ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 3 – No.2, April 2016

Methodology for Mesh Type and Mesh Density Selection

Rishi Saxena
University of Petroleum and Energy Studies, Dehradun
saxenarishi740@gmail.com

Akshansh Seth
University of Petroleum and Energy Studies, Dehradun
akshansh.seth@stu.upes.ac.in

Kanishka Mathur
University of Petroleum and Energy Studies, Dehradun
kanishka.mathur@stu.upes.ac.in

ABSTRACT
The basic idea of FEA is to make calculations at only limited number of points and then interpolate the results for the entire domain. Any continuous object has infinite degrees of freedom and it's not just possible to solve the problem in this format Finite Element Method reduces the degree of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).

We present in this paper a study of flat plate with a hole at its center using 2 dimensional meshing with different element types (quad and tria). The analytical solution was computed first and then compared with the results of the ANSYS FEA. This paper studies the comparison between quad and tria type of meshing with varying number of elements. Conclusion made was that quad elements are better than triangular elements. The greater the number of elements in the critical region the better the accuracy.

Keywords
Meshing Elements, Plate with a hole, Modeling in CATIA V5, FEA Analysis in Ansys.

1. INTRODUCTION
Meshing can be done in 1D , 2D or 3D so there are several types of meshing elements to be used for analysis. Following tables gives an idea about the different meshing elements.

Fig1

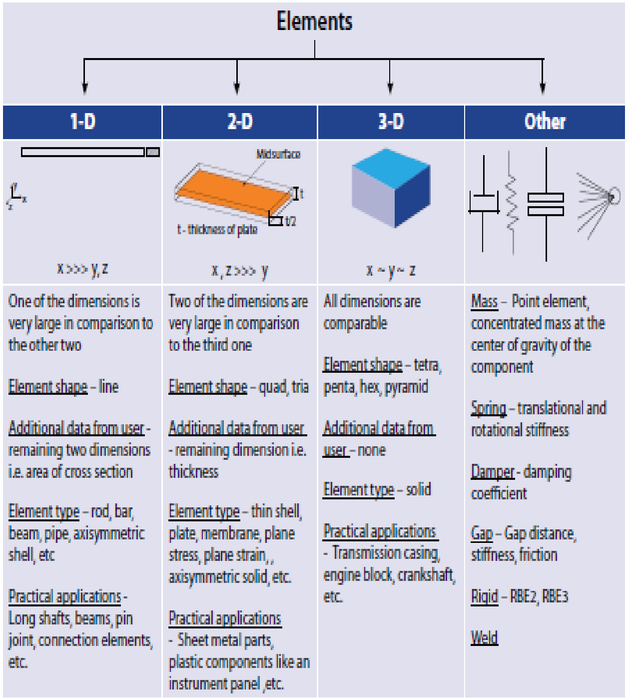

Fig2: Comparison of different types of meshing elements

The objective of present work is to identify the effect of mesh density on the solution for a particular analysis. More the mesh density more would be the accuracy but there are different types of meshing elements so how to decide which one to use or what would be the thumb rule to decide this. So this research is done in order to identify the thumb rule for selection of meshing element.Here we consider a square plate 10mm thick and dimension 1000x1000mm with a hole at center on which a force of 10000N is acting , we are going to calculate the maximum principal stress both analytically and with the help of FEA and then will compare the results.

The force applied on the plate can be calculated analytically as follows-

The Stress Concentration Factor (SCF) is defined as=

Max. Stress/Nominal Stress

Nominal Stress= F/A =10000N/(1000mm*10mm)=1N/mmsq

For an infinite plate SCF=3

Hence, Max stress=
Stress Concentration Factor(SCF)*Nominal Stress= 3N/mmsq

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 3 – No.2, April 2016

2. MODELING AND MATERIAL PROPERTIES

2.1 Modeling

The modeling of plate was done in CATIA V5 software in part design module. CATIA is multi platform CAD/CAM/CAE software developed by Dassault Systems used to design, simulate, analyze and manufacture products in different industries specially aerospace and automotive.

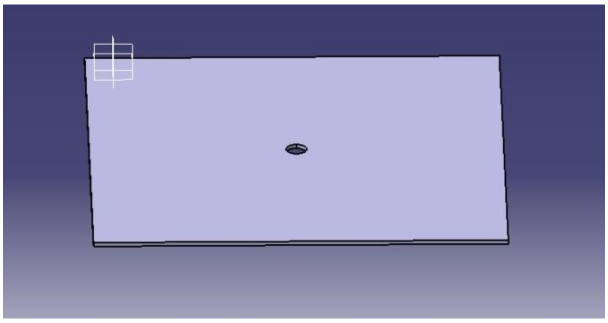

Fig3: Plate model in CATIA V5

2.2 Material Data

Properties of Structural Steel

Density- 7850 kg/m3

Ultimate Tensile Strength- 480 MPa

Yield Strength- 250 MPa

3. FINITE ELEMENT ANALYSIS

The Finite element analysis was done in Static Structure Analysis system of ANSYS Workbench 14.0 software.

ANSYS Workbench combines the strength of our core product solvers with the project management tools necessary to manage the project workflow. In ANSYS Workbench, analyses are built as systems, which can be combined into a project. The project is driven by a schematic workflow that manages the connections between the systems. From the schematic, you can interact with applications that are native to ANSYS Workbench (called workspaces) and that display within the ANSYS Workbench interface, and you can launch applications that are data-integrated with ANSYS Workbench, meaning the interface remains separate, but the data from the application communicates with the native ANSYS Workbench data.

4. MESHING AND ELEMENT QUALITY CHECK

Themeshing of the model was done in Static Structural Analysis system of ANSYS Workbench 14.0

MODEL-1

Mesh type = Tria

No. of elements at hole = 16

Stress = 2.29 N/mm2

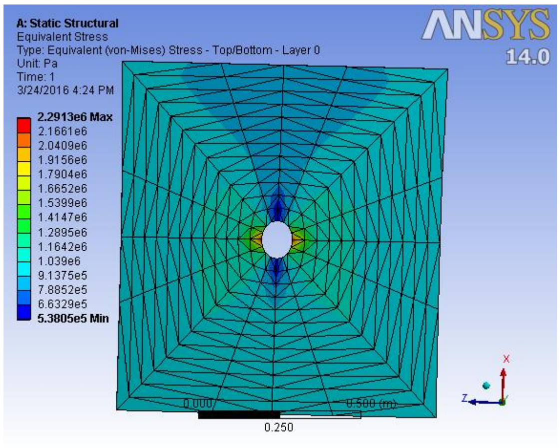

Fig 4: Equivalent Stress Calculation with Tria Element

MODEL-2

Mesh type = Quad

No. of elements at hole = 4

Stress = 1.69 N/mm2

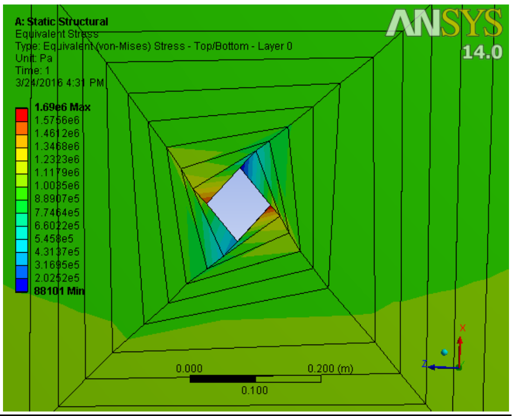

Fig 5: Equivalent Stress Calculation with Quad Element

MODEL-3

Mesh Type = Quad

No. of elements at hole = 8

Stress = 1.99 N/mm2

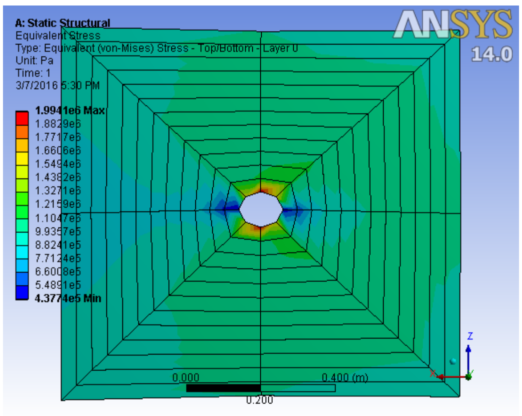

Fig 6: Equivalent Stress Calculation with Quad Element

MODEL-4

Mesh type =Quad

No. of elements at hole = 16

Stress = 2.37 N/mm2

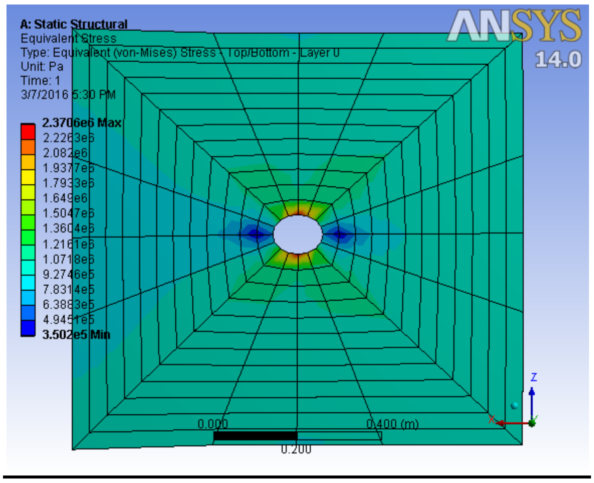

Fig 8: Equivalent Stress Calculation with Quad Element

MODEL-5

Mesh type = Quad

No. of elements at hole = 64

Stress = 2.53 N/mm2

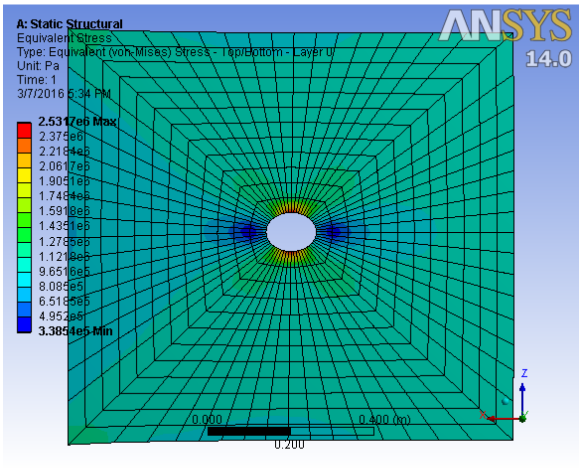

Fig 7: Equivalent Stress Calculation with Quad Element

MODEL-6

Mesh type = Quad

No. of elements = 128

Stress = 2.533 N/mm2

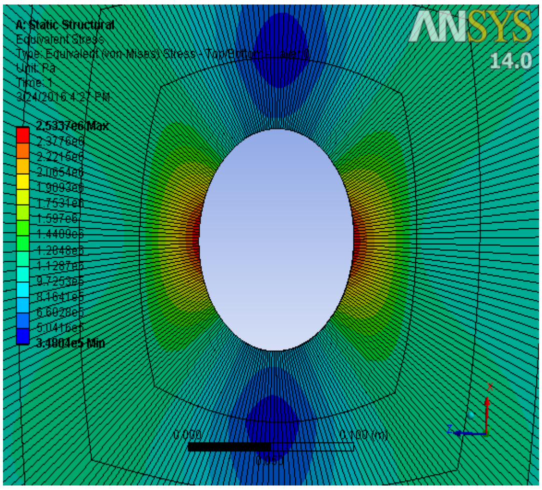

Fig 9: Equivalent Stress Calculation with Quad Element

5. RESULT AND CONCLUSION

Table 1. Comparison of tri and quad elements

	Element Type	No. of Elements	Stress (N/mm2)
MODEL-1	**Tria**	16	2.29
MODEL-4	**Quad**	16	2.37

Table2.Table of stress values for different no. of quad elements

	Element Type	No. of Elements	Stress (N/mm2)
MODEL-2	**Quad**	4	1.69
MODEL-3	**Quad**	8	1.99
MODEL-4	**Quad**	16	2.37
MODEL-5	**Quad**	64	2.53
MODEL-6	**Quad**	128	2.533

Through this research paper we aimed to identify the effect of mesh density and also to derive a thumb rule for the meshing and it is quite clear that,

- Quad Elements will give better results than Tria elements.
- Greater the number of elements in the critical region (i.e. hole) , the better would be its accuracy.

6. ACKNOWLEDGEMENT

We would like to expresses our gratitude to Aerosphere for providing academic inputs, guidance & encouragement throughout this period. Special thanks to our family and friends for the support they had given us throughout our work.

7. REFERENCES

[1] Practical Finite Element Analysis by Nitin S. Gokhale

[2] Metallic Materials Properties Development and Standardization (MMPDS -01)

[3] The Finite Element Method in Engineering (Fifth Edition)

[4] Fig 1 courtesy Tata Johnson Control Automotive India

[5] Elements of Strength of Materials by Stephen P. Timoshenko (Author), Donovan H. Young (Author)