Investigating Meshing Effects and Element Type Selection in Cantilever Beam Simulations with OpenSeesPy

Prathamesh Varma
Student, Government College of Engineering, Aurangabad - 431005
Email: prathamesh66523@gmail.com

Keywords: Finite Element Analysis, Mesh Refinement, Element Type Selection, OpenSeesPy, Structural Analysis

Declarations

Funding

This research did not receive any specific grants from any funding agency in the public, commercial, or not-for-profit sectors.

Conflicts of Interest

The author declares no conflicts of interest relevant to the content of this article.

Ethics Approval

Not applicable. This study did not involve human participants or animals.

Consent to Participate

Not applicable.

Consent for Publication

I have read and approved the final manuscript and consented to its submission for publication.

Availability of Data and Materials

The datasets and codes generated and/or analyzed during the current study are available at: https://github.com/Prathamesh001/openseespy-cantilever-study

Authors' Contributions

Prathamesh Varma: Conceptualization, Methodology, Data curation, Software, Writing—original draft, and

Abstract

This paper presents a comparative study on the effect of mesh density in finite element modeling of a cantilever beam subjected to a point load at its free end. Two modeling approaches in OpenSeesPy were investigated: the ElasticBeamColumn element and the quadrilateral plane stress element. The results indicate that the ElasticBeamColumn element yields accurate deflection and stress results even with a single element, whereas the plane stress model exhibits deviations even at high mesh densities (10×300). These findings highlight the importance of appropriate element selection and meshing strategies in structural analysis. This study also demonstrates the use case for beam and quadrilateral elements in OpenSeesPy.

1 Introduction

Finite Element Analysis (FEA) is a widely used tool in structural engineering. The accuracy of the FEA depends significantly on the type of elements used and the mesh density chosen[3]. In this study, we examine the impact of mesh refinement on the analysis results of a cantilever beam subjected to a point load at its free end. We compared two modeling strategies in OpenSeesPy: one using the line-based ElasticBeamColumn element and the other using a 2D continuum-based quadrilateral plane stress element.

2 Problem Definition

The problem analyzed is a cantilever beam of length 3 m and cross section of $0.1 \times 0.1m^2$, fixed at one end and subjected to a vertical point load of 100 N at its free end. The objective was to study the accuracy of the deflection, bending stress at the section, and maximum bending moment, and to compare the performance of the two modeling approaches under varying mesh densities using this simple problem.

3 Methodology

3.1 Modeling in OpenSeesPy

Two separate models of the cantilever beam were developed in OpenSeesPy[7] based on the OpenSees framework[5, 6].

- **ElasticBeamColumn Element Model**: The beam was modeled as a single line element. This element has three degrees of freedom per node. A sample plot using this element is shown in Figure 1.

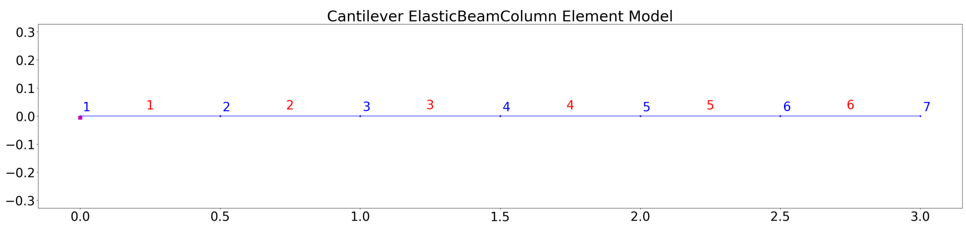


Figure 1: Plot of Beam using ElasticBeamColumn elements

- **Quadrilateral Plane Stress Element Model**: The beam was modeled as a 2D continuum using plane stress quadrilateral elements with varying mesh densities. The quadrilateral element is a 2D element with two degrees of freedom per node. A sample plot using this element is shown in Figure 2.

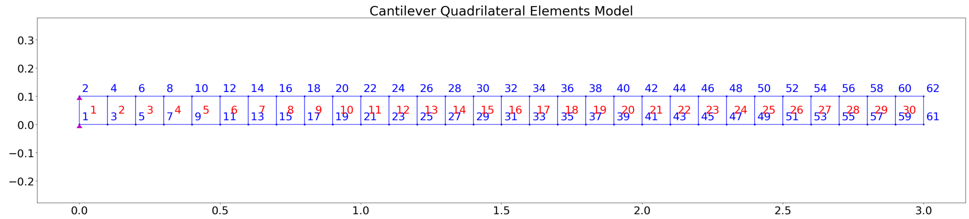


Figure 2: Plot of Beam using Quadrilateral Elements

3.2 Example OpenSeesPy Implementation

A simplified OpenSeesPy script for a cantilever beam with ElasticBeamColumn elements is shown below.

```python
import openseespy.opensees as ops
import matplotlib.pyplot as plt
import opsvis as opsv

# Model and units
ops.wipe()
ops.model('basic', '-ndm', 2, '-ndf', 3)

# Nodes
ops.node(1, 0, 0)
ops.node(2, 3.0, 0)

# Boundary condition
ops.fix(1, 1, 1, 1)

# Section properties
E = 210e9
I = 8.333e-6
A = 0.01

# Define geometric transformation
ops.geomTransf('Linear', 1)

# Define element
ops.element('elasticBeamColumn', 1, 1, 2, A, E, I, 1)

# Loading
ops.timeSeries('Linear', 1)
ops.pattern('Plain', 1, 1)
ops.load(2, 0, -100, 0)

# Analysis
ops.system('BandGeneral')
ops.numberer('RCM')
ops.constraints('Plain')
ops.integrator('LoadControl', 1.0)
```

```
ops.algorithm('Linear')
ops.analysis('Static')
ops.analyze(1)

#Plot the model
fig1, ax1 = plt.subplots(figsize=(30, 12))
opsv.plot_model(node_labels=1, element_labels=1, ax = ax1)
plt.title("Cantilever ElasticBeamColumn Element Model")

# Results
print("Tip deflection:", ops.nodeDisp(2))
```

This snippet demonstrates the minimal setup for modeling a cantilever beam using beam elements. For the plane stress quadrilateral model, the approach was extended to generate 2D meshes, define material properties, and integrate stresses to obtain sectional forces and moments. For details on the available commands, refer to the OpenSeesPy documentation[7].

3.3 Meshing Strategy

In both element-type models, the mesh density was increased per iteration. Mesh refinement is essential for solution convergence in continuum elements[1, 4]. For the plane stress model, multiple mesh densities were tested, with a maximum mesh density of 10 elements along the height and 300 elements along the length of the beam. The ElasticBeamColumn model was also tested with varying numbers of elements, although even a single element was sufficient to provide accurate results per beam theory.

3.4 Loading and Boundary Conditions

- Fixed support at one end.
- 100 N point load applied vertically downward at the free end.

4 Results

4.1 Deflection

Table 1 shows a comparison of the tip deflections obtained at every iteration of increasing mesh density. Such convergence studies are standard practices for verifying the finite element accuracy[3, 8, 2]. The variation in the mesh densities can be observed in Figures 3 and 4.

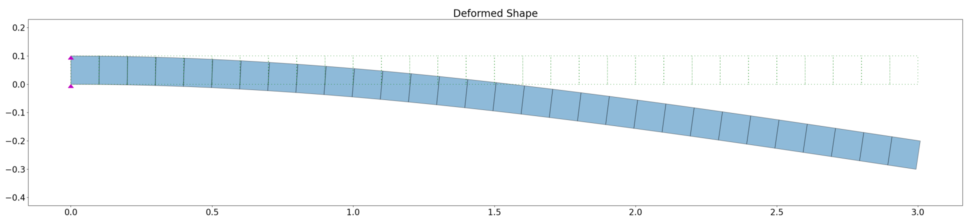


Figure 3: Deformation of Cantilever Beam with Quadrilateral Element Mesh of Density 1×30

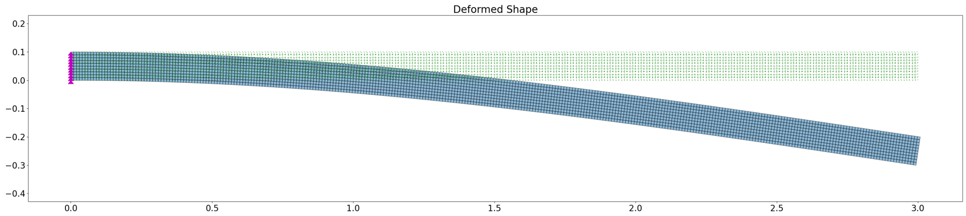


Figure 4: Deformation of Cantilever Beam with Quadrilateral Element Mesh of Density 10×300

Table 1: Simulation Results with Element Size, Mesh Density, and Convenient Units

Element Size (m)	Mesh Density	Deflection (mm)	Bending Moment (Nm)	Max Stress (MPa)
0.1×0.1	1×30	-0.352895	4.105095×10^{-11}	0.000080
0.05×0.05	2×60	-0.461505	100.6410	0.805128
0.033×0.033	3×90	-0.489503	189.4880	1.136928
0.025×0.025	4×120	-0.500138	231.3322	1.310616
0.02×0.02	5×150	-0.505225	253.3962	1.417681
0.0166×0.0166	6×180	-0.508034	266.3032	1.491517
0.01428×0.01428	7×210	-0.509744	274.4743	1.546497
0.0125×0.0125	8×240	-0.510861	279.9677	1.589784
0.011×0.011	9×270	-0.511631	283.8376	1.625308
0.01×0.01	10×300	-0.512183	286.6673	1.655401

4.2 Stress Distribution

Figure 5 shows the bending stress variation across the support cross-section.

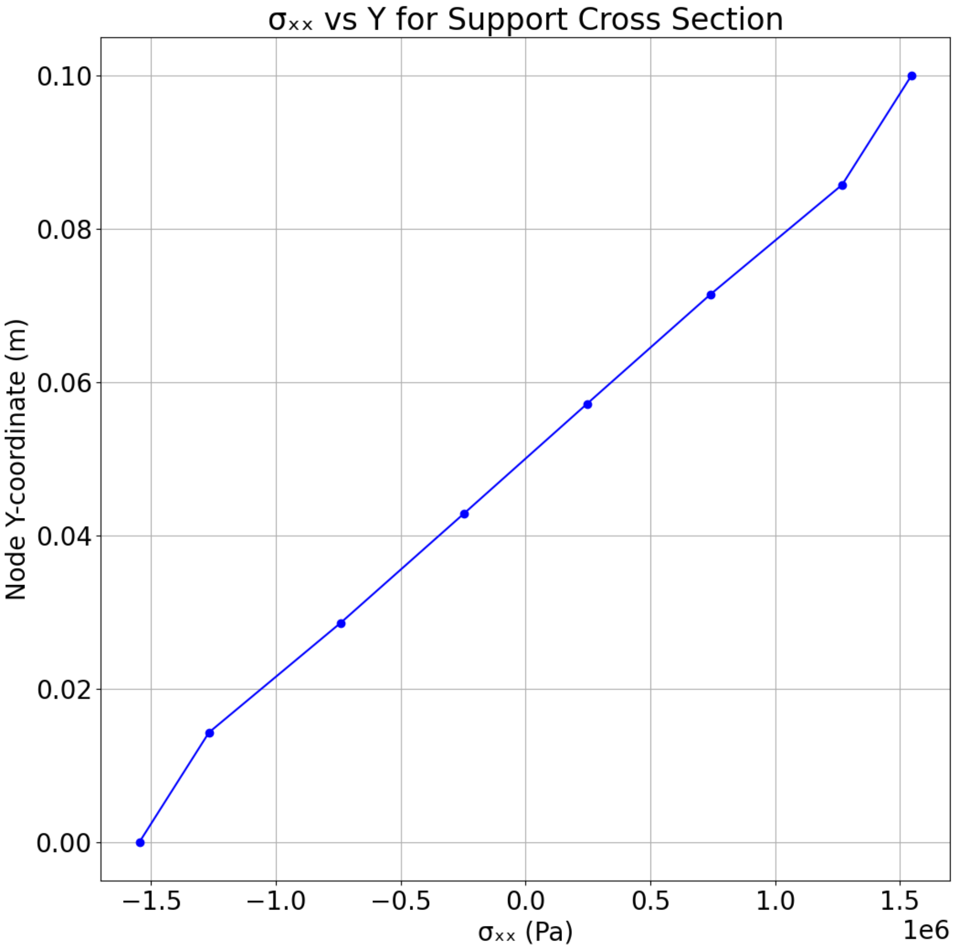


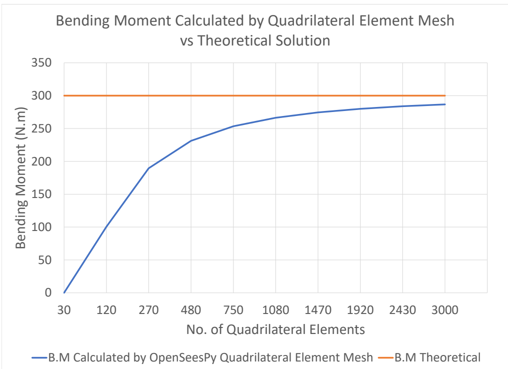
Figure 5: Stress Distribution at Mesh Density 7×210 at Support Section using Quadrilateral Element

4.3 Key Observations

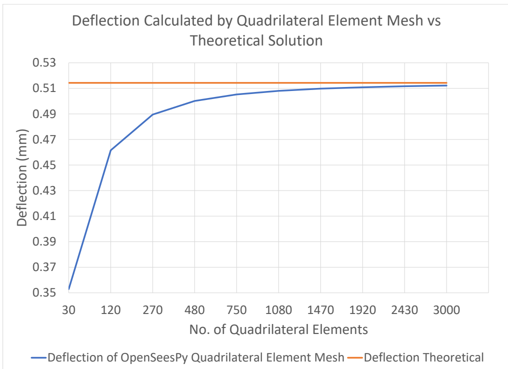
- The ElasticBeamColumn element produced results matching theoretical solution even with a single element. The deviations in the results obtained using the quadrilateral elements are listed in Table 2.
- Plane Stress models required very high mesh densities to approach accurate results, but deviations persisted even then.
- Convergence of quadrilateral element mesh towards the theoretical solution is found to be as shown in Figure 6.

Table 2: Deviation Results with Varying Number of Quadrilateral Elements

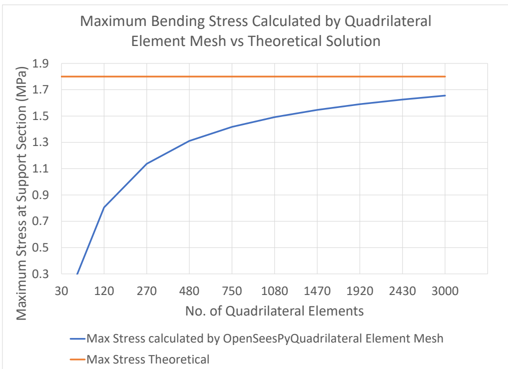
Number in Depth	Number in Length	B.M Deviation (%)	Bending Stress Deviation (%)	Deflection Deviation (%)
1	30	100.0000	99.9956	31.3808
2	60	66.4530	55.2707	10.2619
3	90	36.8373	36.8373	4.8178
4	120	22.8893	27.1880	2.7499
5	150	15.5346	21.2399	1.7607
6	180	11.2323	17.1379	1.2145
7	210	8.5086	14.0835	0.8820
8	240	6.6774	11.6787	0.6648
9	270	5.3875	9.7051	0.5151
10	300	4.4442	8.0333	0.4078



(a) Convergence of Maximum Bending Moment per Increase in Mesh Density



(b) Convergence of Maximum Deflection per Increase in Mesh Density



(c) Convergence of Maximum Bending Stress per Increase in Mesh Density

Figure 6: Variation of Analysis Results per Increase in Mesh Density

5 Discussion

The ElasticBeamColumn element inherently captures the analytical beam behavior under bending, thereby explaining its accuracy with minimal discretization. In contrast, the plane stress element approximates the 2D continuum behavior and is sensitive to mesh refinement. This was the case even with a highly refined mesh. These findings from this simple problem reinforce the need to choose element types appropriate for the problem being solved. Higher mesh densities showed result trends approaching the theoretical solution.
While our study shows the superior efficiency of ElasticBeamColumn elements for this simple beam problem, it is important to recognize that quadrilateral plane stress elements have critical applications in structural analysis, particularly in contexts where beam theory assumptions no longer hold.

Applicability of Quadrilateral Plane Stress Elements

While the ElasticBeamColumn element is efficient for slender beams under bending, Quadrilateral Plane Stress elements are essential in the following scenarios [3, 8]:

- **2D Stress Fields:** Required for capturing in-plane stress distributions in plates, shear walls, and diaphragms.
- **Complex Geometries:** Suitable for regions with cutouts, holes, or variable thickness that cannot be represented by 1D line elements.
- **Stress Concentrations:** Able to model localized stress variations near openings, connections, or discontinuities.
- **In-plane Loading:** Necessary for problems with distributed in-plane loads and supports acting over 2D domains.

6 Conclusion

This study demonstrates that for simple cantilever beam problems under point loading, line elements such as ElasticBeamColumn in OpenSeesPy provide highly accurate results with minimal computational effort. quadrilateral plane stress elements, while capable of modeling complex geometries and stress states, require extremely fine meshes to match beam theory results. Careful consideration of the element type is essential for an efficient and accurate finite-element analysis.

7 Code

Find the code used for obtaining the above results at my repository: `https://github.com/Prathamesh001/openseespy-cantilever-study`

References

[1] Klaus-Jürgen Bathe. *Finite Element Procedures.* Prentice Hall, 2006.

[2] Ted Belytschko, Wing Kam Liu, Brian Moran, and Khalil Elkhodary. Nonlinear finite elements for continua and structures. 2013.

[3] Robert D. Cook, David S. Malkus, Michael E. Plesha, and Robert J. Witt. *Concepts and Applications of Finite Element Analysis*. John Wiley & Sons, 4 edition, 2002.

[4] Thomas J.R. Hughes. *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*. Dover Publications, 2000.

[5] Silvia Mazzoni, Frank McKenna, Michael H. Scott, and Gregory L. Fenves. *OpenSees Command Language Manual*. Pacific Earthquake Engineering Research Center, 2006.

[6] Frank McKenna, Gregory L. Fenves, Michael H. Scott, et al. *OpenSees: A Framework for Earthquake Engineering Simulation*. Pacific Earthquake Engineering Research Center, 2011. `https://opensees.berkeley.edu`.

[7] Dong Zhu et al. *OpenSeesPy Documentation*, 2022. `https://openseespydoc.readthedocs.io`.

[8] Olek C. Zienkiewicz, Robert L. Taylor, and J.Z. Zhu. *The Finite Element Method: Its Basis and Fundamentals*. Butterworth-Heinemann, 6 edition, 2005.