ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 4, Issue 3, May 2015

Effect of Mesh Size on Finite Element Analysis of Plate Structure

Shashikant T. More, Dr. R. S. Bindu

***Abstract* –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.**

Index Terms— **Finite element analysis, mesh size, static analysis, buckling analysis.**

I. INTRODUCTION

In finite element analysis (FEA), the accuracy of the FEA results and required computing time are determined by the finite element size (mesh density). According to FEA theory, the FE models with fine mesh (small element size) yields highly accurate results but may take longer computing time. On the other hand, those FE models with coarse mesh (large element size) may lead to less accurate results but smaller computing time. Also, small element size will increase the FE model's complexity which is only used when high accuracy is required. Large element size, however, will reduce the FE model's size and is extensively used in simplified models in order to provide a quick and rough estimation of designs. Due to its importance, in generating FEA models, the foremost problem is to choose appropriate elements size so that the created models will yield accurate FEA results while save as much computing time as possible. The objective of this paper is to present guidelines for choosing optimal element size for different types of finite element analyses. In order to achieve that goal, in this study, a series of static, and buckling analyses were performed on a structure model made up of plates to reveal the effects of the element size on the accuracy of the FEA results. The solver NX-Nastran and Femap pre and post-possessor, used for modeling and analyses involved in this work.

II. BACKGROUND

A number of investigators have studied the effects of elements size on the accuracy of numerical results of different types of analysis and important conclusions have been drawn from previous research. Yaning Li and Tomasz Wierzbicki have shown that mesh size effects occur when using semi-coupled plasticity/damage theory and Mohr-Coulomb ductile fracture model to simulate ductile fracture of the plane strain flat grooved tensile specimen using shell element model with element deletion. It is found that the stress and strain fields have high gradients in the localization zone and the continuing application of the classical stress-strain relation in the localization zone is the cause for mesh size effects in Finite Element simulations. An equivalent element model is developed to calibrate the non-local stress-strain relation for different mesh sizes. [1] Weibing Liu, Mamtimin Geni, and Lie Yu have obtained different FEA accuracy by different element size and type. It is observed that as curve and surface boundary of higher- order element can approach boundary accurately, calculation accuracy under hexahedral element is higher than tetrahedral element, and calculation accuracy of model analysis can be improved by increasing the number of nodes. [2] Padmakar Raut compared the performance of linear and quadratic tetrahedral elements and hexahedral elements in various structural problems. The problems selected demonstrate different types of behaviour, namely, shear, torsional and axial deformations. It was observed that the results obtained with quadratic tetrahedral elements and hexahedral elements were equivalent in terms of accuracy.[3] Yucheng Liu presented a systematic study on finding the effects of finite element size on the accuracy of numerical analysis results, based on which brief guidelines of choosing best element size in finite element modeling are provided. Static and buckling analyses are involved in this study to discuss effects of element size in numerical analysis. [4]

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 4, Issue 3, May 2015

III. MODEL UNDER STUDY

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 Y-axis and 144KN along X-axis applied at a point at top of model as shown in Fig 1.

Table 1. Steel material properties

Material Properties	
Young's modulus	206GPa
Density	7000kg/m^3
Yield stress	335MPa
Ultimate stress	470MPa
Poisson's ratio	0.3

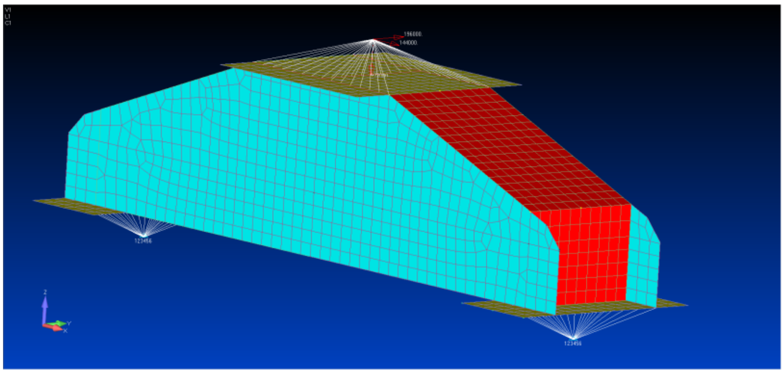
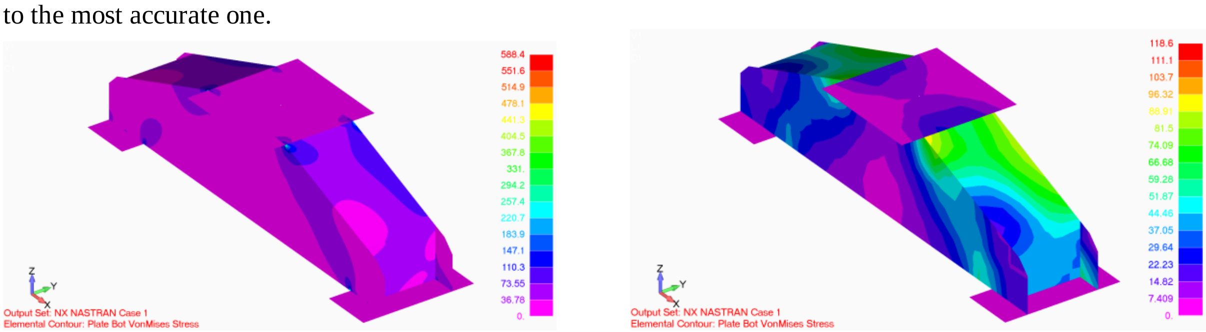


Fig 1. Load and constraints

IV. STATIC ANALYSIS

A series of FE models were generated for structure with mesh size 5 mm and variation from 10 mm to 100 mm with intermediate step of 10 mm using quad shell elements with automatic mesh technique. Fig 2 shows stress distribution of (a) finest mesh model, (b) coarsest mesh model. Fig 6 (a) and (b) shows deformation of coarsest mesh model and deformation of finest mesh model respectively. Von Mises stress and deformation yielded from each model were calculated and compared to study the influence of element size on the static analysis results. Static analysis results and comparisons are listed in Table 2. It is assumed that the FE model with the finest mesh generate the most accurate results and percentage approximate errors were calculated by comparing other results to the most accurate one.



(a) (b)

Fig 2. Stress distribution of (a) finest mesh model (b) coarsest mesh model

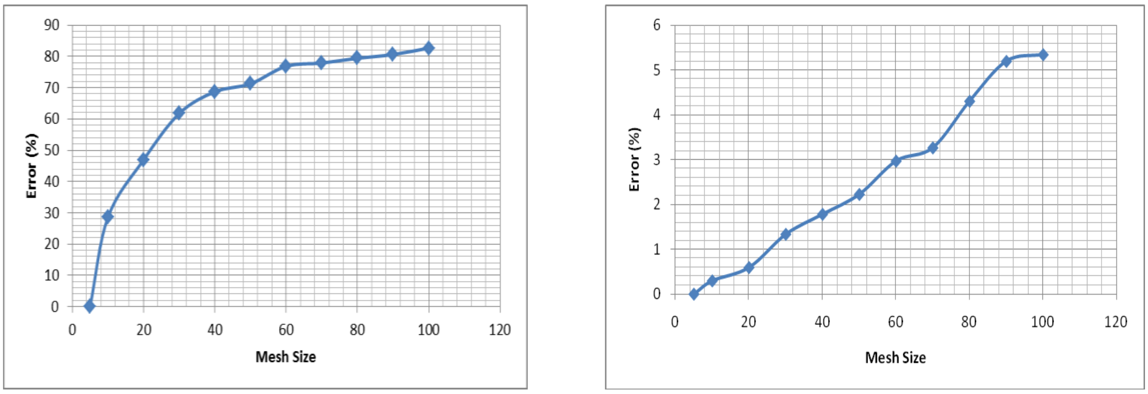
By comparing static analysis results given in table 2 and fig 4, following observations were made

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 4, Issue 3, May 2015

(1) The errors of deformation are far lower than the errors of von Mises stress. According to FEA theory, stresses are not predicted as accurately as the displacements because they are calculated from the displacements and it is assumed that the stresses are constant over the element.
(2) The large variations in the von Mises stresses are at the stress concentration fields in the model.
(3) The computing time for coarse mesh model is only 5 sec, which is less than 1/31 of time by finest mesh model.
(4) It can be concluded that for static analysis, the FE model which is meshed with 40 mm size can give us optimal combination of accuracy and efficiency.

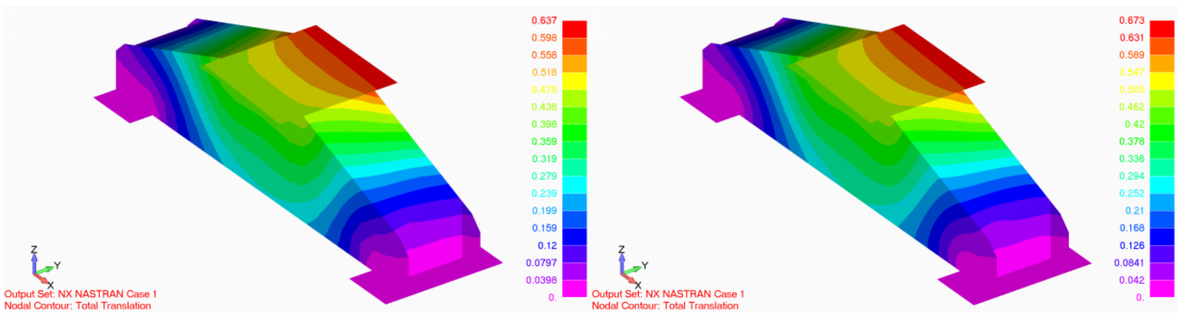
Table 2. Static analysis results and comparisons

Element Size	von Mises stress MPa	%	Deformation mm	%	Computing time Sec.
5	588	0	0.673	0	157
10	420	28.57	0.671	0.29	16
20	312	46.93	0.669	0.59	6
30	225	61.73	0.664	1.33	5
40	184	68.70	0.661	1.78	5
50	169	71.25	0.658	2.22	5
60	136	76.87	0.653	2.97	5
70	130	77.89	0.651	3.26	5
80	121	79.42	0.644	4.30	5
90	114	80.61	0.638	5.20	5
100	102	82.65	0.637	5.34	5



(a) (b)

Fig 3. Element size vs accuracy for (a) maxi.von Mises stress (b) deformation



(a) (b)

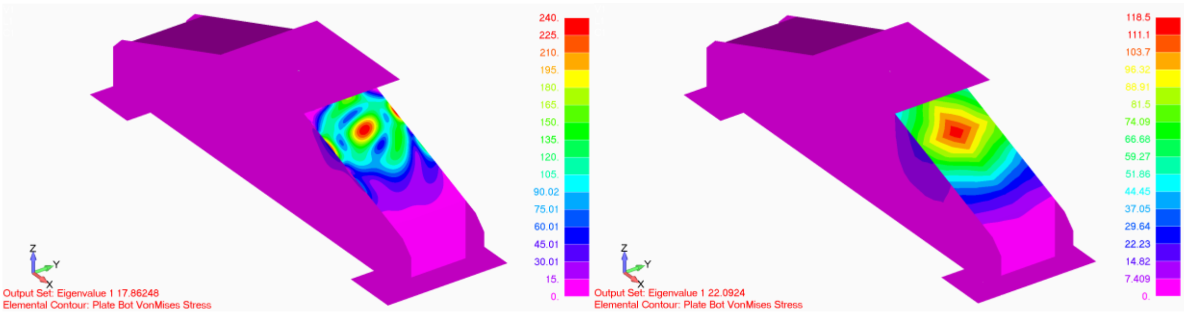
Fig 4. Deformation of (a) coarsest mesh model (b) finest mesh model

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 4, Issue 3, May 2015

V. BUCKLING ANALYSIS

After static analysis, bucking analyses were carried out on same series of FE models as that of created for static analysis. In buckling analysis we solve for the Eigen values which are scale factors that multiply the applied load in order to produce the critical buckling load. In general, only the lowest buckling load is of interest, since the structure will fail before reaching any of the higher-order buckling loads. Therefore, usually only the lowest Eigen value needs to be computed. The set value is the Eigen value and critical buckling factor for a buckling analysis. The model would buckle at a load value of Eigen value times higher than the applied load.

Critical load (buckling load) factor that is Eigen values were computed for each FE model and compared in Table 3, where the approximate error was calculated based on comparing each result to the results yielded from the finest-meshed model. The effects of elements size on the accuracy of buckling analysis results are plotted. Fig 5 shows Eigen value of (a) finest mesh model, (b) coarsest mesh model.



(a) **(b)**

Fig 5. Eigen value of (a) finest mesh model (b) coarsest mesh model

Table 3. Buckling analysis results and comparisons

Element Size	Eigen value	%	Computing timeSec.
5	18.96	19.14	806
10	17.88	19.05	110
20	17.84	19.23	22
30	17.94	18.78	11
40	18.06	18.24	10
50	18.27	17.29	6
60	18.84	14.71	6
70	19.27	12.76	6
80	21.09	4.52	6
90	21.95	0.63	5
100	22.09	0	5

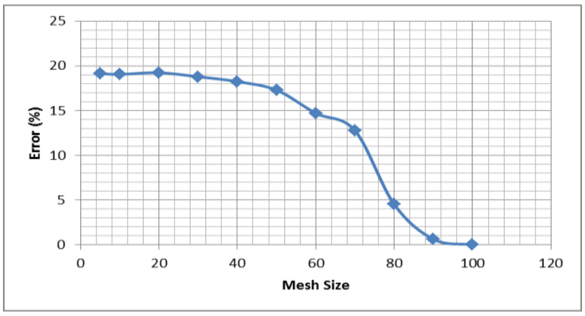


Fig 6. Element size vs accuracy for Eigen value

By comparing buckling analysis given in table 3 and fig 6, following observations were made

(1)Critical buckling load increases with increase in the mesh size due to stress concentrations in elements

(2)Variations in critical buckling load are small for the mesh sizes between 5mm and 50mm.
(3) It can be concluded that for buckling analysis, the FE model which is meshed between 30 and 50mm size can give optimal combination of accuracy and efficiency.

VI. FUTURE SCOPE

As demonstrated in the paper, the element size has different influence on different types of results. Therefore, it is neither possible nor necessary to derive mathematical models for each type of result to show the influences of the element size. Another shortcoming of the FEA models presented in this paper is that those models use automatic mesh only. Advanced mesh techniques such as adaptive mesh are not considered.

The mesh strategy recommended here can be applied to more and more complicated model with irregular shapes for further validation. Also effects of adaptive mesh need to be considered in studying the influences of element size on the accuracy of FEA results.

VII. CONCLUSION

In this study, the effects of element size on accuracy of finite element models and simulation results were investigated through static analysis, and buckling analysis. It was found that for static analysis which assumes steady loading and response conditions, model should be discretized into elements of size 40 mm in order to obtain satisfied results, consuming less computer resources and computing time. For bucking analysis the FE model which is meshed between 30 and 50mm size can give us optimal combination of accuracy and efficiency.

ACKNOWLEDGMENT

Author would like to thank for the valuable suggestions and guidance rendered by Dr. R. S. Bindu. Also thanks to Dr. K Dhande (HOD-Mechanical Engineering) and Dr. R. K. Jain (Principal) for their extreme support. Last but not least many thanks to friends and my colleagues for their help and good will.

REFERENCES

[1] Yaning Li and Tomasz Wierzbicki, Mesh-size Effect Study of Ductile Fracture by Non-local Approach, Proceedings of the SEM Annual Conference, June 1-4, 2009 Albuquerque New Mexico USA.

[2] Weibing Liu, Mamtimin Geni, and Lie Yu, Effect of Mesh Size of Finite Element Analysis in Model Analysis for Periodic Symmetric Struts Support, Key Engineering Materials Vols. 462-463, 2011, pp 1008-1012.

[3] Padmakar Raut, Impact Of Mesh Quality Parameters On Elements Such As Beam, Shell And 3D Solid In Structural Analysis, International Journal of Engineering Research and Applications, Vol. 2, Issue 6, 2012, pp.099-103.

[4] Yucheng Liu, Choose the Best Element Size to Yield Accurate FEA Results While Reduce FE Models's Complixity, British Journal of Engineering and Technology, Vols. 1 No. 7, 2013, pp 13-28.

[5] Nitin S. Godbole, Practicle finite element analysis, First edition, Finite to Infinite, 2008

AUTHOR BIOGRAPHY

Mr. Shashikant Tulshiram More, born on 01-12-1980, completed Diploma in Mechanical Engineering from Rajaram Shinde Institute of Engineering and Technology, Pedhambe, Ratnagiri(MSBTE), in the year 2001 and graduated in Mechanical Engineering from Shri Tuljabhavani College of Engineering, Tuljapur (Aurangabad University),, Maharashtra, India in the year 2004. He is pursuing his postgraduate study in Mechanical Engineering with specialization in Design Engineering from Padmashree Dr. D. Y. Patil Institute of Engineering & Technology (Pune University), Pimpri, Pune, which is affiliated to Savitribai Phule Pune University, Maharashtra, India.

Dr. (Mrs) R. S. Bindu, born on 24-11-1968 graduated in Mechanical Engineering from College of Engineering, Pune, in the year 1987. She did her postgraduate study in Mechanical Engineering with specialization in Heat Power in the year 1994. The author has been engaged in active research in the field of Mechanical Engineering for the last 11 years. The author has 38 research papers to her credit which have been published in various national and international journals. At present, she is working as Professor in the Department of Mechanical Engineering at Padmashree Dr. D. Y. Patil Institute of Engineering & Technology, Pimpri, Pune-411018, Maharashtra, India