Scilab Case Study Project

On

1D Finite Element Analysis of a Suspended Column in Scilab GUI

Submitted by

Koustav Bhattacharjee

B.E. 3rd Year, Department of Mechanical Engineering
Jadavpur University, Kolkata

Domain

Finite Element Analysis

August 11, 2025

Abstract

This case study presents development of a 1D FEM-based solver for a statically loaded column and associated GUI to get list of values for various given inputs - comprising of column length, various types of load. The FEM solver is predominantly built upon the Galerkin Method. The model solves for stress, strain, and load distribution along the column, validated against exact solutions, demonstrating good agreement and improved accuracy with increased node discretization[1] .

Keywords: Graphical User Interface, Finite Element Method, Static Loading, Column Analysis, Stress Distribution, Numerical Modeling, Galerkin Method, One Dimensional FEM

1 Introduction

This project presents the development of a graphical user interface (GUI) application for solving a classical problem in structural mechanics using the Finite Element Method (FEM). The model under consideration is a vertically suspended column that is fixed at one end and free at the other. Subjected to axial loading, this setup is a fundamental case study in uniaxial stress analysis and serves as a valuable example for introducing FEM concepts in structural engineering.

The objective is to create an intuitive, user-friendly interface through which users can input physical and geometric parameters—such as length, cross-sectional area, elastic modulus, and applied force—and visualize the resulting stress distribution along the column. The application discretizes the column into finite elements, assembles the global stiffness matrix, applies boundary conditions, and computes nodal displacements and element stresses accordingly.

By abstracting the underlying numerical procedures into a visual and interactive format, this project bridges the gap between theoretical formulation and practical implementation, making it suitable for educational use and initial design exploration in structural analysis.

2 Problem Statement

The project is focused on overcoming persistent challenges in the computational modelling and user interfacing of finite element analysis for statically loaded columns, specifically within the Scilab environment. The primary goals of this work are outlined as follows:

1. **Bridging the Gap Between Theory and Practice with an Accessible Scilab GUI**: Finite Element Method (FEM) tools are often complex and hard to use due to script-based interfaces. A simple, interactive GUI can make FEM-based column analysis more accessible by allowing visual input and instant feedback—all within the Scilab environment.

2. **Enhancing the Fidelity of Column Analysis with Comprehensive Load Modelling**: Traditional column analysis models often overlook key real-world factors like body forces (e.g., gravity) and surface tractions, or rely on coarse mesh discretization, reducing the accuracy of stress and strain predictions. The challenge lies in integrating these loading conditions with finer discretization to achieve more realistic and reliable simulations of column behavior under static loading.

To tackle these challenges, this project presents a reliable and user-friendly finite element solver built entirely in Scilab, paired with a custom GUI. The interactive tool lets users input key geometric and material properties, apply different loading conditions—including point loads, surface tractions, and body forces—and instantly view results like nodal stress, strain, and load distribution.

Validated against analytical solutions and multi-element tests, the Scilab tool offers improved accuracy while making structural analysis more accessible to students, educators, and engineers. This approach encourages clearer understanding and more practical use of FEM in exploring column behaviour under static loading.

3 Basic Concepts Related to the Topic

3.1 The Scilab Graphical User Interface

A graphical user interface (GUI) is a graphical display that contains devices, or components, that enable a user to perform interactive tasks. To perform these tasks, the user of the GUI does not have to create a script or type commands at the command line. Often, the user does not need to know the internal details of the task at hand. GUI components can include menus, toolbars, push buttons, radio buttons, list boxes, sliders, and others.

Each component, and the GUI itself, is associated with one or more user-defined routines known as *callbacks*. The execution of each callback is triggered by a specific user action, such as a button click, mouse selection, menu item activation, or cursor movement over a component. The programmer provides these callback functions, which define how the GUI responds to user interaction. This kind of programming approach is generally referred to as *event-driven programming*.

The process of building a GUI begins with its design. One must decide what the GUI should accomplish, how the user will interact with it, and what components are necessary. After finalizing the design, an appropriate method for creating the GUI in Scilab must be selected. Scilab provides a GUI builder and supports programmatic creation of interface elements. The chosen technique typically depends on the developer's experience, preferences, and the complexity of the intended GUI.

Scilab's GUI capabilities allow for the development of user-friendly applications that can display data visually, guide users through simulation steps, and automate numerical computations in an interactive manner.

3.2 Stress Distribution

In continuum mechanics, stress is a measure of the average force per unit area of a surface within a deformable body on which internal forces act. In other words, it is a measure of the intensity of the internal forces acting between particles of a deformable body across imaginary internal surfaces. These internal forces are produced between the particles in the body as a reaction to external forces applied on the body. External forces are either surface forces or body forces. Because the loaded deformable body is assumed as a continuum, these internal forces are distributed continuously within the volume of the material body; that is, the stress distribution in the body is expressed as a piecewise continuous function of space coordinates and time.

For the simple case of a body axially loaded, e.g., a bar subjected to tension or compression by a force passing through its centroid, the stress σ, or intensity of the distribution of internal forces, can be obtained by dividing the total tensile or compressive force F_n, by the cross-sectional area A, where it is acting upon. In this case, the stress is represented by a scalar called engineering stress or nominal stress that represents an average stress σ_{avg} over the area, meaning that the stress in the cross-section is uniformly distributed. Thus, we have:

$$\sigma_{avg} = \frac{F_n}{A} \approx \sigma \tag{1}$$

In the design of structures, calculated stresses are restricted to be less than a specified allowable stress, also known as working or designed stress. Allowable stress is chosen as some fraction of the yield strength or of the ultimate strength of the material of which the structure is made. The ratio of the ultimate stress to the allowable stress is defined as the factor of safety.

3.3 Finite Element Formulation

Consider a tapered uniaxial stress suspended column as shown in Figure 1. The cross- sectional area varies with x and may be denoted as $A(x)$. The uniaxial stress is denoted as σ, where σ is positive for tensile stresses and negative otherwise. The axial force σA is assumed to vary according to a first-order Taylor expansion as shown in Figure 2. Also shown are the body force $bAdx$, as a result of gravity, where b is the body force per unit volume and surface traction s, where s is the distributed external or surface loading per unit area.

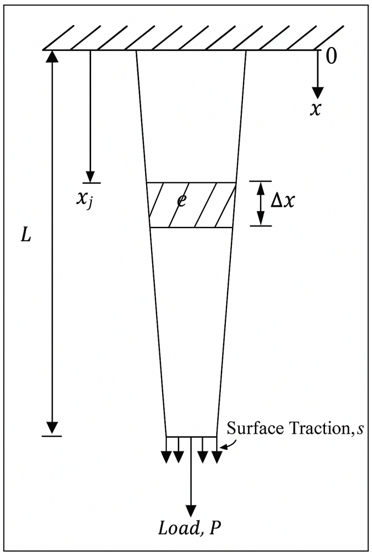


Figure 1: Suspended Column under Applied Load

Resolving the forces on elemental volume of the uniaxial stress column in static equilibrium in the x-direction, we have:

$$\frac{d}{dx}(\sigma A) + sA + bAdx = 0 \tag{2}$$

Solving reduce equation of forced balance using the finite element method by adding the weight function:

$$\int_V bAN\, dV + \int_S sAN\, dS + \int_V \frac{d}{dx}(\sigma A)N\, dV = 0 \tag{3}$$

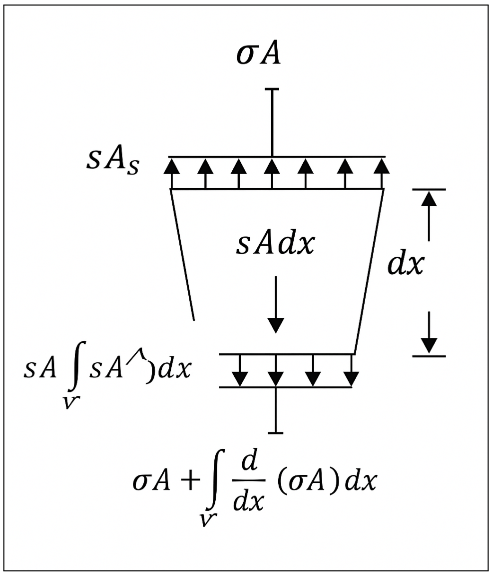


Figure 2: Elemental Portion of the Column

Considering only linear elastic material for which the constitutive relation applies, the stress-strain relationship for the uniaxial stress state is given by:

$$\sigma = E(\varepsilon - \varepsilon_0) + \sigma_0 \tag{4}$$

where E is Young's modulus, ε is the strain, ε_0 is the initial strain (e.g. thermal strain), and σ_0 is the initial stress (pre-stress).

Assuming there is no pre-stress, $\sigma_0 = 0$, and no self-strain, $\varepsilon_0 = \alpha T \Delta T$, due to thermal expansion in the material, the constitutive relationship reduces to a form of Hooke's law:

$$\sigma = E\varepsilon, \quad where \quad \varepsilon = \frac{du}{dx} \tag{5}$$

where u is the axial displacement.

Substituting the Hookes' Law in the Weighted Residual Function, and resolving the first term of equation through integration by parts:

$$\int_V bAN\, dV + \int_S sAN\, dS - \int_V EA\frac{du}{dx}\frac{dN}{dx}\, dV = 0 \tag{6}$$

The vector of nodal unknowns for an element is related to local nodal displacements u_j and u_k at nodes j and k as:

$$\mathbf{a}^{\epsilon} = u_j u_k \tag{7}$$

The strain nodal displacement matrix $B = LN$, where $L = \frac{d}{dx}$ and N is the shape function matrix, and the global nodal displacements are denoted $\mathbf{u} = N\mathbf{a}^\epsilon$. Therefore:

$$\varepsilon = B\mathbf{a}^\epsilon \tag{8}$$

For linear elements, the shape function matrix N is given by:

$$N = \frac{1}{x_k - x_j} x_k - xx - x_j \tag{9}$$

From $B^T = LN^T$ and by replacing the elemental volume $dV = Adx$, and applying the Gauss Divergence Theorem on the third term of equation, we have:

$$\int_V bdVN + \int_S sdSN - \int_V EB^T B\, dV\mathbf{a}^\epsilon = 0 \tag{10}$$

Rearranging weak form equation in the form of $k^\epsilon \mathbf{a}^\epsilon = \mathbf{f}^\epsilon$, where k^ϵ is the element stiffness matrix and $\mathbf{f}^\epsilon$ is the element nodal force vector, it results in:

$$k^\epsilon = \int_V EB^T B\, dV, \quad \mathbf{f}^\epsilon = \int_S sN\, dS + \int_V bN\, dV \tag{11}$$

Therefore, from equation of force and stiffness, the following relationships hold:

$$k^\epsilon = \int_V EB^T B\, dV, \quad \mathbf{f}^\epsilon = \mathbf{f}_s + \mathbf{f}_b, \quad \mathbf{f}_s = \int_S sN\, dS, \quad \mathbf{f}_b = \int_V bN\, dV \tag{12}$$

The element strain, element stress, and element average load on the column can be computed from:

$$\varepsilon = B\mathbf{a}^\epsilon, \quad \sigma = EB\mathbf{a}^\epsilon, \quad F = A\sigma \tag{13}$$

The above formulation enables the study of uniaxial static loading on a suspended column, accounting for surface traction and body forces, using the finite element method.

4 Flowchart

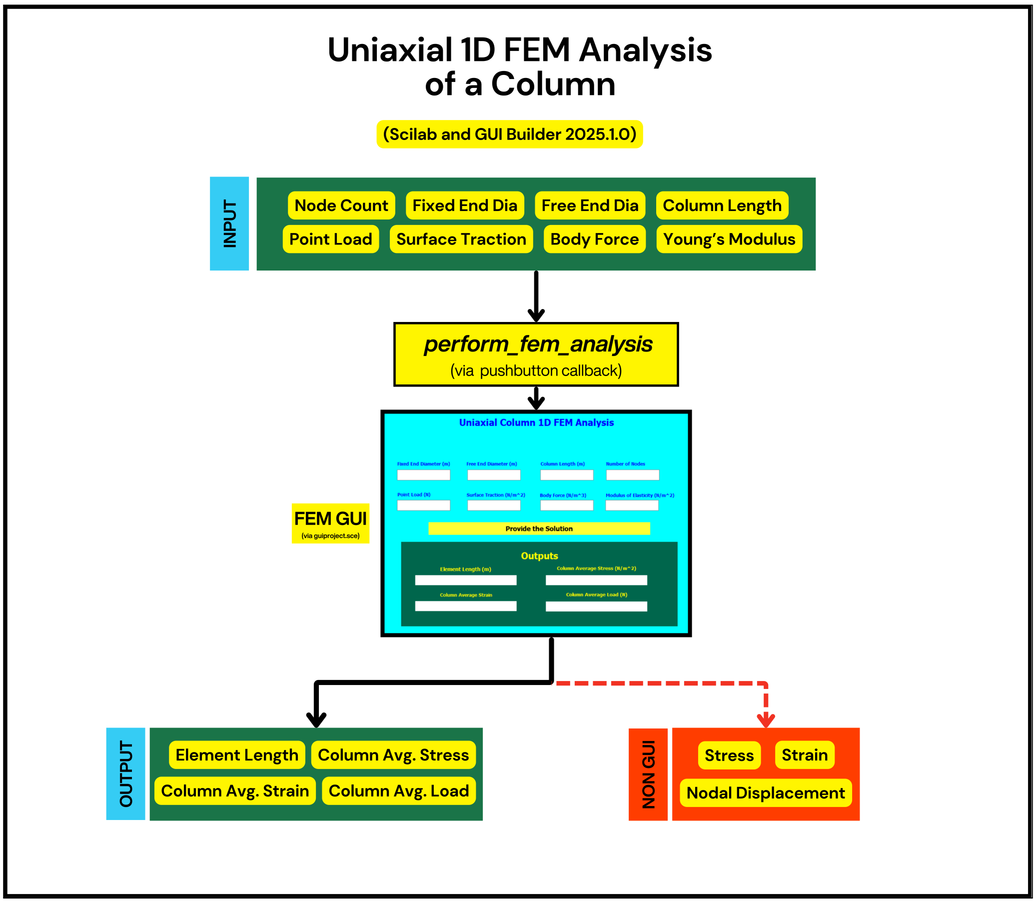


Figure 3: Entire FEM Analysis Code Structure and Execution (Basic Outline)

5 Software/Hardware Used

The software that has been primarily used is the normal **Scilab 2025.1.0** scripting console. Alongside, the GUI builder from the Atoms GUI library of Scilab plugins. The device on which the program has been ran is a **Windows 11 OS** machine.

6 Procedure of Execution

The execution of the code can be done as follows:

1. Open Scilab on desktop. For the GUI for basic four outputs, the **guiproject.sce** is clicked on and opened and the file is executed preferably without echo.

2. This opens the GUI interface, the GUI has the input fields to be filled.

3. Click the solution pushbutton to get the outputs.

4. Tune the requisites as you need.

5. Get the additional stress and strain datapoints and utilize it for tabular data creation on your own.

6.1 Acceptable Input Value Range

The acceptable input value ranges for the analysis to be fruitful are listed for reference as follows:

Table 1: Acceptable Input Ranges for 1D FEM Column Analysis GUI

Parameter	**Minimum**	**Maximum**	**Unit**
Column Length	0.01	10.00	m
Fixed-End Diameter	0.001	1.000	m
Free-End Diameter	0.001	1.000	m
Number of Nodes	2	200	–
Point Load	0	1.0×10^{6}	N
Surface Traction (Distributed)	0	1.0×10^{8}	N/m^2
Body Force (per Volume)	0	1.0×10^{6}	N/m^3
Young's Modulus	1.0×10^{6}	2.1×10^{11}	N/m^2

7 Results

The GUI has been built swiftly using the Scilab GUI builder tool. The GUI includes all the parameters similar to the GUI generated in the reference paper using MATLAB.

The interface has the inputs of the **fixed end and free end diameter of the column, column length, number of nodes, point load value, body force, surface traction and modulus of elasticity** - all of the inputs in SI unit.

The outputs obtained are the average column stress and strain, the element length and the average column load.

The graphs that are obtained are the stress and strain curves from these values. The graph interpretations are written consequently after each diagram.

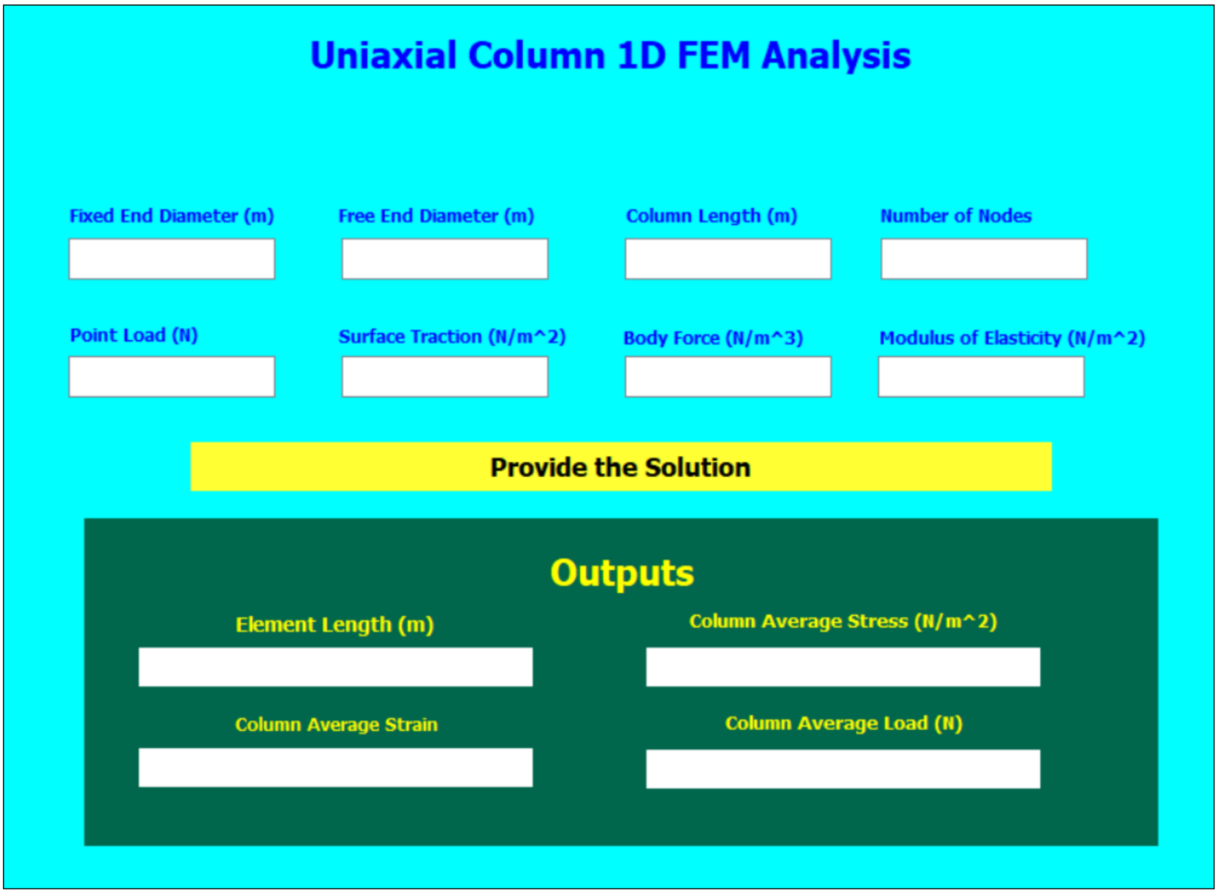


Figure 4: Uniaxial Column FEM Analysis GUI

To obtain a certain standard result, the following parameters have been taken.

Parameter	**Value (SI)**
Column Length	0.6 (m)
Fixed End Diameter	0.1 (m)
Free End Diameter	0.075 (m)
Nodes	20
Point Load	1500 (N)
Surface Traction	$5 \times 10^6 (N/m^2)$
Body Force	77008.5 (N/m^3)
Modulus of Elasticity	20.7×10^{10} (N/m^2)

Table 2: Parameters used for action of all forces

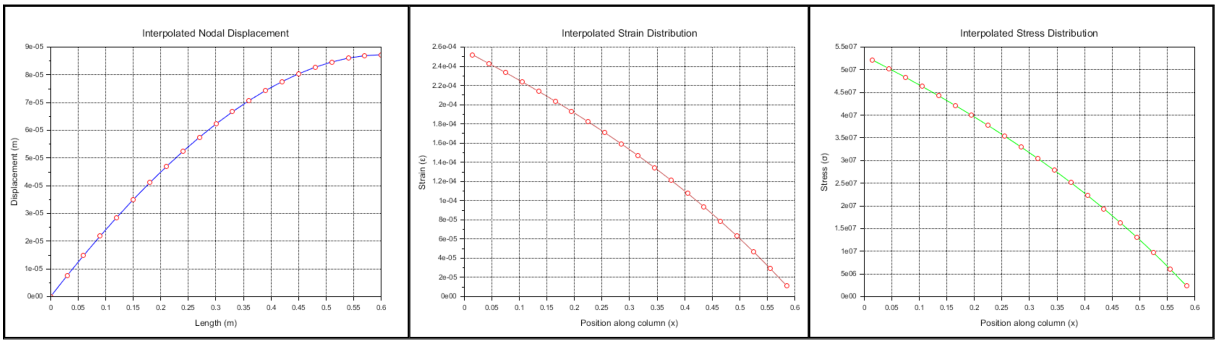


Figure 5: Nodal Displacement. Stress and Strain curves when all forces acting

Parameter	**Value (SI)**
Column Length	0.6 (m)
Fixed End Diameter	0.1 (m)
Free End Diameter	0.075 (m)
Nodes	20
Point Load	1500 (N)
Surface Traction	5 $\times 10^6 (N/m^2)$
Body Force	0
Modulus of Elasticity	20.7×10^{10} (N/m^2)

Table 3: Parameters used for action for zero body force calculation

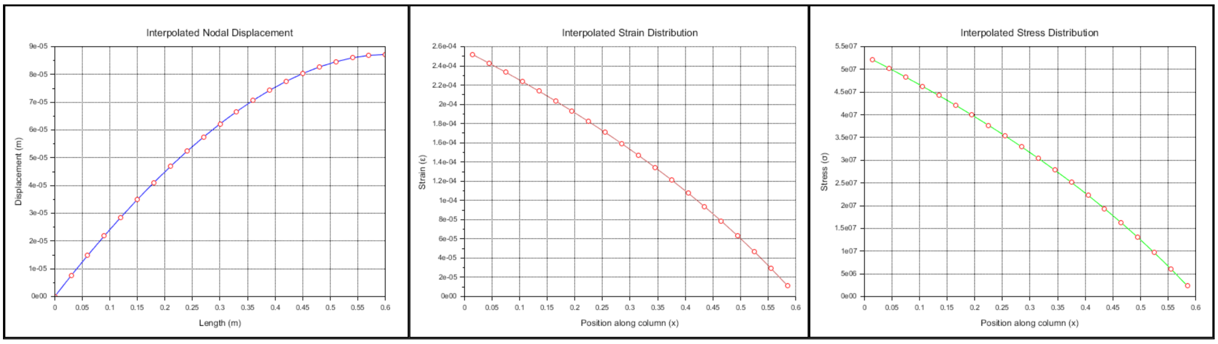
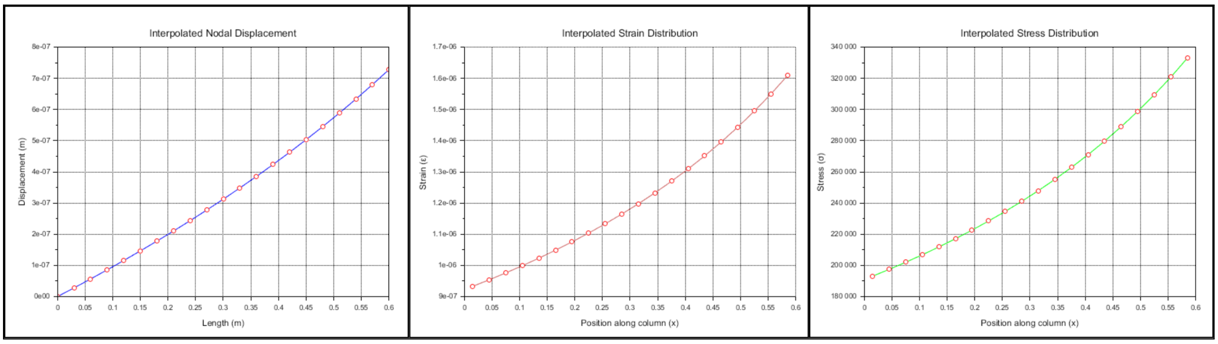


Figure 6: Nodal Displacement. Stress and Strain curves with zero body force



Parameter	Value (SI)
Column Length	0.6 (m)
Fixed End Diameter	0.1 (m)
Free End Diameter	0.075 (m)
Nodes	20
Point Load	1500 (N)
Surface Traction	0
Body Force	0
Modulus of Elasticity	20.7×10^{10} (N/m^2)

Table 4: Parameters used for the action of zero body force and traction

Figure 7: Nodal Displacement. Stress and Strain curves body force and surface traction zero

These curves don't come as axis box in the GUI, the graphs come as separate graphic windows due to the inherent graph plotting functionalities inside the callback code. These are 20 node values.

8 Observations

The analysis done as per[1] was done, for node counts 2 (single element) and 20 (multi-element). The results that were obtained for the column average values are written.

Nodes	Average Load (N)	Average Stress (N/m^2)	Average Strain
2	1500	3.42×10^6	0.0001653
20	1500	3.01×10^6	0.0001454

Table 5: Values obtained on executing function with all loads

The Gauss Quadrature integrals showed errpneous results on testing and hence the basic midpoint integration in performed. [1].

Yet this model distinctively depicts few aspects of an uniaxial column stress analysis.

1. **Increase of strain and stress towards fixed to free end.**
2. **Removal of body force does not heavily impact the dynamic conditions of a column, as it is mostly constant in case of a column.**
3. **The removal of surface traction impacts the dynamic condition to a severe extent. The point load flattens both the stress and the strain graphs.**

The issue regarding the value matches comes as that the curves suggest the strains and stresses of order 1/10 times of the values depicted in the table of the referred paper. Moreover our scheme delves out the value to be close in regard to nodal displacements, but the value increased by 10 times in the current study as well. As the referenced value parameters used for validation are not aligned, even when a book has been cited for validation - it is still questionable and maybe occurring due to differed weigth distribution analysis.
Thus the 1D finite element analysis of a uniaxial column can be concluded.

References

[1] M. O. Petinrin, "Development of graphical user interface for finite element analysis of static loading of a column using matlab," 2010.