

OPTIMIZATION OF BRIDGE STRUCTURE

Introduction

In this project we are going to optimize simply supported beam which we can consider as bridge structure in terms of height and compliance. In this project, a simply supported beam is subjected to a uniformly distributed load. As the beam is symmetric around y axis, we will consider half right side of the beam to optimize. The beam is subjected to uniformly distributed load in downward y direction. The beam after deformation moves in the downward direction on the left side and horizontally towards right on the other support, as there is roller support in x direction at left side of the beam and in y direction at bottom right of the beam. Overview of the three stages of this project is given below.

Stage I: Topology Optimization

We are going to use topology optimization to minimize the compliance of the structure in first stage. By using top.m code we will get geometry which will have less compliance and we can interpret this geometry to optimize it further. This will be like conceptual design of the beam.

Stage II: Weight Optimization

In second stage we will optimize weight of the structure. If we change the height of the cross section it will change the deflection of the beam hence taking height of the cross section of the beam as design variable, we will use FMINCON function in MATLAB to get optimized weight of the beam by considering maximum allowable stress as 250MPa. This stage is similar like detailed designing. We use CALFEM MATLAB code to find the elemental forces instead of finding it analytically. CALFEM will give stiffness matrix and displacement of each element.

Stage III: Sensitivity Analysis

In sensitivity analysis we are going to use gradients of stress with respect to design variable h to optimize the weight of the structure. This will be third stage of the project. Here we will use sensitivity of the stress with respect to design variable as constraint.

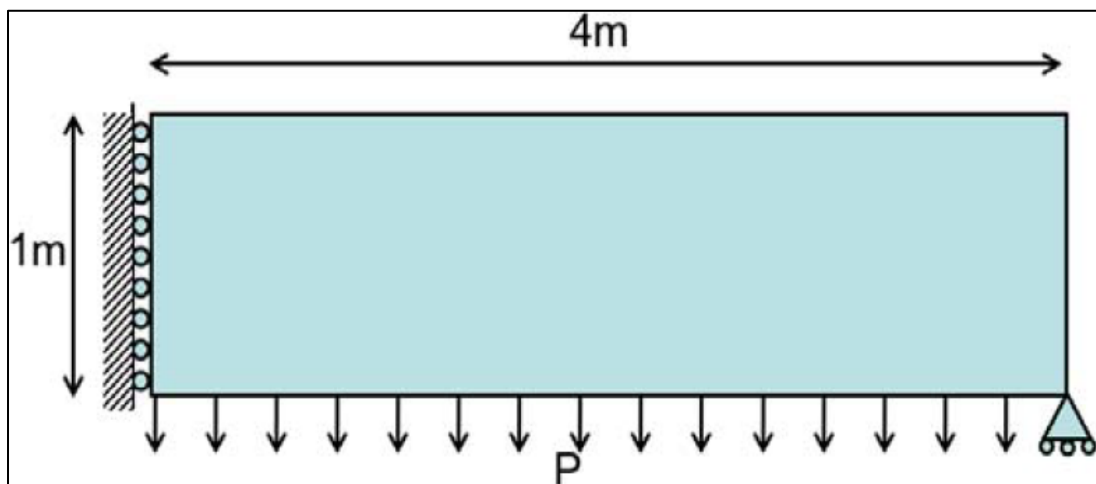


Figure1: Bridge structure

Material Properties:

Density=7835 kg/m³ ,Young's modulus=200Gpa,

Thickness=0.02m, Volume fraction=0.49,

P=20kN/m, Poisson's ratio=0.49

Stage I Topology Optimization

In Topology optimization we used 99-line code Top.m. In that as default boundary conditions and force is for simply supported beam with tip load force. So, we need to change the boundary condition and force.

In this problem, distributed force is applied at bottom of the beam. So, following are the equations which we will modified in Top.m code.

All nodes will share the force, so only at the end nodes force will be 0.5. So, force on all the nodes on the bottom surface will be 1. By finding locations of the nodes of the bottom surface we will get following equations,

$$F\left(\left(2 * (nely + 1): 2 * (nely + 1): (2 * (nelx + 1) * (nely + 1))\right), 1\right) = -1$$

$$F(2 * (nely + 1), 1) = -0.5$$

$$F\left(\left(2 * (nelx + 1) * (nely + 1)\right), 1\right) = -0.5$$

Where nelx is number of elements in x direction and nely is number of elements in y direction.

This beam has fixed at left end, but it can move in y direction and at right end it is fixed in y direction, it can move in x direction. So, fixing those degree of freedom in the Top.m.

$$fixeddofs = union([1: 2: 2 * (nely + 1)], [2 * (nelx + 1) * (nely + 1)])$$

Input for Top.m in MATLAB is given as

Matlab command

`top(nelx , nely , f , p , r )`

↑ ↑ ↑
Volume fraction Exponent Oscillation control

Results:

Volume fraction is considered 0.49, exponent is 3 and for different mesh size with keeping aspect ratio 4:1 and for different oscillation control value we got the results as below

Effect of discretization:

- It is clear from results that fine mesh will give more clear geometry of the structure.
- If we use fine mesh even though we will get more detailed output, but it might not be manufacturable.
- For coarse mesh ,output will be blurred and indistinguishable.
- Although fine mesh size will give us better output in terms of clarity and detailing, but it will take time to give results.

Effect of oscillation control parameter:

- Oscillation parameter will affect clarity of the topology output. Increasing value of rmin will give better results.
- For rmin 0.5 and 1 compliance is almost similar.
- As from results increasing rmin for same mesh size we will get more clear results for topology design.
- If we use 160*40 mesh size with rmin 1.5 , results what we get that is not manufacturable so, choosing 80*20 mesh size with rmin 1.5 design.

h1	h2	h3	h4	h5	h6	h7	h8	h9	h10	h11	h12
0.2	0.2	0.1	0.1	0.15	0.15	0.2	0.2	0.1	0.1	0.15	0.1

Initial height from topology optimization output

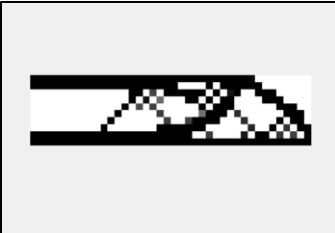
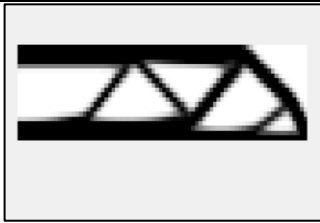
Mesh size	Oscillation control($r_{\min}$)	0.5	1	1.5
40x10				
80x20				
160x40				

Table 1: Results for Topology Optimization for different mesh size and $r_{\min}$

Stage II Weight optimization

First, we will interpret the results we will get it from topology optimization. As shown in the figure 2, defining elements and nodes for the beam. We found the lengths and co-ordinates of the nodes by visual inspection. Table 2 explains elements, nodes and their co-ordinates and length of each beam element. As to calculate the optimized weight of the beam we must define separate 3 functions in MATLAB. First will be objective function which consists our objective which optimization of weight. Second will be constraint function which will have CALFEM code, equality and inequality constraints of stress. CALFEM code will be used to find stiffness matrix, elemental force and displacements of each element which we consider as beam element. Third function will be main FMINCON function where we will call both objective function and constraint function inside FMINCON function which will be used to achieve optimized height of the cross section of the beam. Also plotting of the deformed and undeformed geometry with using CALFEM functions will be written in main FMINCON function.

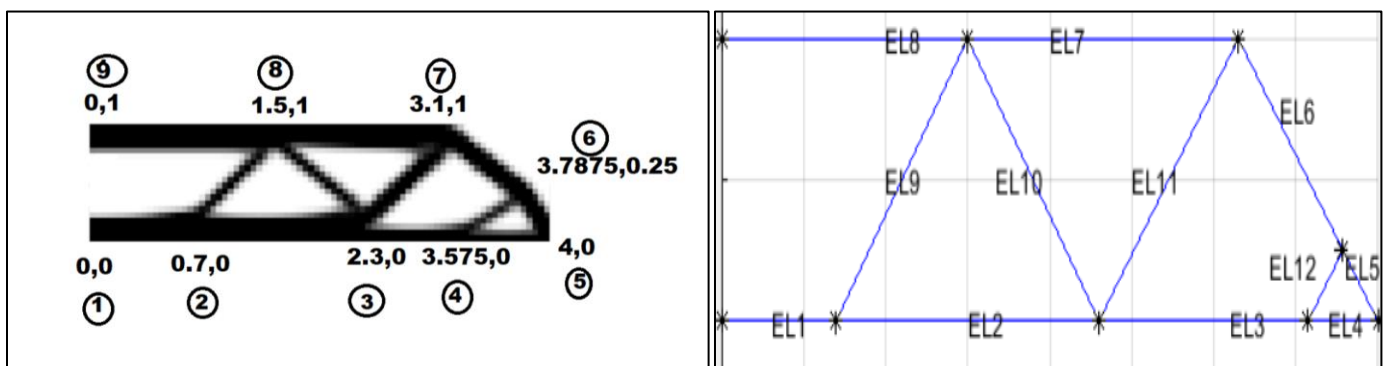


Figure 2: Topology optimization output with co-ordinates of nodes and elements

Element	Nodes	Degrees of Freedom	Node 1	Node 2	Length of Element
1	1-2	1,2,3,4,5,6	0,0	0.7,0	0.7
2	2-3	4,5,6,7,8,9	0.7,0	2.3,0	1.6
3	3-4	7,8,9,10,11,12	2.3,0	3.575,0	1.275
4	4-5	10,11,12,13,14,15	3.575,0	4,0	0.425
5	5-6	13,14,15,16,17,18	4,0	3.7875,0.25	0.32811
6	6-7	16,17,18,19,20,21	3.7875,0.25	3.15,1	0.98433
7	7-8	19,20,21,22,23,24	3.15,1	1.5,1	1.65
8	8-9	22,23,24,25,26,27	1.5,1	0,1	1.5
9	2-8	4,5,6,22,23,24,	0.7,0	1.5,1	1.281
10	3-8	7,8,9,22,23,24,	2.3,0	1.5,1	1.281
11	3-7	7,8,9,19,20,21	2.3,0	3.15,1	1.31
12	4-6	10,11,12,16,17,18,	3.575,0	3.7875,0.25	0.32811

Table 2: Elements and Nodes Numbering

- After interpreting co-ordinates by using output of topology optimization we can create beam model of this in MATLAB. There are total 12 elements and 9 nodes . Defining degree of freedoms for each element. As each node will have six DOF, so stiffness matrix will be 27x27. Distributed force in y direction is there so defining force by using function eq.

$$eq1=[0 \ -20000]$$

- Defining material properties for beam such as young's modulus and area, Moment of Inertia for each element.

$$E=200GPa \ , \quad A = w * h \quad , \quad I = w * \frac{h^3}{12}$$

For example, for element 1 we can write it as $ep1=[2e11 \ A(1) \ I(1)]$.

- Defining elements as ex and ey for each element. For example, for 1st element it will be given as,

$$ex1=[0 \ 4/3], \quad ey1=[0 \ 0]$$

- Using beam2e function to compute stiffness matrix of each element. For element with distributed force we must consider distributed load as follows,

$$[Ke1,fe1]=beam2e(ex1,ey1,ep1,eq1)$$

And for elements without distributed force, we can define as

$$Ke5=beam2e(ex5,ey5,ep5)$$

- For assembly of K matrix following will be the format for distributed force

$$[K, f]=assem(Edof(4,:),K,Ke4,f,fe4)$$

- And for elements without distributed force,

$$K=assem(Edof(5,:),K,Ke5)$$

- Boundary conditions can be given as $bc=[1 \ 0;14 \ 0;25 \ 0]$. As there will not be x-directional movement for node 1 and no y directional movement for node 5 and no x directional movement for node 9.
- From the stiffness matrix and force we can calculate the displacement by using solveq function.
- We can extract this information by using extract function. Also, we can compute elemental forces by using beam2s function.
- To optimize the weight, we are using FMINCON function in MATLAB. Weight can be calculated as

$$Mass = Density * Volume$$

$$Density = 7835 \ kg/m^3$$

$$Volume = w * (h_1 * l_1 + h_2 * l_2 + h_3 * l_3 + h_4 * l_4 + h_5 * l_5 + h_6 * l_6 \\ + h_7 * l_7 + h_8 * l_8 + h_9 * l_9 + h_{10} * l_{10} + h_{11} * l_{11} + h_{12} * l_{12})$$

So objective function will be

$$f=(0.7*h(1)+1.6*h(2)+1.275*h(3)+0.425*h(4)+0.32811*h(5)+0.98433*h(6)+1.65*h(7)+1.5*h(8)+1.281*h(9)+1.281*h(10)+1.31*h(11)+0.32811*h(12))*0.02*7835$$

- Here we are optimizing height of the cross section of the beam, rest of the parameters in the weight of the beam are constant. So, this will be objective function in FMINCON subject to stress constraint. Stress in each element can be calculated by using elemental force which we calculated by CALFEM as mentioned earlier. As there will be 6 forces, we will get for both end of each beam element as follows,

$$\text{Force} = [N, V, M]$$

N=Normal force, V=Shear force, M=Moment

- Considering forces on bottom surface of the beam and calculating stress accordingly. Given allowable axial stress is 250 MPa. For a beam stress due to element axial force and stress due to moment on the beam. Hence total sum of the forces should be within allowable axial stress limit. It can be stated as below

$$\frac{N}{A} + \frac{-My}{I} \leq 2.5 * 10^8$$

- There will total 12 constraints for each element. So, all the CALFEM functions and stress constraints will be in constraint function.
- Let's assume initial height is 0.5 m and lower bound as 0.001 m for each element. In FMINCON function using 'sqp' algorithm and make constraint tolerance equal to 1e-12.

options = optimoptions(@fmincon,'Display','iter','Algorithm','sqp','ConstraintTolerance',1e-12)

- FMINCON function can be stated as below,

[h,fval,exitflag,output,lambda,grad,hessian]=fmincon(@objfun5,h0,[],[],[],[],lb,[],@confun5,options)

- To plot the deformed and undeformed geometry we can use eldisp2 and eldraw2 functions of CALFEM respectively.

Results:

- We will get plot for deformed and undeformed geometry of the structure as shown Figure 3.
- Also, we are getting optimized height of each element and weight of the beam.
- Optimized weight of the structure=84.9911 kg with exit flag 1.
- Optimized Height of each element in meters.

h1	h2	h3	h4	h5	h6	h7	h8	h9	h10	h11	h12
0.0943	0.0937	0.0925	0.0826	0.0282	0.0282	0.0248	0.032	0.0082	0.0079	0.0202	0.001

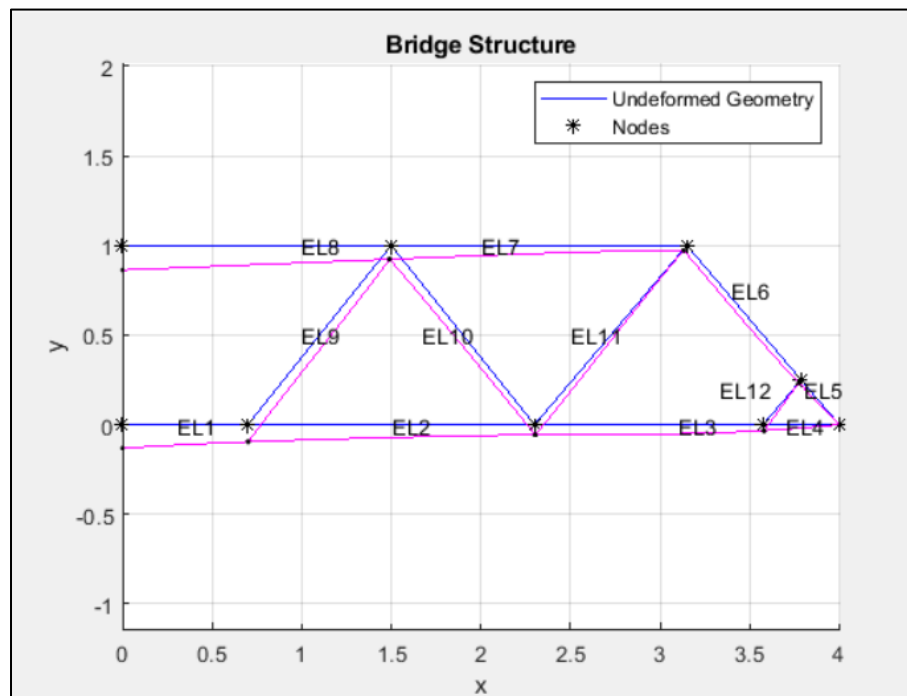


Figure 3: Deformed and Undeformed geometry

Iter	Func-count	Fval	Feasibility	Step Length	Norm of step	First-order optimality
0	12	1.190533e+02	2.917e+08	1.000e+00	0.000e+00	2.586e+02
1	29	9.749611e+01	2.049e+08	3.430e-01	6.178e-02	1.801e+02
2	43	8.066202e+01	1.272e+08	7.000e-01	1.018e-01	2.046e+02
3	57	7.831926e+01	2.585e+08	7.000e-01	8.581e-02	2.112e+02
4	71	7.627392e+01	2.524e+08	7.000e-01	3.977e-02	1.087e+03
5	86	7.841853e+01	1.766e+08	4.900e-01	9.555e-03	2.949e+02
6	99	8.183891e+01	5.873e+07	1.000e+00	1.348e-02	2.586e+02
7	112	8.373697e+01	2.122e+07	1.000e+00	1.281e-02	1.393e+02
8	125	8.529694e+01	4.937e+06	1.000e+00	8.903e-03	2.586e+02
9	142	8.531355e+01	9.503e+05	3.430e-01	1.595e-02	2.586e+02
10	160	8.474368e+01	4.111e+06	1.681e-01	9.559e-03	2.137e+02
11	186	8.475172e+01	4.099e+06	9.689e-03	7.304e-04	2.136e+02
12	200	8.560166e+01	9.257e+05	1.000e+00	1.830e-02	2.586e+02
13	224	8.549133e+01	1.007e+06	1.977e-02	1.830e-03	2.534e+02
14	248	8.539157e+01	1.100e+06	1.977e-02	2.001e-03	2.533e+02
15	273	8.533146e+01	1.148e+06	1.384e-02	1.555e-03	2.499e+02
16	297	8.526248e+01	1.197e+06	1.977e-02	1.499e-03	2.449e+02
17	321	8.520243e+01	1.235e+06	1.977e-02	1.153e-03	2.400e+02
18	345	8.514869e+01	1.276e+06	1.977e-02	1.169e-03	2.352e+02
19	369	8.510198e+01	1.320e+06	1.977e-02	1.184e-03	2.305e+02
20	394	8.507472e+01	1.338e+06	1.384e-02	8.511e-04	2.273e+02
21	418	8.505032e+01	1.353e+06	1.977e-02	9.074e-04	2.228e+02
22	440	8.502540e+01	1.367e+06	4.035e-02	1.182e-03	2.138e+02
23	458	8.499306e+01	1.391e+06	1.681e-01	2.238e-03	1.773e+02
24	471	8.498600e+01	2.766e+05	1.000e+00	3.425e-03	2.363e+01
25	484	8.496839e+01	2.716e+05	1.000e+00	4.149e-03	1.667e+01
26	503	8.497011e+01	2.136e+05	1.681e-01	7.167e-04	4.336e+01
27	516	8.498855e+01	2.962e+04	1.000e+00	1.188e-03	8.959e-01
28	529	8.499108e+01	7.615e+02	1.000e+00	1.621e-04	1.834e-01
29	542	8.499114e+01	3.112e+00	1.000e+00	9.552e-06	6.804e-03
Iter	Func-count	Fval	Feasibility	Step Length	Norm of step	First-order optimality
30	555	8.499114e+01	7.035e-02	1.000e+00	1.523e-06	3.080e-04
31	568	8.499114e+01	2.444e-06	1.000e+00	2.207e-11	1.837e-05

Iteration History

[Local minimum found that satisfies the constraints.](#)

Optimization completed because the objective function is non-decreasing in [feasible directions](#), to within the default value of the [optimality tolerance](#), and constraints are satisfied to within the selected value of the [constraint tolerance](#).

[<stopping criteria details>](#)

Stresses in each member force :

1.0e+08 *|

2.5000	2.5000	2.1423	2.0215	2.5000	2.5000	2.5000	2.5000	2.5000	2.5000	2.5000	0.6061
--------	--------	--------	--------	--------	--------	--------	--------	--------	--------	--------	--------

Height of each member force:

0.0943	0.0937	0.0925	0.0826	0.0282	0.0282	0.0248	0.0320	0.0082	0.0079	0.0202	0.0010
--------	--------	--------	--------	--------	--------	--------	--------	--------	--------	--------	--------

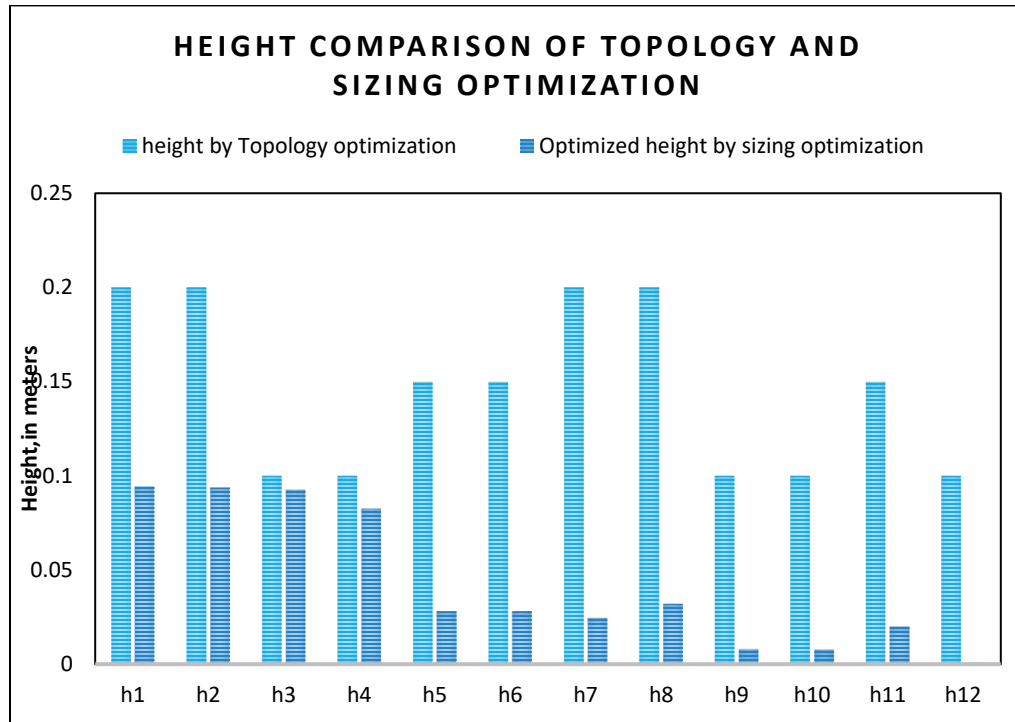
fval =

84.9911

Output of the FMINCON Function

Topology Design	Sizing design
304.37kg	84kg

Table 2: Comparison of topology optimization and sizing optimization



Stage III Sensitivity Analysis

In this stage we calculated sensitivity of stress with respect of height of the cross section of the beam. That sensitivity we can use in FMINCON to calculate optimized weight of the beam. Hence our main objective function will not change which is minimizing the weight of the beam. So, this will be objective function. For Constraint function there will be steps to compute sensitivity by using CALFEM code. We are supposed to consider axial stresses in the element, both the normal force and bending moment, therefore the stress in the beam will be combination of both the stresses.

- To compute stiffness matrix, elemental forces and displacement we can use the same code which is explained in stage II.
- Sensitivity of the stress with respect to design variable h can be calculated by direct differentiation method as,

$$\frac{d\sigma}{dh} = \frac{\partial\sigma}{\partial h} + \frac{\partial\sigma}{\partial d} * \frac{dd}{dh}$$

In this explicit term is $\frac{\partial\sigma}{\partial h}$ and $\frac{dd}{dh}$ is implicit term.

- To find the implicit term $\frac{dd}{dh}$ we know

$$\{F\} = [K]\{d\}$$

where d = displacement, F = force applied, K = stiffness matrix

Differentiating this equation with respect to h ,

$$\frac{d[K]}{dh} * \{d\} + \frac{dd}{dh} * [K] = \frac{dF}{dh}$$

From this equation we can get the implicit term.

- To find $\frac{d[K]}{dh}$ we must consider the bar plus beam component of stiffness matrix which is 6x6 matrix can be given as

$$\bar{K}^e = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}$$

Differentiating each term w.r.t. h we will get K' =

$$\begin{array}{ccccc} \frac{E * w}{L} & 0 & 0 & -\frac{E * w}{L} & 0 & 0 \\ 0 & \frac{3 * E * w * h^2}{L^3} & \frac{1.5 * E * w * h^2}{L^2} & 0 & \frac{-3 * E * w * h^2}{L^3} & \frac{1.5 * E * w * h^2}{L^2} \\ 0 & \frac{1.5 * E * w * h^2}{L^2} & \frac{12 * E * w * h^2}{L} & 0 & \frac{-1.5 * E * w * h^2}{L^2} & \frac{6 * E * w * h^2}{L} \\ -\frac{E * w}{L} & 0 & 0 & \frac{E * w}{L} & 0 & 0 \\ 0 & \frac{-3 * E * w * h^2}{L^3} & \frac{-1.5 * E * w * h^2}{L^2} & 0 & \frac{3 * E * w * h^2}{L^3} & \frac{-1.5 * E * w * h^2}{L^2} \\ 0 & \frac{1.5 * E * w * h^2}{L^2} & \frac{6 * E * w * h^2}{L} & 0 & \frac{-1.5 * E * w * h^2}{L^2} & \frac{12 * E * w * h^2}{L} \end{array}$$

- Finding K' for each element. And then we can use CALFEM to assemble these in one matrix same as we did it in stage II for example for 1st element

$$Kdglobal1 = assem(Edof(1,:), Kdglobal, Kd1)$$

$$[K, f] = assem(Edof(1,:), K, Ke1, f, fe1)$$

Applying boundary conditions to K' matrix of each element. This is important step as we remove DOF which are zero. So, we have three DOF which are zero. Hence K' will be 24*24.

- As distributed load is not going to dependent on height since we are considering sizing parameter so $\frac{dF}{dh} = 0$.
- We can compute the dd/dh term by given formula,

$$\frac{dd}{dh} = [K]^{-1} * \left(-\frac{d[K]}{dh} * \{d\} \right)$$

- To find stress derivative w.r.t. height we must consider stress due to normal and bending force separately and then we can add them later.

First for bar element, using 'for loop' we can compute derivative of stress in bar element w.r.t. d. It can be computed by,

$$\frac{\partial \sigma}{\partial d} = \frac{E}{L} * \frac{dd}{dh}$$

For bar element

- $\frac{\partial \sigma}{\partial h}$ term for bar element is zero as there is no dependence of h in that.
- For beam element,
If $\varphi = \sigma$ due to bending moment then derivative w.r.t. design variable and d can be given as below,

$$\frac{\partial \psi_2}{\partial \mathbf{b}} = \frac{1}{2} \mathbf{EB}(0)^T \mathbf{d}, \quad \frac{\partial \psi_2}{\partial \mathbf{d}} = \frac{b}{2} \mathbf{EB}(0)^T$$

- Using beam interpolation function

$$\mathbf{B}_g = \frac{1}{L^2} [-6 + 12s, L(-4 + 6s), 6 - 12s, L(-2 + 6s)]$$

$$\mathbf{B} = \frac{1}{L^2} [6 - 12s, L(-2 + 6s)]$$

- Product of the two terms $\frac{\partial \sigma}{\partial d} * \frac{dd}{dh}$ can be saved in different matrix for bar and beam element. This product we must add and save it in new matrix which will be our final term $\frac{\partial \sigma}{\partial d} * \frac{dd}{dh}$ which we can save it in A matrix .
- For $\frac{\partial \sigma}{\partial h}$ we can compute stresses for each element by using,

$$\frac{\partial \psi_2}{\partial \mathbf{b}} = \frac{1}{2} \mathbf{EB}(0)^T \mathbf{d},$$

as mentioned earlier. We can save it in B Matrix.

- Finally, we can add these terms to get sensitivity ,

$$C = A + B$$

Defining stress constraint and gradient constraint in constraint function. This will be part of constraint function.

- Now in Main FMINCON function the only difference is in defining options addition is put grad-constraint is on.

```
options=optimoptions(@fmincon,'Algorithm','sqp','Display','iter','ConstraintTolerance',1e-20,'GradConstr','on','FinDiffRelStep',1e-4)
[h,fval,exitflag,output]=fmincon(@objfun5,h0,[],[],[],[],lb,[],@confun6,options)
```

- This is how we will get our sensitivities which will be matrix of 12x12. In sensitivity matrix each row will be sensitivity of the stress in one element w.r.t. heights of the all elements.

Discussion

- In this project we calculated topology design first which is obtained by minimizing the compliance.
- To make it more realistic and manufacturable we interpret the coordinates and elements for the structure we define them as beam elements in CALFEM code.
- Instead of using FEA software to get values for elemental forces , we compute it from CALFEM code.
- FMINCON using CALFEM with stress constraint will give optimized values of weight and height.
- Performing Sensitivity analysis gives a greater number of function evaluations than function evaluations if you obtain optimized weight without performing sensitivity analysis.

- 5.9E+1 1	- -8E+10	5.17E+ 10	1.96E+ 12	4.37E+ 11	- 6.4E+0 9	- 3.7E+0 9	- 3.1E+0 9	- 1.1E+1 0	- 1.1E+1 0	1.19E+ 09	- 2.3E+1 1
- 8.8E+1 1	- 7.7E+1 0	9.37E+ 09	2.14E+ 12	4.27E+ 11	- 1.2E+1 0	- 3.1E+0 9	- 3.1E+0 9	- 7.3E+0 9	- 1.2E+1 0	- 7.9E+0 9	- 1.7E+1 1
4E+11	- 9.7E+0 9	- 7.3E+1 0	3.85E+ 12	1.25E+ 12	- 1.4E+1 0	- 2.3E+0 8	- 1.7E+0 8	- 1.8E+0 9	- 1.7E+0 9	1.22E+ 08	- 6.6E+1 0
- 1.8E+1 1	1.14E+ 10	- 7.3E+1 0	- 1.5E+1 2	3.64E+ 10	1.99E+ 10	4.78E+ 09	2.61E+ 09	6.38E+ 09	6.2E+0 9	1.6E+0 9	1.49E+ 10
1.8E+1 0	6.73E+ 09	8.67E+ 09	6.5E+1 1	2.87E+ 12	8.9E+1 0	8.08E+ 09	7.46E+ 09	1.14E+ 10	1.32E+ 10	2.25E+ 10	2.89E+ 12
- 1.7E+1 1	1.25E+ 10	- 1.7E+1 0	- 1.1E+1 2	- 3.5E+1 2	- 2.61E+ 10	- 5.12E+ 09	- 6.69E+ 10	- 1.13E+ 10	- 1.86E+ 10	- 5.28E+ 10	- 2.7E+1 2
1.35E+ 11	- 6.2E+0 8	8.73E+ 09	1.32E+ 12	4.06E+ 11	- 5.2E+1 0	- 8.8E+1 0	- 5.8E+1 1	- 9.3E+0 9	- 7.9E+0 9	- 3.7E+1 0	- 2.3E+1 1
2.26E+ 11	1.11E+ 09	1.86E+ 10	3.24E+ 12	1.85E+ 12	- 2.7E+1 0	- 5.1E+1 0	- 4.7E+1 1	- 2.3E+1 0	- 2.5E+1 0	-2E+10	- 2.6E+1 1
- 1.1E+1 2	- 5.3E+1 0	4.97E+ 10	9.05E+ 11	3.33E+ 10	4.09E+ 10	- 7.5E+1 0	- 6.6E+1 1	1.88E+ 09	7.79E+ 09	3.81E+ 10	- 8.2E+1 0
5.28E+ 11	- 1.5E+0 9	3.25E+ 09	2.49E+ 12	4.57E+ 11	3.63E+ 10	- 7.5E+1 0	- 6.6E+1 1	3.06E+ 09	1.43E+ 10	4.68E+ 10	- 4.4E+0 9
4.79E+ 11	- -8E+09	3.55E+ 10	3.69E+ 12	1.59E+ 12	- 4.3E+1 0	4.01E+ 09	9.38E+ 10	1.05E+ 10	2E+10	7.32E+ 10	1.91E+ 11
2.22E+ 11	1.04E+ 09	- 2.4E+1 0	6.15E+ 12	4.46E+ 12	- 1.1E+1 1	5.62E+ 09	6.51E+ 09	7.61E+ 09	9.42E+ 09	2.03E+ 10	2.9E+1 2

Stress Sensitivity Matrix w.r.t h

References

1.A 99-line topology optimization code written in MATLAB :O. Sigmund

<https://pdfs.semanticscholar.org/e933/1af59ba3ac3781ca5f036aefba7274a6e9ed.pdf>

2.CALFEM A FINITE ELEMENT TOOLBOX Version 3.4

www.solid.lth.se/fileadmin/hallfasthetslara/utbildning/kurser/FHL064.../calfem34.pdf