Title:
Convergence Study for Material Property Gradient Based Meshing on Analysis of Functionally Graded Materials under Tensile Load

Authors:
Amba D. Bhatt, abhatt@mnnit.ac.in, MNNIT, Allahabad, India
Lokesh Singh, lokeshsingh25@gmail.com, GD Rungta College of Engineering & Technology Bhilai, India

Keywords:
Graded Mesh, Graded Element, FGMs, FEM

DOI:10.14733/cadconfP.2018.214-219

Introduction:
Functionally graded material or FGM are microscopically non-homogeneous composite materials, with material properties varying continuously and smoothly from one end to another [13]. FGM materials offer optimization for the design of components in terms of material usage and performance, and thus have found special applications in diverse fields including aerospace, defence, nuclear, as well as biomedical applications and analysis of FGM structures [4], [6], [13].

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. Some representative works [2], [9] and [10] are essentially geometry based meshing methods with little consideration for the material gradient. The works of [4] and [8] are the representative works for meshing strategies used for FE analysis of FGMs by approximating the FGMs as piecewise homogeneous materials and then messing each region by conventional meshing strategies for homogeneous objects.

FE analysis of heterogeneous object is relatively current topic in research. Some representative works such as [11] and [12] analyse a FGM object by taking some meshing strategy that is based on material gradient. Most of the works stop at adopting some criteria or the other related meshing the object. No study is available for checking the effect of these criterions on the analysis of the object. Present work uses material gradient as one of the parameter to create variable mesh and studies its effect on the convergence characteristics of a FEM analysis procedure.

Materials and Methods:
Let us consider a bar of length L, and the area of cross section A as shown in Fig. 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 and ($E_2 > E_1$) 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 governed by a power law function as follows:

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

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

Where x is termed as non dimensional length and γ as non-dimensional modulus of elasticity. Thus Eqn. (1) reduces to:

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

Where n is any real positive number called power index for material variation. 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.

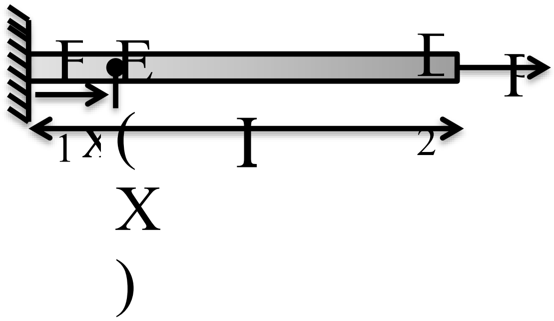

Fig. 1: The Basic Configuration.

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. $\mathrm{E(X)}$. In this context, the following definitions are introduced:

Geometric Mesh
Meshing strategy based on only the geometry of the component with no consideration for material property gradient is termed as geometric mesh. The elements thus formed are referred as geometric element.

Assume that the FGM rod under consideration is to be divided into N elements, based on geometric mesh. Since the area of cross section the rod is constant, the non-dimensional length of each element will be 1/N.

Finite Element Analysis with Geometric Element
A standard finite element formulation was done and displacement *u,* at the load end of the bar was checked for convergence study.

Graded Mesh
In graded mesh, the element size depends on the equal increment / decrement of the material property value across each of the elements.

For N number of elements, and graded mesh, the element size will be determined by equal increment in modulus of elasticity (E) along length of the rod. Each node will have an increment of $\mathrm{(E_2 - E_1)/N}$. So the nodes will be placed at the locations of successive increments of $\mathrm{(E_2 - E_1)/N}$ in the value of E. Let the value of E at a node at a distance X is $\mathrm{E(X)}$. The non-dimensional distance x corresponding to non-dimensional modulus of elasticity, can be evaluated from Eqn. (2) as follows:

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

Obliviously, this style of meshing strategy will give unequal element length for the value of n≠1.

FE Analysis with Graded Element:
The variation in elastic property E with an element can be handled in different ways. The conventional method is to assume the value of E constant within an element as shown in Fig. 2 and the value of E is approximated as the average of the nodal values. So for element e, the modulus of elasticity E (e) is:

$$E(e) = \frac{E_{X_1} + E_{X_2}}{2} \tag{4}$$

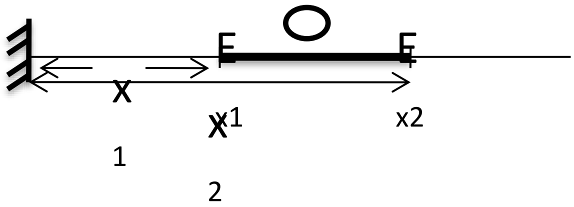

Fig. 2: Constant E, Average Element.

The element with averaged E value is termed here as average element. The average element meshing strategy is equivalent to dividing the region into number of presumed homogeneous regions and then analysing each region as a homogeneous region.

Another method, which may look more attractive and accurate, is taking into account the variation of E across the element, E(X) into consideration while analysing the model. This method we are terming as "graded element" method. In this case the elemental stiffness matrix $[k]_e$ is expressed as:

$$[k]_e = \int_{X_1}^{X_2} [B]^T [D][B] A \, dX \tag{5}$$

Where $[B]$ is strain displacement matrix, D=E(x) and A is the area of cross-section of the element. Elemental Equations are assembled to get global properties of structure. The global equation can be presented as:

$$[K][\delta] = P \tag{6}$$

Where [K] is the global stiffness matrix [δ] is the global displacement vector and [P] is the global load vector. Thus the displacement at every node can be computed.

Closed form Solution of the Problem

For average element solution, the close form solution comparison was taken from literature [14]. For graded element, the closed form solution of Eqn. (6) was found using a symbolic computation tool.

<u>Comparison of Results for Average Element with Graded Element:</u>

For constant γ_{max} the numerical simulation to find out the displacement at the load end was done. The variables were the power law index n, and number of elements.

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 difference between exact value of displacement at the free end and from the numerical simulation is evaluated for different values of the n. It was found that for same number of elements, graded element has superior convergence characteristics. The error increases as n deviates from 1 on either direction.

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 earlier. For comparison purpose the mesh size obtained for material based graded meshing is taken equal to that for the average element analysis. The results 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. It also indicates that as n deviates from the value 1, the material based graded meshing gives faster convergence. So for further studies, material based graded mesh approach is adopted as a better method for convergence.

<u>Comparison of the Convergence Characteristic of 'Material Mesh' Element with Geometric Mesh (Equal Length Element):</u>

Convergence Comparison for Constant γ_{max}

For constant γ_{max}, for example (γ_{max} =2), the convergence of the FE analysis was studied by changing the power varying n. 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. So to further analyse the cause, it is considered necessary to see the effect of γ_{max} variation on the convergence characteristics.

Effect of Variation of γ_{max}

By varying, γ_{max} from 1 to 100, numerical simulations were conducted. The variables were different type of meshing techniques (geometric mesh and material mesh), both of the analysis techniques (average element and graded element) and the power index n. It was evident from the results that, for different γ_{max} ratios, the results obtained in both cases for 1<n<5 and 0<n<1 have the same pattern as we have indicated in earlier sections. However, this analysis gives us a direction that probably meshing the element with the same power 'n' needs to be revisited. Thus an effort was done to take a different index for meshing than n.

<u>Taking Different Power Index for Material Meshing:</u>

For creating mesh, a power ‘m’ that in general is different from the power index n is used to create material mesh. Keeping γ_{max} constant, for the same value of power index n, the convergence was tested by varying m. Simulations were done by changing γ_{max} and repeating the procedure again. Interestingly, for each combination of γ_{max} and n, there is a value of the index m for which the convergence is fastest. The value of index ‘m’ for the fastest convergence is termed as m_{opt}.

Similar numerical study is done for varying γ_{max}. It was interesting to note the relationship between m_{opt}, and index n, is fairly linear as shown in Fig. 4. It shows a relationship between material 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 Fig. 3 (for γ_{max} = 100). For n>1, the relationship between m_{opt} and n can be expressed as:

$$m_{opt} = 0.52n + 0.32 \tag{7}$$

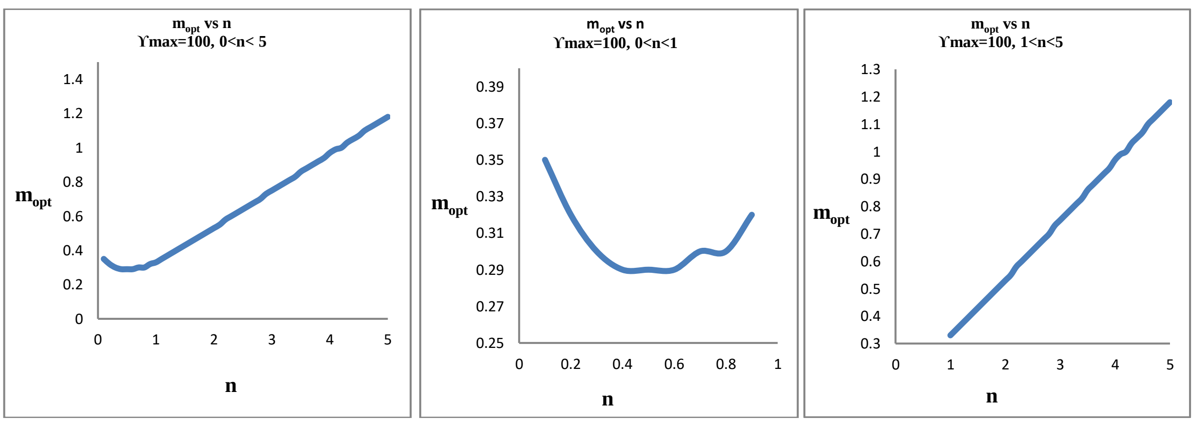

Fig. 3: Actual Powers used for Defining FGMs (n) versus Power used for Mesh Creation (m_{opt}) for Gradient Mesh Generation γ_{max} = 100: (a) 0<n< 5, (b) 0<n< 1, (c) 1<n<5.

<u>Results using Optimum Value of m (m_{opt}) for Meshing:</u>

To see the effectiveness of this approach, for further analysis, m_{opt} is taken the index for meshing for different values of n. The result for different values of n and γ_{max} = 2 are shown in Fig. 5. All the results indicate that the convergence using material mesh is superior for all the cases. It was also shows that the convergence is now not affected by the power index n. The interesting point to note is that the value of m_{opt} is fairly independent of the value of influence of γ_{max} .

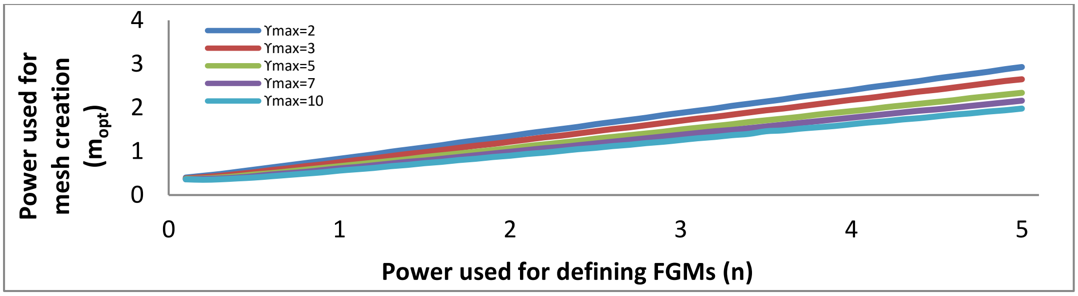

Fig. 4: Relationship between Powers used for Defining FGMs (n) versus Power used for Mesh Creation (m_{opt}) for Gradient Mesh Generation.

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 of huge computational advantage.

Conclusion:
Based on the simulation-results it can be concluded that material based graded element where the element size is dependent on the power 'n' that defines material property gives faster convergence. 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 fastest convergence. A linear relationship is also developed for finding out the optimum power index for meshing from the power index n.

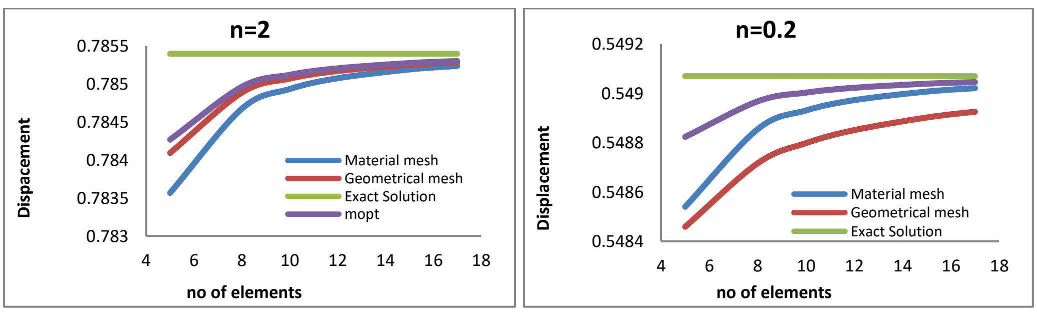

Fig. 5: Displacement Results between Geometrical Mesh, Material Based Gradient Mesh, Meshing Based on the m_{opt} for n=2 and for n=0.2 (From left to right).

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.

References:

[1] Birman, V.; Byrd, L.-B.: Modeling and analysis of functionally graded materials and structures, Applied Mechanics Reviews, 60 (5), 2007, 195-216.https://doi.org/10.1115/1.2777164
[2] Cheng, G.; Li, H.: New Method for Graded mesh Generation of Quadrilateral Finite Elements, Computer and Structures, 59 (5), 1996, 823-829.https://doi.org/10.1016/0045-7949(95)00316-9
[3] 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.https://doi.org/10.3722/cadaps.2011.289-300
[4] Gasik, M.-M.: Micromechanical modeling of functionally graded materials, Computational Materials Science, 13 (1-3), 1998, 42-55.https://doi.org/10.1016/S0927-0256(98)00044-5
[5] Gere, J.-M.: Mechanics of Materials, 2nd edition, CBS publisher, 2004.

[6] Kallemeyn, N.-A.; Tadepalli, S.-C.; Shivanna, K.-H.; Grossland, N.-M.: An Interactive Multi-block Approach To Meshing The Spine, Computer Methods And Programs in Biomedicine, 95(3), 2009, 227-235. https://doi.org/10.1016/j.cmpb.2009.03.005

[7] Mezzanotte, P.; Roselli, L.: On the accuracy of the finite difference method using mesh grading, Microwave Symposium Digest, IEEE MTT-S International, 2, 1995, 781-784. https://doi.org/10.1109/MWSYM.1995.405991

[8] Peters, T.- J.: A quasi-interactive graded mesh generation algorithm for finite element/ moment method analysis on NURBS-Based Geometries, Antennas and Propagation Society, international symposium, AP-S digest, Volume 3, 1994, 1390-1393.

[9] Pfeiler,T.-W.; Lalush, D.-S.; Loboa,E.-G.: Semi automated finite element mesh generation method for long bone, Computer Methods and Programs in Biomedicine, 85 (3), 2007, 196-202. https://doi.org/10.1016/j.cmpb.2006.10.009

[10] Pise, U.-V.; Bhatt, A.-D.; Srivastava, R.-K.; Warkedkar, R.: A B-Spline based heterogeneous object modeling and analysis for proximal femur with graded element, Journal of Biomechanics, 42 (12), 2009, 1981-1988. https://doi.org/10.1016/j.jbiomech.2009.05.019

[11] Santare, M.-H.; Thamburaj, P.; Gazonas, G.-A.: 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 (21), 2003, 5621-5634.https://doi.org/10.1016/S0020-7683(03)00315-9

[12] Sullivan Jr, J.-M.; Charron G.; Paulsen, K.-D.: A three-dimensional mesh generator for arbitrary multiple material domains, Finite Element in Analysis and Design, 25 (3-4), 1997, 219-241. https://doi.org/10.1016/S0168-874X(96)00027-3

[13] Yang P.; Xiaoping Q.: A B-spline Based Approach to Heterogeneous Object Design and Analysis, Computer-Aided Design, 39 (2), 2007, 95-111. https://doi.org/10.1016/j.cad.2006.10.005

[14] Zhang, Y.; Hughes, T.-J.-R.; Bajaj, C.-L.: An automatic mesh generation method for domain with multiple material, Computer Methods in Applied Mechanics and Engineering, 199 (5-8), 2010, 405 -415. https://doi.org/10.1016/j.cma.2009.06.007