

# Comparison of static analysis of a pipe under 1-D, 2-D, 3-D Meshing

Rajat Garg  
University of Petroleum &  
Energy Studies, Dehradun  
rajatgarg95@live.com

Siddharth Negi  
University of Petroleum &  
Energy Studies, Dehradun  
negisiddharth95@gmail.com

## ABSTRACT

Finite element analysis (FEA) gives solution of engineering problems using computers. FEA technique facilitates an easier and a more accurate analysis. In this technique, meshing of the whole structure is done in which the structure is divided into very small simple shapes/elements, such as triangles or quadrilaterals in two dimensions and tetrahedra or hexahedra in three dimensions. Individual behaviour of these elements is known and, based on this knowledge; behaviour of the entire structure is determined.

In FEA, type of meshing is a critical issue. It closely relates to the accuracy, computing time and efforts required for meshing of finite element models, which determines their complexity level. This paper presents study of the effects of type of meshing on accuracy of numerical analysis results. Based on these results the guidelines for choosing the appropriate mesh strategy in finite element modelling are provided. The analysis of the hollow tube under different condition is carried out by using ANSYS. The model under study is made up of structural steel.

## 1. INTRODUCTION

In FEA, almost all structures can be approximated by the following basic elements:

**1-D Line elements:** Element consisting of two nodes. Example: Truss and beam elements. In computers, a line, connecting two nodes at its ends as shown, represents a line element. The cross-sectional area is assumed constant throughout the element.

**2-D solid elements:** Elements that have geometry similar to a flat plate. Example: Plane stress, plain strain, plates, shells, and axisymmetric elements. 2-D solid elements are plane elements, with constant thickness, and have either a triangular or quadrilateral shape, with 3 nodes or 4 nodes. For higher order 2-D elements, the number of nodes can vary.

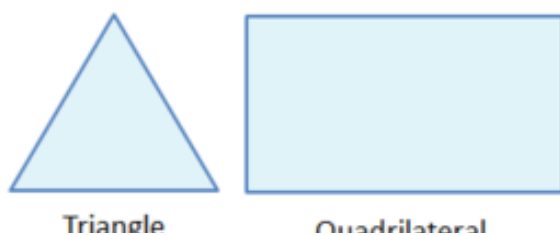


Fig1: Types of 2-D Elements

**3-D solid elements:** Element that have a 3-D geometry. Example: Tetrahedron and hexahedron elements. The basic 3-

D solid elements have either a tetrahedral (4 faces) or hexahedral (6 faces) shape. The basic elements have corner nodes and straight edges, but the number of nodes and edge geometry can vary.

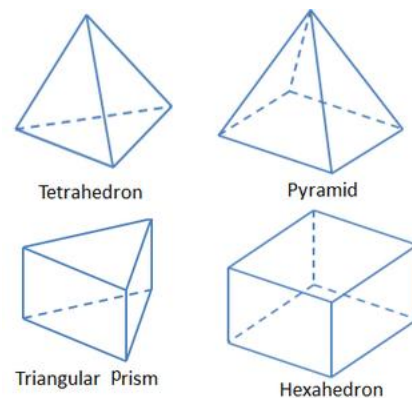


Fig2: Types of 2-D Elements

Table 1.

| Type of Meshing | Elements           |
|-----------------|--------------------|
| 1-D             | 1-D Line Elements  |
| 2-D             | 2-D Solid Elements |
| 3-D             | 3-D Solid Elements |

## 2. MODEL UNDER STUDY

Model under study is made up of structure.

The overall dimensions of model are-

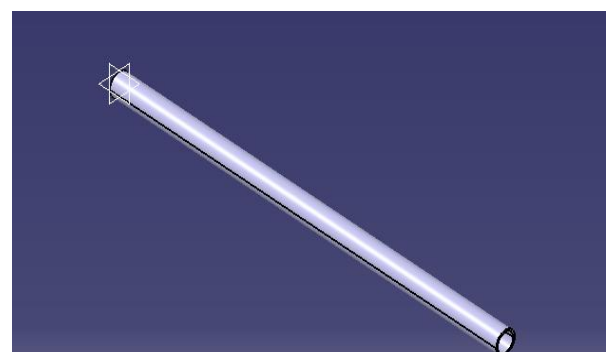


Fig3: CAD Model of a pipe

Length- 500mm

Outer Radius-12mm

Inner Radius- 10mm

Material properties of the steel are listed in below Table

**Table 2. Structure Steel Material Properties**

|                 |                       |
|-----------------|-----------------------|
| Young's Modulus | 200 GPa               |
| Density         | 7850 kgm <sup>3</sup> |
| Yield Stress    | 250 MPa               |
| Ultimate Stress | 460 MPa               |
| Poisson's Ratio | 0.3                   |

## 3. THEORITICAL SOLUTION

### 3.1 Concentrated load at free end

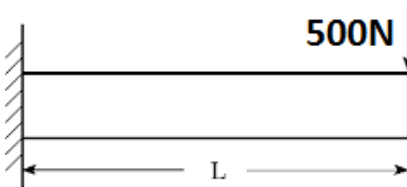


**Fig4: Concentrated load at free end**

Formula used to calculate the stress is,  $\sigma = MY/I$

Where, M= Bending Moment Y= distance from end to the neutral axis of cross section

I= Moment of Inertia of circular tube

Hence,  $\sigma = 355.7860 \text{ MPa}$

Formula used to calculate deformation is,  $D = WL^3/3EI$

Where, W=500N L= Length of circular tube E=Young's Modulus of the material

Hence,  $D = 12.3536 \text{ mm}$

### 3.2 Tensile load at free end

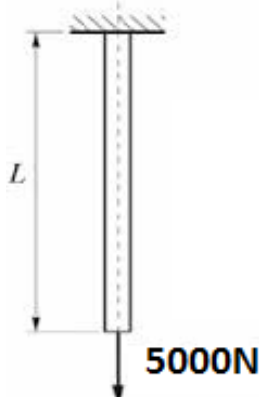


**Fig4: Concentrated load at free end**

$$\sigma = F/A$$

Where, F=5000N A= Area of cross section of circular tube Fig5: Total Deformation in 1-D Meshing

$$\sigma = 36.1715 \text{ MPa}$$

Formula used to calculate deformation is,  $\Delta l = \sigma l/E$

$$\text{Hence, } \Delta l = 0.090429 \text{ mm}$$

## 4. STATIC ANALYSIS

### 4.1 Concentrated load at free end

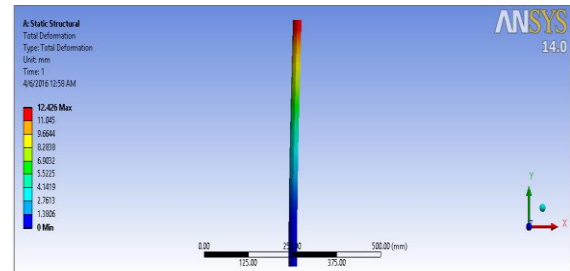


**Fig5: Total Deformation in 1-D Meshing**

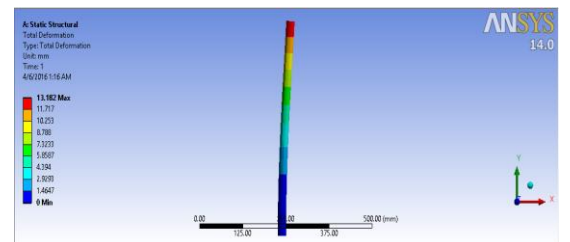


**Fig6: Total Deformation in 2-D Meshing**

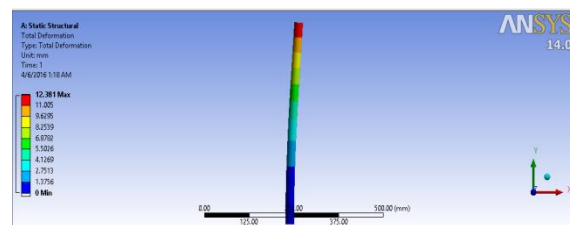


**Fig7: Total Deformation in 3-D Meshing**

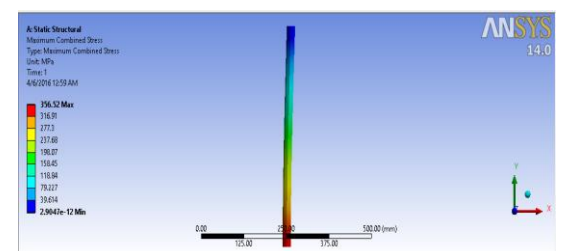


**Fig8: Stresses in 1-D Meshing**

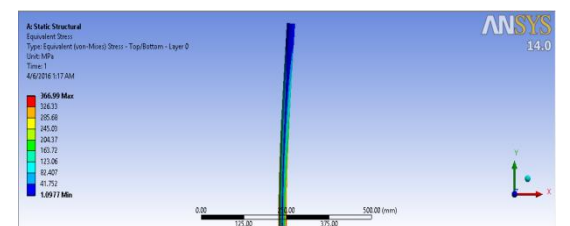


**Fig9: Stresses in 2-D Meshing**

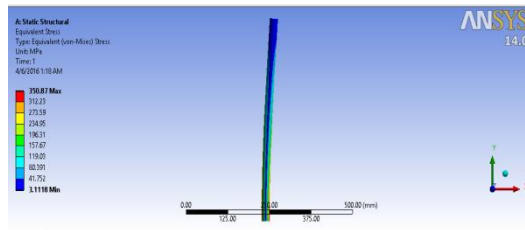


Fig10: Stresses in 3-D Meshing

Table 3: Computing time in different types of meshing

| Type of Meshing | Meshing Time (sec) | Solving Time (sec) |
|-----------------|--------------------|--------------------|
| 1-D             | 2.60               | 4.91               |
| 2-D             | 3.86               | 5.35               |
| 3-D             | 4.34               | 8.24               |

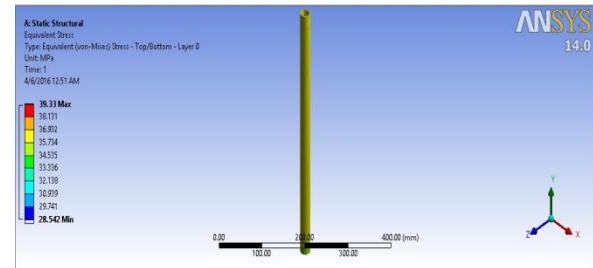


Fig15: Stresses in 2-D Meshing

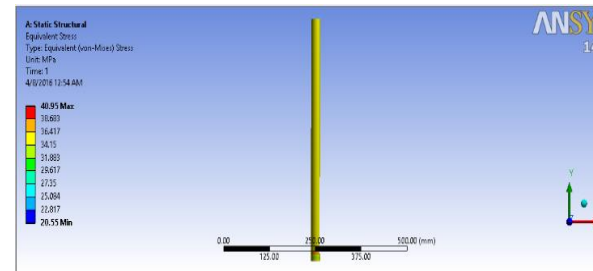


Fig16: Stresses in 3-D Meshing

## 4.2 Tensile load at free end

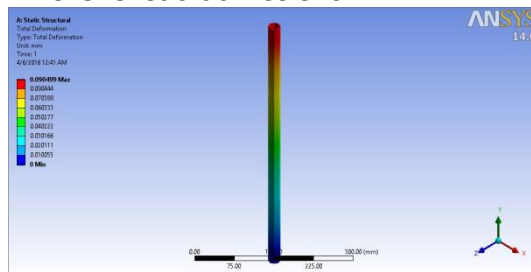


Fig11: Total Deformation in 1-D Meshing

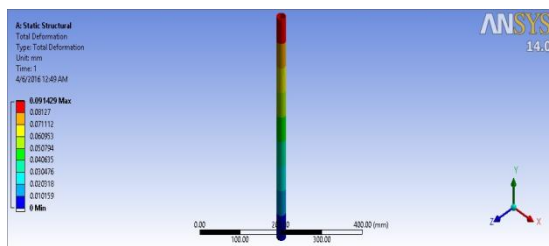


Fig12: Total Deformation in 2-D Meshing

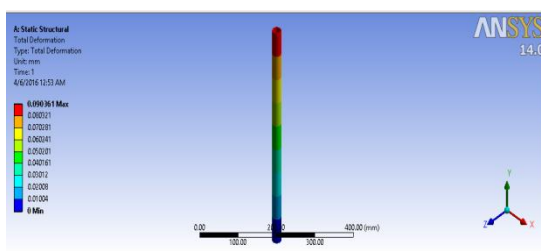


Fig13: Total Deformation in 3-D Meshing

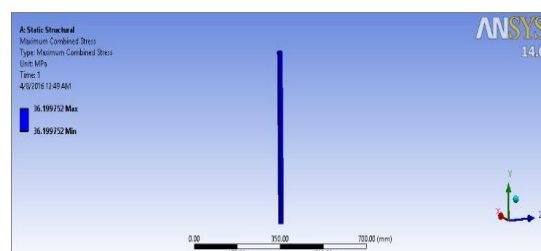


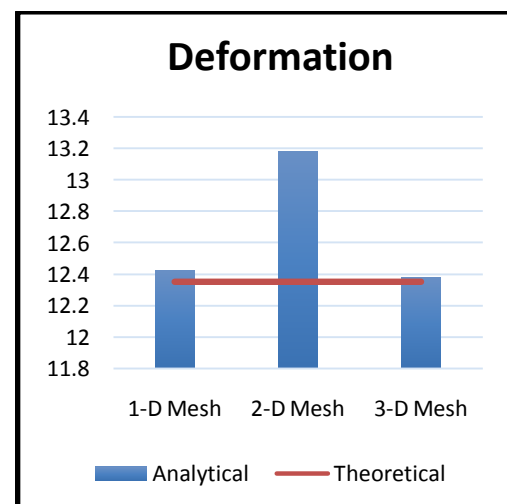
Fig14: Stresses in 1-D Meshing

Table 4: Computing time in different types of meshing

| Type of Meshing | Meshing Time (sec) | Solving Time (sec) |
|-----------------|--------------------|--------------------|
| 1-D             | 2.36               | 6.09               |
| 2-D             | 3.88               | 7.31               |
| 3-D             | 4.12               | 9.11               |

## 5. COMPARISON

### 5.1 Concentrated load at free end

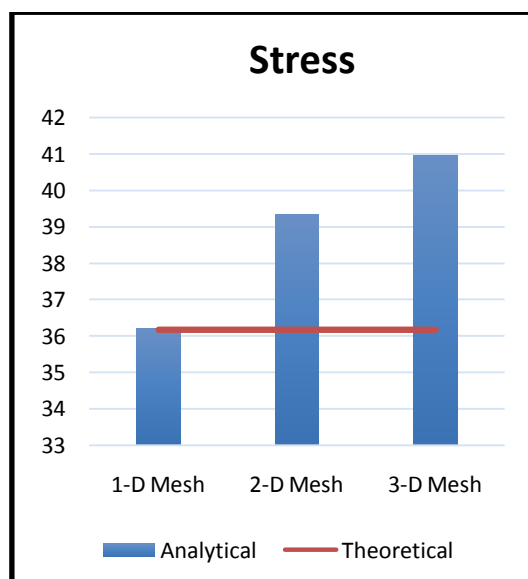
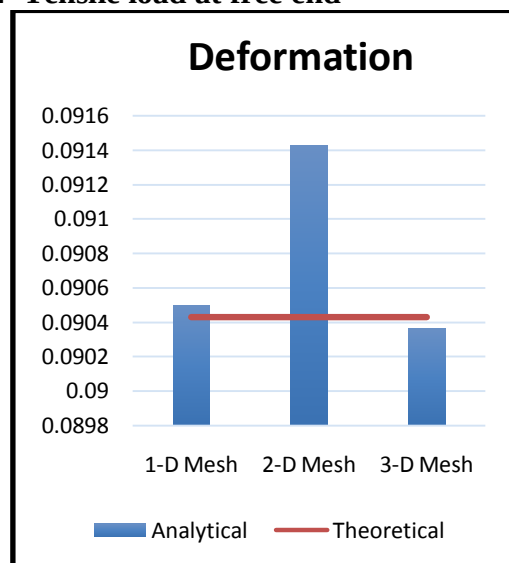


through static analysis. It was found that for static analysis the 1-D meshing of model having constant cross section gives more accuracy when compared to theoretical result.

## 7. REFERENCES

- [1] <http://www.ewp.rpi.edu/hartford/users/papers/engr/ernesto/hillb2/MEP/Other/Articles/MeshingGuide.pdf>
- [2] [http://www.engr.uconn.edu/~cassenti/AnsysTutorial/Module%201\\_Workbench/Module%201\\_3\\_W\\_1D%20Workbench%20distributed.pdf](http://www.engr.uconn.edu/~cassenti/AnsysTutorial/Module%201_Workbench/Module%201_3_W_1D%20Workbench%20distributed.pdf)
- [3] Strength of Materials, R.K. Rajput, Fifth Edition 2010
- [4] Metallic Materials Properties Development and Standardization (MMPDS) Handbook, 10th Edition, 10th Edition 2015

## 5.2 Tensile load at free end



## 6. CONCLUSION

In this study, the effects of type of meshing on accuracy of finite element models and simulation results were investigated