EFFECT OF MESH DENSITY ON FINITE ELEMENT ANALYSIS AND ESTABLISH AN OPTIMAL MESH DENSITY- CASE STUDY

ANTHATI MEENA
Assistant Professor (MSC MATHS)
Nova College of Engineering and Technology

Abstract

Analytical modelling is the initial step in a mathematical-based finite element analysis of any sophisticated structure associated with irregular shape, complex loading pattern and combination of different loads along with complex boundary condition. In the modelling of structure, mesh density plays a major role and it becomes a critical issue of finite element analysis, which closely relates to the accuracy of finite element model while directly determine their complexity level. This project report presents a systematic study on finding the effects of mesh density on accuracy of numerical analysis results, based on which brief guidelines of choosing the best mesh strategy in finite element modelling is provided. In the present work, for studying the effect of mesh density, a bridge deck has been considered and analysis has been performed separately for self-weight of the structure as well as vehicular loading which is dynamic in nature.

1.0 INTRODUCTION

In finite element analysis (FEA), the accuracy of the FEAresults and requested computing time are determined by thefinite element size (mesh density). According to FEA theory,the FE models with fine mesh (small element size) yield tohighly accurate results but may take longer computing time.On the contrary, those FE models with coarse mesh (largeelement size) may lead to less accurate results but do savemore computing time. Also, small element size will increasethe FE model's complexity which is only used when high accuracy is required. Large element size, however, willreduce the FE model's size and is extensively used insimplified models in order for providing a quick and roughestimation of designs. Due to its importance, in generatingFEA models, the foremost problem is to choose appropriateelements size so that the created models will yield accurateFEA results while save as much computing time as possible.

The objective of this paper is to present guidelines forchoosing optimal element size for different types of finiteelement analyses. In order to achieve that goal, in this study,a series of static, modal, and impact analyses were performedon thin-walled beam and plate models to reveal the effects ofthe element size on the accuracy of the FEA results. Anexplicit solver, LS-DYNA, was used for modeling andanalyses involved in this work.

Finite Element Method is a numerical method for solving problems of engineering and mathematical physics. The application of Finite Element method is spread over a large area ranging from Structural mechanics, Fluid mechanics, and thermodynamics to electromagnetic potential. FEA has been gaining its popularity due to its unique nature of analyzing multiple complex problems easily and it has given a new direction in the field of engineering analysis. One of its major applications is in Structural Mechanics problems. Structural Analysis becomes a major issue in front of Professional engineering or even in academic research when it is a complex situations like analyzing a multidimensional extremely irregular shape, structure associated with different elements with different material properties, sophisticated boundary condition and complex loading pattern with combined effect of different kinds of loads. To overcome with these kinds of problems, we have the tool Finite Element Method. At present time, there is lots of analysis software for structural mechanics problems based on finite element method. These all helps in dealing with complex structural engineering project.

2.0 LITERATURE SURVEY

LS-DYNA(2007) In finite element analysis, mesh density is a critical issue which closely relates to the accuracy of the finite element models while directly determines their complexity level. This paper presents a systematic study on finding the effects of mesh density on the accuracy of numerical analysis results, based on which brief guidelines of choosing the best mesh strategy in finite element modeling are provided. Static, modal, and impact analysis are involved in this study to discuss the effects of element size in finite element analysis.

Brocca M. and Bazant Z.P(2001) Analytical Modelling is the initial step in a software-based Finite Element Analysis of any sophisticated structure associated with irregular shape, complex loading pattern and combination of different loads along with complex boundary condition. In the modelling of structure, Mesh Density plays a major role and it becomes a critical issue of finite element analysis, which closely relates to the accuracy of finite element model while directly determine their complexity level. This project report presents a systematic study on finding the effects of Mesh density on accuracy of numerical analysis results, based on which brief guidelines of choosing the best mesh strategy in finite element modelling is provided. In the project work, for studying the effect of Mesh Density, a bridge deck has been considered and analysis has been performed separately for Self-weight of the structure as well as Vehicular loading which is dynamic in nature. The modelling of the Bridge deck is performed several times with different mesh sizes for studying the analysis results to accomplish the best Mesh strategy that maintain the complexity level of the modelling without compromising in the accuracy of the analysis results. In addition, a case study on similar kinds of attempts to establish perfect Mesh size has been discussed briefly and a comparison has been made between the conclusions of this project work with the case study results, which are in line up to a great extent.

Ashford and Sitar(2001) In finite element analysis, mesh size 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 mesh size 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 static and buckling analysis is carried out to know the effects of mesh sizes by using Femap and NX-Nastran. The model under study is of a structure made up of steel plate.

Saouma V.E., Natekar D. and Sbaizero O(2002) This paper investigates the feasibility of updating finite element models with coarser-than-usual meshes and examines the consequences of using such meshes from a model updating viewpoint. A channel-section structure with relatively simple geometry was modelled using three different mesh densities and the resulting analytical models were correlated with the experimental model obtained from measurements on an actual test structure. All three finite element models were then updated using the inverse eigensensitivity method and the success of the updated models was investigated with particular emphasis on mesh density. Although the model with the highest mesh density produced the best updated model, because of the good initial agreement between the analytical and experimental modal models, it was concluded that the coarse mesh route was worth pursuing, especially in the light of the prohibitive computational requirements associated with large finite element models.

Masakazu T.,(2001) In this work, in order to investigate a modeling technique of the structure with bolted joints, four kinds of finite element models are introduced; a solid bolt model, a coupled bolt model, a spider bolt model, and a no-bolt model. All the proposed models take into account pretension effect and contact behavior between flanges to be joined. Among these models, the solid bolt model, which is modeled by using 3D solid elements and surface-to-surface contact elements between head/nut and the flange interfaces, provides the best accurate responses compared with the experimental results. In addition, the coupled bolt model, which couples degree of freedom between the head/nut and the flange, shows the best effectiveness and usefulness in view of computational time and memory usage. Finally, the bolt model proposed in this study is adopted for a structural analysis of a large marine diesel engine consisting of several parts which are connected by long stay bolts.

Perillo-Marcone A(2003) Simulation-based medicine and the development of complex computer models of biological structures is becoming ubiquitous for advancing biomedical engineering and clinical research. Finite element analysis (FEA) has been widely used in the last few decades to understand and predict biomechanical phenomena. Modeling and simulation approaches in biomechanics are highly interdisciplinary, involving novice and skilled developers in all areas of biomedical engineering and biology. While recent advances in model development and simulation platforms offer a wide range of tools to investigators, the decision making process during modeling and simulation has become more opaque. Hence, reliability of such models used for medical decision making and for driving multiscale analysis comes into question.

3.0 METHODOLOGY

Model under study is a structure made up of steel plates, which is selected from one of the bulk material handling machine. The overall dimensions of model are 2080 mm X 600 mm X 500 mm. Material properties of the steel are listed in Table 1. Amount of load and constraint applied on model are actual loads acting on the structure selected. At two points at the bottom, model was fully constrained, and load of 972 KN along Z-axis, 1960KN along Yaxis and 144KN along X-axis applied at a point at top of model as shown in Fig 1.

Table Steel material properties

Young's modulus	206GPa
Density	7000kg/m3
Yield stress	335MPa
Ultimate stress	470MPa
Poisson's ratio	0.3

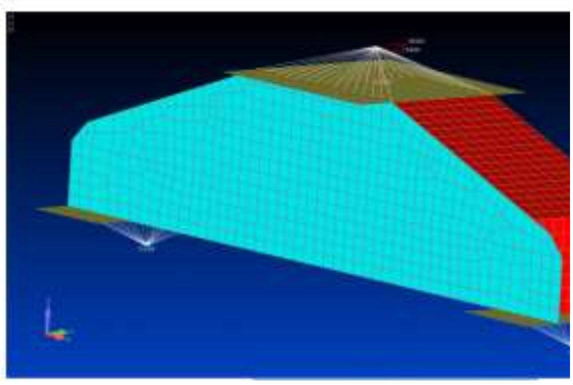

Fig 1. Load and constraints

4.0 RESULTS

Data Assumption and Model Development Following the results of the case study, on a similar track, finite element analysis results have been studied for a bridge deck corresponding to the different analytical modeling of same bridge associated with different mesh density. The basic assumptions made for developing the bridge super structure model are as follows-

• The span of the bridge is 20.0m with a distance of 19.0m between center to center of supports.

• The bridge is a 2- Lane bridge with outer width 8.5m.

• The super structure of the Bridge consists of three RCC I girders with constant sectional properties though out the length of the bridge, connected with bridge deck and analysis results of the central girder have been used in this experiment.

• Each end of the bridge have three supports connected the girder to the bearing pedestal and it gives total six supports in the bridge system and all the supports are assumed to be pinned. • End diaphragm & intermediate diaphragm in girders and crash barrier and sidewalk on the bridge deck has been neglected for simplification of analysis and calculation.

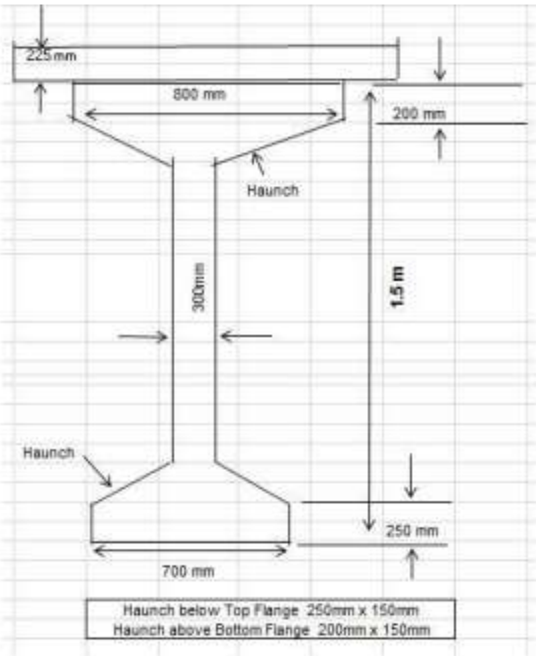

Figure Cross Sectional Details of I Girder

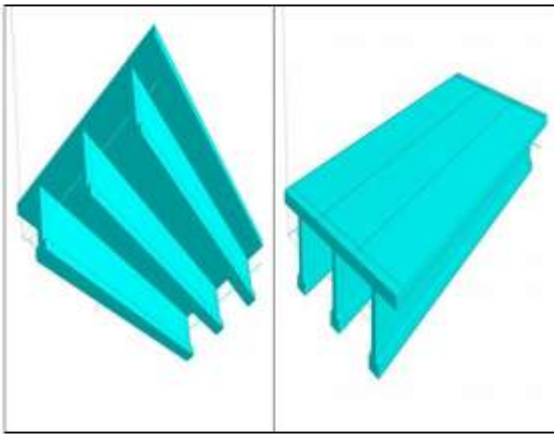

Figure:3D View of the Bridge Deck with Girders

The plan area of the bridge deck has been kept constant (20x8.5m2) and modelling has been performed six different times with a mesh density ranging from 1 to 72. As mentioned earlier, all girders are assumed to be pinned supported and therefore different mesh density has been arrived for the analytical model by diving beam girder in to different numbers of divisions. For example, if all girders in between the supports are represented by a single beam element is corresponding to a mesh density of 1. Hence in other modelling, the girder are divided into 3,6,12,36 and 72 numbers of elements in between supports to generate mesh density of 3,6,12,36 and 72 respectively. Out of these mesh density, 1 to 12 can be treated as coarser, 36 finer and 72 divisions as finest meshing

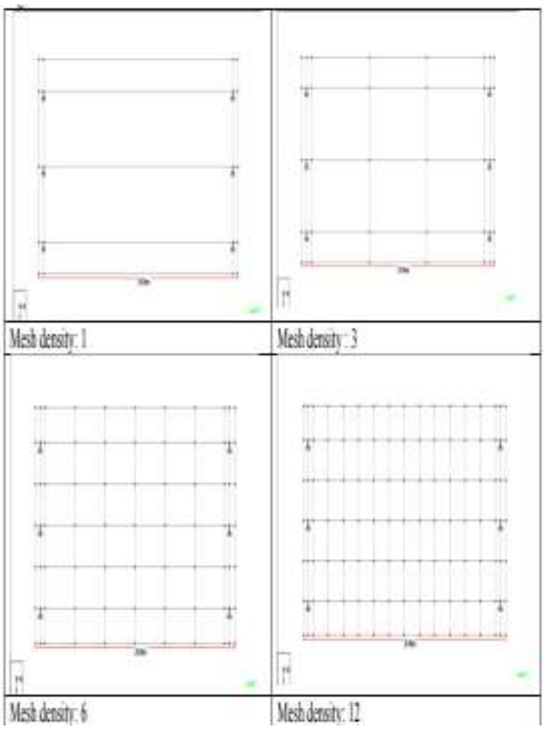

Figure: Modelling with Different Mesh Density

5.0 CONCLUSION

This article presented reporting considerations of FEA studies of biological structures with a focus on solid mechanics ofS biomechanical systems. In the short-term, we hope this compilation will encourage collaborative evolution and adoption of these parameters by investigators, journals, funding agencies, and societies and more thoughtful documentation and review of modeling studies. In return, the uncertainty in the review process of FEA models might be diminished due to utilization of a systematized approach to understand the content of a modeling-based investigation. In the scientific community, this will likely establish confidence through model reproducibility, reusability, and accountability.

We are certainly aware that these parameters are only considerations and are not complete. Our hope is they will evolve along side the evolution of scientific research in computer modeling of biological structures. We further hope that these parameters will be utilized as the basis for customization in other disciplines, for different types of modeling modalities, and also for multidomain, multiphysics, and multiscale analysis. In biomechanics for example, musculoskeletal models commonly used in movement simulations (Erdemir et al., 2007) and fluid-solid coupling utilized in multiphysics investigations of the cardiovascular system (Humphrey and Taylor, 2008) may utilize these parameters for their specific discipline. An online version for prospective editing, adapting, and feedback is available in http://www.imagwiki.nibib.nih.gov/mediawiki/index.php?title=Reporting_in_FEA.

REFERENCES

1. *LS-DYNA Theoretical Manual, Livermore SoftwareTechnology Corporation, May 2007.*

2. *Brocca M. and Bazant Z.P., "Size effect in concretecolumns: finite-element analysis with microplane model",Journal of Structural Engineering, 127(12), 2001,1382-1390.*

3. *Ashford and Sitar, "Effect of element size on the staticfinite element analysis of steep slopes", International Journalfor Numerical Analytical Methods in Geomechanics, 25(14), 2001, 1361-1376.*

4. *Saouma V.E., Natekar D. and Sbaizero O., "Nonlinearfinite element analysis and size effect study in a metal-reinforced ceramics-composite", Materials Science andEngineering A, 323(1-2), 2002, 129-137.*

5. *Masakazu T., "The finite element analysis about sizeeffect of concrete members", Memoirs of the Faculty ofEngineering, Fukuyama University, 25, 2001, 49-54.*

6. *Perillo-Marcone A., Alonso-Vazquez A. and Taylor M.,"Assessment of the effect of mesh density on the material property discretisation within QCT based FE models: apractical example using the implanted proximal tibia",Computer Methods in Biomechanics and BiomedicalEngineering, 6(1), 2003, 17-26.*

7. *Zmudzki J., Walke W. and Chladek W., "Influence ofmodel discretization density in FEM numerical analysis onthe determined stress level in bone surrounding dental implants", Information Technologies in Biomedicine, 47,2008, 559-567.*

8. *Roth S. and Oudry J., "Influence of mesh density on afinite element model under dynamic loading", Proceedings of3rd European Hyperworks Technology Conference,November 2nd-4th, 2009, Ludwigsburg, Germany.*

9. *Li Z.-P., Kindig M.W., Subit D. and Kent R.W.,"Influence of mesh density, cortical thickness and materialproperties on human rib fracture prediction", MedicalEngineering & Physics, 32(9), 2010, 998-1008.*

10. *Liu Y.-C. and Day M.L., "Simplified modeling of thin- walled box section beam",International Journal ofCrashworthiness, 11(3), 2006, 263-272.*