Task 3

In this task, structural topology optimization of some 2D structures will be performed using VARTOP method, a topology optimization technique developed by our research group at the UPC. The implementation in MATLAB of this topology optimization approach can be downloaded from VARTOP_GitHub_repository. If you have a GitHub account (and git installed in your computer), you can directly clone the repository to your local machine by executing the following command line:

```
$ git clone https://github.com/DanielYago/UNVARTOP.git .
```

Alternatively, if you do not have a GitHub account, you can simply download the repository in a zip file, by clicking on the green `Code` button and then selecting **`Download ZIP`**. Then, you can manually add all the folders to the MATLAB Search Path or run `install_UNVARTOP.m` script, which will automatically add the folders to the path. This repository provides you with all the MATLAB scripts and functions required to optimize structural problems. In particular, it can be used to maximize the whole stiffness of a structure or design compliant mechanisms. For further information on this topology optimization approach, you can download the paper from the ResearchGate page.

This assignment will be focused on the topological optimization of a set of 2D examples using the provided mathematical tool, with the aim of showing you the potential of these (topology optimization) techniques in the development and design of components in a wide range of applications. In particular, the results obtained from this tool will be used to perform a structural sizing analysis.

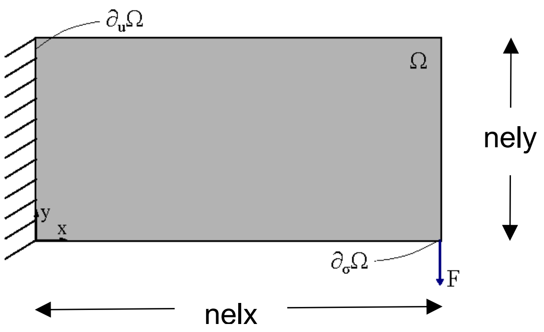
As an end user of this computational tool, you have to define the boundary conditions of the corresponding problem (i.e., the nodes in which the displacements are prescribed as well as the nodes in which the forces are applied), along with the 2D finite element mesh defined with the number of elements in both x and y-axes (unit square finite elements). In addition, other parameters related with the optimization must be defined: initial volume percentage of void material (`Vol0`), target volume percentage of void material (`Vol`), number of steps from `Vol0` to `Vol` (`nsteps`), exponential factor of the volume evolution (`k`) and regularization parameter (`tau`). A general Matlab call of the code would be

```
UNVARTOP_2D_compliance(nelx, nely, nsteps, Vol0, Vol, k, tau)
```

with k normally being equal to 0 and tau equal to 0.5.

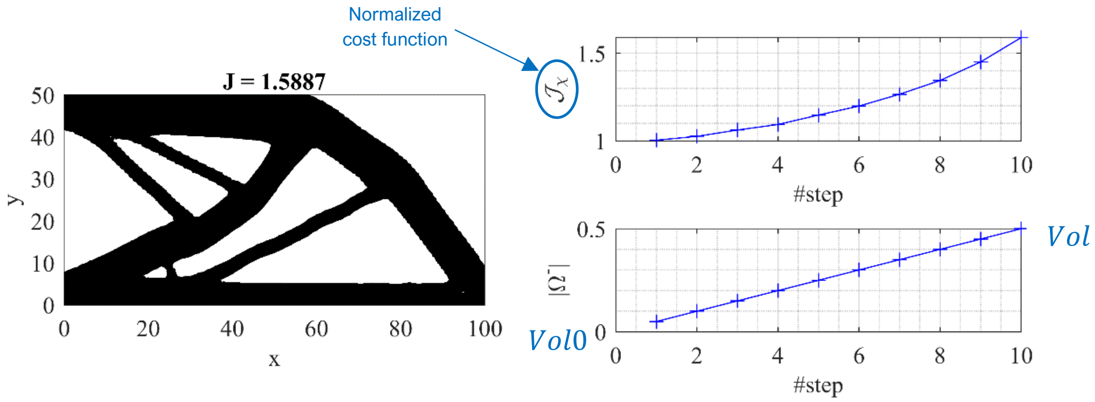
Test example

Consider a cantilever beam with a vertical pointwise load F applied on the bottom right corner of the design domain Ω. The displacements are prescribed on the left side of the domain.

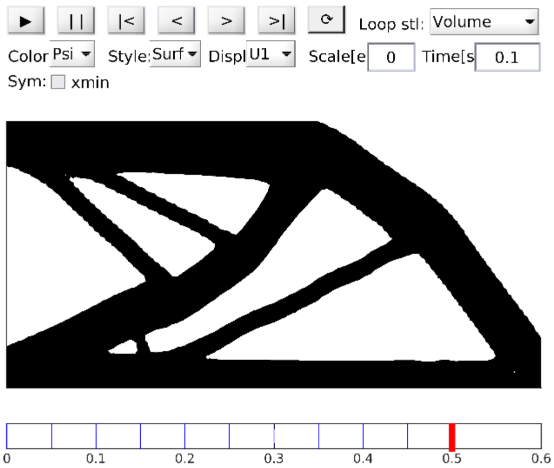


Equivalently, the displacements for both degrees of freedom are prescribed for $x = 0$, and a vertical force $F = 0.01$ is applied at $x = nelx$ and $y = 0$. The list of fixed degrees of freedom can be easily obtained by `2*find(coord(:,1)==0)+(-1:0)`, while the one for the load can be found with `2*find(coord(:,2)==0 & coord(:,1)==nelx)`. This information will be used in the resolution of the equilibrium system.

As a test, run this example with $nelx = 100$, $nely = 50$, $n_{steps} = 10$, $Vol0 = 0$, $Vol = 0.5$, $k = 0$ and $tau = 0.5$. The obtained results should be the same as those shown below:



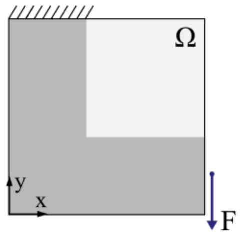
The left figure illustrates the optimal topology for the target volume ($Vol = 0.5$), while the right one displays the evolution of the cost function (expression to be minimized) with the steps (i.e. with an increasing void percentage). A graphical user interface is also generated with the set of optimal solutions where the deformed topology can be displayed, increasing the scale factor. In addition, the design can be animated as a loop with respect to the volume or the scale, by pressing the ⟳ button.



PROBLEM 1: L-shape structure

Consider the design of a simplified hook of aspect ratio 1:1 with a single load applied on the right side of the domain at $y = 0.2$. The nodes on the top-left boundary ($y = 1$ and $x < 0.4$) are fixed (the displacement of the corresponding degrees of freedom is prescribed to 0). In addition, the domain has a prescribed void zone in the top right area, defined by $xi \geq 0.4$ and $yi \geq 0.4$. All the nodes included in this area are listed in `passive_node` and the characteristic function of these elements is therefore set to β.

According to the definition of the previous problem, perform an optimization with 140 elements in each direction in 10 steps, from $Vol0 = 0.36$ to $Vol = 0.75$. Set the exponential parameter k and the regularization parameter tau to 0 and 0.5, respectively.



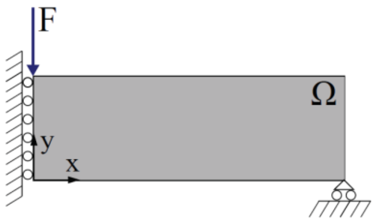

```
F(2*find(coord(:,2)==round(0.2*nely)&coord(:,
1)==nelx),1)=-0.01*nelx;
fixed_dofs = reshape(2*find(coord(:,1) <=
0.4*nelx & coord(:,2)==nely)+(-1:0),1,[]);
active_node = [];
passive_node = find(coord(:,1)>ceil(nelx*0.4)
& coord(:,2)>ceil(nely*0.4));
```

Problem setting **Problem definition**

PROBLEM 2: MBB beam

Optimize half of the MBB beam with an aspect ratio of 2.5 to 1 (x-axis and y-axis, respectively), and a single vertical force F applied at the top-left corner. Symmetry must be assumed on the left side of the domain (horizontal displacement is prescribed to 0) and the vertical displacement at the bottom-right corner is also constrained.

Discretize the domain with 190 and 76 elements in the x-axis and y-axis, respectively. Perform an optimization from $Vol0 = 0$ to $Vol = 0.5$ in 10 equally spaced steps ($k = 0$) and $tau = 1$.



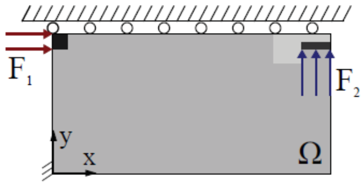

```
F(2*find(coord(:,2)==nely & coord(:,1)==0),1)
= -0.01*nelx;
fixed_dofs = [reshape(2*find(coord(:,1)==0)-
1,1,[]), reshape(2*find(coord(:,1)==nelx &
coord(:,2)==0),1,[])];
active_node = [];
passive_node = [];
```

Problem setting **Problem definition**

PROBLEM 3: Gripper (compliant mechanism)

Use the numerical tool to optimize the design of a compliant mechanism with an aspect ratio 2.5:1, in which the vertical displacement at the output port (jaws of the gripper) is maximized when a horizontal force is applied at the input port (left side of the domain). The domain is supported by a small area in the bottom-left corner ($y < 0.1$ and $x = 0$) and symmetry is applied on the top side of Ω ($y = 1$). A small area in the output port is set to soft material (i.e. included in `passive_node` list) in order to represent the gap in the jaws of the gripper. In addition, some stiff material areas are restricted in both ports (`active_node` list), and an additional spring stiffness must be included in both ports to ensure convergence, the values being equal to 0.05 and 0.005 for the input port and output port, respectively. It can be directly added to the stiffness matrix in the corresponding degrees of freedom.

Note that `UNVARTOP_2D_complmechanism(nelx,nely,nsteps,Vol0,Vol,k,tau)` must now be used for this example. Using a 200x80 finite element mesh, perform the optimization of this example in 12 steps from $Vol0 = 0$ to $Vol = 0.7$, using an exponential evolution with $k = -2$ and a regularization parameter of $tau = 0.5$.




```
F(2*find(coord(:,2)>=0.9*nely & coord(:,1)==0) -
1,1) = 0.0001*nelx;
F(2*find(coord(:,2)==round(0.9*nely)&coord(:,1)>=
0.9*nelx),2)=0.0001*nelx;
fixed_dofs =
[reshape(2*find(coord(:,2)==nely),1,[]),
reshape(2*find(coord(:,1)==0 &
coord(:,2)<=0.1*nely)+(-1:0),1,[])];
active_node = [find(coord(:,2)>0.9*nely &
coord(:,1)<0.05*nelx); find(coord(:,2)> 0.9*nely
& coord(:,2)<=0.95*nely & coord(:,1) >=
0.9*nelx)];
passive_node = [find(coord(:,1)>0.8*nelx &
coord(:,1)<0.9*nelx & coord(:,2)>0.8*nely);
find(coord(:,1)>=0.9*nelx &
coord(:,2)>0.95*nely)];
```

Problem setting **Problem definition**

PROBLEM 4: Structural sizing analysis of a cantilever beam

The topology optimization technique can be used to determine which is the minimum domain size (minimum vertical size when the horizontal size is kept constant) subject to a maximum acceptable vertical displacement for the cantilever beam. A vertical point-wise load is applied on the middle of the right side of the domain while the displacements in the left side are prescribed to 0, as illustrated below:


```
F(2*find(coord(:,2)==round(0.5*nely)&coord(:
,1)==nelx),1) = -0.01*nelx;
fixed_dofs =
reshape(2*find(coord(:,1)==0)+(-1:0),1,[]);
active_node = [];
passive_node = [];
```

Problem setting **Problem definition**

Then, perform the optimization of the previous problem for a set of `nely` values (ranging from 20 to 160), keeping constant the value of the load F and the number of elements in the x-direction (e.g. $nelx = 100$). Note that the actual size in this implementation is measured with the number of elements in each direction, since the finite elements are squares of unit size. Therefore, an increase in the number of elements in a given direction implies an increase of this length and a variation in the aspect ratio.

In addition, the amount of stiff material (in number of elements) must be also kept constant. Therefore, keep in mind that the target void fraction Vol should change with the number of elements $nely$ (or the vertical size) to keep the total number of stiff elements constant. The corresponding target volume percentage Vol is given by

$$Vol(nely) = 1 - \frac{(1 - \overline{Vol})\overline{nely}}{nely}$$

where $\overline{Vol}$ and $\overline{nely}$ are reference values. Consider the example with $nelx = 100$, $nely = 50$, $Vol0 = 0$ and $Vol = 0.7$ as the reference case to determine the exact Vol fraction.

As for the number of steps `nsteps,` it can be set to 14, although it can be increased to 20 in case the optimization does not converge. The other two parameters can be set to 0 and 0.5 for the evolution factor `k` and the regularization factor `tau`, respectively.

From each topology optimization problem, you must obtain the maximum displacement value as well as the optimal topology layout so that you can plot the maximum displacement versus the number of elements in the vertical direction with the corresponding topologies. Determine then the optimal topology with the minimum vertical size that does not exceed a maximum displacement of 80 units.

Submit a synthetic report including the following points:

a. A plot of the optimal solution of the test example and a comparison with the results provided in this paper. Discuss the results.
b. Plots of the optimal solutions of problems 1 and 2 including the deformed structures.
c. A plot of the deformed compliant mechanism (problem 3) at the appropriate scale, showing the vertical displacement at the jaws of the gripper. Analyse and discuss the obtained movement.
d. A plot of the attained maximum deflections versus the width of the beams, measured in terms of the number of elements used in the vertical direction (problem 4). Include a comparison of the optimal topologies and also the optimal solution satisfying the maximum allowed deflection.