IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 17, Issue 2 Ser. V (Mar - Apr 2020), PP 44-48
www.iosrjournals.org

The Mesh Quality significance in Finite Element Analysis

Amit Nemade[1], Azim Shikalgar[2]
[1, 2]School of Mechanical Engineering, MIT World Peace University, Pune, India.

Abstract:*Computer Aided Engineering is a vital simulation software widely used in mechanical, biomechanics and many other applications. CAE plays a crucial role in product designing which obtains the stress, displacement, elastic strain and fatigue in the component. The reliability of the product's structure being analyzed via Finite Element Analysis is highly dependent on the accuracy of the analysis execution. Discretization of the domain is one of the crucial issues in Finite Element Analysis, which is also known as meshing. A well discretized mesh is necessary in order to obtain the more accurate and precise results within the less computational time. The present work aims at testing a finite element model of a plate with four bolt holes using various meshing strategiesand the results are investigated after numbers of numerical simulations to conclude the best choice for meshing.*

Key Word*:FEM,Discretization,Mesh quality, Aspect ratio, Jacobian ratio*

Date of Submission: 16-04-2020 Date of Acceptance: 01-05-2020

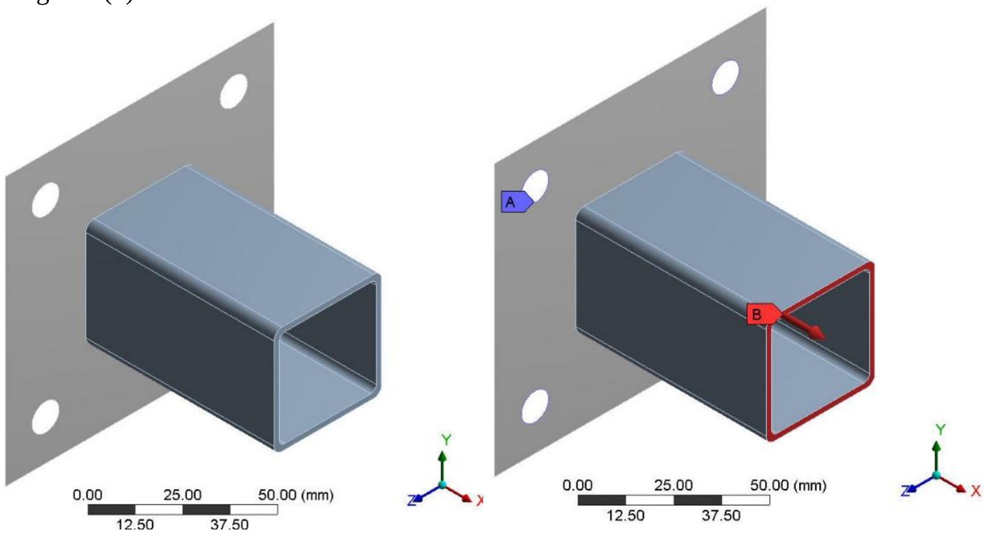
I. Introduction

Finite element Analysis is a simulation software tool which has been widely used in many applications since few decades to study the stress, elastic strain, displacement and fatigue in the field of mechanics. In FEA, the degree of freedom is reduced from infinite to finite by discretization. With the help of discretization, the main domain is break in to number of pieces where each piece represents an element. The process of discretization is also called as "meshing". The meshing of the geometry plays a crucial role in stability and accuracy of numerical computation. The result computing time and the accuracy of the results are dependent on the mesh density also means the size of the element. The FEA theory states that a Finite Element Model with fine meshed elements yield to obtain high accuracy but relatively the process may take more computing time. The fine mesh may be used only when the high accuracy and precision is required. The Finite Element Model with coarse mesh may simulate the results in short computing time but the results obtained may not be accurate in comparison with the fine mesh FE model. It becomes necessary to choose appropriate mesh and element size to obtain the FEA results having high accuracy with minimum possible commuting time. The quality of the mesh during FEM simulation can be checked by the count of mesh elements, checking the Aspect ratio and the Jacobian ratio for elements [100].The element quality is the value ranges from 0 to 1 with higher values meaning higher quality elements and looking at the minimum values and making changes in the mesh to try to increase the minimum element quality in the mesh. The Aspect ratio is the ratio of the longest dimension of the individual element to its shortest dimension and the mesh quality can be increased by decreasing the maximum value of the ratio. Jacobian Ratiois a mathematical quality based on the determinant of the Jacobian matrix at various sampling points on each element usually the nodes or integration points. For the Jacobian ratio, to obtain the quality mesh, you will be wanting to decrease the maximum ratio. The histogram is obtained for checking the mesh quality, basically graphs out mesh metrics so element quality on the X-axis and the number of elements on the Y-axis.

The objective of the present work is to propose the guidelines for choosing the optimum method of meshing. In order to achieve this objective, series of analysis studies are carried out on a plate with four bolt holes. The analysis is performed to discuss the effect of various meshing strategies and element size in FEA. The relationship between the quality of the mesh and the simulation results is obtained from the variation in the dimension of the elements and the mesh strategy used which is further assessed through the number of nodes and elements, the Aspect ratio and Jacobian ratio.

II. Method and Analysis

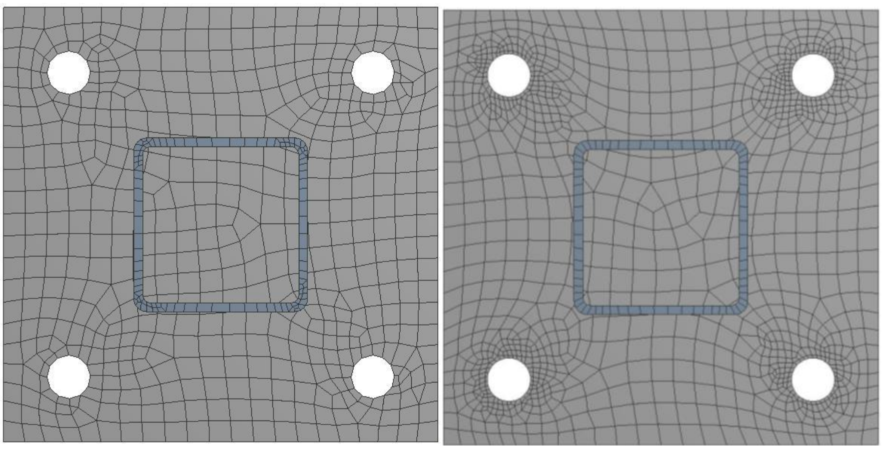
The plate with four bolt holes having 3 mm thickness ss considered for the evaluation of the mesh strategy focusing on the induced stress and deformation under the static loading. The plate is consideredrigidly fixed through four bolts and the static load is applied parallel to the axis of the bolts on the surface of the square section. The geometry is modelled in the ANSYS Design Modeler as shown in figure 1(a). Since the plate is bolted through the holes, the holes edges are considered to be rigidly fixed and the pulling load is applied at the centre parallel to the axis of the bolts through the welded square tube. The applied boundary conditions are proposed in figure 1(b).



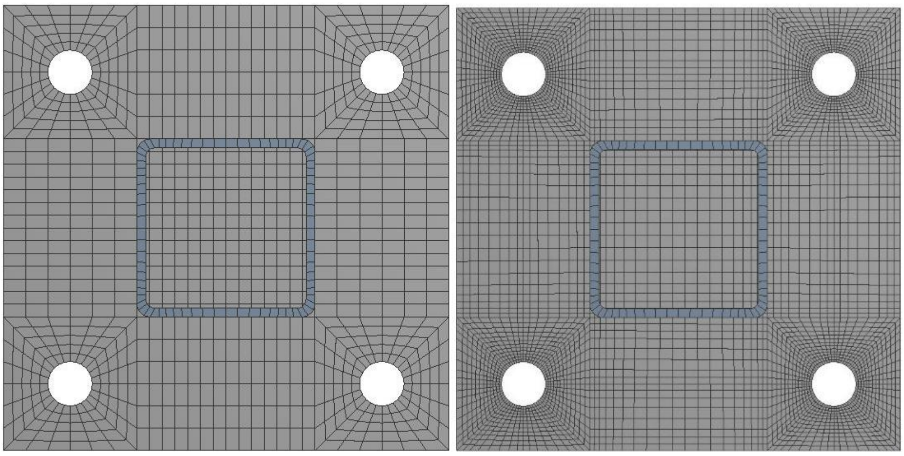
(a) (b)

Figure 1.(a)Plate with four bolt holes, (b)Applied boundary conditions

The plate material is considered to be structural steel. Four types of meshing techniques regeneratedand the results are evaluated to elaborate the dependency of type of mesh, element size on the FEA results. The first mesh is generated with free mesh of Hex20 and Quad4 elements having fairly equal size of 5. 086mm.Figure2aillustrates the generated free mesh. The mesh quality, aspect ratio and jacobian ratio can be checked via "Mesh-Display Style". The maximum mesh quality of the generated mesh is 0.99723 and minimum is 0.21482, the maximum aspect ratio is 3.4589while the maximum jacobian ratio is obtained as 6.5008. The same mesh is generated with high density elements at the holes since the area experiences high stress concentration. For obtaining the high-density mesh at the holes, the hole edge sizing is generated with element size equals 3mm as shown in figure 2b.The structured mesh is generated by dividing the edge of hole and the face around the hole into eight faces and providing the same body sizing for each hole edge and the opposite edge. Thethird mesh is generated with mapped mesh of Hex20 and Quad4 elements having low aspect ratio that is, the ratio of the longest dimension of the individual element to its shortest dimension. Figure 2c illustrates the pattern of the elements in the structured mesh. The maximum mesh quality the generated mapped mesh is 0.99947 and minimum is 0.53499, the maximum aspect ratio is 2.2748 and while, the maximum jacobian ratio is obtained as 2.0884. The fourth mesh is generated with the same structured elements but have high aspect ratio. The elements follow the uniform pattern as illustrated in figure2d. The maximum mesh quality the generated mapped mesh is 0.99947 and minimum is 0.5141, the maximum aspect ratio is 2.7737 while the maximum jacobian ratio is equals to2. 0884.The number of nodes and elements are varied according to the density and aspect ratio of the mesh. Figure 3(a) and figure 3(b)illustrates the comparison four mesh in terms of the total number of nodes and elements.

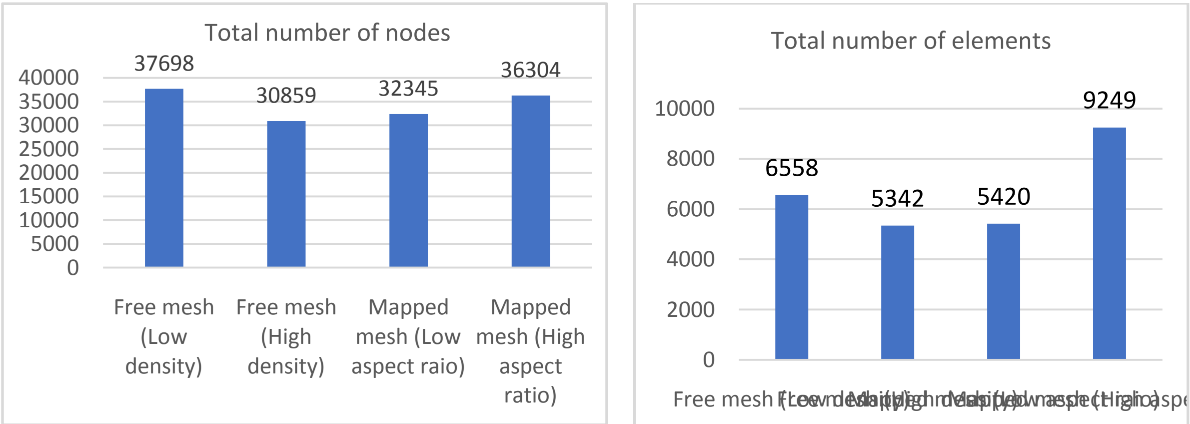


(a) (b)



(c) (d)

Figure 2. (a)Low density free mesh, (b)High density free mesh, (c)Mapped mesh with low aspect ratio and (d)Mapped mesh with high aspect ratio.

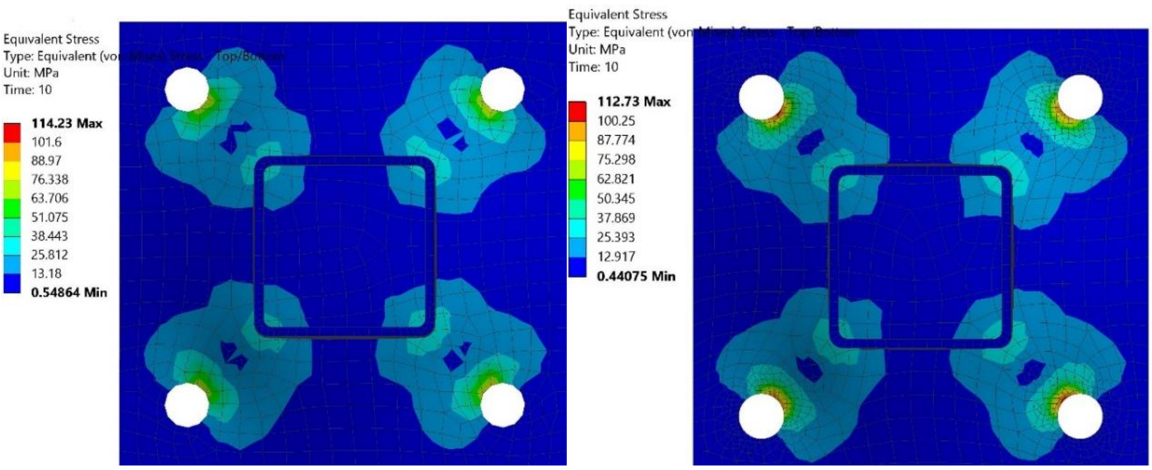


(a) (b)

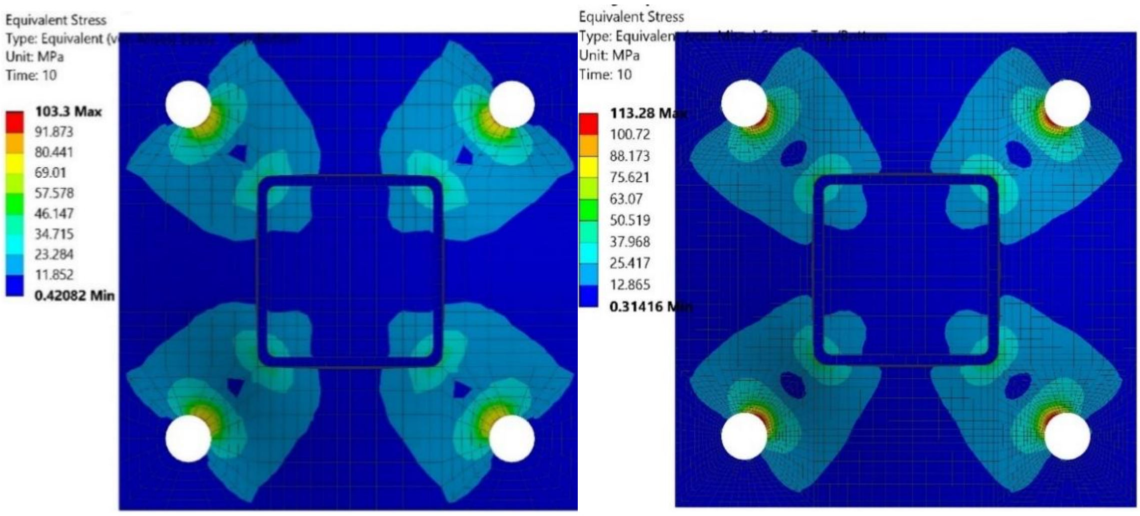
Figure 3. Comparison of the meshes in terms (a)Total number of nodes and (b)Total number of elements.

III. Results and Discussion

The load is applied in 10 steps varying from 100N to 1000N. The results of the FEM simulation are presented and evaluated on the plate with holes in terms of equivalent Von-Mises stress (Figure 4) and total deformation (Figure 5). The stress contour presents the maximum stress is induced at the highly stress concentrated area at the cylindrical edge of the holes. The simulation results are further compared to study the influence of the mesh technique on the analysis results. Figure illustrates the close comparison of Von-Mises stresses and the total deformation for all four mesh models.

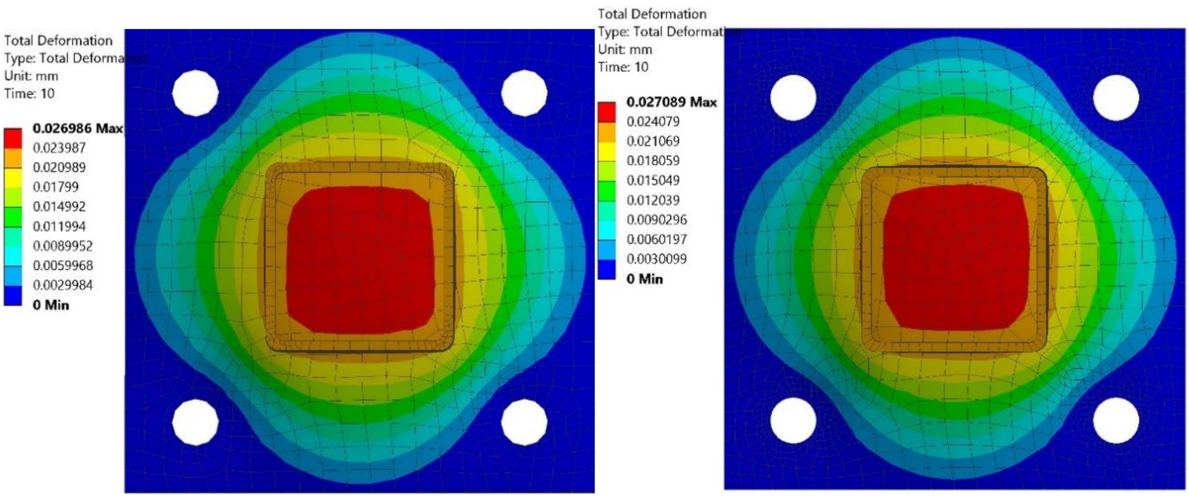


(a) (b)

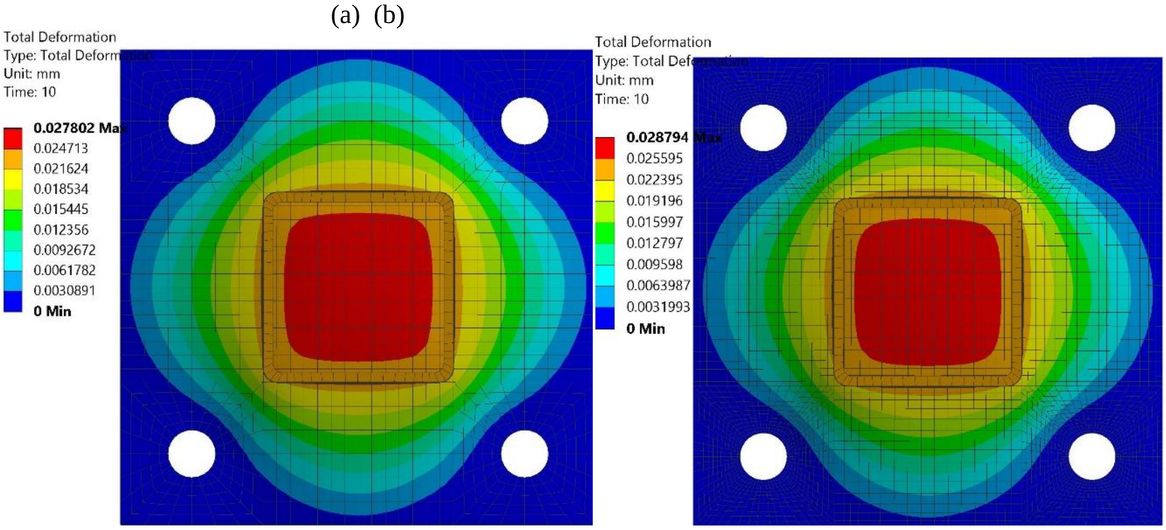


(c) (d)

Figure 4. Equivalent Von-Mises stress for (a)Low density free mesh, (b)High density free mesh, (c)Mapped mesh with low aspect ratio and (d)Mapped mesh with high aspect ratio models.



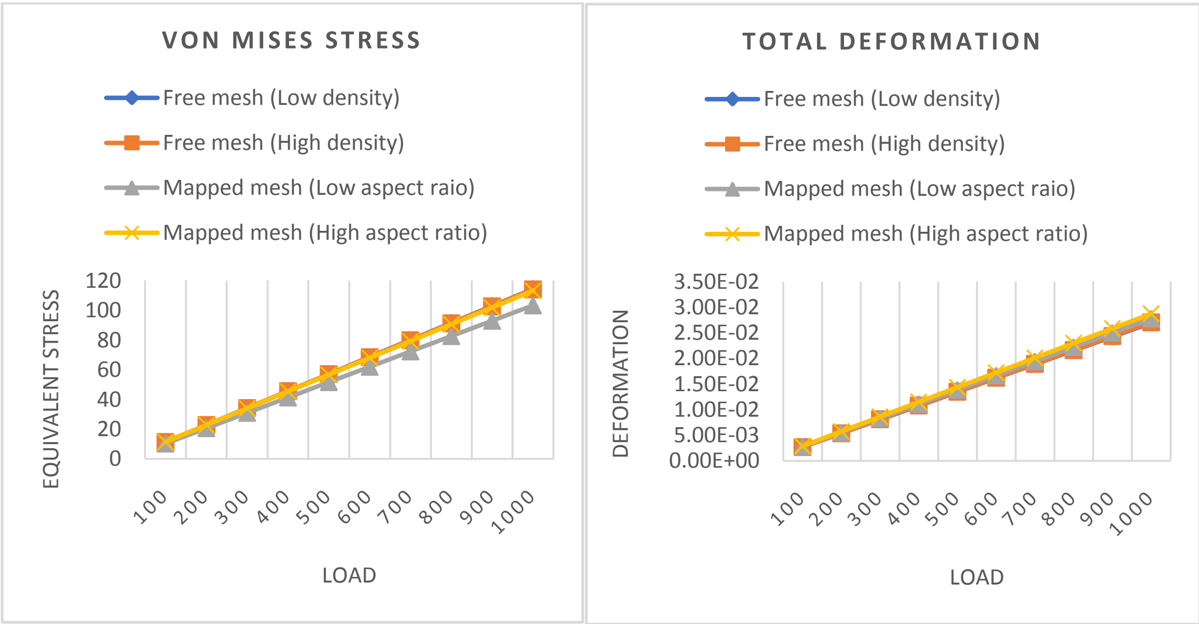
(a) (b)



(c)(d)

Figure 5. Total deformation for (a)Low density free mesh, (b)High density free mesh, (c)Mapped mesh with low aspect ratio and (d)Mapped mesh with high aspect ratio models.

Figure6illustrates the close comparison of Von-Mises stresses and total deformation for all four mesh models. For "low density free mesh", "high density free mesh" and "high aspect ratio mapped mesh", the stress contour results are fairly close and consistent throughout. The stress behaviour of the low aspect ratio mapped mesh is slightly different in comparison with the other three mesh models with a difference of 8.81% in the maximum Von-Mises stress at the same loading of 1000 Newtons. If the high-density free mesh and high aspect ratio mapped mesh are compared, the maximum stress magnitude for both the mesh are close with a small magnitude difference of 0.485%. Also, there is very less difference between the maximum stress results of low density and high-density mesh where the low-density free mesh over estimates the high-density free mesh stress value by 1.313%. The low aspect ratio mapped mesh having aspect ratio 2.2748 differ by 8.81% with respect to high aspect ratio structured mesh having aspect ratio equals 2.7737 hence it could be concluded that the proper mapped mesh with an adequate aspect ratio does not propose the precise results. From the obtained results, it can also be said that the adequate element sizing is required to obtain the accurate results.



(a) (b)

Figure 6. Comparison of (a)Von-Mises stresses and (b)The total deformation for all four mesh models

IV. Conclusion

The four types of meshing techniques low density free mesh, high density free mesh, low aspect ratio structured mesh and high aspect ratio structured mesh were considered for the analysis of plate with four bolt holes. The mathematical simulation was carried out for all four mesh to obtain the Von-Mises stress and total deformation and the results were compared. It was concluded that the mesh density should be high at the high stress concentrated area. The proper structured mesh with an adequate aspect ratio does not propose the accurate and precise FEA results but it is more depend upon the sizing of the element that is the mesh density.

References

[1]. Nitin S. Godbole, Practical finite element analysis, First edition, Finite to Infinite, 2008
[2]. Amir Javidinejad, "FEA Practical Illustration of Mesh-Quality-Results Differences between Structured Mesh and Unstructured Mesh", International Scholarly Research Network, Volume 2012.
[3]. Yucheng Liu and Gary Glass, "Effects of Mesh Density on Finite Element Analysis", SAE International, 2013-01-1375.
[4]. 12. ANSYS Meshing User's Guide. Available online: https://www.ansys.com/products/platform/ansys-meshing (accessed on 18 May 2019)
[5]. Saouma V.E., Natekar D. and Sbaizero O., "Nonlinear finite element analysis and size effect study in a metal reinforced ceramics-composite", Materials Science and Engineering A, 323(1-2), 2002, 129-137.
[6]. M. Picasso, "Adaptive finite elements with large aspect ratio based on an anisotropic error estimator involving first order derivatives," Computer Methods in Applied Mechanics and Engineering, vol. 196, 14–23, 2006.

Amit Nemade,etal. "The Mesh Quality significance in Finite Element Analysis."*IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),* 17(2), 2020, pp. 44-48.