International Journal of
Advanced Structures and Geotechnical Engineering
ISSN 2319-5347, Vol. 06, No. 01, January 2017

Effect of Mesh Size of Floor Slab against Lateral Loads while using Etabs Program

MD. MAHMUD SAZZAD, MD. SAMDANI AZAD, MD. TARIQUL ISLAM, FAYSAL IBNA RAHMAN
Department of Civil Engineering, Rajshahi University of Engineering & Technology, Bangladesh
Email: tareq.ruet71@gmail.com

Abstract: This paper represents an analytical overview of using different mesh size while analysis of building frame consisting of area elements in Etabs. It sometimes becomes a challenge for Engineers to arrogate a certain mesh size for floor elements. Meshing is a very important prospect in finite element analysis. Mesh affects not only in statics, but also dynamics of structures as well. All fields of civil engineering faces impressive differences on analytical results due to divergent mesh elements. The boundary value problems greatly reliy on it. Lessening of mesh sizes improves the overall efficiency of building frame by increasing the accuracy of yielding. About 60 models have been analyzed to cope with the optimum outcomes. The outcome of the research guides effective mesh divisions for the Structural Engineers to corroborate the analyses result.

***Keywords**: Mesh, Etabs, Finite Element Analysis and Displacement.*

Introduction:

Size of mesh has considerable importance in finite element analysis. The convergence of finite element analysis depends upon the size of mesh of an element in structural mechanics. In Civil Engineering, researchers use a number of programs for finite element analysis purposes such as Ansys, Abacus, and Adina. On the other hand, Engineers use Etabs, SAP2000, Staad.pro, Prokon for commercial purposes. The use of mesh element is also in commercial programs especially in area elements. But it becomes laborious to make any decision about taking a definite mesh size. Trial and error is a time consuming method for the convergence test. On the contrary, very large elements may the cause of less accurate results while small elements can lead to more accurate outcome. But sometimes taking small element becomes difficult as it increases number of divisions which consumes a lot of time. For the purpose of reducing these problems, an optimum size of mesh or an optimum mesh division is needed.

The aim of present research is to compare the maximum lateral displacements of some selected frames using varying mesh sizes and mesh divisions. Numeral researches have already been conducted by different researchers. More and Bindu [1] discussed on the effect of mesh size on finite element analysis of plate structure. By using Femap and NX-Nastran, they have conducted a series of analysis comprising of static and buckling analysis. In static analysis, they have discussed on deformation and its percentage of error, Von Mises stress and its percentage of errors and computing time for simulation. In buckling analysis, they studied on eigen value, percentage of errors and required time for simulation. The recommendation of this study follows that mesh size between 40-50 mm might be optimum for both analysis and outputs. Effect of mesh size on finite element analysis of beam has been delineated by Dutt [2]. They have used differential mesh sizes consisting of 2, 3, 4, 5 and 6 mm in a beam element. Von Mises stress and deflections have been calculated from analysis and represent that mesh size of 2mm have the output with no error. Liu et al. [3] illustrated on effect of mesh size of finite element analysis in modal analysis for periodic symmetric struts support. They have considered a number of element types. It is delivered that calculations are more correct for hexahedral element. Raut [4] discussed on impact of mesh quality parameters on elements such as beam, shell and 3d solid in structural analysis. The main issue of this study is to discuss on using tetrahedral elements and hexahedral elements. Bending, shear, torsion and axial deformations have been measured for differential mesh sizes. They also discussed about the quality of tetrahedral elements in thin-walled structures. Sazzad et al. [5] studied on mesh effect on the stability of slopes. Mesh elements of dissimilar sizes have been taken in to consideration for the stability analysis of slopes by conducting Finite element analysis. 6 node triangle elements and mixed elements comprising of triangle and quadrilateral had been assumed in this study. The outcome of the research depicts that using 6-nodded triangle element shows less factor of safety.

The aim of present research includes a suggestion to use an optimum meshes division and size for both accuracy and less time consumptions. In present research, Etabs 9.7 have been used as the program for analysis. In case of commercial programs, the meshing is mainly done in area elements as stated above. In present study, different mesh divisions have been adopted for analysis. Furthermore, different mesh sizes have also been adopted. The reasons for these two parameters are that for commercial purposes different area elements may be taken. In such case, a certain mesh size can generate different mesh sizes for different elements. On the contrary, a certain mesh divisions can generate different mesh size in different element. For this purpose, two parameters have been taken into contemplation for the better accomplishments.

In case of residential buildings, the area elements are mainly slab or shear walls. Present research deals with the slab elements. Slab elements in such buildings are dimension in between 3 to 6 meters or 7 meters. For parametric study, 3 different sizes of slab have been adopted for analysis purposes. All these frames are of single bay and single storied which contain a single slab element. Adopted slabs are square in sizes. As the lateral displacements are considered here, earthquake analysis has been carried out. Earthquake load has been considered here as the lateral displacements due to earthquake forces is greater than wind loads [6]. Analytical outputs depict a variation of lateral displacements due to earthquake loads from which a practical decision can be made by considering optimum mesh divisions and sizes.

Finite Element Analysis: Finite element method is a very comprehensive method for structural analysis. This method has been derived for the purpose of numerical solutions. Is this method, the structural frames are accompanied by a combination of nodes and elements. Elements should have certain properties consisting thickness, coefficient of thermal expansion, density, Young's modulus, shear modulus and Poisson’s ratio. The theoretical implementations of finite element analysis have been derived from a group of equations [7]. The beginning of finite element analysis can be found from matrix analysis of structures. The virtual system of any structural frame can be expressed by external and internal virtual work.

External virtual work= $\int \boldsymbol{\delta} \in^{\mathbf{T}} \boldsymbol{\sigma} \mathbf{dV}$ -------------------- (1)

The virtual internal work in the right-hand-side of the above equation may be found by summing the virtual work in the individual elements—this is the crucial step where we will need displacement functions written only for the small domain rather than over the entire system. As shown in the subsequent sections, Eq.(1) leads to the following governing equilibrium equation for the system

$$\mathbf{R} = \mathbf{K}\mathbf{r} + \mathbf{R^0} \text{ -------------------------- (2)}$$

Where, **R**= vector of nodal forces, representing external forces applied to the system's nodes

r= vector of system's nodal displacements, which will, by interpolation, yield displacements at any point of the finite element mesh.

$\mathbf{R^0}$ = Vector of equivalent nodal forces, representing all external effects other than the nodal forces which are already included in the preceding nodal force vector **R**. These external effects may include distributed or concentrated surface forces, body forces, thermal effects, initial stresses and strains.

K= system stiffness matrix, which will be established by assembling the elements' stiffness matrices : $\mathbf{k^e}$

By solving equation to,

$$\mathbf{r} = \mathbf{K^{-1}}\ (\mathbf{R}-\mathbf{R^0}) \text{ ------------------------- (3)}$$

On the other hand, the strains and stresses in individual elements can be calculated as follows

$$\in = \mathbf{Bq} \text{ --------------------------------- (5)}$$

$$\sigma = \mathbf{E}(\in - \in^0) + \sigma^0 \text{ ------------------ (6)}$$

$$\sigma = \mathbf{E}(\mathbf{Bq} - \in^0) + \sigma^0 \text{ -------------(7)}$$

Here, **q**= vector of element's nodal displacements--a subset of the system displacement vector **r** that pertains to the element under consideration.

B= strain-displacement matrix that transforms nodal displacements **q** to strains at any point in the element.

E= elasticity matrix that transforms effective strains to stresses at any point in the element.

$\in^0$ = Vector of initial strains in the element.

σ^0 = Vector of initial stresses in the element.

By applying the virtual work equation (1) to the system, we can establish the element matrices **B**, $\mathbf{k^e}$, as well as the technique of assembling the system matrices $\mathbf{R^0}$ and **K**.

Methodology:

Seismic analysis provisions from UBC1997 have been followed for performing necessary calculations. The cross sectional area of beams, columns and slabs has been taken to maintain similarity for all models. Relevant material properties are also same. The used frames contain 3X3, 4X4 and 5X5 m slabs.

Table 1: Different specifications used in analysis

Contents	**Specifications.**
Beam Size	250X300 mm
Column Size	250X250 mm
Slab Size	125 mm thick
Height	3m
Support Conditions	Restraint

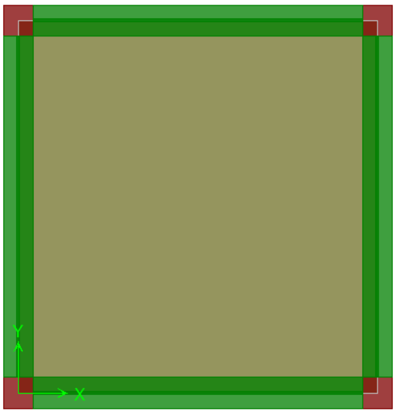

Figure 1: Plan of square model

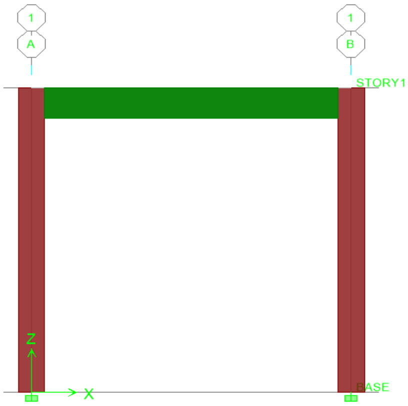

Figure 2: Elevation of square model

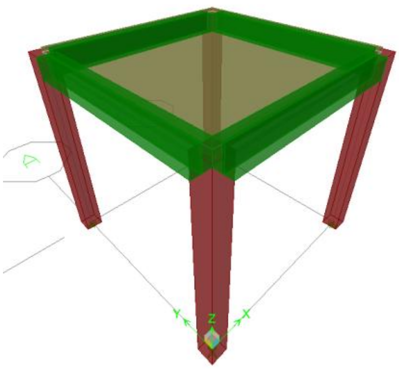

Figure 3: 3D view of square model

For conventional analysis, Equivalent static force method has been adopted.

$$V = \frac{CvI}{RT}W \text{---------------------------- (8)}$$

$$V = \frac{2.5CaI}{R}W \text{-------------------------- (9)}$$

$$V = 0.11C_aIW \text{----------------------------- (10)}$$

$$T = C_t(h_n)^{3/4} \text{----------------------------- (11)}$$

The value of base shear obtained from equation (8) should be less than the value acquired from equation (9) and should be greater than the value of equation (10).

C_t = 0.0731 (0.030) for reinforced concrete moment-resisting frames and eccentrically braced frames.
W= Total seismic weight of the building.
C_v=0.32
R=Response modification factor=5.5
Soil profile=S_c
T=Structural period, which can be calculated from equation (11)
I= Structural importance factor=1

Table 2: Seismic Parameters

Parameters	Specifications
Base Shear, V	Calculated from equation (8)
Zone factor, Z	0.20
Structural importance , I	1
Response modification factor, R	5.5
Soil Profile, S	S_c
Building Coefficients. Ct	0.073

In table 2, the Zone factor (Z) has been taken from earthquake zoning map of Bangladesh [8]. Present study has taken zone 2 for performing analysis. The other properties have been taken from UBC1997 [9].

Effect of Mesh Size:

In present study, the analyses have been conducted in two steps. In first step, differential mesh divisions have been used. In this case, mesh divisions of 20 to 200 have been adopted considering an interval of 20. For each consecutive frame, analyses have been conducted for ten times. In the second step, mesh size has been taken as the variable. Mesh sizes of 20-100 mm have been contemplated with an interval of 10 mm.

Table 3: Table for lateral displacements (D) for different mesh divisions

Mesh Divisions	D (3X3) (mm)	D (4X4) (mm)	D (5X5) (mm)
20	0.5155	0.8052	1.1711
40	0.5226	0.8088	1.1716
60	0.5258	0.8132	1.1774
80	0.5273	0.8154	1.1802
100	0.5282	0.8166	1.1819
120	0.5288	0.8175	1.183
140	0.5292	0.818	1.1837
160	0.5294	0.8184	1.1843
180	0.5297	0.8188	1.1847
200	0.5298	0.819	1.185

Table 3 illustrates the lateral displacements due to earthquake forces for different mesh divisions. Substantial differences can be observed from the table. It is clear that the value of D increases with the intensifying value of mesh divisions. If the mesh division expanded, the value of D will be raised. Though the differences are very low, but the difference might be enlarged for multistoried buildings.

Table 4: Table for lateral displacements (D) for different mesh sizes

Mesh Sizes (mm)	**D (3X3) (mm)**	**D (4X4) (mm)**	**D (5X5) (mm)**
20	0.5293	0.819	1.1858
30	0.5282	0.8179	1.1845
40	0.527	0.8166	1.1832
50	0.5258	0.8154	1.1819
60	0.5245	0.8141	1.1806
70	0.5233	0.8129	1.1793
80	0.5221	0.8114	1.1779
90	0.5209	0.8103	1.1765
100	0.5194	0.8088	1.1751

It is observed from table 4 that the value of D increase with the decreasing values of mesh sizes. In this case, the differences are also negligible as our models are very simple. But the differences might be higher for multistoried building.

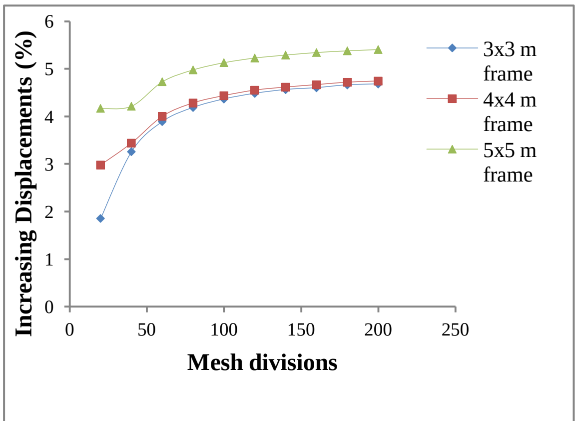

Figure 4: Percentage of increasing displacements with respect to mesh divisions

Fig. 4 depicts the increasing percentage of displacements with respect to mesh divisions. It is understandable that the percentages increase at a high rate up to 40 or 50 divisions. The percentages have been calculated by adopting 20 divisions as value. After 50 divisions, the curve gets flattened. It is also evident that for 4x4 bay and 5x5 bay frame, the percentage have a up and down value up to 50 divisions. So this can be a good option to take mesh divisions around 50 or 60.

Table 4: Determination of mesh sizes from mesh divisions

Specimen	50 divisions	60 Divisions
3X3	60 mm	50 mm
4X4	80 mm	66.67 mm
5X5	100 mm	83.33 mm

Following table 4, the determined mesh sizes can be used for different sized area elements to reduce errors.

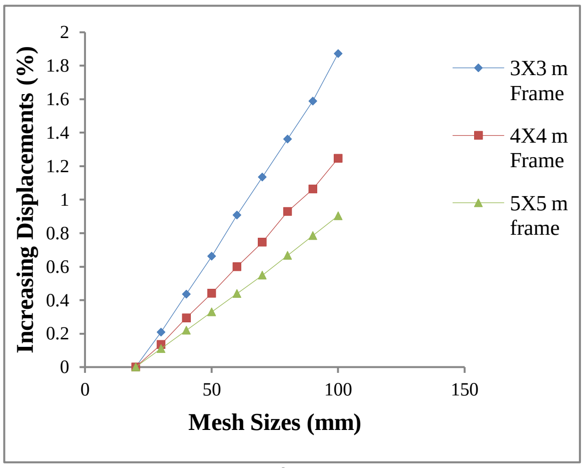

Figure 5: Percentage of increasing displacements with respect to mesh sizes (mm)

Fig. 5 deliberates the chart of increasing percentages of displacements with respect to mesh sizes. From the graph, it can be remarked that the percentage increases rapidly if the mesh sizes are increased. These percentages have been enumerated by appraising the base value for 20 mm mesh size. From previous graph, 50 mm to 60 mm mesh size might be optimum for 3X3 m frame and it is true for Fig 5 too. On the contrary, 67-80 mm mesh can be used for 4x4 m frame and 83 to 100 mm mesh size can be used for 5x5 m frame. By adopting these different mesh sizes, the error percentage will be in the same level in fig 5 and the percentages will below 0.8.

Conclusion:

From the numerical inspection presented above, it is very difficult to make a choice of a certain mesh size. Because, a certain mesh size may generate different mesh divisions for different sized frame. In such case, it will be difficult to understand about the accuracy of mesh size. But for all dimensioned frame, an average mesh divisions of 50 or 60 can solve this issue which has been stated above clearly. So while using Etabs for structural frame analysis, it is recommended to keep the mesh divisions limited within 50 to 60 divisions for area elements to secure optimum output.

References:

[1] More, S., T. and Bindu, R., S. 2015. "Effect of Mesh Size on Finite Element Analysis of Plate Structure", International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 4, Issue 3.

[2] Dutt, A. 2015. "Effect of Mesh Size on Finite Element Analysis of Beam", SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 12.

[3] Liu, W., Geni, M. and Yu, L. 2011. "Effect of Mesh Size of Finite Element Analysis in Modal Analysis for Periodic Symmetric Struts Support", Key Engineering Materials Vols. 462-463 (2011) pp 1008-1012

[4] Raut, P. 2012. "Impact Of Mesh Quality Parameters On Elements Such As Beam, Shell And 3D Solid In Structural Analysis", International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 2, Issue 6, November- December 2012, pp.099-103

[5] Sazzad, M., M., Rahman, F., I. and Mamun, M., A., A. 2015. "Mesh Effect on FEM based Stability analysis of Slope", International Conference on Recent Innovation in Civil Engineering for Sustainable Development (IICSD-2015)

[6] Sazzad, M., M. and Azad, M., S. 2015. "Effect of building shape on the response to wind and earthquake", International Journal of Advanced Structures and Geotechnical Engineering ISSN 2319-5347, Vol. 04, No. 04

[7] Finite Element Method in Structural Mechanics (https://en.wikipedia.org/wiki/Finite_element_method_in_structural_mechanics)

[8] Bangladesh National Building Code 2014

[9] Uniform Building Code 1997