Testing For Subtlety in Analysis Software

Sai Praneeth Jasti[1] And Avinash Kumar Ray[2]

[1]*Final Year mechanical student, Amrita School of Engineering, Coimbatore, India.*

[2]*Final Year mechanical student, Amrita School of Engineering, Coimbatore, India.*

ABSTRACT:

This paper studies and reviews on the nuances of the Analysis software like ANSYS in performing Finite Element Analysis on different CAD model formats like IGES, PARASOLID.

KEYWORDS: Analysis, ANSYS, Computer Aided Engineering, PARASOLID, IGES

1. INTRODUCTION:

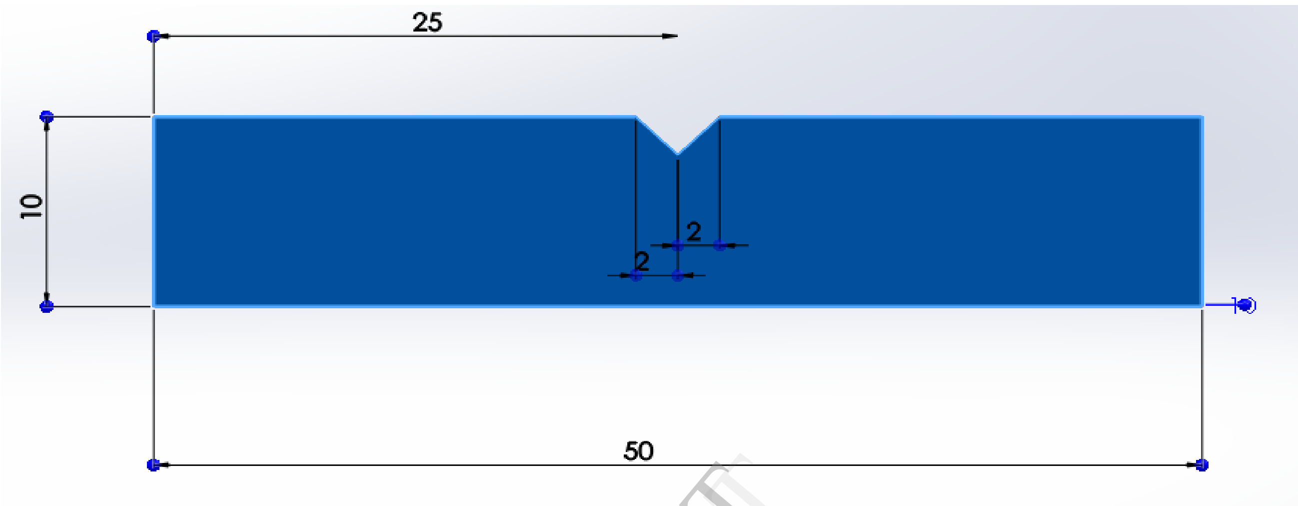
Solid modelling is widely used by every designer. There are many Solid Modelling software packages available that are facilitating aspiring designers. Every solid modelling package has its own format for storing the file. Now-a-days softwares are contemplating the users to view and edit other format files. The next stage of computer modelling is computer based analysis. Various analysis conditions are verified like structural, thermal, fluid etc. In most of the analysis process CAD model generated in any solid modelling packages are imported.

It is evident that when a CAD model built in particular software and in particular format opened in another different software, even it is compatible with that format, and we find that the model has lost some feature distorted in dimensions and render quality. Analysis software is designed in scope of being compatible to various such kinds of solid modelling packages. But it is not evident that whether there are any errors produced if same analysis is applied on same CAD model which was built in different leading modelling software in their respective formats.

Our paper tries to investigate this scenario using leading analysis software ANSYS by analysing the maximum deflection of notched specimen CAD model generated in popular Solid Modelling Softwares like SOLID WORKS, CATIA and PRO/E in various formats like IGES, PARASOLID, CATIA, PRO/E etc. Sampling the results obtained and comparing of the results is done to effectively determine the variation attained in different scenarios and carefully examine the reasons behind it.

2. SPECIMEN:

The specimen used in this analysis is a standard specimen used in Izod and Charpy test. It is a 50 mm length specimen with 10*10 mm^2 cross section. There is triangular notch exactly in the middle of the specimen that act as a stress inducer. The below figure is the specimen modelled. It is modelled in solid works, Catia, and Ansys softwares in different formats.



3. ANALYSIS PROCESS:

ANSYS is a commercial finite-element analysis software with the capability to analyse a wide range of different problems.Like any finite-element software, ANSYS solves governing differential equations by breaking the problem into small elements. The governing equations of elasticity, fluid flow, heat transfer, and electro-magnetism can all be solved by the finite-element method in ANSYS.

This paper considers all the different formats that can be imported into ANSYS mainly IGES, PARASOLID, SAT, CATIA V5 and also create the specimen in software itself. The steps involved in simulation are given below.

1. Open ANSYS Multiphysics
2. Click import option fromFile
3. Select the format type in imports and select the file
4. Once the geometry is imported select structural from Preferences in ANSYS main menu
5. In pre-processor
    a. Element type: The specimen used in the analysis is a homogenous structural solid. Hence Solid 20 node 186 element type is most suitable for simulating deformations.

ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013

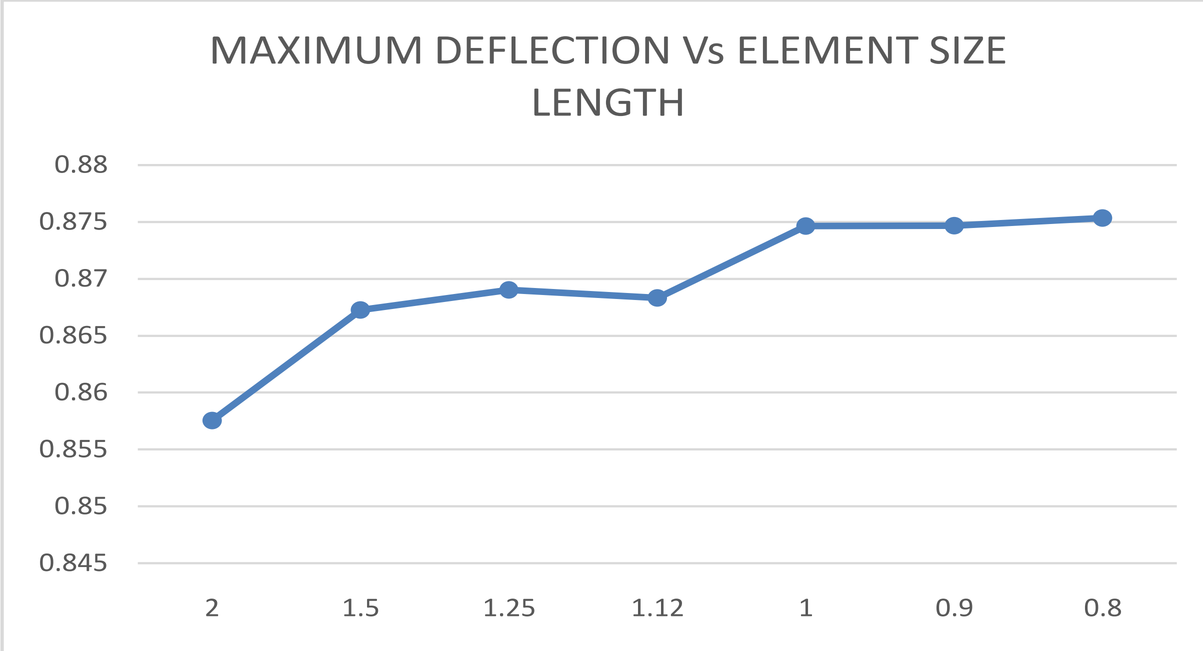
b. Material Props: Standard Aluminium is selected as the material of the specimen with properties as
    i. Young's Modulus :70GPa
    ii. Poison ratio :0.35
    iii. Density :2.17×10^{-6}Kg/mm^3
c. Meshing: Element edge length in size controls is derived based on mesh convergence and mesh was performed on entire volume
d. Loads: The specimen is subjected to cantilever beam conditions with one side fixed and a uniformly distributed tensile pressure of 1000N/mm^2 on other side of the specimen.

6. The solution of the analysis is performed and reviewed in post processing
7. Maximum deflection observed is noted and results were tabulated
8. Record these values for different edge element size, check for the convergence and note down for that particular edge element different results.
9. We repeat these steps for types of file formats such as:
    a. CATIA:
        1. IGES
    b. SOLID WORKS:
        1. IGES
        2. PARASOLID.

4. MESH CONVERGENCE:

Mesh convergence is used for generalising the size of element to be used in meshing. It involves the study of standardising the element size by the gradual decrease of element size with respect to maximum deviation, minimum deviation, maximum stress and minimum stress.

In our analysis the results were compared against the solution attained for model created in ANSYS. Hence mesh convergence was performed by starting element size length as 2.



After performing the mesh convergence studies, we found the element size of .8 to be most appropriate.

5.INFERENCE:

The results were initially compared between specimen modelled in Ansys and that of other specimen subjected to meshing size of 0.8 and subjected to same boundary conditions.

MODEL FORMAT	MAXIMUM DEFLECTION (mm)	PERCENTAGE DEVIATION (%)
ANSYS	0.87534	0
Solid Works IGES	0.877001	0.1897
Catia IGES	0.865994	-1.067
Solid Works PARASOLID	0.809065	-7.5713

Of all the formats IGES is comparatively more reliable than PARASOLID in terms of its consistence to give precise solution even it is modelled in two different softwares.

This deviation in maximum deflection is evident in concluding the fact that the solution of finite element analysis varies in Ansys software with format used. But before we conclude, the possibility of convergence has to be checked and hence the convergence of each solution isdetermined by taking different mesh size lengths in order to achieve more accurate solution.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013

5.1 Convergence Verification of Catia IGES Format

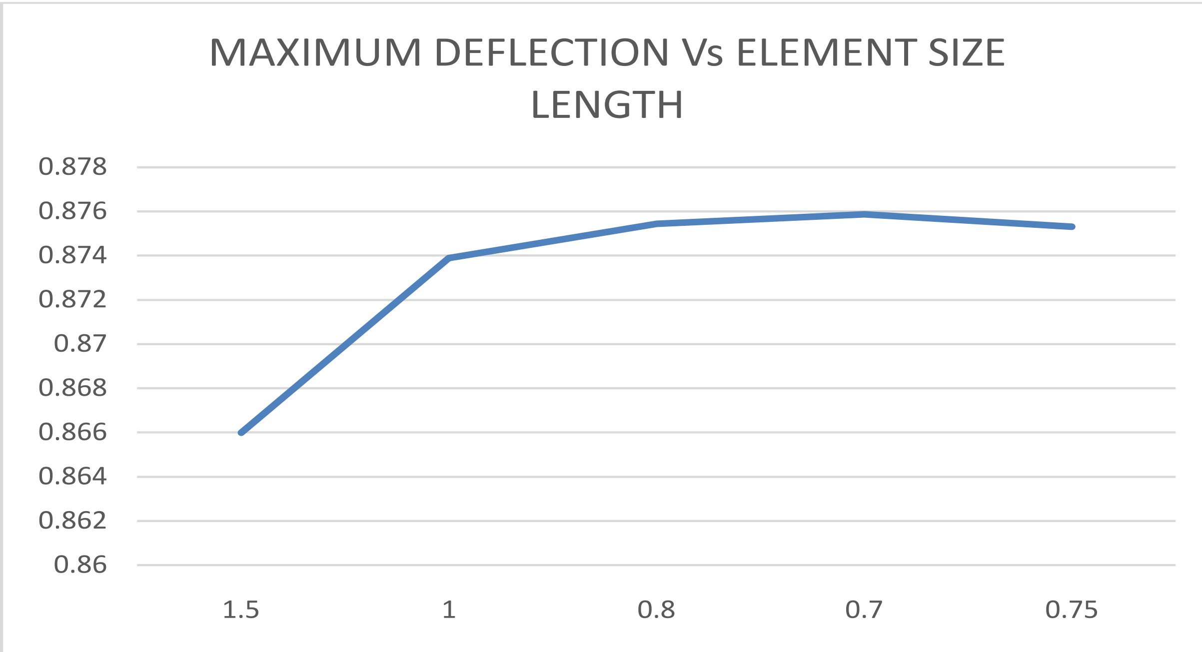


Fig 2: Maximum Deflection of specimen modelled in Catia IGES format

Convergence was achieved at mesh size of 0.7 with a maximum deflection of 0.8758 mm which gives a precise solution with an accuracy of 0.5255%.

5.2 Convergence Verification of Solidworks PARASOLID Format:

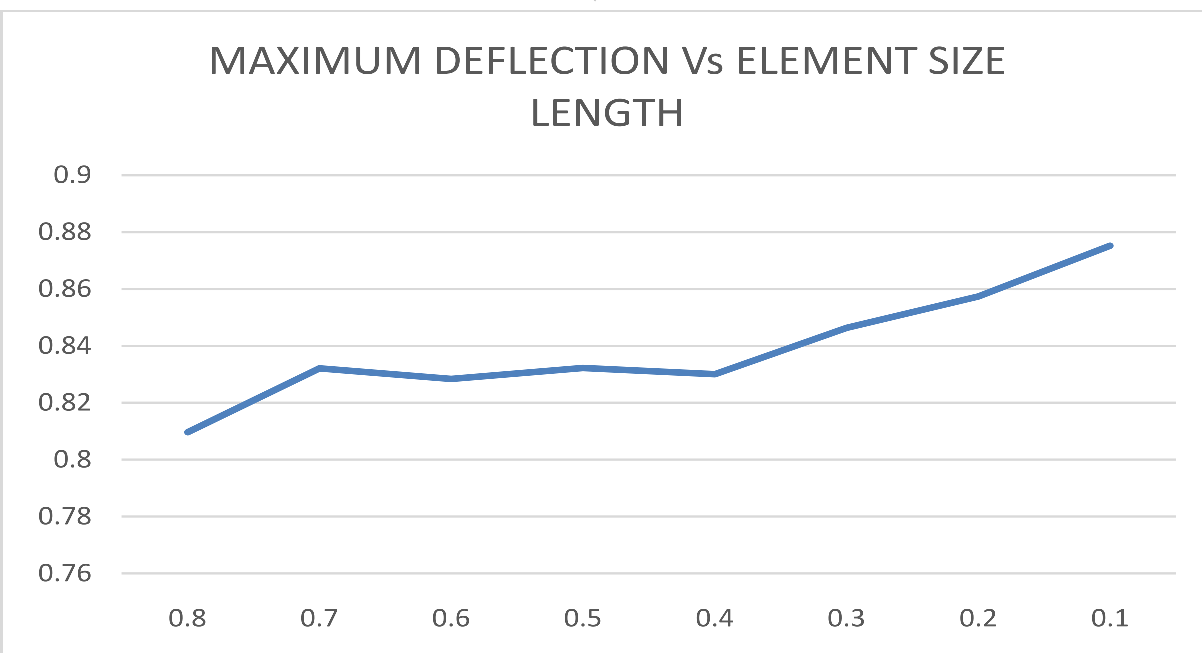


Fig 3: Maximum Deflection of Specimen Modelled in Solid Works PARASOLID Format

Convergence was achieved at mesh size of 0.6 with a maximum deflection of 0.8283 mm which gives a precise solution with an accuracy of -2.94%.

6. CONCLUSION:

In this paper the analysis was performed on a simple specimen in different CAD formats and the results obtained were having significant deviations. When complicated analyses are to be performed the designer should be aware of in which format he will get best desirable and accurate results possible.

7. REFERENCES:

- PaletiSrinivasan,KrishnaChitanyaSambana, Rajesh kumar Datti, *Finite element analysis using ansys11.0*, PHI Learning Private Limited, New Delhi, 2010.
- Ramamurty, G., *Applied Finite Element Analysis*, 2nd ed., I.K. International Publishing House Pvt. Ltd., New Delhi, 2010.