorithm, implementation and testes.

Under development.

References

[1] Demi Fang, Nathan Brown, Catherine De Wolf, and Caitlin Mueller. Reducing embodied carbon in structural systems: A review of early-stage design strategies. 76:107054.

[2] Ernest Ching and Josephine V. Carstensen. Truss topology optimization of timber–steel structures for reduced embodied carbon design. 252:113540.

[3] Y M Xie and G P Steven. *Evolutionary Structural Optimization*. Springer-Verlag, London, 1997.

[4] Xiaodong Huang and Mike Xie. *Evolutionary Topology Optimization of Continuum Structures: Methods and Applications*. Wiley, 2010.

[5] O Sigmund and K Maute. Topology optimization approaches. *Struct. Multidisc. Optim.*, -:–, 2013.