

Convergence Study for Material Property Gradient Based Meshing on Analysis of Functionally Graded Materials, Part I - Under axial Load

Lokesh Singh

*Department of Mechanical Engineering,
GD Rungta College of Engineering & Technology
Bhilai, 490023, India
lokeshsingh@gmx.com*

Abstract

Generally finite element mesh generation work is well established for homogenous objects where object-geometry is the main criteria for meshing an object. The effect of variation of material property as happens in case of FGMs, on mesh size is largely remains untouched. Present work considers the elastic property gradient as criteria for meshing and studies its impact on the convergence of the analysis result. A one dimensional functionally graded bar was considered, and the meshing is done for different elastic modulus variation within the bar. It was observed that elastic gradient basis for element size will have positive effect on convergence if a different gradient relationship is chosen for the meshing. The relationship between the elastic property variation function and the mesh size variation function was established. It was observed that for an optimum convergence result, the gradient power taken for meshing is in general different from the power that represents the material property variation. This work is a unique and novel study that is expected to have significant in improvement in the finite element meshing and convergence of FGM.

Keywords: Graded mesh, Graded element, FGMs, FEM.

1. Introduction

Functional graded material is composite material form of two or more constituent phases with a continuously variable composition. Many live systems have mechanical properties that vary continuously as a function of position such as bones, wood, cellulose etc. The mechanical benefit of such system becomes obvious to designer and consequently there has been a growing interest in producing man made non-homogeneous material for specific application/s, often referred as functional graded material or FGM in short. Graded material with a gradual variation of material and material properties have come into play to replace homogeneous materials in the situation where there is a need to control property such as in the case of rocket engine or turbine blade [1-3].

Similar to design methods, the computational simulations are important tool for development because of their potential to reduce expensive experimentation. Finite element simulation is a vital tool for analysis and mesh generation is an essential part of it. Extensive work is available in the area of mesh generation algorithms and handling complex geometric objects but only few papers are available on meshing based on varying material property termed here as graded mesh. The summary of the papers related to mesh generation of FGMs is as follows Cheng et al. [4] has put parent pattern module method to generate the graded mesh by allowing for gradation in both coordinate directions. They generated quadrilateral elements, with no restriction on the distribution of mesh density. Peter et al. [5] presents an algorithm which can be used to triangulate geometries represented by parametric coordinates. The parameterization used to define all curves as non uniform rational B-Spline (NURBS) to automatically triangulate a region with a graded mesh. Mezzanotte et al. [6] focused on a graded meshing strategy to minimize the coarseness error. They defined two types of cell sizes, large and small and graded mesh is applied from small size cell that increased with a constant factor to large cell size. Chiv et al. [7] implemented quad-tree mesh generation method to separate the interface region of different materials within an object and generated a triangular mesh. Zhang et al. [8] describes an approach for automatic unstructured tetrahedral and hexahedral meshes of composite domain made up of heterogeneous materials by introducing the notion of a material change edge and minimizer point method for identifying the interface boundary and interface node between different materials.

Nicole et al [9] separated different component of the cervical spine with a triangulated surface region of the structure and using a multi-block method for mesh generation. Sullivan et al. [10] developed a three dimensional mesh generation method which is well suited for adaptive situations. A template of elements is superimposed upon the boundary of the model and elements that are straddled at the boundary are adapted to conform to the models boundaries. Internal and distinct materials are retained in the final mesh.

FE analysis of heterogeneous object is relatively current topic in research. Pise et al. [11] simulated static loading of bio-objects like human femur with B-Spline based modelling, meshing and its 3D finite element analysis with material based graded element. Pfeiler et al. [12] employed a direct conversion of CT Hounsfield units to material property (Young's modulus and Poisons ratio) and minimize user interaction for mesh smoothing to produce FEM analysis model. Yang et al. [13] proposed heterogeneous lofting for modeling of a multi-material object and for analysis graded B-Spline finite element solution procedure.

The work done by researchers so far available in public domain is focused on utilizing certain material based criterion for meshing the object. No systematic study is available so far to study the effects of the material based meshing methods on the convergence characteristics of a FEM analysis procedure.

The present work is aimed at study the effect of material gradient based meshing on the convergence of the FE analysis. The first part of the paper considers a FGM bar of varied material composition and simulates material gradient based meshing strategy and its effect on the convergence characteristics of the FE analysis.

Part II of the paper does the similar analysis for FGM beam under transverse loading condition. Though the problem looks simple at first look but the analysis yield some interesting and new conclusions.

Part III of the paper does the analysis for arbitrary material property variation FGMs bar and beam under axial and transverse loading condition.

2. Materials and Methods

2.1. Problem Statement

We start with a very basic and well known one dimensional bar example. Let length of bar be L, and the area of cross section A. As shown in Figure 1, one end of the bar is fixed and a tensile load of P is applied at another end. The bar is composed of two materials, with the elastic properties assumed to be E_1 and E_2 respectively ($E_2 > E_1$) at the end. It is assumed that the variation of modulus of elasticity $E(X)$ at a distance X from the fixed end within the bar is govern by a power law function as follows

$$E(X) = E_1 + (E_2 - E_1) \left(\frac{X}{L}\right)^n \quad (1)$$

$$\text{Let } x = \frac{X}{L}, \gamma_{\max} = \frac{E_2}{E_1}, \text{ \& } \gamma(x) = \frac{E(x)}{E_1}$$

Where x is termed as non dimensional length and γ as non dimensional modulus of elasticity using non dimensional parameters Eq. (1) reduces to

$$\gamma(x) = 1 + (\gamma_{\max} - 1)x^n \quad (2)$$

Where n is any real positive number called material variation power. For all of the simulation it is presumed that the load P is 1unit and the area of cross section A is also 1 unit.



Figure 1: The basic configuration

The material property distribution represented as $\gamma(x)$ within the longitudinal direction for different values of n is shown in Figure 2.

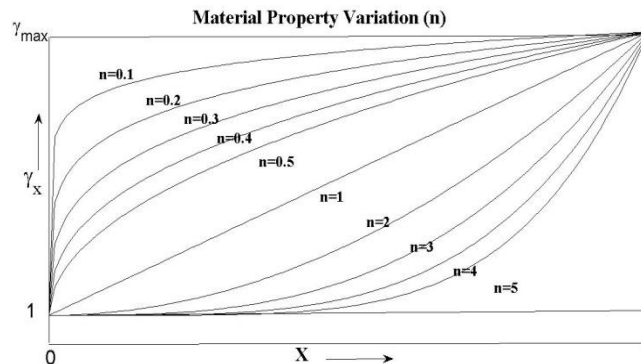


Figure 2: Material property variation

2.2. Meshing style

Most of the meshing style available so far take FGM material as a combination of piecewise homogeneous region and then meshing each region separately. For homogeneous objects, the mesh size is dependent primarily on the geometry of the object. It is presumed that for FGM, the appropriate mesh size is dependent on the material property gradient. The material property considered here is the modulus of elasticity of the object, i.e. $E(X)$

Let the rod be divided into N number of elements for geometric mesh, since the area of cross section is constant, all elements are considered of equal size. Thus the non dimensional length of each element is $1/N$. A standard finite element formulation was done and displacement u at the load end of the bar was checked for convergence study.

The second study was done using the material based graded meshing approach as follows. Let the number of elements be N . Now the element size will be determined by equal increment in modulus of elasticity (E) along X direction. Each node will have an increment of $(E_2 - E_1)/N$. So the nodes will be placed at the locations of successive increment of $(E_2 - E_1)/N$ in the value of E . Let the value of E at a node at a distance X is $E(X)$. The non dimensional distance corresponding to non dimensional modulus of elasticity, can be deduced from equation 2:

$$x = \left(\frac{\gamma(x)-1}{\gamma_{\max}-1} \right)^{\frac{1}{n}} \quad (3)$$

Figure 3(a) and (b) indicate the length of each element $e(i), i = 0.1$ to 5, for different values of n (2 & 1/2 respectively)

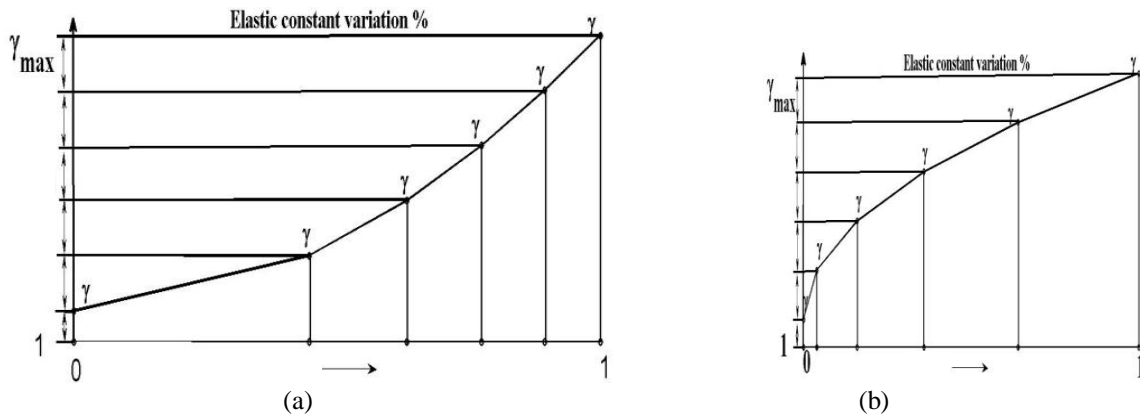


Figure 3: Location of nodes based on equal variation in E (a) for n=2 and (b) n=1/2.

So it is evident that this style of meshing strategy will give unequal element length for the value of n≠1.

3. FE Analysis with Graded Element

The variation in elastic property E with an element can be handled in different ways. The convention method is to assume the value of E within the element constant is shown in Figure 4 and its value is the average of the nodal values of E thus:

$$E_e = \frac{E_{x_1} + E_{x_2}}{2} \tag{4}$$

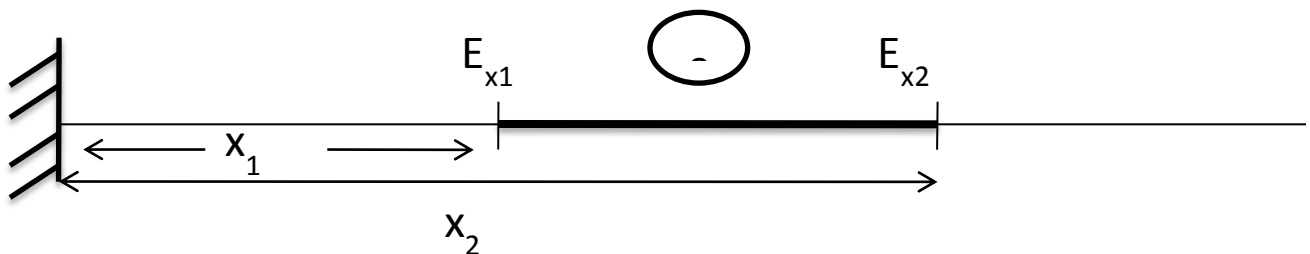


Figure 4: Constant E, average element

The element with averaged E value is termed as average element. The average element meshing strategy is equivalent to the homogeneous region based meshing method that is used by most of the researchers. Another method, which may look more attractive and accurate, is taking with account the variation of E across the element, E(X). This method we are terming as “graded element” method. In this case the elemental stiffness matrix $[k]_e$ is expressed as

$$[k]_e = \int_{x_1}^{x_2} [B]^T [D] [B] dx \tag{5}$$

Where [B] is the strain displacement matrix.

$$B = \frac{1}{x_2 - x_1} [-1 \ 1]$$

$$D = E(X)$$

$$\text{So } [k]_e = \int_{x_1}^{x_2} \frac{E(X)A}{(x_2 - x_1)^2} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dX \quad (6)$$

$$[k]_e = \int_{x_1}^{x_2} \frac{E_1 \gamma(x)L}{(x_2 - x_1)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx$$

$$[k]_e = E_n L \int_{x_1}^{x_2} \frac{\gamma(x)}{(x_2 - x_1)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx \quad (7)$$

Elemental equations are assembled to get global properties of structure by using following system equation

$$\{k\}_e \{\delta\}_e = \{P\}_e \quad (8)$$

Where $\{\delta\}_e$ is nodal displacement vector of the element and $\{P\}_e$ nodal load vector.
 The global equation can be presented as

$$[K][\delta] = [P] \quad (9)$$

Where [K] is the global stiffness matrix $[\delta]$ is the global displacement vector and [P] is the global load vector

3.1. Closed Form Solution of the Problem

For validation purpose closed form solution of the problem can be taken from standard literature [14]. The displacement (u) on the free end can be found by the equation 10.

$$u = \int_0^L \frac{P}{AE(X)} dX \quad (10)$$

Since A=1, L=1

$$u = \int_0^1 \frac{P}{E_1 \gamma(x)} dx$$

$$u = \frac{P}{E_1} \int_0^1 \frac{1}{\gamma(x)} dx \quad (11)$$

Equation 11 is solved using the symbolic computation tool available in Matlab for different value of the power n, and value of E_1 & E_2 . The value of displacement thus obtained will be used to compare convergence results of the FE analysis.

4. Comparison of results for average element with graded element.

In this section, we present some numerical examples which show the behavior of the graded element vis-a-vis average element. For numerical study in this section γ_{\max} is taken as 2.

4.1. Comparison of results for average element with graded element with equal element size.

The first numerical simulation is done to understand the effect of the graded element vis a vis the average element with equal mesh size.

The absolute error defined as:

$$\text{Error \%} = \left| \frac{\text{Simulated value} - \text{Exact value}}{\text{Exact value}} \right| * 100 \quad (12)$$

The value of Error % is plotted in Figure 5 for different value of n. For same number of elements, graded element has superior convergence characteristics. The error increases as n deviates from 1 on either direction i.e. $n < 1$ & $n > 1$ as shown in Figure 5.

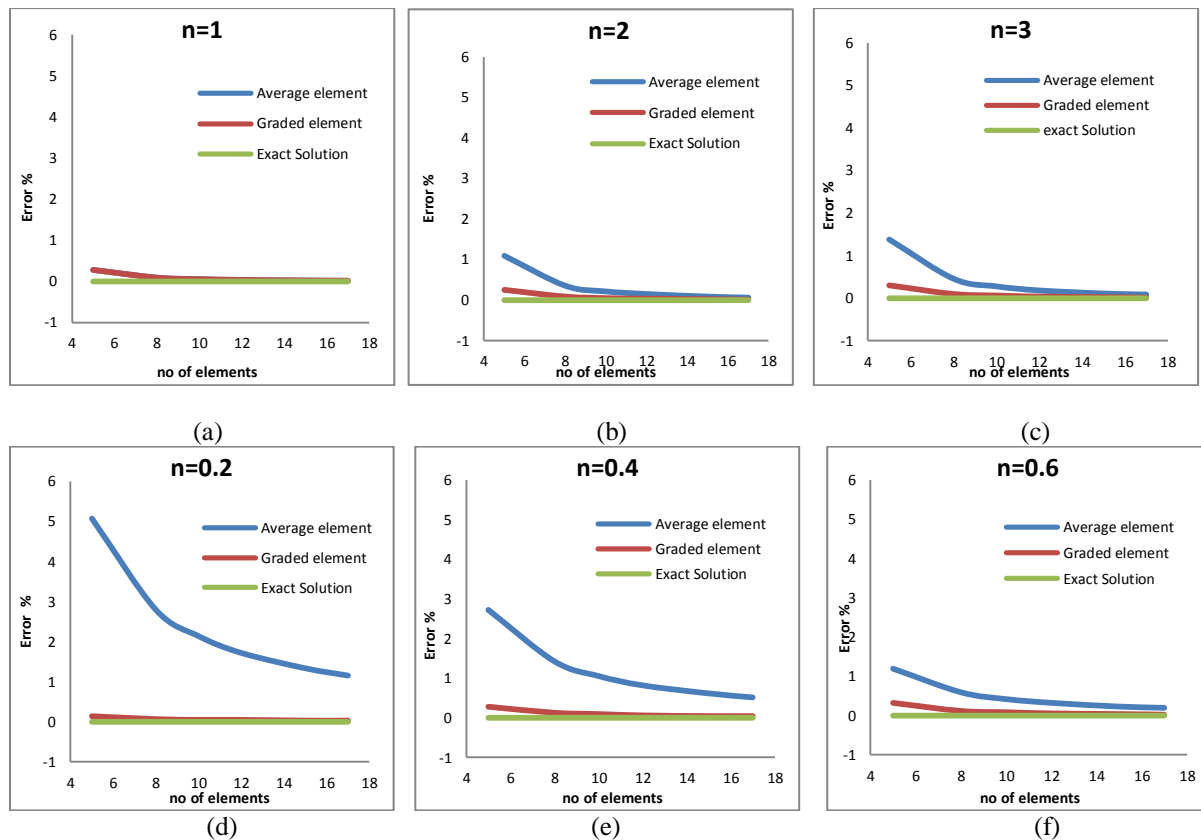


Figure 5: Error plot for average element, graded element and closed form solution for $\gamma_{\max} = 2$ (a) $n=1$. (b) $n=2$. (c) $n=3$. (d) $n=0.2$. (e) $n=0.4$ and (f) $n=0.6$.

4.2. Comparison of results for average element with graded element with unequal element size.

The second simulation is done to understand the effect of changing the mesh size in accordance with the material based graded meshing approach described in Section 2. For comparison purpose the mesh size obtained for material based graded meshing is taken equal to that for the average element analysis. The results Figure 6 indicate that for the same mesh size, the graded element approach is superior to average element approach, so taking graded element (that is varying material property within an element) is advantageous for convergence.

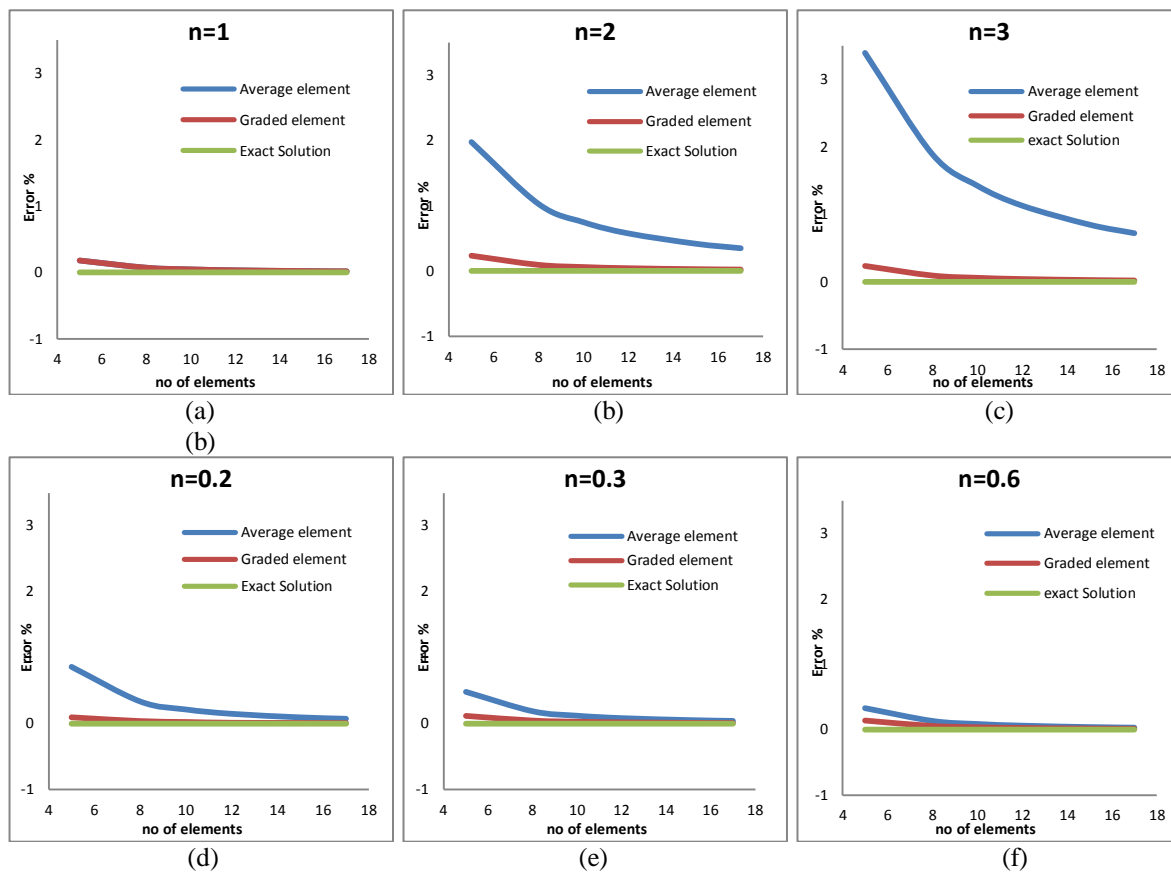


Figure 6: Error plot for average element, graded element and closed form solution for $\gamma_{\max} = 2$ (a) $n=1$. (b) $n=2$. (c) $n=3$. (d) $n=0.2$. (e) $n=0.3$ and (f) $n=0.6$

It also indicates that as n deviates from the value 1, the material based graded meshing gives faster convergence.

5. Comparison of the Convergence Characteristic of ‘Material Mesh’ Element with Geometric Mesh (Equal Length Element)

This section compares the convergence characteristic of material mesh and geometrical mesh. In both cases, the element is considered graded.

5.1. Convergence comparison for constant γ_{\max}

For constant γ_{\max} , the convergence of the FE analysis was studied by varying n . The results for $\gamma_{\max} = 2$ are shown in Figure 7. It is interesting to note that geometrical mesh is effective for $n > 1$ whereas material mesh was effective for $0 < n < 1$. The reason for this effect can be attributed to the effect of different mesh size and different γ_{\max} values on the convergence. To further analyse the cause, it is considered necessary to see the effect of γ_{\max} variation on the convergence characteristics.

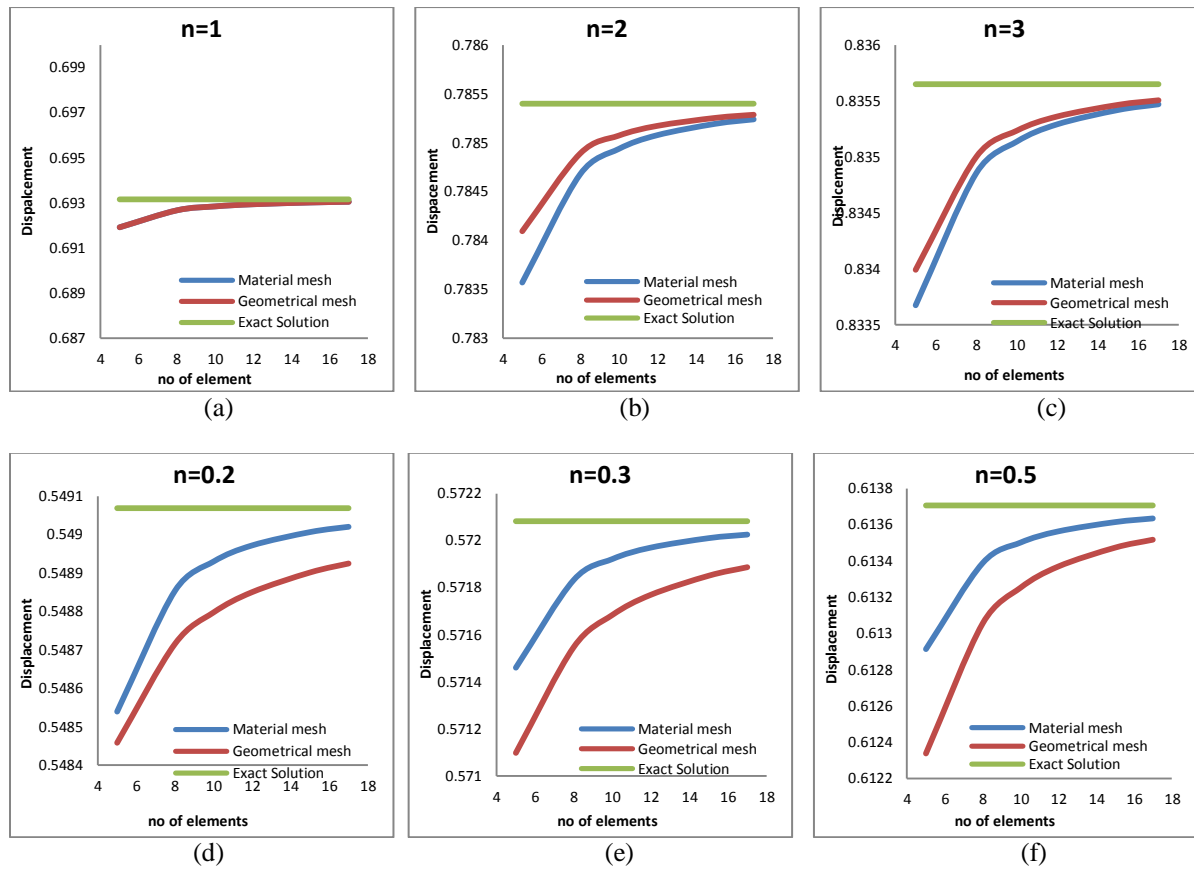
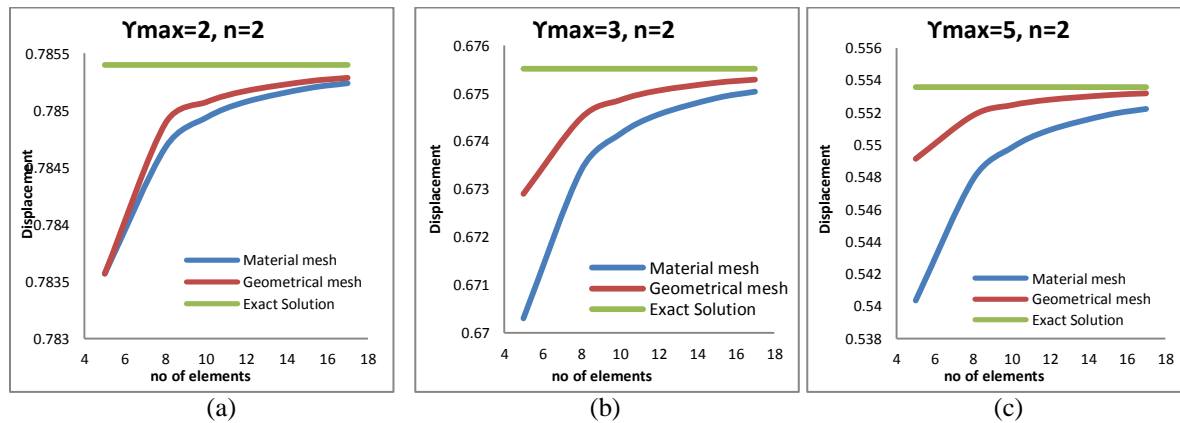


Figure 7: Displacement results (a) $n=1$. (b) $n=2$. (c) $n=3$. (d) $n=0.2$. (e) $n=0.3$ and (f) $n=0.5$.

5.2. Effect of variation of γ_{max}

This simulation is similar to that in Section 5.1, except that value of γ_{max} is also varied from 1 to 100; material mesh, geometrical mesh and exact solution were compared for different values of n . Some representative results are shown in Figure 8.



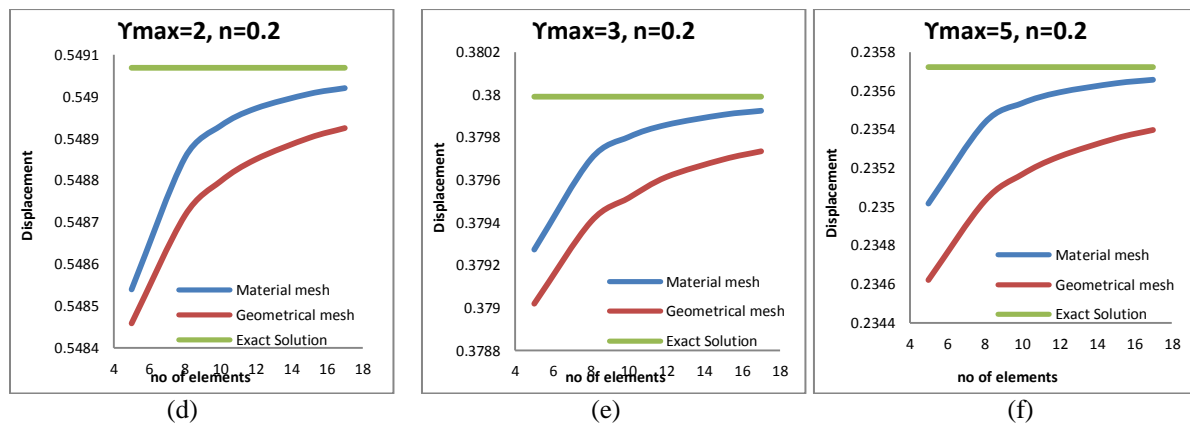


Figure 8: Displacement results for $\gamma_{max} = 2, 3 \text{ \& } 5$ (a) (b) (c) $n=2$, Displacement results for $\gamma_{max} = 2, 3$ and 5 (d) (e) (f) $n=0.2$

It seems that, for different γ_{max} ratios, the results obtained in both cases for $1 < n < 5$ & $0 < n < 1$ have the same pattern as given in Section 5.1. This analysis gives us a direction that probably meshing the element with the element with the same power 'n' needs to be revisited.

6. Taking Different Power Index for Material Meshing.

Section 4 & 5 also indicate that if we take the material mesh based on the power-law index n for material distribution, the convergence is general may not be optimal. Thus it makes sense to use some other power say 'm' which is in general different from 'n'.

For creating mesh, a power 'm' is used to create material mesh and the results in Figure 9 shows convergence result in between power-law index n versus m for constant γ_{max} . The value of m for best convergence is taken from the graph and termed as m_{opt} .

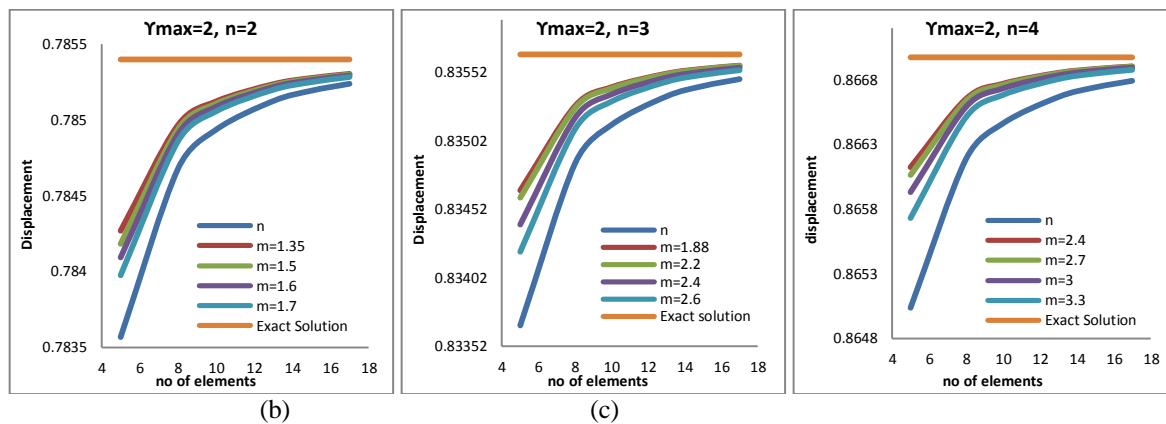


Figure 9: Displacement results for $\gamma_{max} = 2$ (a) $n=2$ versus $m=1.35, 1.5, 1.6,$ and 1.7 (b) $n=3$ versus $m=1.88, 2.2, 2.4,$ and 2.6 (c) $n=4$ versus $m=2.4, 2.7, 3,$ and 3.3 .

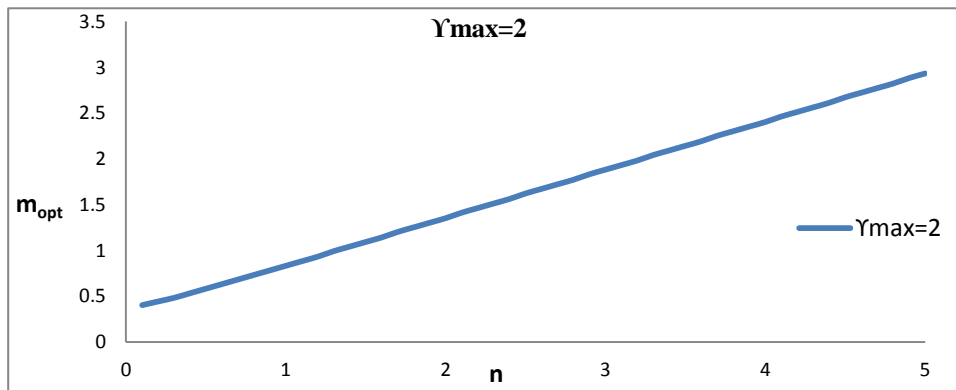


Figure 10: Actual powers used for defining FGMs n versus optimal power used m_{opt}

Figure 10 shows the plot between n and m_{opt} for γ_{max} . This plot is fairly linear and can be expressed as

$$m_{opt} = 0.52n + 0.32 \tag{13}$$

Similar numerical study is done for varying γ_{max} and it is shown in Figure 11.

Figure 11 shows a relationship between n and m_{opt} for different values of γ_{max} . The nature of the graph is linear for n>1. And becomes non linear for large γ_{max} in the range of 0<n<1 indicates in Figure 12 (for $\gamma_{max} = 100$)

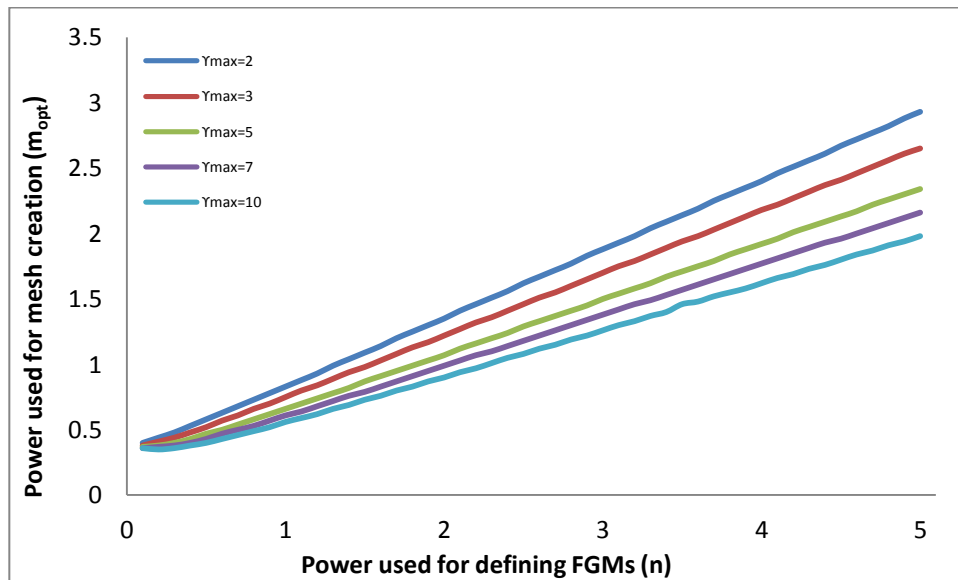


Figure 11: Relationship between powers used for defining FGMs (n) versus power used for mesh creation (m) for gradient mesh generation

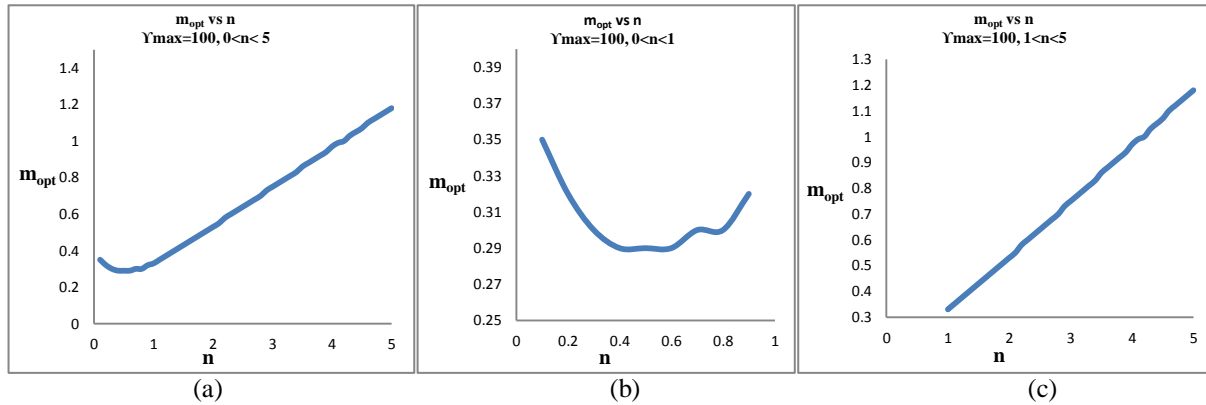
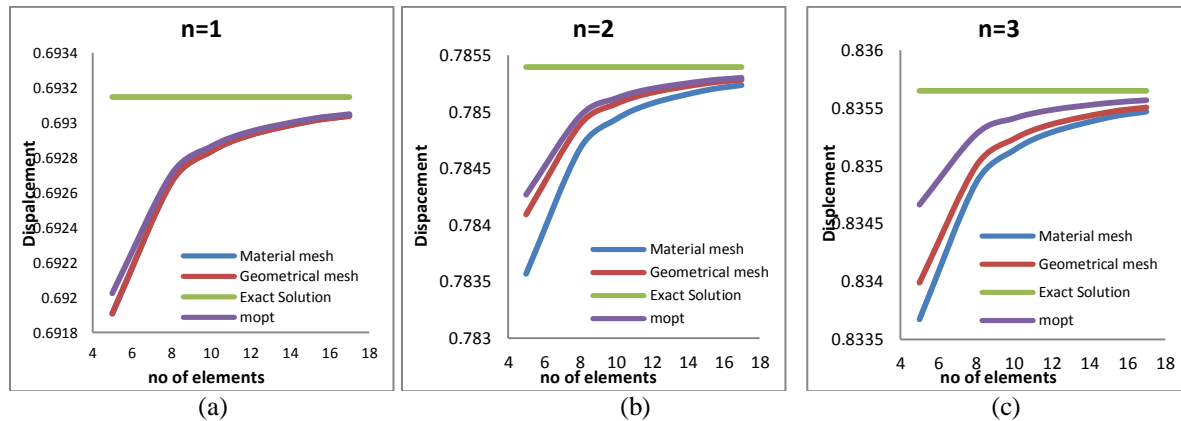


Figure 12: Actual powers used for defining FGMs (n) versus power used for mesh creation (m_{opt}) for gradient mesh generation $\gamma_{max} = 100$. (a) $0 < n < 5$. (b) $0 < n < 1$ and (c) $1 < n < 5$.

7. Results using optimum value of m (m_{opt}) for meshing

In this section, we implement the meshing based on the power for meshing m_{opt} as explained in Section 5. The result for different values of n and $\gamma_{max} = 2$ are shown in Figure 13. All the results indicate that the convergence using material mesh is superior for all the cases. Similar results were optimal for higher volumes of γ_{max} but not shown here because space constraint. It also shows that the convergence is now not affected by the power n , which is in contrast to Section 5.2 where we have different results $n < 1$ and $n > 1$.



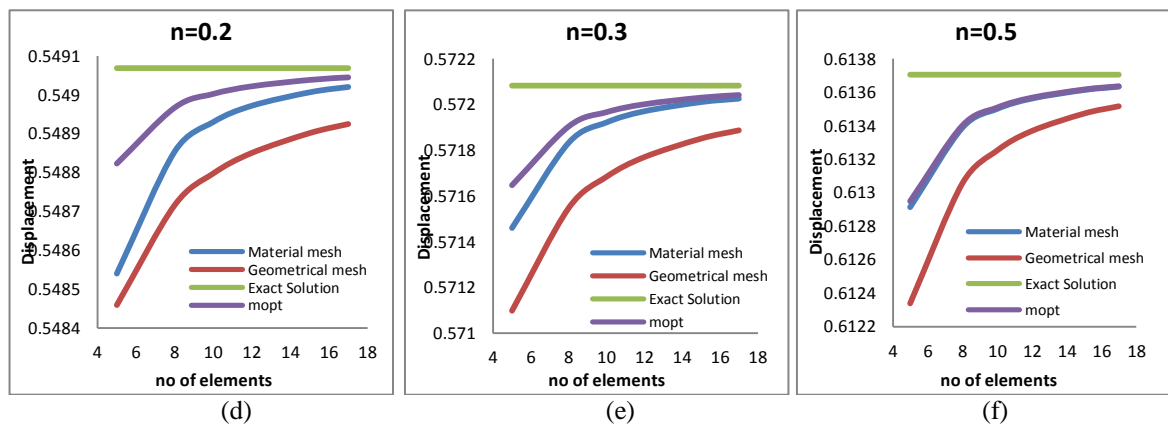


Figure 13: Displacement results between geometrical mesh, material based gradient mesh, meshing based on the m_{opt} (a) $n=1$. (b) $n=2$. (c) $n=3$. (d) $n=0.2$. (e) $n=0.3$ and (f) $n=0.5$.

8. Discussion

The goal of this study was to study the effect of the material based meshing on convergence of FE analysis results. Our study clearly indicate the superiority of the material based meshing vis a vis conventional meshing with m_{opt} as a basis for meshing, the number of element for same acceptable error can be reduced to more than 50% in some cases, which is a huge computational advantage

9. Conclusion

Based on the simulation it can be concluded that material based graded element where the element size dependent on the power 'n' that defines material property gives faster convergence most of the cases. It is also concluded that for material based graded meshing, it is recommended to use m_{opt} for meshing in place of power n, and taking m_{opt} as power for meshing gives faster convergence.

This work was done for one dimensional stress analysis under tension only. Work is in progress to extend this philosophy for higher dimensional stress analysis and for different type of load conditions.

10. References

- [1] Victor Birman, Larry W. Byrd, Modeling and analysis of functionally graded materials and structures, Applied Mechanics Reviews. 60 (2007) 195-216.
- [2] Michael H. Santare, Priya Thamburaj, George A. Gazonas, The use of graded Finite elements in the study of elastic wave propagation in continuously non-homogeneous materials. International Journal of Solids and Structures. 40(2003) 5621-5634.
- [3] Michael M. Gasik, Micromechanical modeling of functionally graded materials. Computational materials science. 13 (1998) 42-55.
- [4] Hua Li, Gengdong Cheng, New Method for Graded mesh Generation of Quadrilateral Finite Elements, Computer and structures. 76 (2000) 729-740.
- [5] Timothy J. Peters, A quasi-interactive graded mesh generation algorithm for finite element/ moment method analysis on NURBS-Based Geometries. Antennas and propagation Society international symposium, 1994 AP-S digest (Volume: 3) 1390-1393.

- [6] Paolo Mezzanotte, Luca Roselli, on the accuracy of the finite difference method using mesh grading, Microwave Symposium Digest. 1995 IEEE MTT-S International 781-784 volume 2.
- [7] Chiu W. K., Kou X.Y., Tan, S.T., Adaptive meshing of 2D heterogeneous object using material Quad tree, Computer aided design and application. 8(2) (2011) 289-300.
- [8] Yongjie Zhang, Thomas J.R. Hughes, Chandrajit L. Bajaj, An automatic mesh generation method for domain with multiple material, Journal of computational methods and application mechanical engineering. 199 (2010) 405 - 415.
- [9] Nicole A. Kallemeyn, Srinivas C. Tadepalli, Kiran H. Shivanna, Nicole M. Grossland, An Interactive Multi-block Approach To Meshing The Spine, Computer Methods And Programs In Biomedicine. 95(2009) 227-235.
- [10] John M. Sullivan Jr., Geoffrey Charron, Keith D. Paulsen, A three-dimensional mesh generator for arbitrary multiple material domains, Finite element in analysis and design. 25 (1997) 219-241.
- [11] Uday V. Pise, Amba D. Bhatt, R.K. Srivastava, Ravi Warkedkar, A B-Spline based heterogeneous object modeling and analysis for proximal femur with graded element, Journal of Biomechanics. 42 (2009) 1981-1988.
- [12] T.W. Pfeiler, D.S. Lalush, E.G. Lobo, Semi automated finite element mesh generation method for long bone, Computer methods and programs in biomedicine. 85 (2007) 196-202.
- [13] Pinghai Yang, Xiaoping Qian.: A B-spline Based Approach to Heterogeneous Object Design and Analysis, Computer Aided Design 39. (2007) 95-111.
- [14] James M. Gere, Mechanics of Materials 2nd edition CBS publisher, 2004