“Implementation of the Pushover Analysis in MatLab for 2D and 3D Steel Trusses”

Luis Fernando Verduzco Martínez

luiz.verduz@gmail.com
Faculty of Engineering, Autonomous University of Querétaro, Santiago de Querétaro, Mexico

Overview

This software is the implementation of the Pushover analysis for 2D and 3D structural trusses fully developed in MatLab as functions. Such functions are based entirely on axial stresses. A uniform load incrementation is carried out on each step until a collapse or stiffness degradation criteria is reached. Such function has proved to have a great potential for its implementation in teaching the Pushover method in structural mechanics and applied sciences through the simulation of collapse mechanisms. Not only $P - \Delta$ collapse graphics are obtained, but also the axial load history of each bar during the analysis process.

Keywords: Pushover, Steel Trusses, Teaching, Pushover, Collapse Mechanisms , Stiffness degradation

1 The Pushover analysis

The *Pushover* analysis is used in structural engineering for many tasks when it is required to evaluate the response of a structure in the plastic range behaviour, for instance, when evaluating performance of structures and estimating potential Damage States under certain load conditions. For Steel Trusses the analysis process consists of analysing a structure by incremental punctual loads initially imposed, defining step by step axial plastic-formations on the element's or bars until reaching a collapse mechanism. The analysis is terminated when the collapse mechanism is reached, either by stating a certain state of damage for the structure or a degree of stiffness degradation. This way, Collapse Safety Factors can be determined. A condition of stiffness degradation can be imposed as (1) to stop the analysis process, where K_j is the last stiffness matrix, K_0 the initial under elastic behaviour and deg is a stiffness degradation factor (for which values between $0.003 - 0.005$ are recommended).

$$\frac{det(K_j)}{det(K_0)} < deg \tag{1}$$

During the analysis process, when an axial plastic formation is generated in a bar, then an equivalent structure is generated by substituting such plastified elements for their yield resistance as punctual forces with contrary signs acting at each of both ending nodes of the bars **Fig. 1**. This such equivalent nodal forces at each will remain constant through out the subsequent analysis steps to simulate the present plastified bar that no longer behaves in the elastic range, and whose cross-sectional area or module of elasticity are taken to a very low value so that (2) may comply, and hence the subsequent truss's stiffness may consider such plastification and simulate the degradation of the force-displacement capacity.

$$\frac{EA}{L} \approx 0 \tag{2}$$

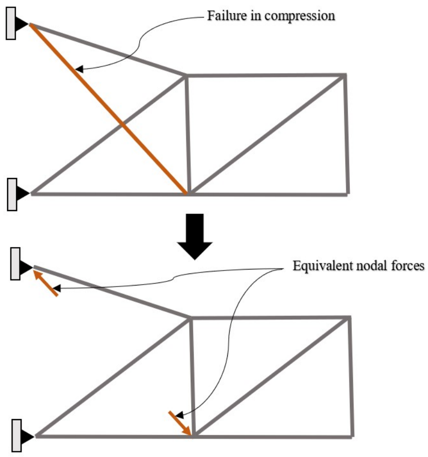

Figure 1. Equivalence of a plane truss structure with a plastified member.

This mechanism is congruent with the hypothesis established for the method, which are listed next:

1.1 Hypothesis

1. It is supposed that the bars have a perfect elasto-plastic mechanical behaviour **Fig. 2**

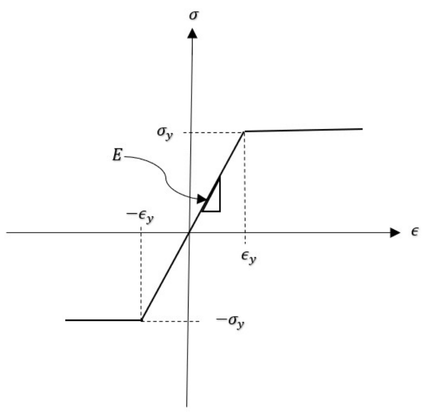

Figure 2. Depiction of an elasto-plastic mechanical behaviour.

1.2 Algorithmic process

Algoritmo 1.1: Pseudo-code for the Pushover analysis method.

1. Initialize load incremental factor as λ =1.0
2. Compute mechanic elements of bars en check the structural efficiency of each bar element either in compression or tension to verify that $F \leq F_y$
   - If any member has overpassed such yield stress limit, then change that corresponding element's cross-section area or Modulus of Elasticity to very small value close to 0, so that $EA/L \approx 0$ and apply the corresponding equivalent nodal forces at each of element's nodes.
   - If no member has suffered overpassed such plasticity limit then apply increment di in the λ factor as $\lambda = \lambda + di$ and return to step 2
   - Verify if the termination damage condition is complied or not $K_i/K_o < deg$
     - If such condition is complied, then stop the program
     - Otherwise, apply the load increment di in the λ factor as $\lambda = \lambda + di$ as return to step 2
3. Plot the deformed structure before collapsing
4. Plot the stress and strain bar history

1.3 Teaching the Pushover analysis

Teaching the Pushover analysis in Postgraduate program courses of engineering is still a challenging task by many Professors. The inherent dualism of the method similar to the Finite Element method requires a balance between the mathematical theory and physical understanding, and can only be applied through a computer. Even though many textbooks related with the subject contain didactic exercises treating theoretical issues and simple sample problems, they do not contain full sections that may guide the student on how to compute the method by oneself, thus lacking the motivating factor for students to fully appreciate the intimate relationship between the theoretical issues of the method, its physical interpretation and its computer implementation.

On the other hand, there is a tendency by educational practitioners of courses related to this topic to leave the students to operate the method by themselves in any way possible, not really paying attention on the programming skills of each individual. Therefore, those students who lack the good programming knowledge base requirement tend to get frustrated by the so many operations and code required, putting in risk their own learning capabilities.

It has been recommended by some researchers and academics that in order to make a computed aided program pedagogical effective such program should be written in such a manner that all the usual subroutines related to the method in question are present; the student should be able to assemble all of such functions and operations so that a calculation is possible, without requiring a large amount of programming effort or computational skills. This way, a much more time-efficient learning environment is enhanced, requiring only for each student to build their own code for every problem to solve, instead of coding and programming the whole computational process with the risk of losing track in the run. Thus, the physical interpretation along with the solution strategy of any problem are continuously connected to the mathematical and numerical treatment of such problem at every moment.

When learning the Pushover analysis method or any other engineering process it is required to review whether or not the calculations are correct, which task is usually observable by the collapse mechanism. It is thus, advantageous for students to be able to visualize the evolution of the deformed structure as it gets closer to the collapse mechanism. Even though software like Microsoft Excel could also provide good means for the computation of the Pushover method, such package is limited in graphic tools, as they lack of dynamism compared to MatLab graphic functions, for instance.

2 Axial resistance of steel structural elements

Due to design factors imposed by design and construction codes both the tension and compression axial stress resistance of a truss' bars are reduced, in such a way that the tension resistance may always be greater than the compression resistance as for the latter not only reduction resistance factors are imposed but also buckling effects.

2.1 Resistance in tension

The AISC 360 establishes that the yield tension resistance of a steel element might be calculate simply as (3):

$$F_t = \frac{A_s \cdot F_y}{1.67} \tag{3}$$

2.2 Resistance in compression

On the other hand, for elements subject to compression loads, the AISC 360 estblished their nominal resistance as (4), in which the slenderness ratio of the element kL/r is considered according to the Euler's stress, and thus the design resistance can be calculated as (5):

$$P_n = 0.877 \cdot \frac{\pi^2 \cdot E}{(\frac{kL}{r})^2} \cdot A_g \tag{4}$$

$$F_c = \frac{P_n}{1.67} \tag{5}$$

3 Stress and deformation history

It is very helpful to not only identify those elements that yield at each load step but also the chronological order in which they do so. For this purpose, the stress and deformation history of each element during the whole non-linear analysis process. This information can be used by engineers to optimize the structural design at the very early stages when performing this type of simulations so that the structure may withstand the largest possible stresses/forces with the less possible amount of material.

In order to obtain a clear picture of a structure's collapse behaviour it is usually of interest for engineers to (1) assess the displacement history of specific nodes of the structure as the elements yield, (2) assess the stress and deformation history of specific bar elements until the collapse mechanism or at the given yielding load steps (see **Fig. 3**).

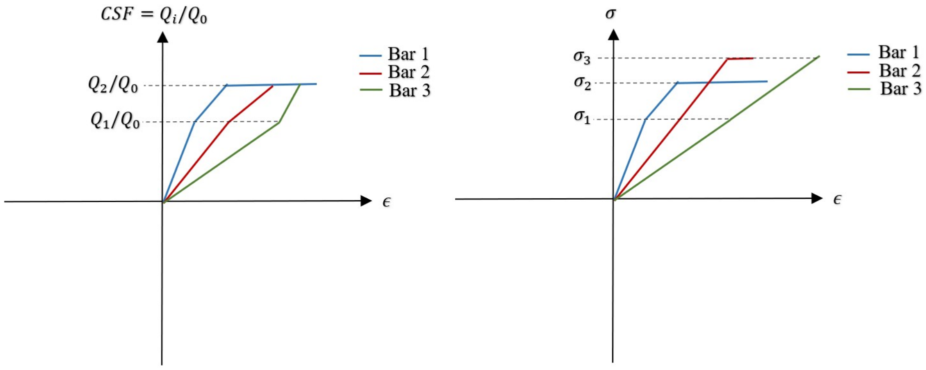

Figure 3. Examples of stress and deformation history graphs for the assessment of a collapse mechanism of a truss structure. The left panel shows depicts the deformation history of certain bars of a truss at each yield step. The right panel shows the stress-deformation history of certain bars at each yield step.

4 Residual stresses

When any of the structure's members has undergone plastic stresses and if all of a sudden the structure is totally unloaded then some residual strains and/or stresses would remain in those member of the structure that had not incurred in the inelastic range so that a new compatible system of internal forces is reached.

During the unloading process it is assumed that each of the elements have a pure elastic linear behaviour so that the total reduction of strains and stresses corresponding to a total unloading have the same amount as those suffered in the elastic solution for any element (see **Fig. 4**).

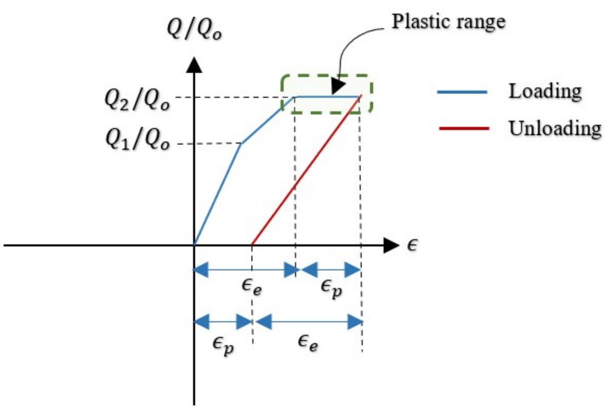

Figure 4. Load and unload graph of a plastified bar element for the determination of residual stresses.

5 Illustrative examples

As following, two examples will be shown-case to illustrate how to use the built-in functions.

5.1 Ex01: Plane steel truss

$Truss2D\_Ex01$

5.1.1 Problem

Let us consider the following plane truss in cantilever shown in **Fig. 5** subject to the depicted punctual forces. It is required to determine the load increment factor at which the structure will collapse, having on consideration that the each of the applied punctual loads are increment in the same rate on each load step. It is also of interest to identify which bar fails first.

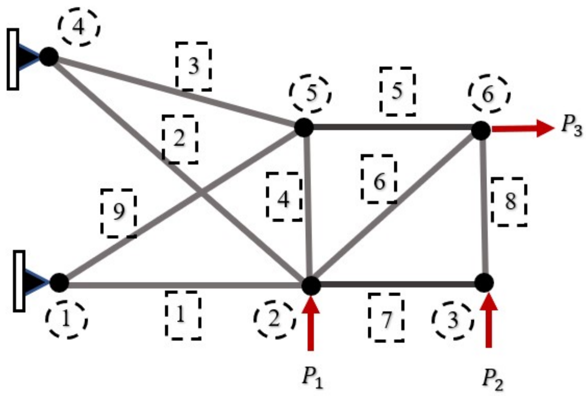

Figure 5. Topology of the 2D truss.

5.1.2 Solution

Let us begin by assigning the material's mechanical properties to each bar element as well as the cross-section geometrical properties with the following vectors:

```
%% Materials
E=[29000*1000*0.4536/2.54^2; % Elements' Modulus of Elasticity (Kg/cm2)
    29000*1000*0.4536/2.54^2;
    29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2];
```

```
Fy=[3515; % Yield stress (Kg/cm2)
    3515;
    3515;
    3515;
    3515;
    3515;
    3515;
    3515;
    3515];

%% Geometry
area1=30.26; r1=5.16; % OC 152 x 7.1
area2=50.71; r2=5.59; % OC 168 x 11.0

A=[area2; % element's cross-section area (cm2)
    area1;
    area1;
    area1;
    area1;
    area2;
    area1;
    area1;
    area2];

r=[r2; % element's cross-section radius rotation (cm)
   r1;
   r1;
   r1;
   r1;
   r2;
   r1;
   r1;
   r2];
```

Now, the basic topology data must be entered, such as the nodal coordinates and the initial and final node of each member:

```
%% Topology
% Node coordinates (cm)
coordxy=[0,0;
         600,0;
         1100,0;
         0,600;
         600,500;
         1100,500];
```

```
% Initial and final node of each bar
ni=[1;4;4;5;5;2;2;6;1];
nf=[2;2;5;2;6;6;3;3;5];
```

Note that the DOF numbering is not given, as it is considered that each node's dof are function of its node's number (i)*:* $dof_i = [2 \cdot i - 1, 2 \cdot i]$.

At last, the punctual loads are set, as well as each of their respective DOF at which they act. For this purpose, the vectors $initialLoads$ and $edofLoads$ are created:

```
%% Loads
load=200; %kgf
load01=load;
load02=2*load;
load03=2*load;

initialLoads=[load01; load02; load03];
edofLoads=[4;6;11];
```

Finally, the function $ElastoPlasticPushover2DTruss$ to perform the Static Non-Linear Pushover method is called as:

```
%% Pushover analysis
[collec_dx_nodes,collec_dy_nodes,collection_FS,Uglobal,...
 critical_bars_global,critical_ten_comp_bars_global,dam,defBarHistory]=...
 ElastoPlasticPushover2DTruss(E,Fy,A,r,coordxy,ni,nf,...
 bc,edofLoads,initialLoads,10,1,[1,2,3,4,5,6,7,8,9],0.001)
```

Note that the penultimate input argument corresponding to those bars whose load history is of interest was set so that all bars are included. Also note that the load incremental step was set to 10 units (Kgf in this case):

5.1.3 Results

When running the program with the previous parameters set, the graphical results are:

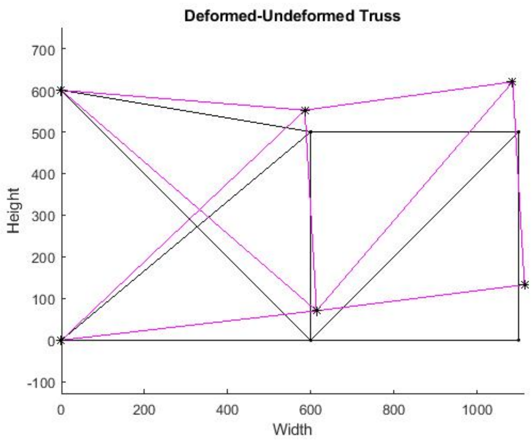

Figure 6. Deformed structure before reaching the collapse mechanism.

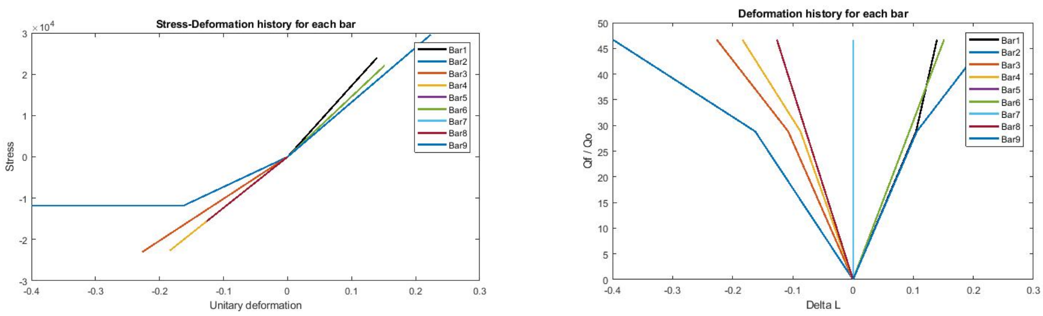

Figure 7. Left panel: Stress-deformation history of each bar. Right panel: deformation history at each yield load step.

Note in the left panel of the latter plot that the only bar that incurs in its plastic range is the number 2 given that it is the first one to yield, followed by the bar 3 which produces the collapse mechanism right after it yields. Note also that the bar number 7 is not working efficiently as a system given that its stresses are very close to 0 during the whole process.

5.2 Ex02: 3D steel truss

$Truss3D\_Ex01$

5.2.1 Problem

It is common in Industrial Parks or Onshore Wind Turbine Towers to have all sorts of steel structures. A common and simple one might be a steel tripod structure such as the one shown in **Fig. 8** which are usually used as a platform to withstand central nucleus structures or simply heavy hanging objects.

Given the extreme heights that these sort of structure might have the wind loads become of paramount importance at the top of them, and thus, to provide auxiliary legs or flanges turns out to be a good choice for their design.

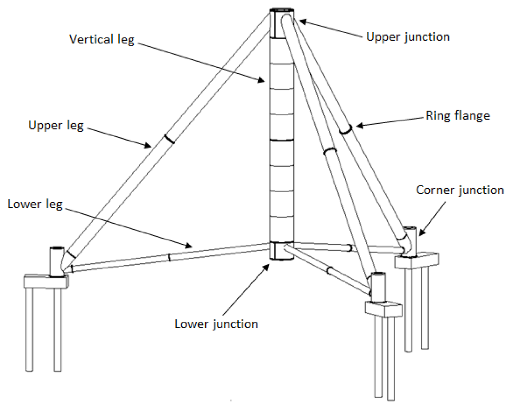

Figure 8. Example of a steel tripod structure used as a platform in Onshore Wind Towers.

Let us consider for instance, the following tripod steel structure of **Fig. 9** with an additional leg to provide extra resistance to the structure in case that non-considered wind or impact loads might be present.

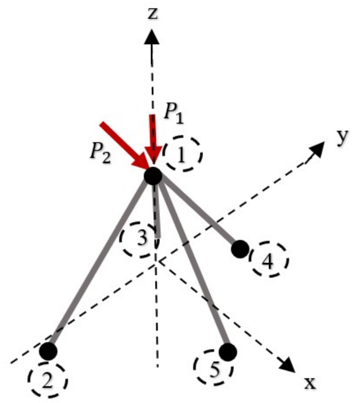

Figure 9. Topology of a structural tripod structure with an auxiliary leg.

5.2.2 Solution

Let us begin by assigning the material's mechanical properties to each bar element as well as the cross-section geometrical properties with the following vectors:

```
%% Materials
E=[29000*1000*0.4536/2.54^2; % Elements' Modulus of Elasticity
    29000*1000*0.4536/2.54^2;
    29000*1000*0.4536/2.54^2;
   29000*1000*0.4536/2.54^2];

Fy=[3515; % Yield stress
    3515;
```

```
     3515;
     3515];

%% Geometry
area1=30.26; r1=5.16; % OC 152 x 7.1
area2=50.71; r2=5.59; % OC 168 x 11.0

A=[area2; % element's cross-section area
     area1;
     area2;
     area1];

r=[r2; % element's cross-section radius rotation
    r1;
    r1;
    r2];
```

Now, the basic topology data must be entered, such as the nodal coordinates and the initial and final node of each member:

```
%% Topology
% Node coordinates
coordxyz=[0,0,500;
          -400,-200,0;
          0,400,0;
          400,400,0;
          400,-400,0];

% Initial and final node of each bar
ni=[1;1;1;1];
nf=[2;3;4;5];
```

Note that the DOF numbering is not given, as it is considered that each node's dof are function of its node's number (i)*:* $dof_i = [3 \cdot i - 2, 3 \cdot i - 1, 3 \cdot i]$.

At last, the punctual loads are set, as well as each of their respective DOF at which they act. For this purpose, the vectors $initialLoads$ and $edofLoads$ are created:

```
%% Loads
load=200; %kgf
load01=load;
load02=-2*load;

initialLoads=[load01; load02];
edofLoads=[1;3];
```

Finally, the function $ElastoPlasticPushover3DTruss$ to perform the Static Non-Linear Pushover method is called as:

```
%% Pushover analysis
[collecDxNodes,collecDyNodes,collecDzNodes,collectionFS,Uglobal,...
 criticalBarsGlobal,criticalTenCompBarsGlobal,dam,defBarHistory,...
 strBarHistory]=ElastoPlasticPushover3DTruss(E,Fy,A,r,coordxyz,ni,nf,...
 bc,edofLoads,initialLoads,10,1,[1,2,3,4],0.001)
```

Note that the penultimate input argument corresponding to those bars whose load history is of interest was set so that all bars are included. Also note that the load incremental step was set to 10 units (Kgf in this case):

5.2.3 Results

When running the program with the previous parameters set, the graphical results are:

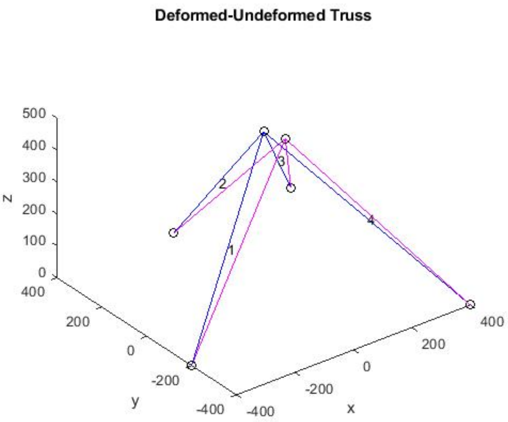

Figure 10. Deformed structure before reaching the collapse mechanism.

Note in the upper panel of the latter plot that the only bar that incurs in its plastic range is the number 4 given that it is the first one to yield, followed by the bar 3 which produces the collapse mechanism right after it yields.

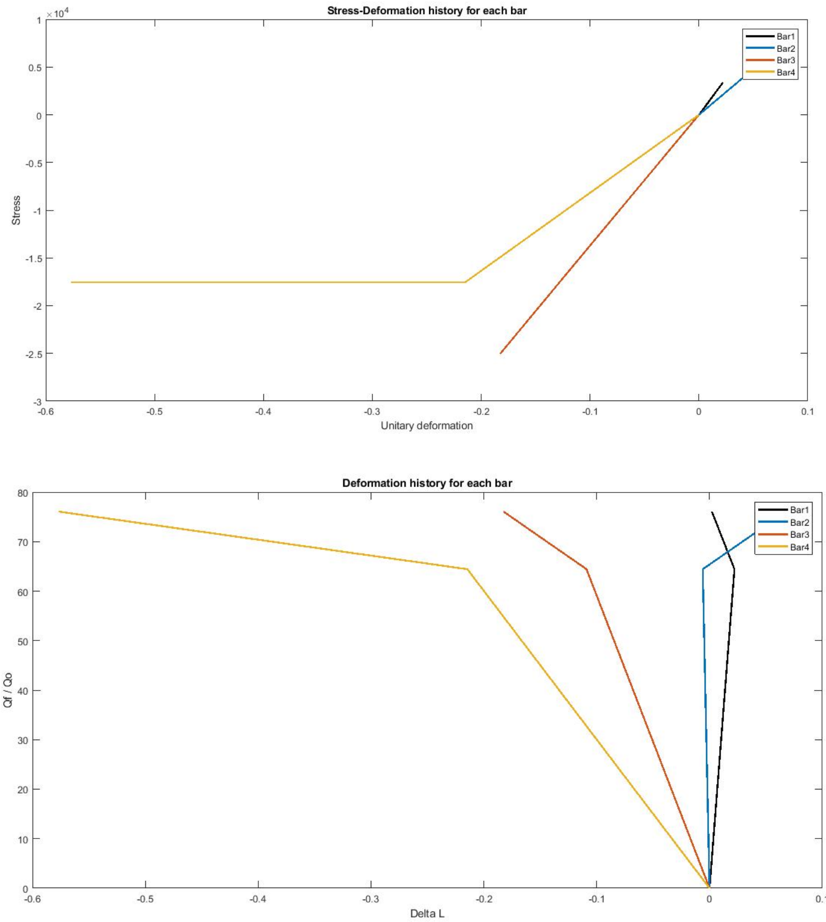

Figure 11. Upper panel: Stress-deformation history of each bar. Lower panel: deformation history at each yield load step.