ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 6, December 2013

Study of the Effect of Finite Element Mesh Quality on Stress Concentration Factor of Plates with Holes

Sharaban Thohura, Dr. Md. Shahidul Islam
Department of Mathematics, Jagannath University Dhaka, Bangladesh.
Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering & Technology, Dhaka, Bangladesh

Abstract— ***The distributions of stresses in an infinite rectangular plate with central circular holes subjected to uniform tensile force at the edges of the plate have been studied by using Finite element Method. Four-node quadrilateral membrane element (2 degrees of freedom per node) has been used in meshing the domain. The aim of the authors is to make a guideline for proper meshing of the computational domain. For numerical investigations different models have been used in this study with variable number and quality of elements from model to model. The mesh quality has been defined as a combination of two criteria such as element quality and enough number of elements in the mesh. The quality of the surface mesh should be good to get acceptable finite element results. In this study the quality of the quadrilateral element has been carried out using two factors namely: Distortion factor and Aspect ratio. Based on these two factors an object oriented program in C++ has been developed. We have determined the stress concentration factor through finite element analysis using PATRAN pre-processor and NASTRAN solver combination and have compared the same with published theoretical results.***

Index Terms—**Stress Concentration Factor, Mesh quality, mesh seed, Finite element Analysis.**

I. INTRODUCTION

Configuring the structures with discontinuities is one of the most important topics in the construction of ships, aero-planes, cars etc. Whenever the cross–section of a structural member changes suddenly, a structural discontinuity arises. The application of finite element method (FEM) to the analysis of discontinuous structural systems has received a significant interest in recent years. Examples of problems in which discontinuities play prominent role in the physical behavior of a system are numerous. From mathematical point of view, analytical solutions are possible only for a limited class of such problems. The complexities of the structures, material properties and of boundary conditions, have progressively led to the predominance of numerical models based on finite elements and finite differences. For cases in which discrete representation of discontinuities is required, the finite element approach provides the best modeling to date. In the past, stress analyses were performed analytically or experimentally, which could be both difficult and time consuming, especially when dealing with discontinuities. Many times an accurate solution was not possible due to the complexity of the discontinuity configuration. However, with the advent of Finite element method (FEM), theses analyses can now be performed with a degree of accuracy. Considering the importance of structural discontinuity, an extensive research work has been carried out by naval architects, offshore and ocean engineers, hydro dynamists and mathematicians. Until now, the stress concentration factor for various isotropic structures is investigated. Heywood [1], Peterson [2] and Pilkey [3] have investigated various isotropic shapes with wide range of holes. Howland [4] determined the solution of the problem of a long isotropic rectangular plate with a centered hole subject to a tension load. Peterson has developed good theory and charts on the basis of mathematical analysis and presented excellent methodology in graphical form for evaluation of stress concentration factors in isotropic plates with different types of abrupt change. Heywood [1] introduced various equations for long length and finite width plate with different opening shapes. In Finite Element Method (FEM) a complex region defining a continuum is discretized into simple geometric shapes called finite elements or meshes. The generation of a finite element mesh during modeling is one of the uttermost challenging tasks [5]. Recently; most finite element codes in the modeling market are capable of producing automatic meshes. The art of designing a suitable finite element mesh still needs constant human intervention, especially to problem associated with discontinuities such as cracks and holes[5], [6]. Generation of poor mesh would lead to the formation of incorrect output with elegant graphic display. So the effect of mesh quality plays a significant role in Finite element analysis. Without this knowledge erroneous result can be produced which can lead to design of a faulty structure. To get accurate result in less modeling time is also very important in the industries now a day because it saves computing ability and time which optimizes production cost. Hence it needs attention for proper design and analysis of meshes of such structural discontinuities. In the present study, the stress concentration around circular holes in an infinite plate subjected to uniform tensile force at the edges of the plate has been studied by using Finite element Method and the

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 6, December 2013

effect of mesh quality on stress concentration factor will has been studied as well. The aim of this study is to make a guideline for proper meshing of the domain. In the first part a finite element model of a rectangular plate with a circular hole has been made and analyzed it to find out stress concentration factor and compared it with the value found from the formula given by Pilkey [3]. After finding out the correct approach of meshing the model has been extended for multiple holes.

II. FORMULATION OF THE PROBLEM

To study the distribution of stresses and determination of SCF, a thin steel plate with a central single hole and plate with multiple holes have been modeled as static, solid, structural, linear and isotropic material model. To define the material property we have taken the Modulus of Elasticity E is 30×10^6 psi and Poisson's ratio υ is 0.3. The element behavior is a plane stress with thickness. The Figure 1 below is a display of a two-dimensional model of a thin rectangular steel plate with a central circular hole. The length A, width H and the thickness t of the plate are 20 inch, 20 inch and 0.4 inch respectively and the diameter of the circular hole is 2 inch. The plate has been placed under constant tensile stress σ (400psi), acting in a perpendicular direction to the width at the edges of the plate.

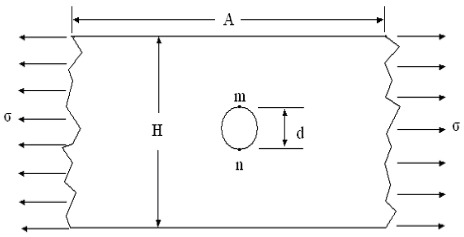

Fig.1: Plane stress of a finite width element with a circular hole.

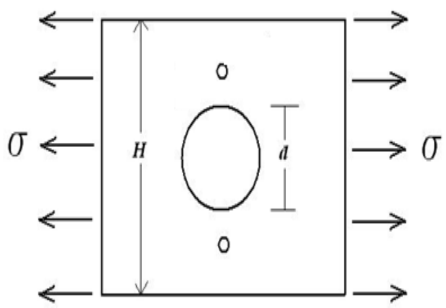

Fig. 2. A plate with a center hole and additional holes in a plane perpendicular to the loading direction

Figure 2 shows a plate with a center hole and two additional smaller holes in a plane perpendicular to the loading direction. For modeling and analysis we placed an additional hole of 0.5 inch diameter (in the quarter mode) and placed its centre 5.5 inch away from the centre of the central hole (remaining length above center hole/2 = 9/2 = 4.5). All other dimensions and parameters are kept same.

III. THEORETICAL STRESS CONCENTRATION FACTOR

As is well known the theoretical stress concentration factor (SCF) for normal stress is defined [2], [7] as

$$K_{tg} = \frac{\sigma_{\max}}{\sigma} \qquad (1)$$

where $\sigma_{\max}$ is the maximum stress in the body and σ is the stress on gross section taken as reference stress. The value of this factor depends very much on the abruptness of discontinuity, and it follows that it is desirable to design the structures in the neighborhood of a discontinuity so as to keep this factor as low as possible.

Stress concentration Formula for plate with a hole is defined as Pilkey & Pilkey [3]

$$K_{tg} = 0.284 + \frac{2}{1 - d/H} - 0.600\left(1 - \frac{d}{H}\right) + 1.32\left(1 - \frac{d}{H}\right)^2 \qquad (2)$$

where K_{tg} is the stress concentration factor based on gross stress.

IV. MESH GENERATION AND MESH QUALITY

The creation of meshing in finite element models is the most important step in the entire analysis since the decision made at this point will affect the accuracy and the economy of the solid model. Thus the choice of mesh for various analyses is crucial. An element or mesh that is good in one problem area such as magnetic field may be poor in another such as stress analysis. Even in a specific problem area an element may behave well or badly, depending on particular geometry, loading and boundary conditions. It is generally accepted that simplex triangular elements are inferior when compared to bilinear quadrilaterals. According to Brauer[8], for reasons of better accuracy and efficiency, quadrilateral elements are preferred for two- dimensional meshes and hexahedral elements for three-dimensional meshes. This preference is clear in structural analysis and seems to also hold for other engineering disciplines [8]. In view of that four-node quadrilateral membrane element (2 degrees of freedom per node) has been used in meshing the computational domain. The mesh quality has been defined as a combination of two criteria such as element quality and enough number of elements in the mesh. The quality of the surface mesh should be good to get acceptable finite element results. Element quality for quadrilateral element is defined by two factors namely: Distortion factor and Aspect ratio. Zhu et al.[9] considered a quadrilateral element satisfactory if all its internal angles θ fall within $90° \pm 45°$ and was considered as unsatisfactory if θ exceeds the limit $90° \pm 60°$. Lo and Lee[10] found that the first condition appeared to be too strict, so a more flexible range of $90° \pm 52.5°$ was used for quadrilateral interior angles. In the present study Lo and Lee's range is chosen for acceptable quality of a quadrilateral element. The optimum shape for a quadrilateral is a square with interior

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 6, December 2013

angles 90º. The distortion factor for quadrilateral element, F_q is defined as,

$$F_q = \sqrt{\sum_{i=1}^{4}(\delta\theta_i)^2} \quad where\ \delta\theta_i = \left|\frac{\pi}{2}-\theta_i\right|, i=1,2,3,4 \qquad (3)$$

It can be seen that F_q would attain a minimum value of zero for a perfect square and the acceptable range of 90º ± 52.5º defined by Lo and Lee[10] would correspond to $F_q \leq 105^\circ$. In this study any element exceeding this range is considered unacceptable. In contrast, Aspect ratio of a quadrilateral element is defined as the ratio of the largest to the smallest length of the sides of the element. The best possible value of the aspect ratio is 1. In meshing the computational domain our aim is to obtain the value of the aspect as much as close to 1. Based on these two factors an object oriented program in C++ has been developed. It calculates each element internal angles and computes the distortion factor. Finding out the maximum and minimum edge length of each element it also calculates the aspect ratio. Finally the average distortion factor of the mesh and element of worst distortion factor is found out. The output of the program for different models in our analysis for determining element quality is summarized
in the Table 1.

V. FINITE ELEMENT ANALYSIS

To conduct the Finite element analysis different models have been used with variable number and quality of elements from model to model. For modeling purposes, one quarter of plate is being subjected to the analysis due to the symmetrical appearance of the structure about horizontal and vertical axis. Application of uniform stress at the edges was also being applied with the help of symmetrical boundary conditions.

VI. RESULTS AND DISCUSSION

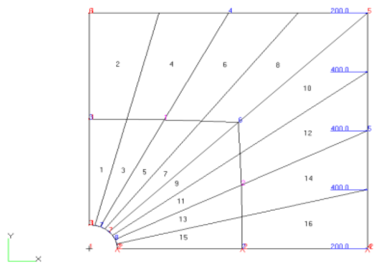

Fig. 3: Mesh of Model 1

Our first finite element model-1(Fig. 3) for the problem has 27 nodes and 16 quadrilateral elements. From the output of the program in c++, it is seen that the distortion factors of the elements are all within the acceptable range but the aspect ratio of all the elements are too high. Where a value of 5 may be too high, here only 8 elements have aspect ratio below 5.The stress distribution for the model 1 is shown in Fig. 9. From the figure it is seen that the maximum stress developed is 589 psi. So the stress concentration factor is 589/400 = 1.4725, where the value of theoretical stress concentration factor is 3.0354. From this result it is clear that quality of the mesh is not good enough and we have to refine our mesh. To improve our result the seeding (seeding simply means locating the coordinate of nodes) biased nodes criteria has been used. Biased nodes are placed along a line in increasing or decreasing space with respect to one another This method has been used to refine the mesh at a high stress concentration area of a model for better analysis. A model with biased nodes results in decreased computing time since fewer elements are used. One method to increase the accuracy is to make the elements as geometrically uniform as possible. This was done by seeding the right number of nodes at the right place.

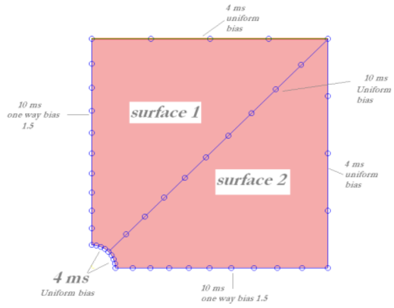

Fig. 4 Mesh seed for model 2

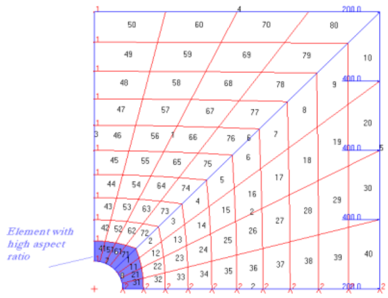

Fig. 5 : Mesh of Model 2

To create second finite element model for our problem two surfaces were created as in Fig. 4. Each edge is then divided as shown in Fig. 4. The word ms means mesh seed which represents number of subdivisions. One way bias 1.5 means largest subdivision is 1.5 times larger from the smallest one. Based on theses seeding the mesh of model 2 (Fig. 5)) is generated. The model has 99 nodes and 80 quadrilateral elements. From the output of the program in c++ for model 2 it is seen that all the elements have acceptable distortion factor but the elements surrounding the hole (the quad no.1, 11, 21, 31, 41, 51, 61, 71) have larger aspect ratio 3.8 to 5.4. The result file (distribution of stresses) is shown in Fig 10.
From the figure it is seen that the maximum stress developed is 1040 psi. So the stress concentration factor is 1040/400 = 2.6.

Since the mesh quality of the elements near the hole is not satisfactory in Model 2 and also we know that the stress concentration occurs at that region, we are intended to refine our mesh. For this reason, in our next model we rearrange mesh seeds though the number of nodes and elements are same as Model 2.

Table 1: Summary of the output of the program for determining aspect ratio and distortion factor

Model no.	Bad quad faces, distortion factor more than 105	Average distortion factor	Average Aspect ratio	Quad with worst distortion factor	Quad with worst aspect ratio
1	nil	46.93	17.21	10: 79.89	7: 33.53
2	nil	46.07	2.06	10: 82.17	71:5.43
3	nil	43.48	1.65	40: 87.39	1: 4.79
4	nill	40.68	1.49	35: 87.39	1: 3.45
5	368: 113.67 369: 23.09 373: 115.27 403: 112.89	31.84	1.55	369:123.1	107:3.92

Quad means 4-noded quadrilateral membrane element

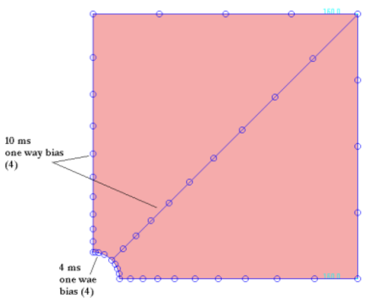

Fig. 6 Mesh seed for model 3

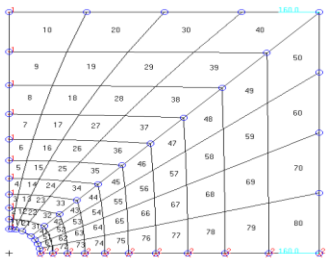

Fig. 7: Mesh of Model 3

The next model has mesh seed as Fig 6. The mesh of model 3 is shown in Fig. 7. The model has 99 nodes and 80 quadrilateral elements. From the output of the program in c++ , we see that only four members (fewer than Model 2) have aspect ratio larger than 3. The result file is shown in Fig. 11. From the figure it is seen that the maximum stress developed is 1180 psi. So the stress concentration factor is 1180/400 = 2.95.

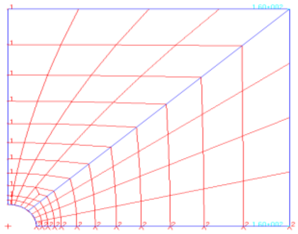

Fig. 8 : Mesh of Model 4

The mesh remaining same (same seeding as in Fig. 6) as model 3 but the elements near the hole are subdivided (along the larger length of the elements) to modify the aspect ratios. As a result, the number of nodes in the next model (Model 4) becomes 117 and the number of element is 96.The mesh of the Model 4 is shown in Fig. 8. From the output of the program, we see that only one element has aspect ratio larger than 3. The result is shown in Fig. 12. From the figure it is seen that the maximum stress developed is 1040 psi. So the stress concentration factor is 1230/400 = 3.075. The result our analysis for different model has been shown in the tabular form in Table 2.

Table 2: Summary of models and their out put

Model No.	Mesh generation & mesh quality						SCF calculation & comparison				
	No. of Nodes	No. of elements	Avg. DF	Avg. AR	Max. AR	No. of elements (AR more than 3)	σ (psi)	σ_{max} (psi)	Numerical K_{tg}	Theoretical K_{tg}	% difference
1	27	16	46.93	17.21	33.52	16	400	589	1.472	3.0354	51.49
2	99	80	46.07	2.06	5.43	9	400	1040	2.6	3.0354	14.34
3	99	80	43.48	1.65	4.80	4	400	1180	2.95	3.0354	2.8
4	117	96	40.68	1.50	3.44	1	400	1230	3.075	3.0354	1.3
5	492	416	31.84	1.55	3.92	8	400	1250	3.125	unknown	

DF = Distortion factor; AR = Aspect ratio

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 6, December 2013

The graphical representations of the result files (stresses in X-component) for different models for single hole are shown in the following figures:

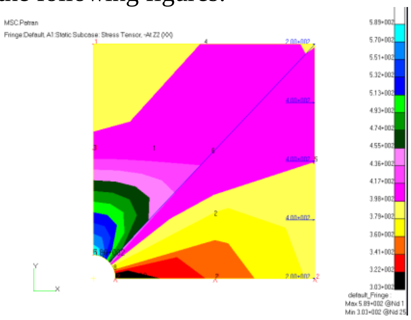

Fig. 9 : Result of Model 1

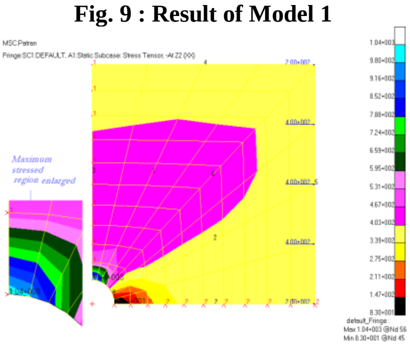

Fig. 10 : Result of Model 2

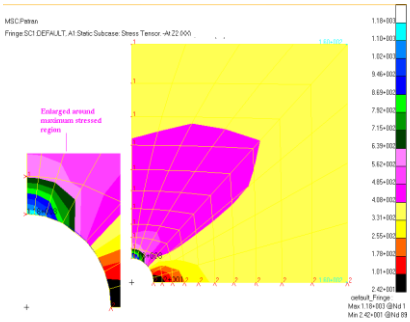

Fig. 11 : Result of Model 3

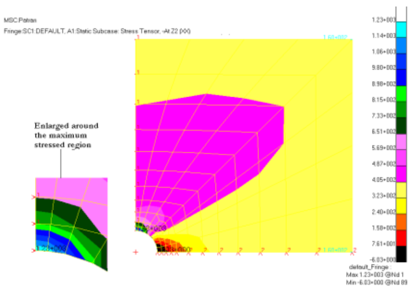

Fig. 12 : Result of Model 4

After finding out the correct approach to mesh a model, our next problem which is not listed in Pilkey[3] is a plate with a center hole and two additional smaller holes in a plane perpendicular to the loading direction. We placed the additional holes to see the effect of stress concentration in the region where originally the maximum stress developed. The mesh seed and mesh of the model with multiple holes are shown in the Fig. 13 and Fig. 14 respectively.

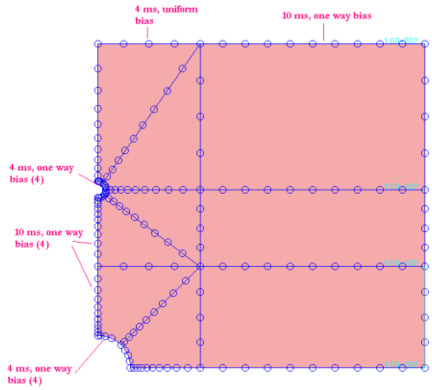

Fig.13 Mesh seed for model 5(multiple holes)

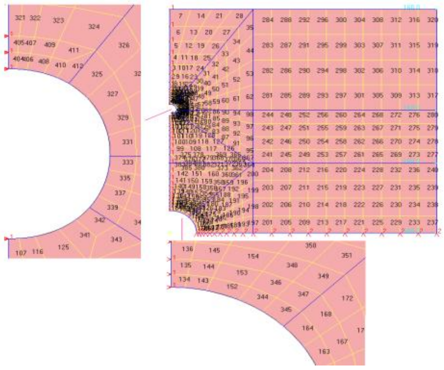

Fig. 14. Mesh of model 5(multiple holes)

The result is shown in Fig. 15. From the figure it is seen that the maximum stress developed is 1250 psi. So the stress concentration factor is 1250/400 = 3.125 (more than 3.075, the stress concentration found from previous problem). So we see that the addition of holes will not make the situation better but will worsen the case. Now we get two additional regions with stress concentration (extreme edges of the smaller hole).

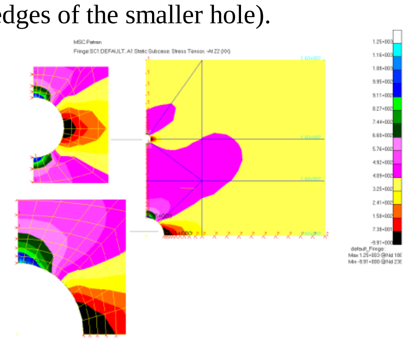

Fig. 15: Result of the model 5(multiple holes)

VII. VON MISES STRESS

In this section the results are shown (for all models) in terms of Von Mises stress. Von Mises stress is a geometrical combination of all the stresses (normal stress in the three directions, and all three shear stresses) acting at a particular location. If the Von Mises stress exceeds the ultimate strength, the material ruptures at that location. The graphical representations of the result files in case of von mises stress for different models for single hole and multiple holes are shown in the following figures:

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 6, December 2013

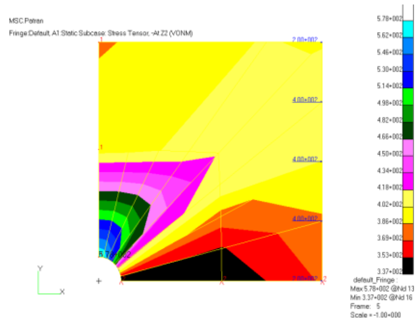

Fig. 16 Von mises stress for model 1

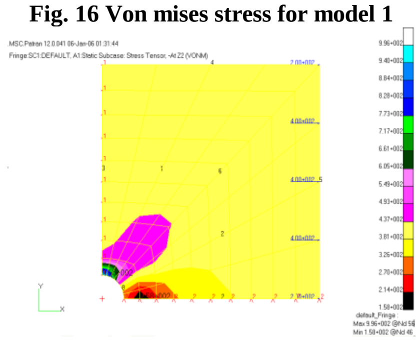

Fig. 17 Von mises stress for model 2

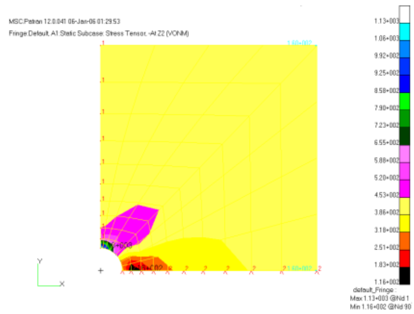

Fig. 18 Von mises stress for model 3

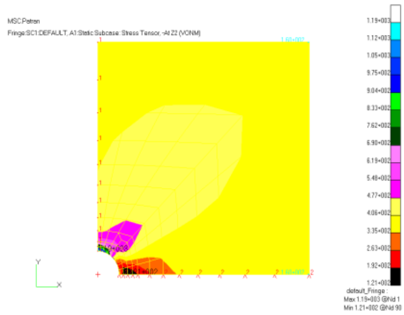

Fig. 19 Von mises stress for model 4

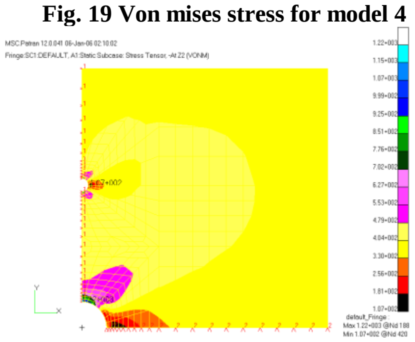

Fig. 20 Von mises stress for model 5(multiple holes)

So we now summarize the results in the following table.

Table 3: Stress concentration factor in terms of Von Mises stress

Model No.	σ_{max} (Maximum Von Mises stress)	σ (Von Mises stress at the edge nodes where pressure applied)	K_{tg} (Numerical)	K_{tg} (theoretical)	% difference
1	578.00	400	1.445	3.0354	52.40
2	996.00	400	2.490	3.0354	17.97
3	1130.0	400	2.825	3.0354	6.900
4	1190.0	400	2.975	3.0354	1.990
5	1220.0	400	3.050	unknown	0.480*

* With respect to the K_{tg} of column 5

VIII. CONCLUSION

This work clearly demonstrates the robustness of the finite element method in handling real life problems. Finite element analyses of holes are imperative because holes are used in engineering components and structures for bolts, rivets etc., and we need to know the stresses and deformation which occur near them. When generating a finite element mesh, one can expect more accurate results with more refined mesh (smaller size but larger number of elements within a confined area). However as the model gets larger (In FEA, larger model does not mean larger geometry, but rather the complexity due to the number of elements used), the computer will spend more time to generate the results of the analysis. It is often very important to minimize the computing time without a significant loss in the accuracy of the solution. In order to achieve satisfying results with minimum computing time, there are several techniques. One method that we have used in our analysis seeding biased nodes. In this study the main target was to perform a FEM analysis on a structural problem concerning stress concentration around a hole in the middle of an infinitely long plate to validate the formula for the problem and by this to find out correct meshing of the domain. The following conclusions can be drawn from the present study:

1. A reliable program in c++ is developed which can effectively check the quality of 2-dimensional quadrilateral finite element mesh. To achieve correct result a good quality mesh is necessary and for evaluating if the mesh is good enough, this program is very much essential.
2. After performing a number of investigations (only four are given) a reliable meshing technique is proposed, using which a mesh consisting of a minimum number of elements can be generated for achieving accurate results.
3. It is found from the analyses that element aspect ratio has a great impact on the finite element results. Our aim is to achieve the best value of the aspect ratio which is 1 but in our analysis the maximum value of the aspect ratio is 3.44 in high stress region. Also care should be given to keep elements having higher aspect ratio away from region of high stress concentration and numbers of these elements should be diminutive.
4. Though the distortion factor of value zero means perfect square (the best possible), but for geometric constraints achieving this value is not always possible. From the analyses we made, it is found that the average distortion factor of the mesh should not be more than 40.
5. After investigating the results, the formula (equation 1) given by Pilkey[3] is found reasonably accurate.

REFERENCES

[1] R. B. Heywood, "Designing by photo elasticity". Chapman and Hall, 1952.

[2] R. E. Peterson, "Stress concentration factors". John Wiley & Sons, 1974.

[3] W.D. Pilkey, "Peterson's stress concentration factors". New York: John Wiley & Sons Inc., 3rd edition, January 14, 2008.

[4] R.C.J. Howland, "On the stresses in the neighborhood of circular hole in a strip under tension". Phil. Trans. Roj. Soc., vol. 229, pp. 49-86, 1929.

[5] T.L. Anderson, "Fracture Mechanics Fundamentals and Application", 3rd Edition: CRC press, 2005.

[6] W. L. Roland, N. Perumal, and N.S. Kankanhalli, "Fundamentals of the Finite Element Method for Heat and Fluid Flow": John Wiley & Sons Ltd, 2004.

[7] Q. Z. Wang, "Simple formulae for the stress concentration factor for two – and three-Dimensional Holes in Finite Domain". J. of strain analysis, vol. 37, pp. 259-264, 2002.

[8] J.R., ed. Brauer, "What Every Engineer Should Know About Finite Element Analysis", Marcel Decker Inc., 1993.

[9] J. Z. Zhu, O.C. Zienkiewicz, E. Hinton, and J. A. Wu, "A new approach to the development of automatic quadrilateral mesh generation", International Journal for Numerical Methods in Engineering, vol. 32, pp.849-66, 1991.

[10] Lo, S. H. and Lee, C. K., "On using meshes of mixed element types in adaptive finite element analysis", Finite Elements in Analysis and Design, vol. 11, pp.307-36, 1992