ELASTOPLASTIC STEEL BEAM BENDING ANALYSIS BY USING ABAQUS

Biswajit Jena*

***Corresponding author: biswa.tech88@gmail.com**

Abstract: In recent years tremendous efforts have been made in development of numerical methods for the analysis of solids and materials. The elasto-plastic behaviour of beams has been analysed by a numerical method called Finite Element Method. The engineering significance of material nonlinearities varies greatly across the disciplines. They seem to occur most often in civil engineering, that deals with inherently nonlinear material such as concrete, soil and low strength steel. In mechanical engineering creep in plasticity in combination with strain hardening and thermal effect are most important. Material nonlinearity give rise to very complex phenomena like hysteresis localisation, fatigue and progressive failure.

Key words: Beam, Beam sections, ABAQUS, elasto-plastic behavior

1. Introduction:

The analysis explains the non-linear response of a structure when subjected to loads. There are three sources of nonlinearity in structural mechanics simulations: Material nonlinearity – Material nonlinearity can be a function of strain, temperature etc. Boundary nonlinearity – Boundary non-linearity occurs if the boundary conditions change during the analysis. Geometric nonlinearity – It deals with change in geometry of structure caused due to large deflection, initial stress. Problems on non-linear analysis can characterized into two types. Non-linear elastic problem – Here stress-strain curve of the material is non-linear, but material behaviour is elastic with all deformations and deflections recoverable on unloading. Elasto-plastic problem – It is characterized by an initial elastic material response onto which a plastic deformation is superimposed after a certain level of stress has been reached.

Abaqus is a suite of powerful engineering simulation program, based on the finite element method, that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus offers a wide range of capabilities for simulation of linear and nonlinear applications. Problems with multiple components are modeled by associating the geometry defining each component with the appropriate material models and specifying component interactions. In a nonlinear analysis Abaqus automatically chooses appropriate load increments and convergence tolerances and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently.

D. R. J. Owen and E. Hinton, "Finite Elements in Plasticity: Theories and Practice" – the book has presented and demonstrated the use of FEM based method for solution of problems involving plasticity. The detail theory and algorithm in the form of modular coding written in FORTRAN is also given. HuiShenShen [1998] Nonlinear bending of simply supported rectangular Mindlin plate under transverse and inplane loads and resting on elastic foundation. Formulations are based on considering first order shear deformation effect. Large deflections are significantly influenced by loaded area, foundation stiffness, amount of initial pressure load.

2. METHODOLOGY:

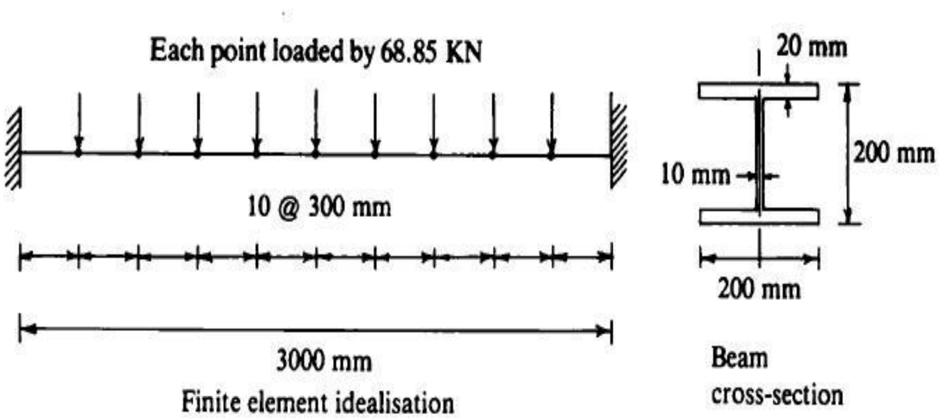
Modeling and analysis of beam in Abaqus software includes the following steps: (1)Pre-processor (2)Solutions (3)General post-processor (4)Time History post-processor

For the analysis of structural member, ABAQUS has been chosen for the purpose of modeling and analyzing the Steel beam with steel in this study due to its flexibility in creating geometry and material modeling.

3. PROBLEM STATEMENT:

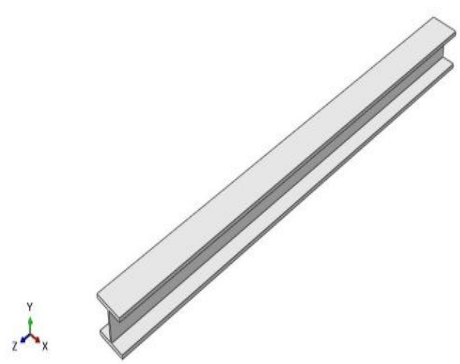
Before doing the non-linear mindlin plate bending analysis, the example problem on Timoshenko beam given in the book "Finite Elements Plasticity " by D.R.J. Owen and E. Hinton has been analyzed by using Abaqus software.

A simply supported beam is subjected to udl. The beam properties and proportions are given below. Loading on the beam has been shown in figure. E = 2.1×105 N/mm2, υ = 0.3, σ 0 = 250 N/mm2 ,udl = 2.295 kN/mm^2, depth of flange = 20mm, width of flange = 200mm, depth of web = 160mm, width of flange = 10mm, L = 3000 mm, I = 68.48×10^4 mm^4

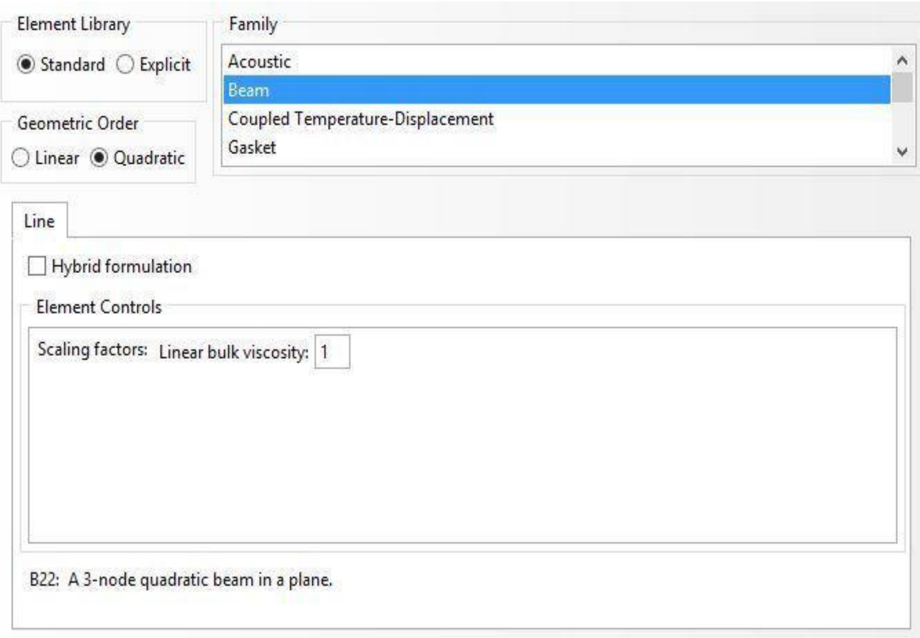


4. Experimental Procedure & Modeling

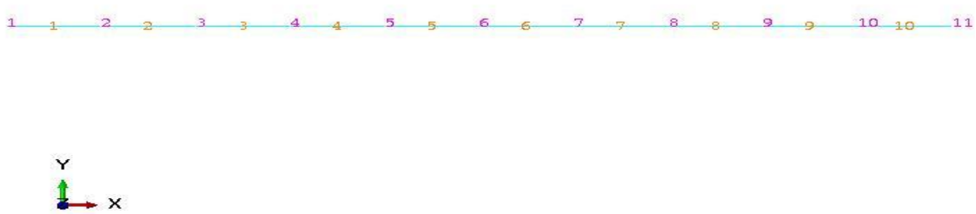
Step 1 : creating beam element, assigning material properties, load, boundary condition



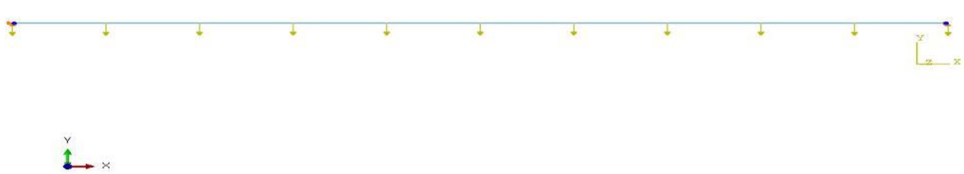
Here the Timoshenko beam element taken is B22 since we are considering transverse shear deformation



Step 2 : Meshing the beam element



Step 3: load on each element



Copyright©2013 DRIEMS

5. Result Analysis & Discussion

Table-1: Non-linear analysis: plastic properties

True stress	**Plastic strain**
0	0
250	0.0012
255	0.007

Putting time period = 1
Time period increment size = 0.01
Minimum increment = 10-5
Maximum increment = 1
Number of iterations = 100

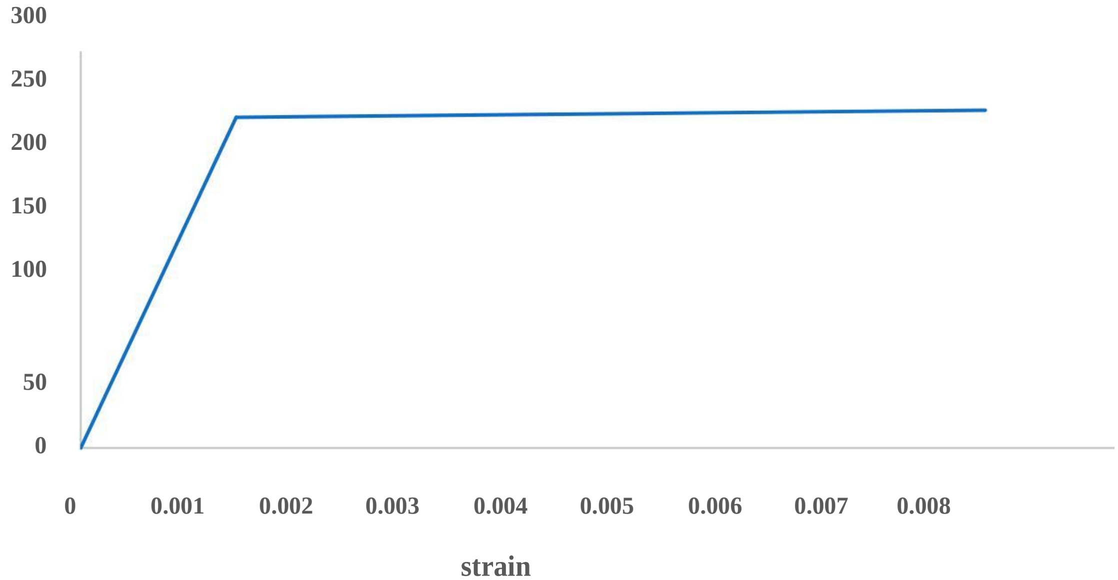


Fig-1: true stress vs plastic strain diagram

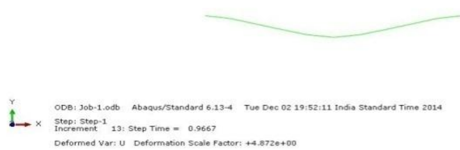


Fig-2: Deflected shape of beam(non-layered approach)

Copyright©2013 DRIEMS

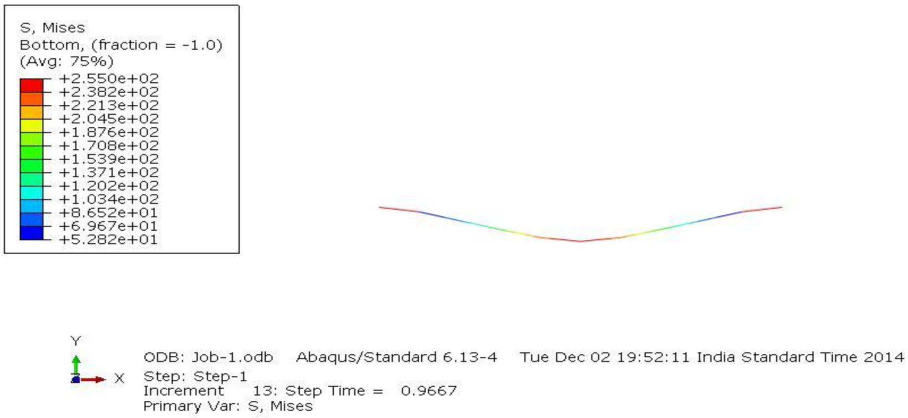


Fig-3: Stress distribution diagram (non-layered approach)

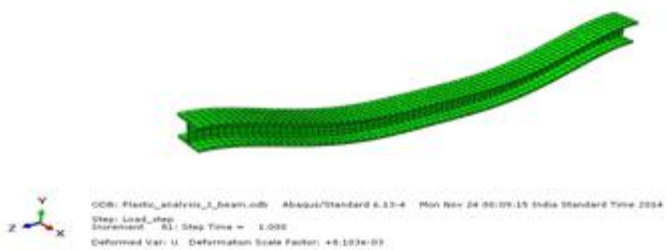


Fig-4: Deflected shape of beam (layered approach)

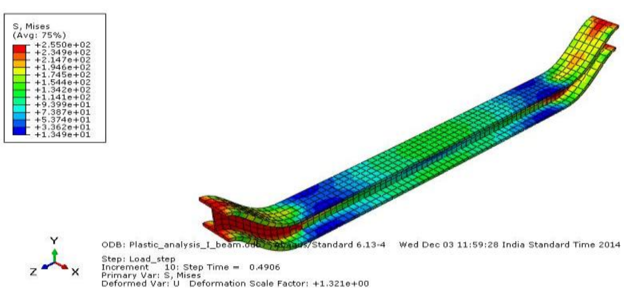


Fig-5: Sress values After 20th increment of load (load at each node= 13.77kN)

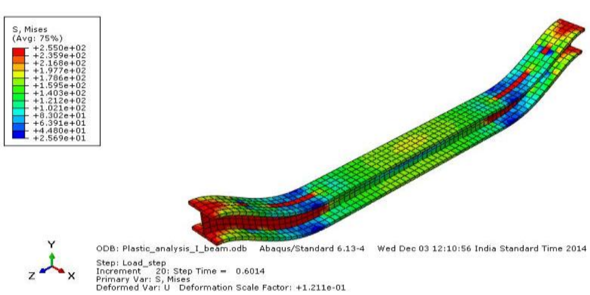


Fig-6: After 30th increment of load(at each node=20.655kN)

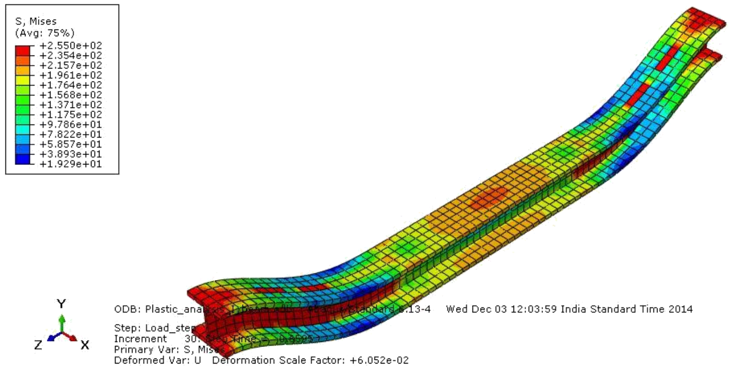


Fig-7: After 50th increment of load(at each node = 34.425kN)

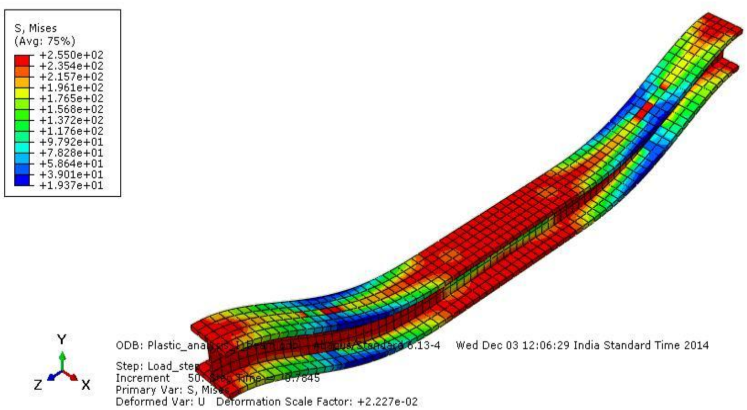


Fig-8: After 80th increment of load(at each node=55.08kN)

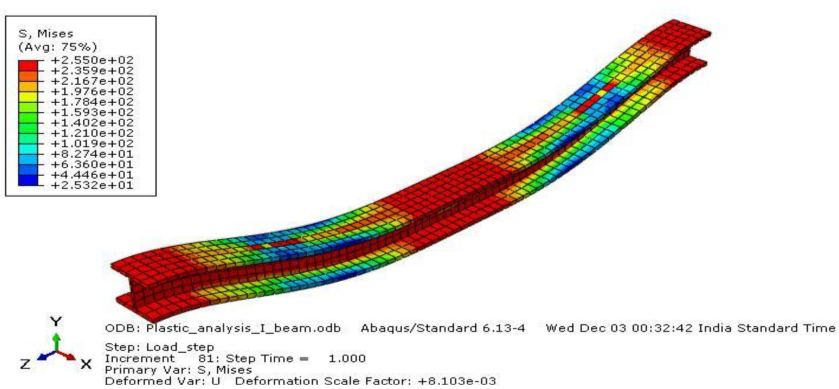
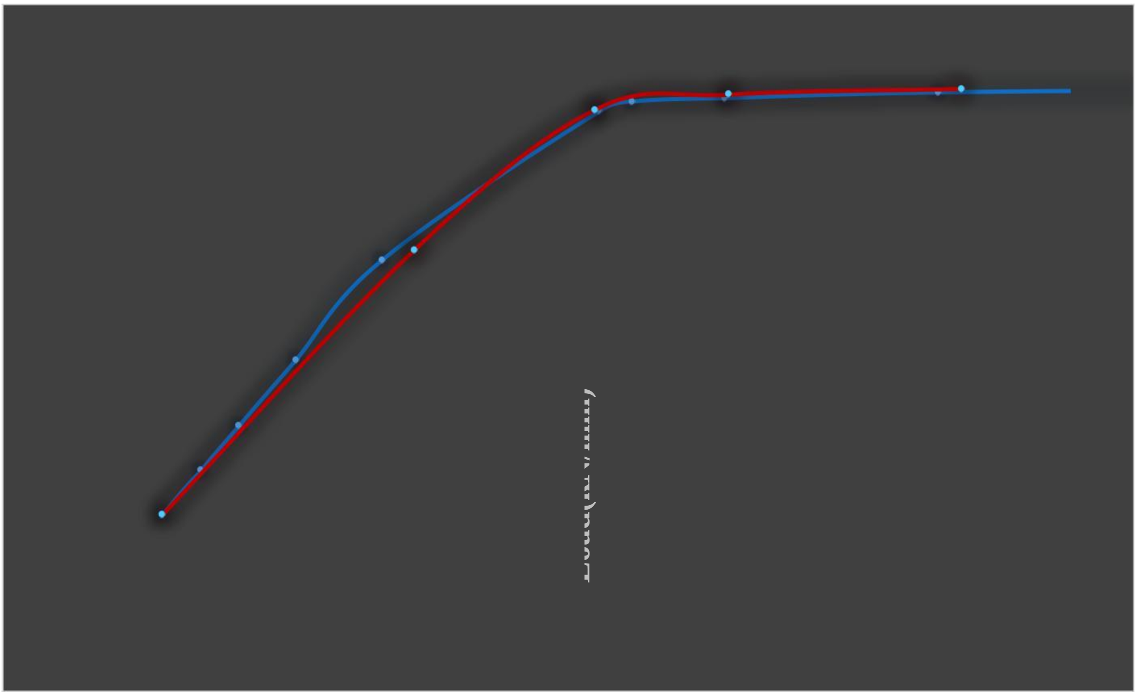


Fig-9: After 100th increment of load(at each node=68.85kN)



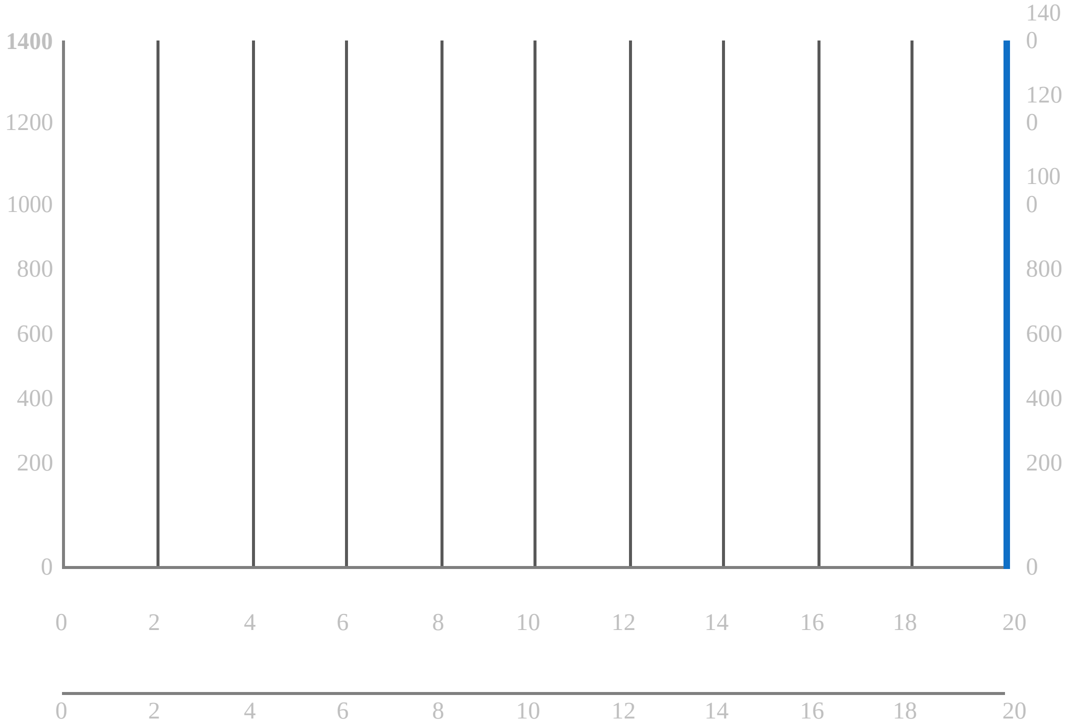


Fig-10: Load ~ deflection curve at centre point

6. Conclusion

The results obtained from Abaqus software is compared with the results obtained from Matlab software and it is found that both values are in good response with each other.

References

1. D. R. J. Owen and E. Hinton, "Finite Elements in Plasticity: Theories and Practice" , 1980, page 1-31, 121-152. – the book has presented and demonstrated the use of FEM based method for solution of problems involving plasticity. The detail theory and algorithm in the form of modular coding written in FORTRAN is also given.
2. M. AsgarBhatti, " Advanced Topics in Finite Element Analysis of structures" [publication in 2005 by John Willey and Sons].
3. Abaqus 6.12 – getting Started with abaqus- manual
1. JN Reddy , " An Introduction to nonlinear Finite Element Method" , OXFORD University, UK,[2004] Chandramani Mishra @ all [2013] : "Failure Analysis of Laminated Composite Beam by FEM", (Vol 2 , Issue 10, October 21013, page 198-204)
5. Shahrestany A. and Fallah N. [2013] : a novel finite volume based formulation for the elasto-plastic analysis of plates.
6. "Experimental abd Theoretical analysis of Composite Beam" (AI Khwarizmi Engineering Journal, Vol. 8, No. 3, PP 12-23)

Biographical Notes

Biswajit Jena is Assistant Professor, Department of Civil Engineering, DRIEMS, Cuttack. He did his M. TECH in Structural Engineering from NIT, Rourkela. He has more than 4 years of teaching & industrial experience. He has published journal papers & attended many conferences on concrete technology.

Copyright©2013 DRIEMS