Assignment in
Structural Optimization, 2020

Division of Solid Mechanics

The task is to implement and analyze different optimization methods. The theory related to the methods and the results should be presented in a well structured report. The methods should be implemented in Matlab where use can be made of suitable subroutines included in the CALFEM toolbox. The developed computer code should be attached as an appendix in the report.

Problem description

Topology optimization of a simple geometry should be performed. The L-bracket geometry and associated boundary conditions are illustrated in Fig. 1. A distributed load is applied at the right end. The material is homogeneous, isotropic and linear elastic.
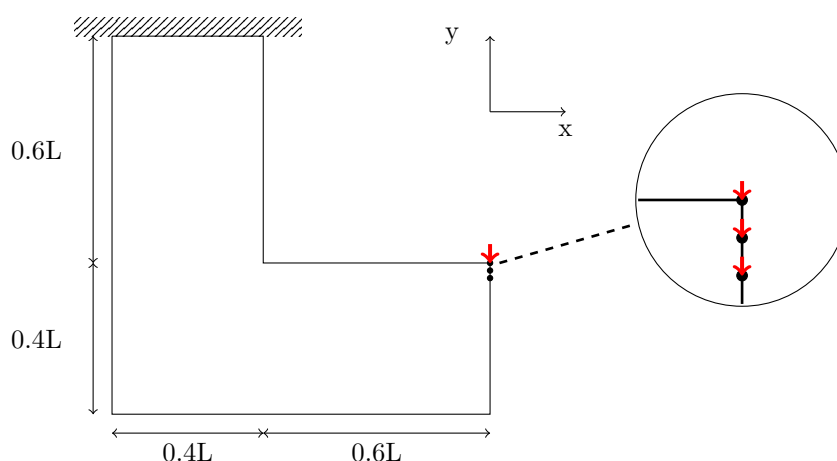

Figure 1: L-bracket with distributed load at the right end.

Problem formulation

The optimization tasks should be solved by the finite element method. The problem should be solved for two finite element discretizations, `LbracketCoarse.mat` and `LbracketFine.mat` (available on the course homepage). First, you should solve the problem for the coarse mesh, `LbracketCoarse.mat`. When you are confident that your implementation works well, solve the same problems for the finer discretization, `LbracketFine.mat`.

In task a), the compliance of the L-bracket in Fig. 1 should be minimized, and in task b), a filtering technique should be implemented to solve the optimization problem in a). In these first two tasks you should implement the CONLIN method for updating the design.

In task c), the L-bracket should optimized by minimizing the maximum element von Mises stress, and you should use the function `mma_solver` (also available on the course homepage) for updating the design. See the Appendix for a description of the function. The density filter implemented in task b) should also be included in this problem.

Plane stress conditions should be used in all optimization problems. The volume of the design domain is V_{box}, defined by Fig. 1 together with $L = 35$ mm and the thickness $t = 1$ mm. Use the material parameters $E = 210$ GPa and $\nu = 0.3$. The magnitude of the external load is given in the input `.mat`-file(s).

Compliance minimization

a) Use the SIMP algorithm to derive a design optimized for maximum stiffness. You should investigate the two different discretizations and different initial design values. The maximum allowed volume of the structure is $V_{max} = 0.35V_{box}$.

Filtering

b) Include a *density filter* in the SIMP algorithm developed in a). The theory describing density filters can be found in e.g. [2]. Examine the influence of the length scale parameter, r. It is sufficient to solve this task only for the finer finite element mesh.

Stress minimization

c) Optimize the structure using stress minimization. The maximum von Mises stress of the structure should be minimized.

Theory

Stress measure

In this task, we are interested in an optimal structure without high stress concentrations, and to that end we minimize the maximum von Mises stress. The von Mises stress, for plane stress conditions, is calculated as

$$\sigma^{vM} = \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + \sigma_{xx}\sigma_{yy} + 3\sigma_{xy}^2}. \tag{1}$$

Because stress is a local quantity, i.e. it is evaluated in each point of the domain (and in this task, in each element centroid), it is difficult to identify and minimize the highest element stress. This is partly due to that, for stress minimization, we need to formulate an objective function that include the stress measures of each point of the domain. One way to aggregate all the N_{elm} von Mises stresses into *one single function* is to use the p-norm of the von Mises stress

$$\sigma_{PN} = \left(\sum_{j=1}^{N_{elm}} (\sigma_j^{vM})^P \right)^{1/P} \tag{2}$$

where $P \geq 1$ is the p-norm exponent and σ_j^{vM} is the von Mises stress computed in the centroid of element j. It can be shown that

$$\lim_{P \to \infty} \sigma_{PN} = \max(\sigma^{vM}), \tag{3}$$

i.e. σ_{PN} approaches the maximum von Mises stress as $P \to \infty$.

Stress, volume and stiffness interpolation

In order to obtain a 0/1 solution, you should use different interpolation schemes for the element stress, volume and stiffness. The theory of these interpolation schemes is described in [1]. Use the SIMP scheme with penalty exponents 0.5, 1 and 3, respectively for the stress, volume and stiffness.

Task

Minimize the p-norm of the von Mises stress, i.e. minimize (2). A recommended value of the p-norm exponent is $P = 14$. Solve the optimization problem for $V_{max} = 0.35V_{box}$. Use the move limit $m = 0.05$ to stabilize the MMA optimizer. Set the lower bound of the design variables to $\delta_o = 10^{-3}$.

Some hints

- Finish the size optimization computer lab BEFORE you start with this assignment.

- It is preferable to scale the objective function and the constraint. Normalize the volume constraint.

- It is crucial that the sensitivities of the objective and constraint functions are correctly computed. One way to verify that the sensitivities are correctly computed is to compare them to numerical sensitivities (see Section 6.1 of the course book).

- The solution of task c), for the finer discretization, is expected to take some time (2 hours or more).

Procedure

The analysis is to be performed in CALFEM. A well structured concise report of your findings should be returned to the Div. of Solid Mechanics no later than **2020-03-13 16.00**. The results should be presented in the form of illustrative graphs and tables. Note that it should be possible to generate the results from the information provided in the report, i.e. numerical parameters used should be clearly stated in the report. MATLAB/CALFEM files (appendix) should be well structured and carefully commented. The reader of the report is assumed to have the same knowledge level as the author before taking the structural optimization course. It is possible to obtain up to 30 points. The task should be solved in groups of *two* (or *individually*). Keep the report as concise as possible. It is strongly recommended that you keep the report well below 10 pages excluding the appendix containing the code.

Submission

You should submit your report in **PDF** to FHLN01@solid.lth.se. In addition to your report you should also attach your m-files. Moreover, a **paper** version should be handed in to the Division of Solid Mechanics.

References

[1] Erik Holmberg, Bo Torstenfelt, and Anders Klarbring. Stress constrained topology optimization. *Structural and Multidisciplinary Optimization*, 48(1):33–47, Jul 2013. ISSN 1615-1488. doi: 10.1007/s00158-012-0880-7. URL https://doi.org/10.1007/s00158-012-0880-7.

[2] Ole Sigmund. Morphology-based black and white filters for topology optimization. *Structural and Multidisciplinary Optimization*, 33(4):401–424, 2007.

Appendix

MMA

The solver `mma_solver` should be called via the command
`[xmma,low,upp] = mma_solver(iOpt,RHO,xold1,xold2,df0dx,fval,dfdx,low,upp,paramVec)`,
where the input arguments are the current optimization iteration `iOpt`, the design variables `RHO`, the design variables one iteration ago `xold1`, the design variables two iterations ago `xold2`, the sensitivity of the objective function `df0dx`, the values of the constraint `fval`, the sensitivity of the constraint (size: $1 \times nelm$), the lower and upper asymptotes `low` and `upp` and the parameter vector $\texttt{paramVec} = \begin{bmatrix} nelm & s_{init} & s_{faster} & s_{slower} & \delta_o & move \end{bmatrix}^T$, where $nelm$ and δ_o is the number of elements and the lower bound of the design variables, respectively. $move$ is the move limit. The output arguments are the updated design variables `xmma`, and the updated lower and upper asymptotes `low` and `upp`.

Use the MMA parameters $s_{init} = 0.5$, $s_{faster} = 1.2$, and $s_{slower} = 0.7$.