Problem 1:

The modelization of the frame is done accordingly to assignment 5, with an introduced bi-linear behavior in the columns. The beam is modelized as an elastic element with two zero-length elements at its extremities with tri-linear constitutive laws, as shown in figure 1.

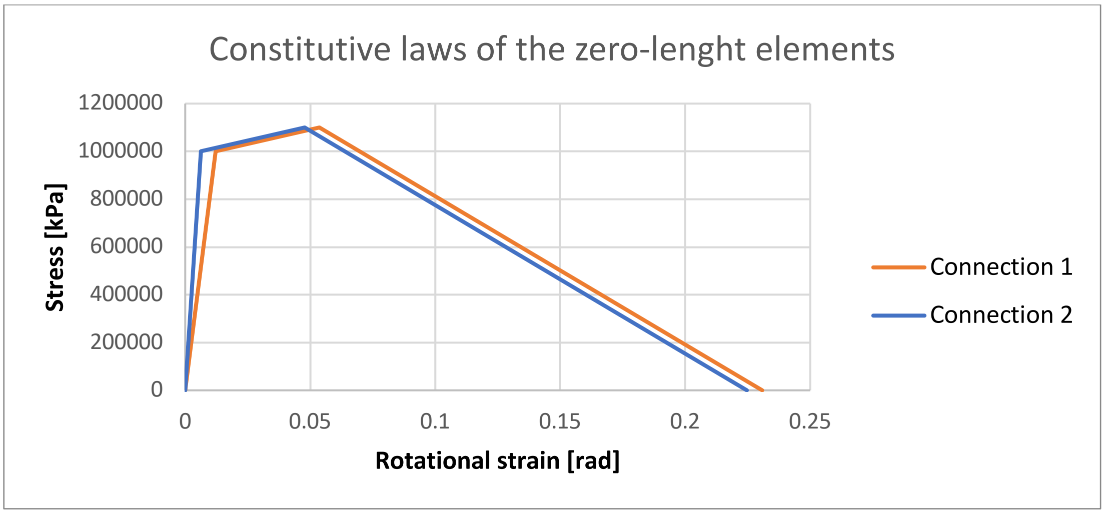


Figure 1: Beam zero-length elements constitutive laws

The constitutive law for the columns is modelized with:

$f_y = 355\ MPa$

$E = 200\ GPa$

$\alpha = 0.03$, the hardening ratio

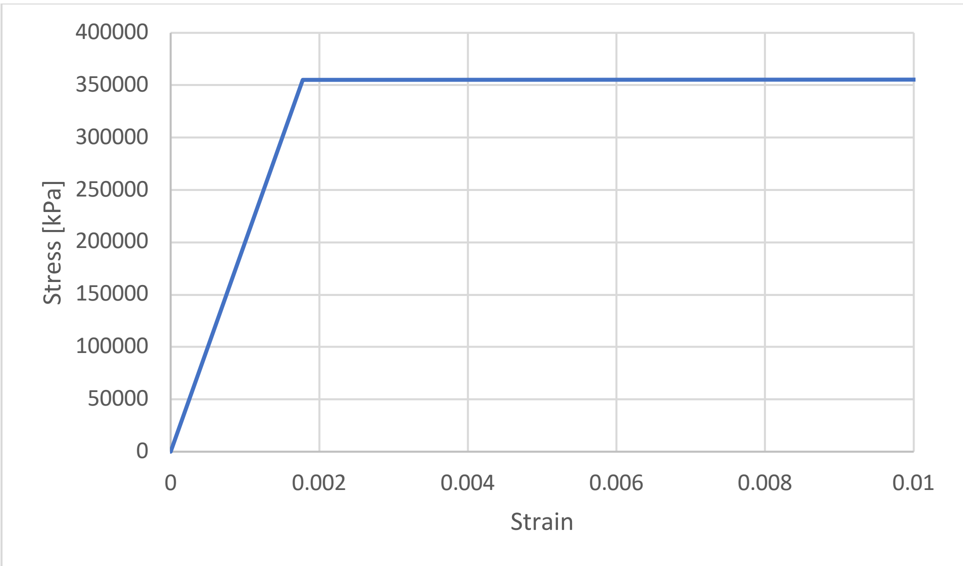
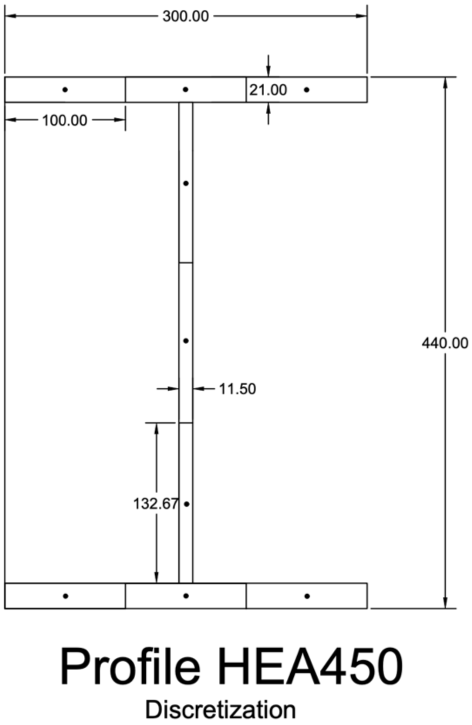


Figure 2: Column constitutive law

Column elements discretization

In order to capture the inelastic behavior of the steel columns, composed of HEA450 cross section, the cross section is discretized into a set number of fibers.

As it is assumed that the column will be stressed not only in the axial direction, but also in bending and in shear, it appears that a minimum of 3 elements in each direction is required. In order to mitigate the computation time, it is decided to discretize the section into 9 fibers as a first approach. The discretization is the following (unit : [mm]) :



The columns are implemented in OpenSees as displacement-based beam column elements with Steel01 material, and discretized cross section as defined.

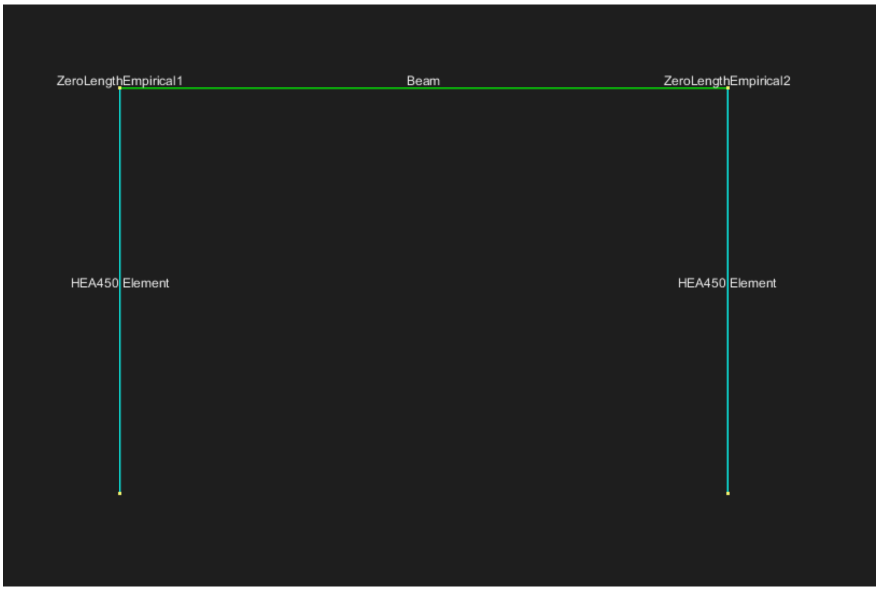


Figure 3: Elements defined in OpenSeesNavigator for the steel MRF

The column elements are modelized with 5 integration points.

Displacement controlled incrementation is applied on the frame, at the node corresponding to the top left corner of the frame.

A total displacement of 600 mm is applied, with 1mm increments.

The final deformed shape is obtained:

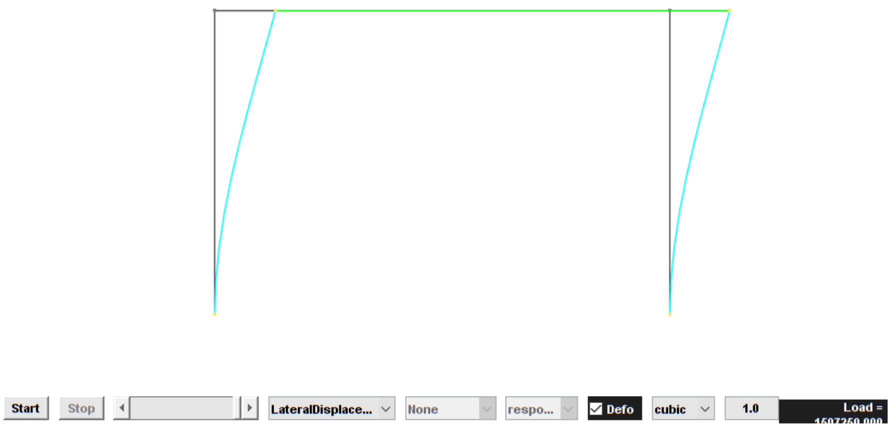
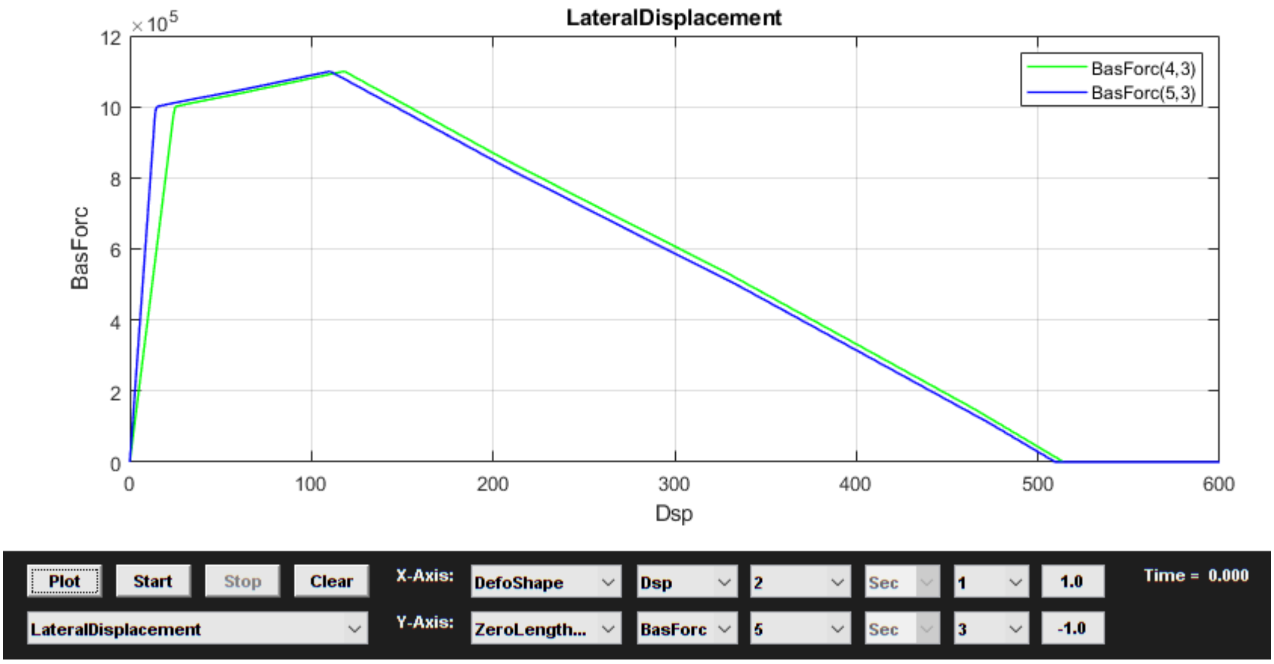


Figure 4: Deformed shape, total displacement incrementation

Beam strength degradation

The beam strength degradation is observed on the moment-displacement curve of the zero-length elements of the beam. Element 4 correspond to the left one, and 5 to the right one.



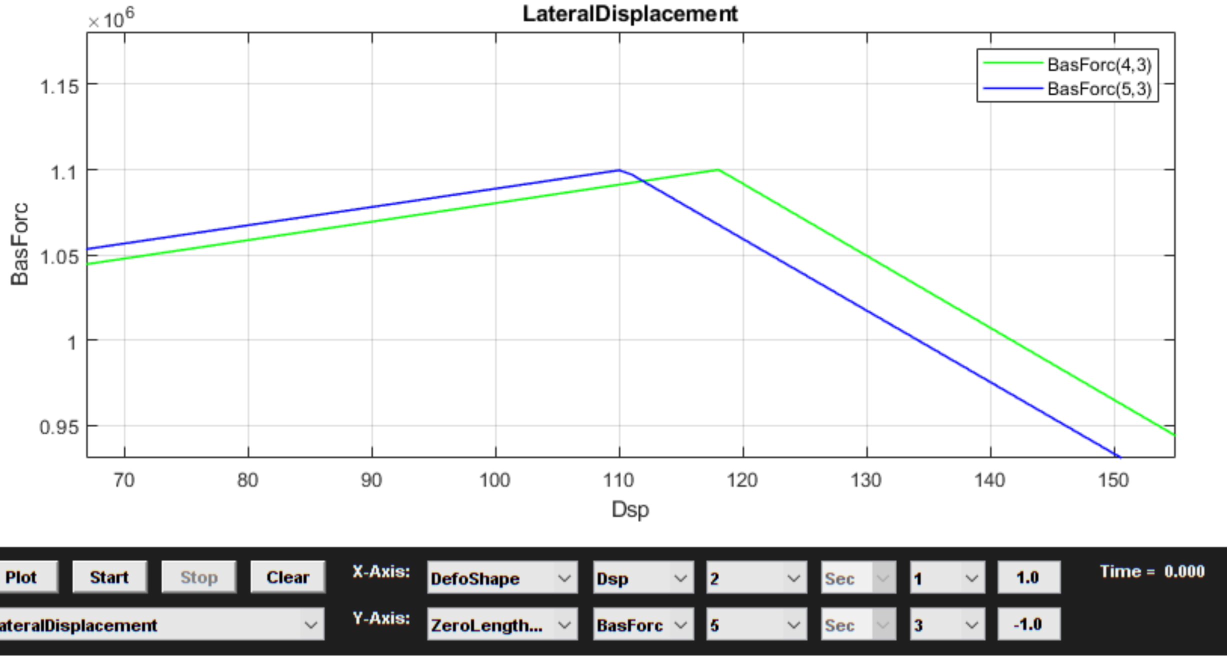


Figure 5: Moment-displacement curve of the zero-length elements. Force in Nmm, dsp in mm.

The right connection is the first to degrade, for a horizontal roof displacement of 110mm. In order to observe the effect of the columns discretization and bi-linear material modelization, the obtained results are compared with the one from assignment 5. In this assignment, the columns were considered as simple linear-elastic elements.

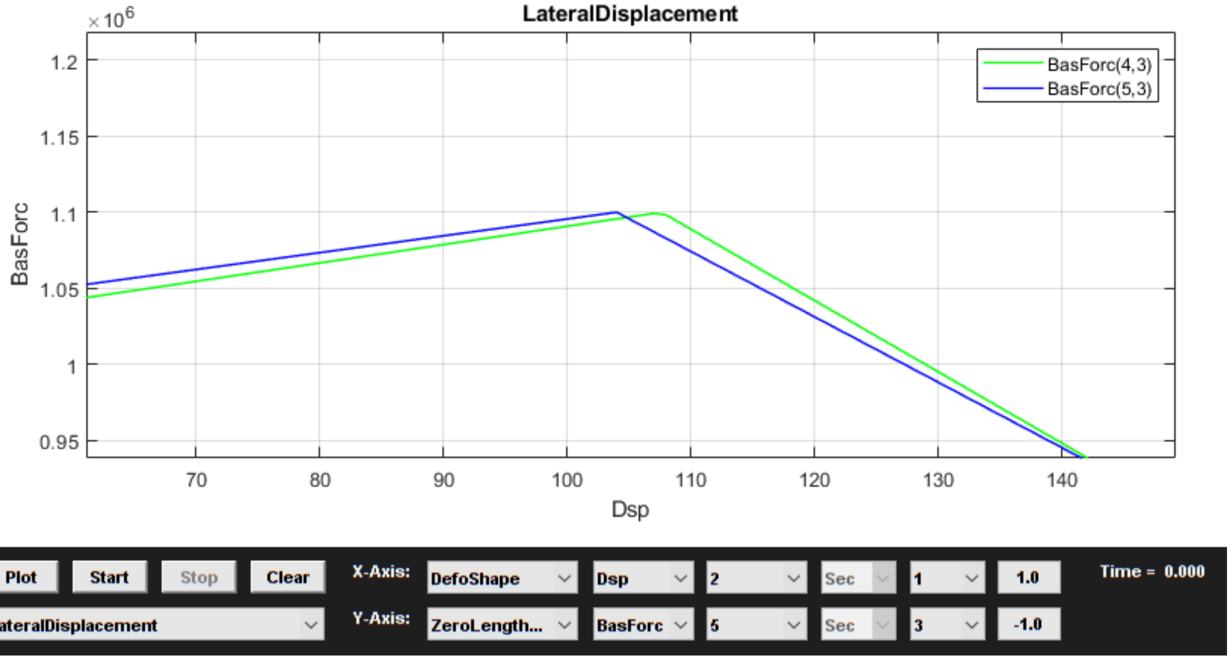


Figure 6: Moment-displacement curve of the zero-length elements, linear-elastic columns. Force in Nmm, dsp in mm.

With a linear-elastic modelization, the strength degradation first appears for a roof displacement of 105 mm.

Therefore, the implementation of the strength hardening behavior in the column and of the fiber discretization results in an increase of the beam computed strength, relative to the displacement. It is also closer to real life behavior of a steel column, as the cross section would plastify progressively.

If the columns were experiencing only elastic deformation, the observed displacement at the zero-elements strength degradation should be the same as in assignment 5. Therefore, we can assume that plastic deformation is computed in the column elements, allowing larger deformations before that the strength peak of the connections is reached.

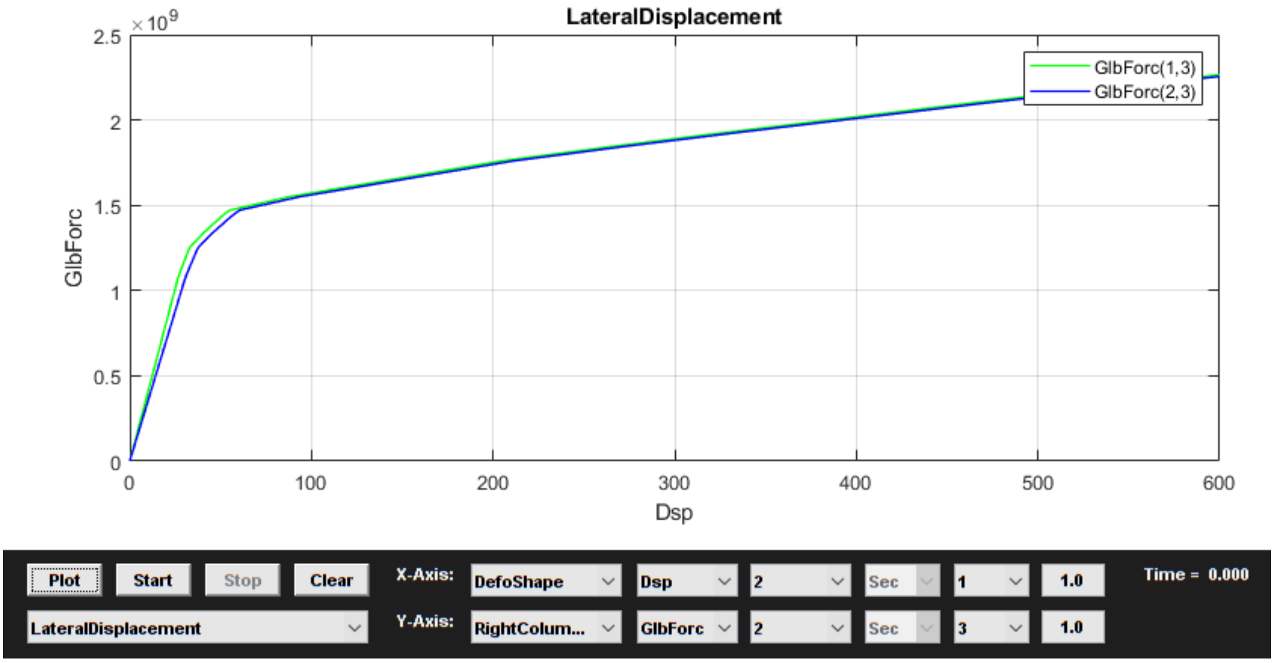


Figure 7: Moment-displacement curve of the columns elements. Force in N, dsp in mm.

It is indeed observed in the moment-displacement curve of the columns. For a roof displacement of 26mm, a first change in the slope of the curve is observed, followed by multiple others. These changes in slope are due to the discretization of the cross-section. When a fiber reaches the yielding stress, it plastify (i.e. enter the hardening state), resulting in a change of the slope.

As the columns plastify before the strength degradation of the connection, the hypothesis formulated before is confirmed.

It is observed that the quasi-totality of the fiber plastify for a roof displacement of around 70 mm.

Storey shear strength of the steel beam

The story shear strength of the steel beam reduce to zero when the moments computed in two connection elements are reduced to 0. From figure 5, it is observed that the moment in the left connection reaches 0 for a roof horizontal displacement of 514 mm (where the last connection strength is reduced to 0).

Problem 2:

Step 1:

For this first step, the stiffness matrix is calculated using the tangent element stiffness matrix of a displacement-based fiber beam-column element. It is then compared with the stiffness matrix calculated for an elastic beam element.

To compute the stiffness matrix using the tangent element stiffness of a displacement-based fiber beam-column element, a numerical integration is used to obtain a numerical estimate of the integral: $Ke = \int_0^l B^T(x) * k^S(x) * B(x)\,dx$ [N/m].The Gauss-Lobato numerical integration is used. Five integration points are considered. The conditions which satisfy all error partial derivatives for five integration points are:

r=[-1,-sqrt(21)/7,0,sqrt(21)/7,1]

ω=[0.1, 49/90, 32/45, 49/90, 32/45, 0.1]

B is calculated as follows:

$$\begin{pmatrix} -1/L & 0 & 0 & 1/L & 0 & 0 \\ 0 & \frac{12x}{L^3} - \frac{6}{L^2} & \frac{6x}{L^2} - \frac{4}{L} & 0 & \frac{-12x}{L^3} + \frac{6}{L^2} & \frac{6x}{L^2} - \frac{2}{L} \end{pmatrix}$$

To calculate the integral, the domain is normalised from [0,L] to [-1,1], using a coordinate transformation: $x = \frac{L}{2} * r + \frac{L}{2}$ and $dx = \frac{L}{2} * dr$.

The cross-section is considered constant along the length of the cantilever. Moreover, the stress in fibers is constant and equal to 1 MPa.

Therefore in N/mm:

```
The tangent element stiffness matrix of a displacement based fiber BC elemnt is:
[[ 4.00e+06  0.00e+00  0.00e+00 -4.00e+06  0.00e+00  0.00e+00]
 [ 0.00e+00  3.96e+04  3.96e+07  0.00e+00 -3.96e+04  3.96e+07]
 [ 0.00e+00  3.96e+07  5.28e+10  0.00e+00 -3.96e+07  2.64e+10]
 [-4.00e+06  0.00e+00  0.00e+00  4.00e+06  0.00e+00  0.00e+00]
 [ 0.00e+00 -3.96e+04 -3.96e+07  0.00e+00  3.96e+04 -3.96e+07]
 [ 0.00e+00  3.96e+07  2.64e+10  0.00e+00 -3.96e+07  5.28e+10]]
```

Computing the stiffness matrix for an elastic beam element, we get in N/mm:

```
K_classic=[[ 4.00000e+06  0.00000e+00  0.00000e+00 -4.00000e+06  0.00000e+00  0.00000e+00]
 [ 0.00000e+00  4.00000e+04  4.00000e+07  0.00000e+00 -4.00000e+04  4.00000e+07]
 [ 0.00000e+00  4.00000e+07  5.33333e+10  0.00000e+00 -4.00000e+07  2.66667e+10]
 [-4.00000e+06  0.00000e+00  0.00000e+00  4.00000e+06  0.00000e+00  0.00000e+00]
 [ 0.00000e+00 -4.00000e+04 -4.00000e+07  0.00000e+00  4.00000e+04 -4.00000e+07]
 [ 0.00000e+00  4.00000e+07  2.66667e+10  0.00000e+00 -4.00000e+07  5.33333e+10]]
```

Computing the error between both stiffness matrix, we get an error of 1%.

The minimum number of fibers to have an error lower than 2% is 8 fibers (error of 1.562%).

The element resisting force Q is also using the Gauss Lobato numerical integration:

$$Q = \int_0^l B^T(x) * f^S(x)\, dx \text{ [N]}$$

```
The element resisting force vector is
[[-40000.]
 [     0.]
 [     0.]
 [ 40000.]
 [     0.]
 [     0.]]
```

We can verify that Q1=-a*b=-200*200=40000 N and Q4=a*b=40000 N, and that the remaining entries are equal to 0 when the stress in the fibers is equal to 1 MPa.

Step 2:

For this second step, the nodal force and the displacements are solved using a Newton-Raphson algorithm.

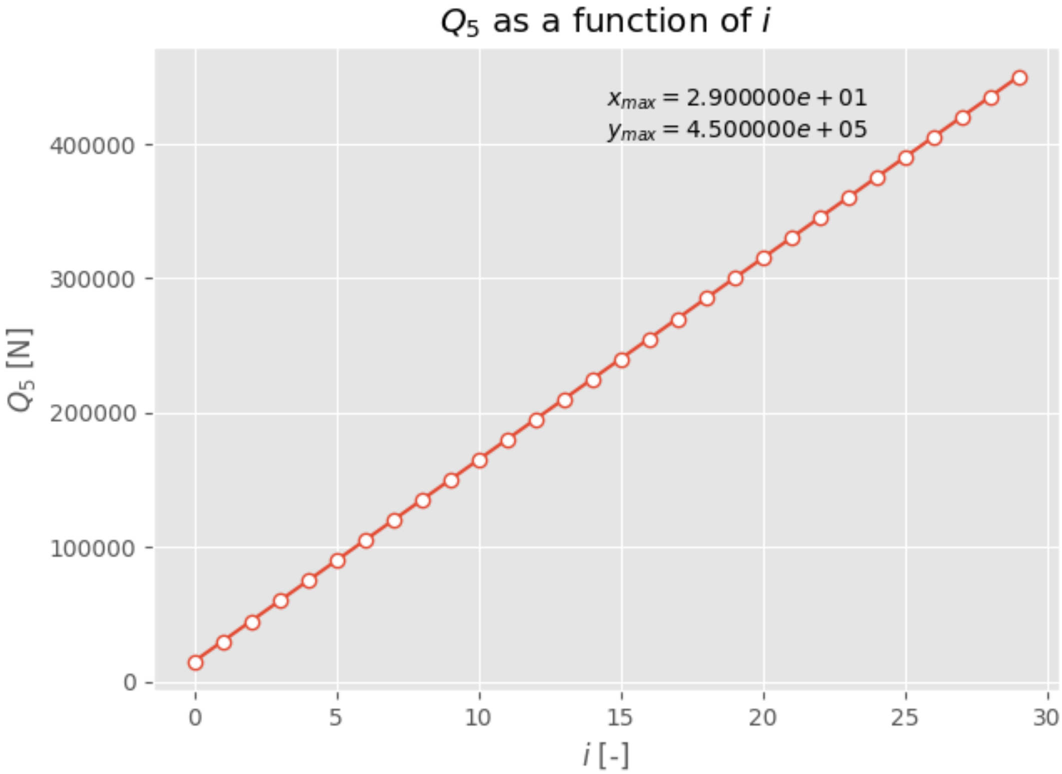
The external load applied on the cantilever is applied in 30 steps. At each step, the load is applied with a maximum number of iterations of 16:

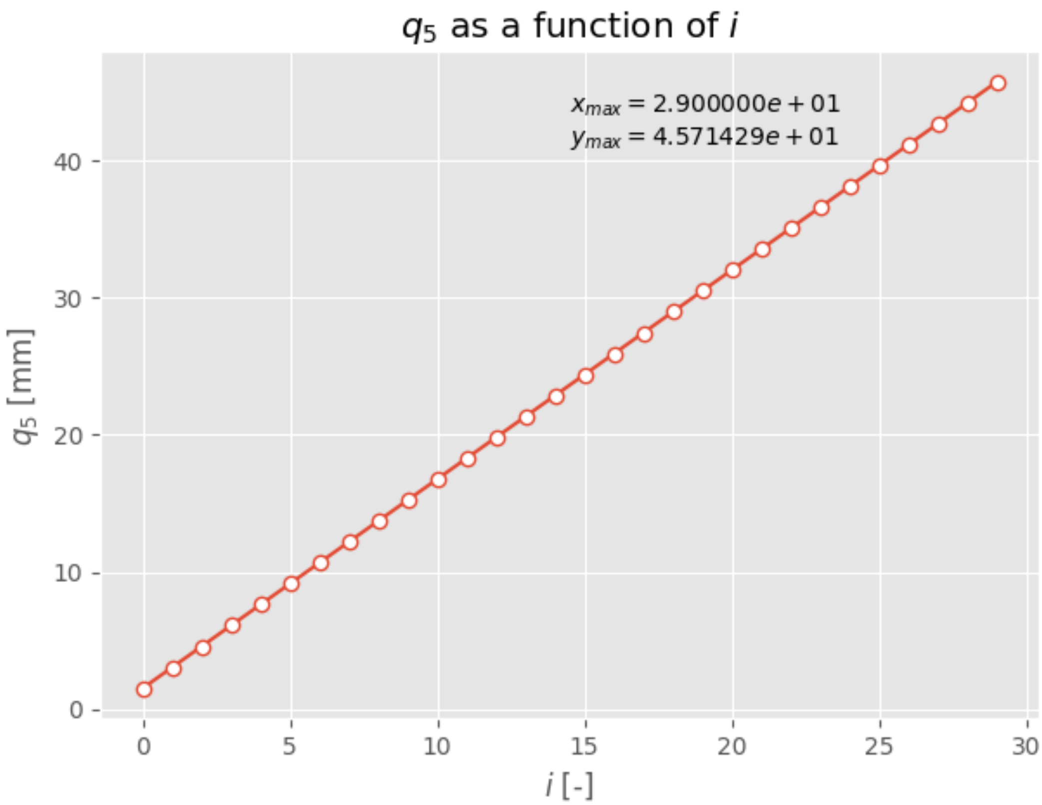
1- The tangent stiffness matrix is calculated for the free degree of freedom of the cantilever.
2- The residual force resulting from the difference between the external force applied and the element resisting force vector is calculated (only for the free degrees of freedom)
3- From this, we can solve the equation K_ff*q_f=R, and find the increment of displacement of the nodes.
4- From the displacements of the nodes, the section deformation is calculated at each point of the Gauss Lobato integration.
5- From this, the strains and stresses at each point of the Gauss Lobato integration are calculated.
6- With the new stresses in the fibers, the element resisting force vector is recomputed.

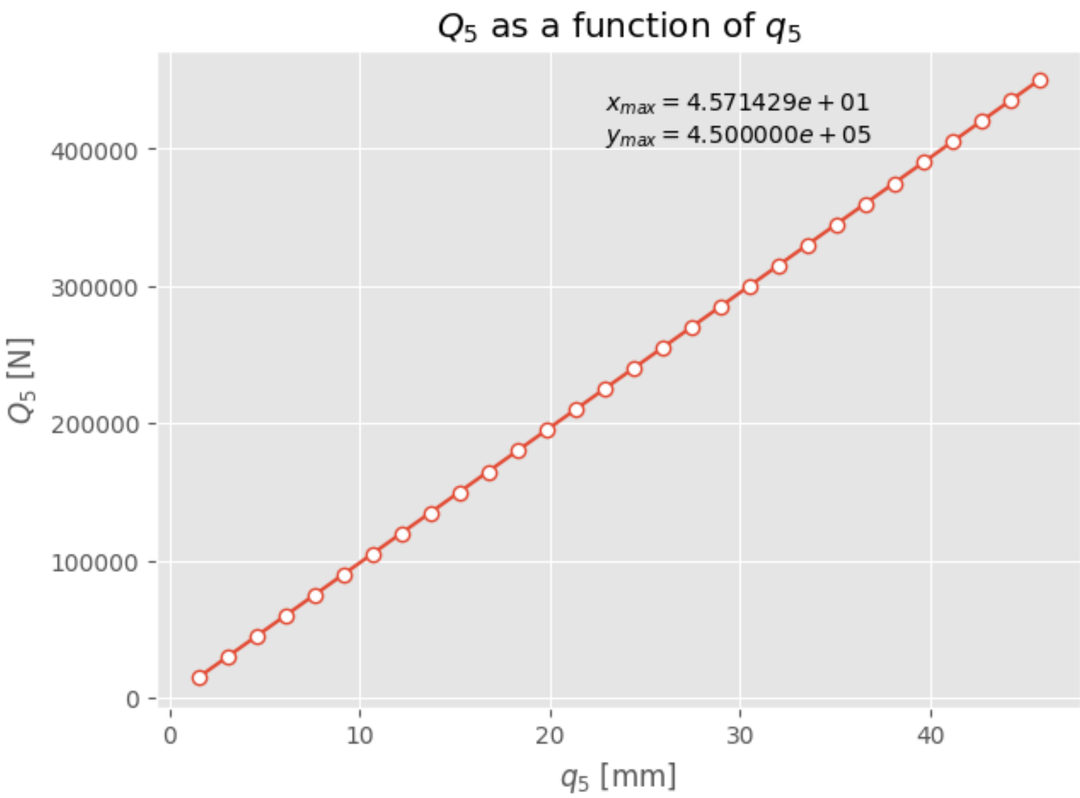
7- The residual force is calculated again for the free degree of freedom of the element.
8- If the tolerance is reached, the next step of the external load applied is performed.

From this, the maximum displacement for the maximum load applied are obtained:

- Maximum displacement: 47.5 mm
- Maximum load: 450000 kN



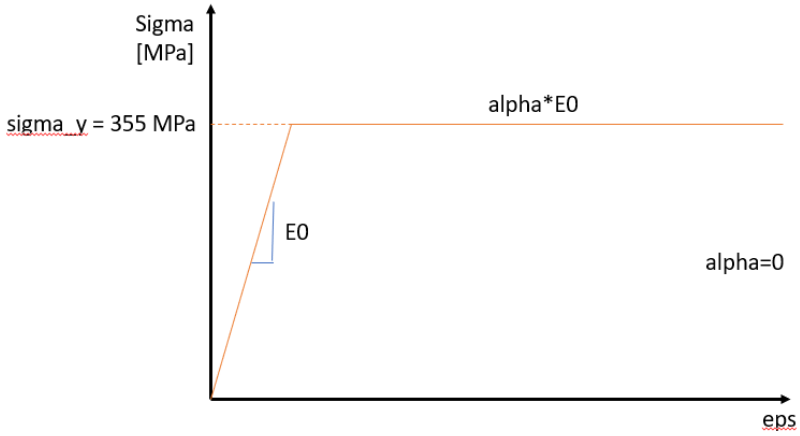




The displacement of the far-end of a cantilever , due to a point load acting at its end is written as: $\omega = \frac{Q}{K} = \frac{QL^3}{3EI} = \frac{450\,000*2000^3}{3*200\,000*200^4/12}$ =45 mm. This deformation value is close to the one obtained with the Gauss-Lobato integration method. The difference can be due to the approximation made using the Gauss-Lobato integration method.

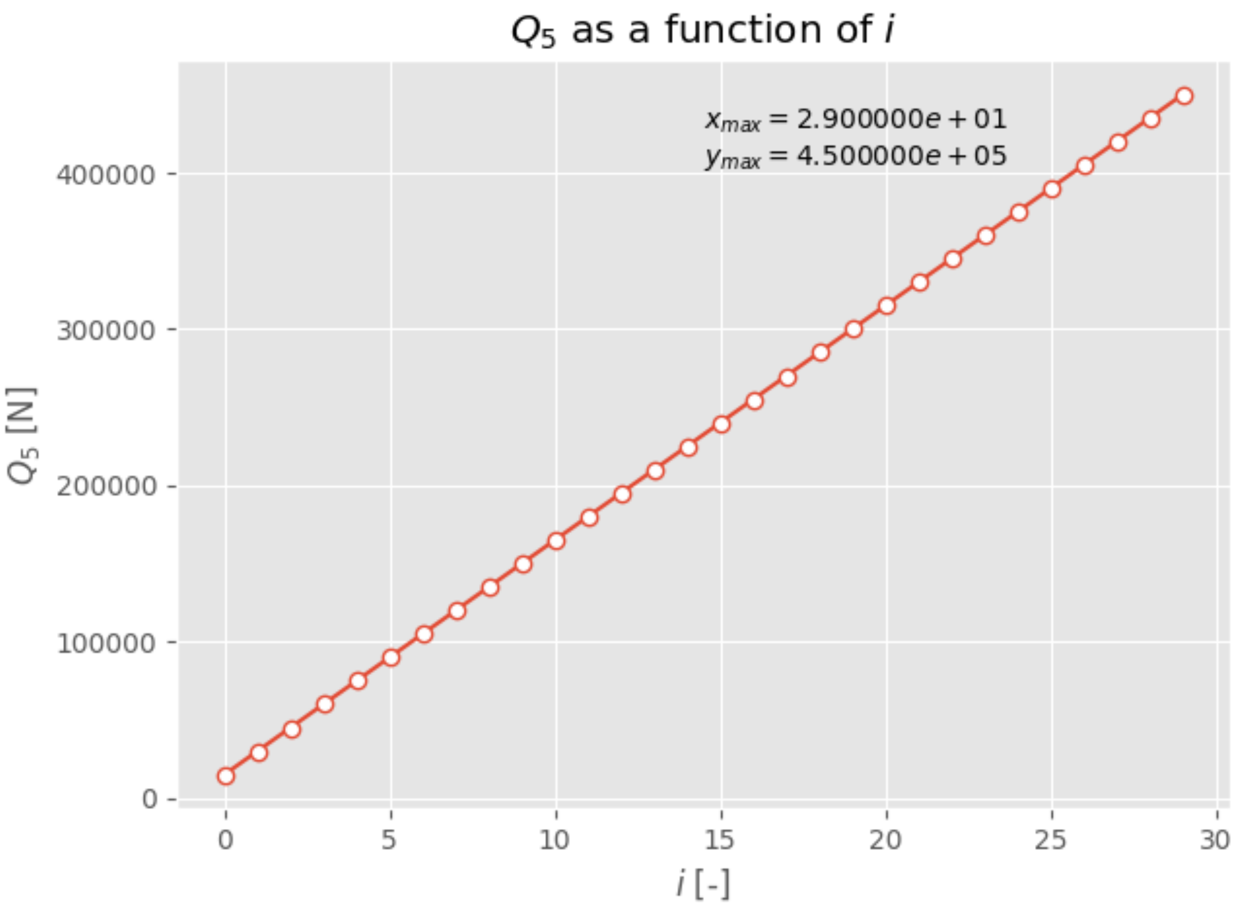
Step 3: Adding an elastic-perfectly plastic material

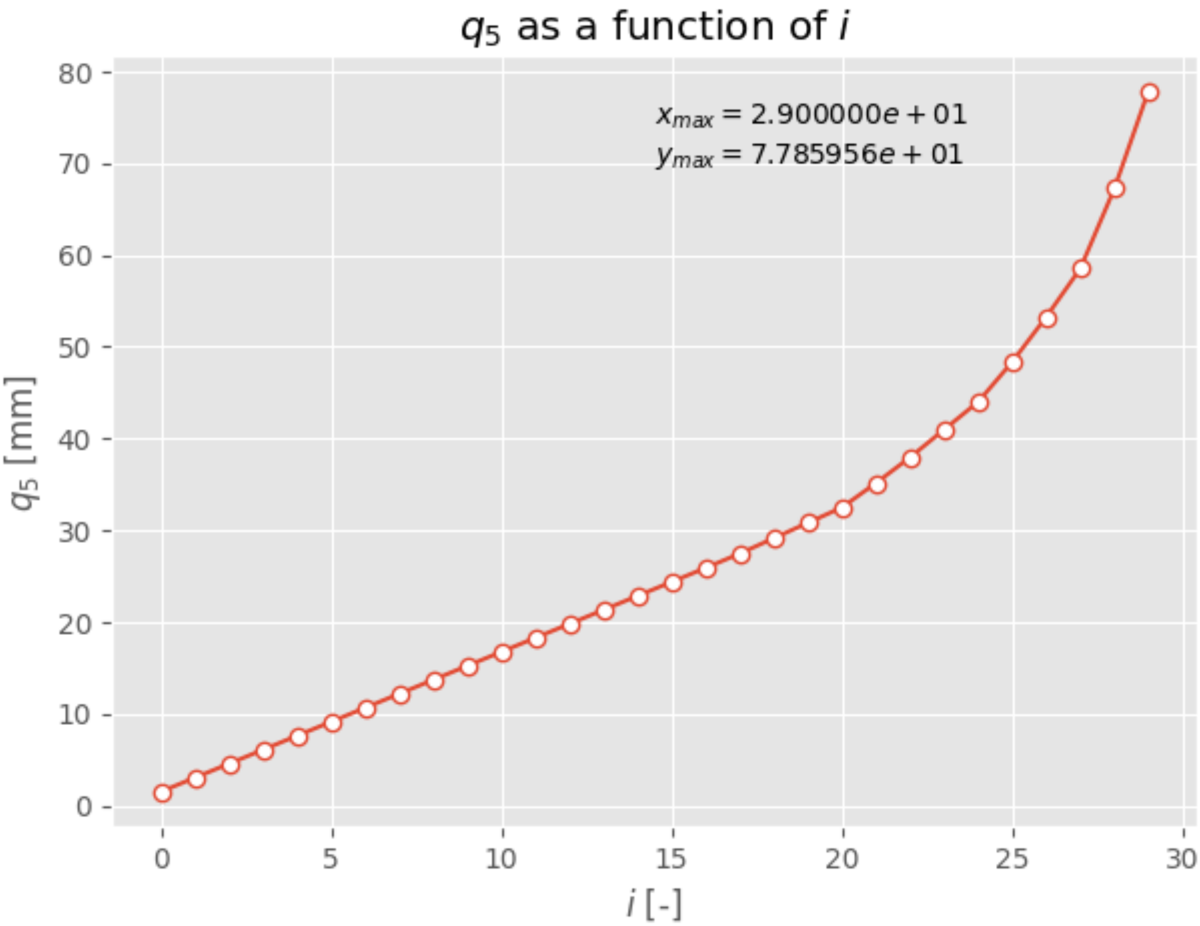
An elastic perfectly plastic material is now considered:

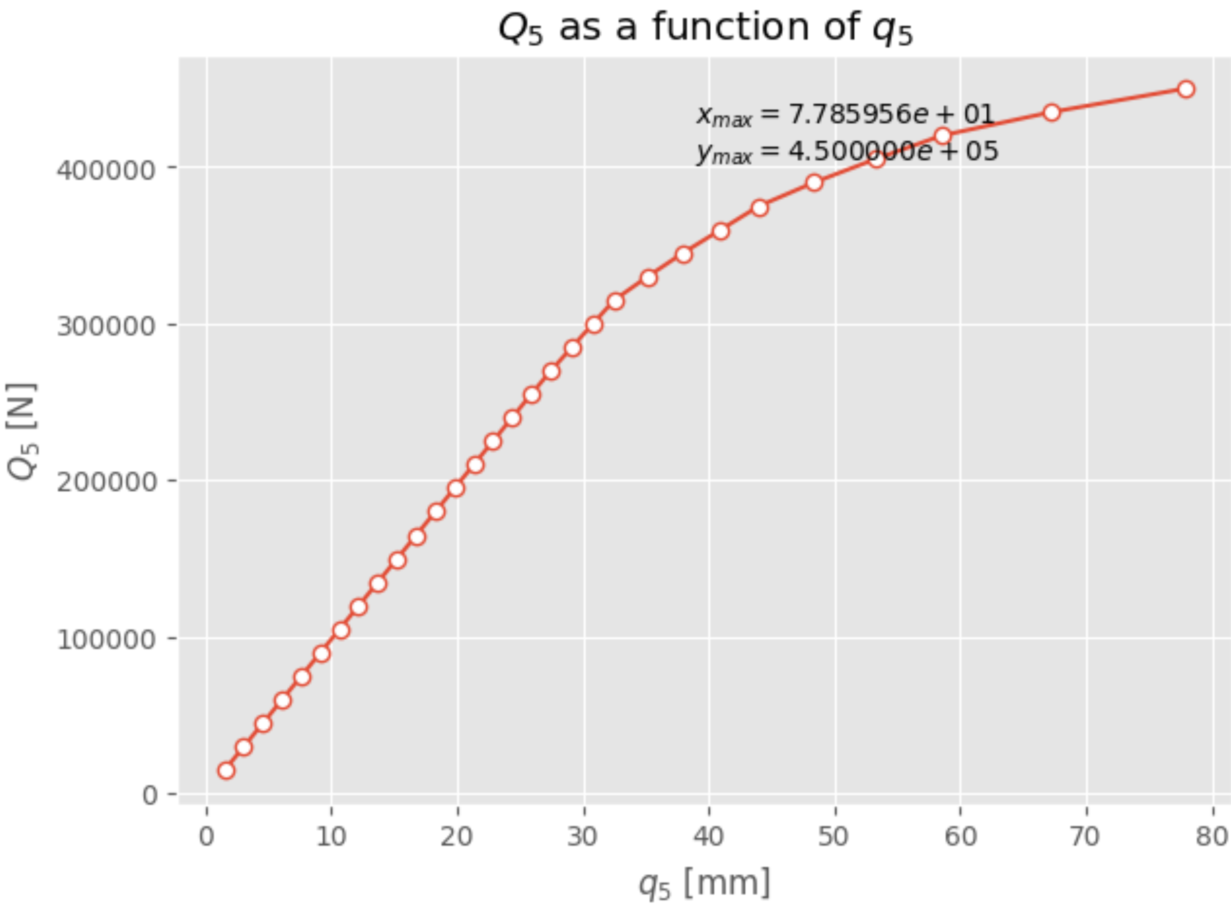


Implementing this new material, the maximum displacement for the maximum load applied are obtained:

- Maximum displacement: 77.86 mm
- Maximum load: 450000 kN




q_5 as a function of i
$x_{max} = 2.900000e+01$
$y_{max} = 7.785956e+01$
q_5 [mm]
i [-]
0
10
20
30
40
50
60
70
80
0
5
10
15
20
25
30


Q_5 as a function of q_5
$x_{max} = 7.785956e+01$
$y_{max} = 4.500000e+05$
Q_5 [N]
q_5 [mm]
0
100000
200000
300000
400000
0
10
20
30
40
50
60
70
80

From the graph above (load as a function of the displacement), the first fiber starts yielding when the graph is no more linear, which corresponds to $V_{end,y}$ =315000 kN.

$Q_y=V_{end,y} *L=315*2=630$ kNm

$M_y= \sigma_y *W_{el}= \sigma_y*\frac{b*h^2}{6} = 355 * \frac{200*200^2}{6} = 473.3\ kNm$

There is a difference of 33%, which may be due to approximations made with the Gauss-Lobato integration method.