ISSN: 2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 2, Issue 12, December 2013

ELEMENTAL BEHAVIOUR IN STATICS ANALYSIS FOR A SIMPLY SUPPORTED BEAM BY USING FEM SOFTWARE

Uttam Y. Siddha[1], Nawaj A. Mulani[1], Rakesh Magmare and Girish C. Mekalke

[1] Asst.Professor in Mechanical Engineering Department at Manajiraje Bhosale Technical Campus, Urun-Isampur415409,(MS) India.

***Abstract*— The simply supported beams are involved in many engineering application. These beam systems suffer from the occurrence of deflection due to initial loading. These deflections have been investigated to avoid possible resulting fatigue. This paper discusses the application of finite element methodology for the analysis of a simply supported beam subjected to a mid length load. The element configurations that are studied range from one dimensional to three dimensional type and various mesh configurations. The Finite Element results are compared with exact analytical solution of the simply supported beam and this shows the elemental behavior of the beam. The report discusses the comparison of analytical exact solution and FEA result. The HYPERMESH pre-processor and post processor are used for finite element and the solver is used as MSC NASTRAN.**

***Index Terms*—simply supported beam (SSB), finite element analysis (FEA).**

I. Introduction

When loads are applied on a beam then immediately axes become bent [1].Displacements from the initial axes are called bending deflections

- To observe, evaluate and report on the load deflection relationship of a simply supported beam.
- To determine the mesh parameter as close as analytical solution of the beam and what the parameter qualities is made for simply supported beam.
- To verify both analytical and MSC NASTRAN FEA solution.

Numerical examples are given in order to determine the effects of various mesh parameters on the response of the simply supported beam.

II. Procedure for ANALYTICAL SOLUTION for a Simply Supported Beam

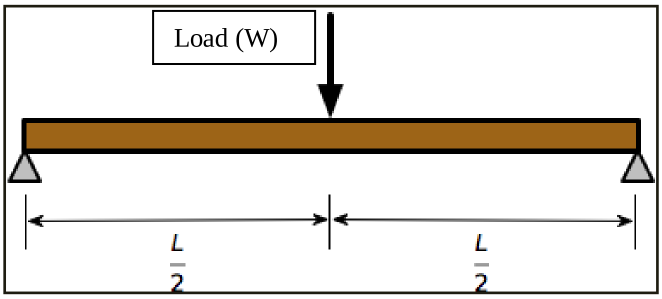
A. *Define Problem*

A cast iron beam 40mm wide and 80 mm deep is simply supported on a span 1.2m .the beam carried a point load of 15KN at the centre find maximum deflection and stress (E=1080000 N/mm2).

Suppose a concentrated load, W is applied to the centre of the simply supported beam [2].

1) The beam will bend or deflect downward as a result of the load W

2) Deflection is a measure of the vertical displacement of the beam as a result of the load W



B. *Analytical solution*

The beam will bend and deflect downward as a result of the load W and Δ is a measure of the vertical displacement of the beam as a result of the load W. The amount of flexural deflection in a beam is related to the beams area moment of inertia (I), the single applied concentrated load (W), length of the beam (L), the modulus of elasticity (E), and the position of the applied load on the beam. The amount of deflection due to a single concentrated load W .The deflection of a simply supported, centre loaded beam can be calculated from the following formula:

$$\text{Deflection } (\Delta) = \frac{WL^3}{48EI}$$

w = concentrated load (N)

L = span length of beam (mm)

E = modulus of elasticity (N /mm 2)

I = moment of inertia of axis perpendicular to load W

C. *Solution*

A. Displacement:

$$\text{Deflection } (\Delta) = \frac{WL^3}{48EI}$$

$$EI = 108000 \times \frac{40 \times 80^3}{12}$$

$$= 1.84 \times 10^{11} \text{ N/mm}^2$$

$$\text{Deflection } (\Delta) = \frac{15000 \times 1200^3}{48 \times 1.843 \times 10^{11}}$$

$$\text{Deflection } (\Delta) = 2.93\text{mm}$$

B. STRESS:

$$\text{Stress} = \frac{F}{A}$$

$$= \frac{15000}{40 \times 80}$$

$$=4.68 N/mm^2$$

III. SSB MODEL IN HYPERMESH

Suppose a concentrated load, W is applied to the center of the simply supported beam in a CAD model in hyper mesh is shown in figure.

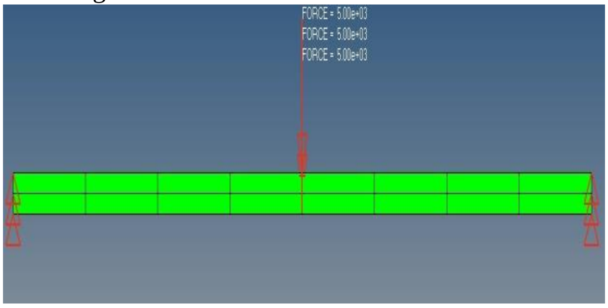


Fig1-simply supported beam in HYPERMESH

IV. INPUT DATA FOR ANALYSIS BY FEM

A. Geometry Requirement

In FEA analysis we require the CAD model of simply supported beam. The FE analysis process starts with CAD geometry of SSB into pre-processor i.e. Hypermesh. The model of SSB is crated in Hypermesh v11.0, which is required for the purpose of further analysis.

B. Material properties all SSB

A materials property is an thorough, often quantitative, property of a material. The ability of a material to stand up to forces being applied without it bending, breaking, shattering or deforming in any mode.

Material	Young's modulus (N/mm^2)	Possions ratio	Density $(tone/mm^3)$
Steel	E= 2.1e5	N=0.3	ρ =7.8 e –09

Table2-Material properties

The mechanical properties of a material are those properties that involve a response to an applied load. The mechanical properties of metals determine the range of value of a material and establish the service life that can be expected. Mechanical properties are also used to help classify and recognize material.

C. Pre-Processing

Meshing is the center step in the finite element analysis as the quality of the mesh directly reflects on the quality of the obtained results[6]. Deciding the element type and generating the mesh using manual or special meshing operations .

Shell element is used for thin wall part having small thickness. It is the most commonly used plate element. It is a 4-noded flat plate element It is capable of resisting both, in-plane and out-of-plane loads.

D. Necessity for Meshing

Acceptance criteria of seat model quality are considered acceptable when meets the body mesh model Quality check list about the various mesh quality parameters such as length , aspect ratio, min and max angles of tria and quad along with percentage of trias in the mesh and the mainly essential parameter, the Jacobian or distortion of the mesh from an ideal shape.

E. Meshing the Components of SSB

Analysis of SSB for the deflection it is required to be meshed with elements to get the component mesh.When the entire element in the beam is meshed, they are all different type meshes are taken for same loading condition on same beam. Show the various types of mesh[4]. They may be wire frame and should show exactly the model.

Types of mesh –

1.]SSB_Split_All_Sides

2.]SSB_Mid_Point_to_Quads

3.]SSB_Divide_Quad_to_Aligned_Diagonal

4.]SSB_Mid_Point_to_Trias

5.]SSB_Divide_Quad_Divide_Shortest_Diagonal

6.]SSB_Divide_Quads_Unionjack_Diagonals

7.]SSB_Mid Pt to Trias

The elements that were used for analysis were CTRIA3, CTIA 6, CQUAD4 and CQUAD8 for 2 dimensions.

F. Element Quality Checks

Maximize element quality (throughout the mesh): Individual element quality directly determines the overall mesh quality. Element quality across a given mesh is evaluated by extracting results from shape.

G. Boundary Conditions

Various loads and constraints are added to the model to represent the loading conditions. The Different load cases can be defined to represent different loading conditions on the same model.

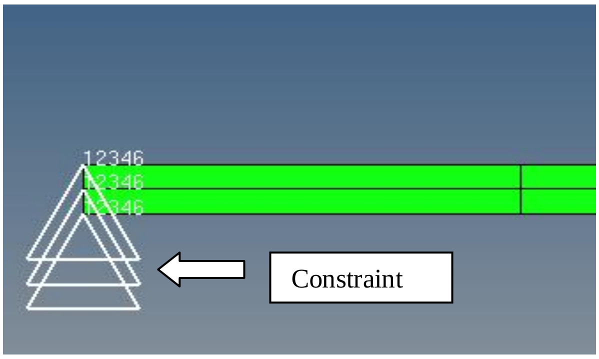


Fig2-constraint in HYPERMESH on SSB

ISSN: 2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 2, Issue 12, December 2013

H. Solution with NASTRAN

The meshing and quality checks are complete then export the mesh file in NASTRAN format for post processing. The FEM model is consisting of nodes, elements, material properties and constraints.

I. Post Processing

Deflection and stress are plotted and examined to see how the part responded to the various loading conditions. Based on the results, modifications may be made to the part and a new analysis may be run to view how the modifications affected the part.

V. FEA ANALYSIS OF SEAT FOR NATURAL

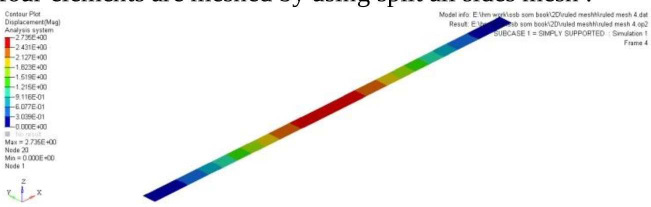
The meshing process included the development of a finite element model using Hypermesh that was appropriate for computing statics analysis individual members of simply supported beam under static loading condition .

The finite element analysis provided a measure of the sensitivity of the simply supported beam in support conditions, as well as a basis for selecting dissimilar material for loading.

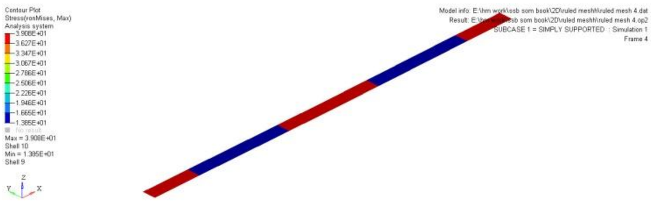
A. Plots of displacement and stress

VI. SSB_SPLIT_ALL_SIDES

The below results were obtained for 4 elements SSB. The four elements are meshed by using split all sides mesh .

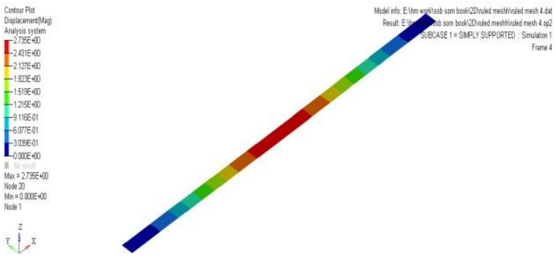


Plot1-4 element displacement

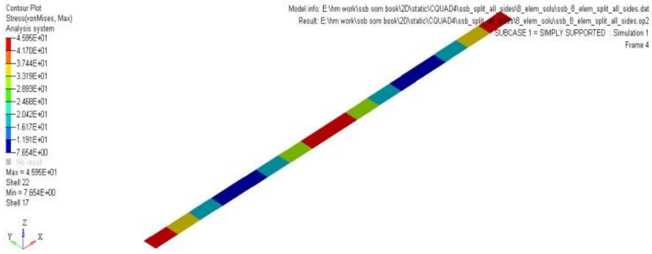


Plot2-4 element stress

The below results were obtained for 8 elements SSB. The 8 elements are meshed by using same method i.e split all sides mesh .



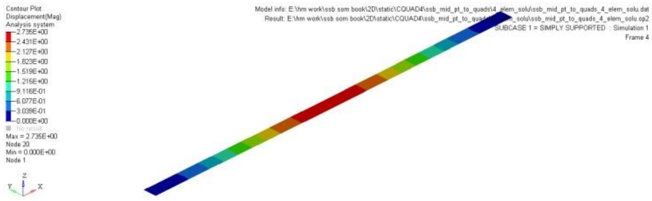
Plot3-8 element displacement



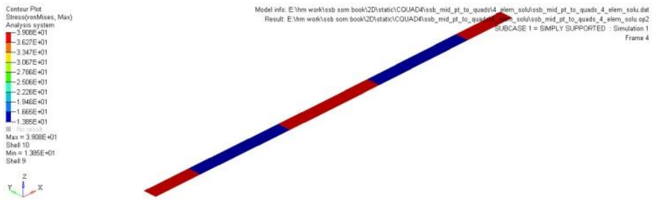
Plot4-8element stress

B. SSB_MID_POINT_TO_QUADS

The below results were obtained for 4 elements SSB. The four elements are meshed by using midpoint to quad mesh.

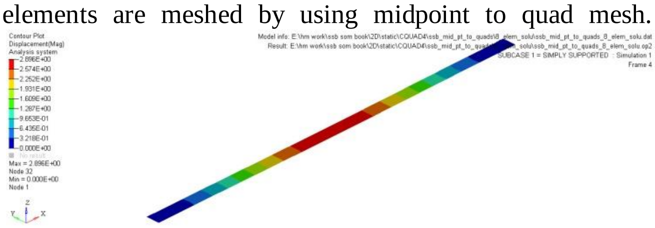


Plot5-4 element displacement

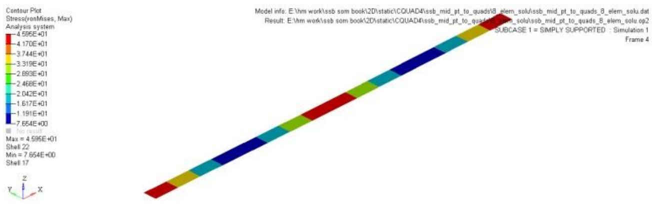


Plot6-4 element stress

The below results were obtained for 8 elements SSB. The 8 elements are meshed by using midpoint to quad mesh.



Plot7-8 element displacement



Plot8-8element stress

In this way, we use the all meshes type for simply support beam for further analysis and obtained the results as shown in bellow comparison table2.

VI. COMPARISON TABLE

Type of mesh	No Ele.	FEA Disp. (mm)	Analytical Disp.	FEA Stress (N/mm^2)	Analytical Stress (N/mm^2)
SSB_split_all_sides	4	2.73	2.93	3.90	4.68
	8	2.89	2.93	4.59	4.68
SSB_mid_point_to_quads	4	2.73	2.93	3.90	4.68
	8	2.89	2.93	4.59	4.68
SSB_divide_quad_to_aligned_diagonal	4	0.066	2.93	7.96	4.68
	8	0.176	2.93	14.75	4.68
SSB_mid_point_to_trias	4	0.095	2.93	11.85	4.68
	8	0.336	2.93	24.83	4.68
SSB_divide_quad_divide_shortest_ diagonal	4	0.066	2.93	7.96	4.68
	8	0.176	2.93	14.74	4.68
SSB_divide_quads_unionjack_diagonals	4	0.066	2.93	7.98	4.68
	8	0.181	2.93	15.10	4.68

Table2- COMPARISON TABLE

The finite element calculations were done by using SOL 103 of MSC NASTRAN and results for various mesh configurations were obtained.

In this simply supported beam we use 2D element usually called as shell used for thin walled part where length and width of the part is at least 10 times greater than thickness. i.e Quads and trias.

The pre-processing complete then export the mesh file in NASTRAN format for post processing. The FEM model (consisting of nodes, elements, material properties, loads and constraints) is then exported from within the pre-processor HYPERMESH. The exported FEM model, normally called solver input deck, is an ASCII file based on the specific syntax of the NASTRAN solver. Solution phase solve the deck prepared in preprocessing using NASTRAN solver.

After the run is complete, the easiest way to access the results is by using the HYPERVIEW. The open style of HYPERVIEW allows for loading and viewing result files obtained from several sources. Based on the solver type of the files and the results you would like to visualize and analyze, there are different ways to load the input deck and their corresponding results into HYPERVIEW.

Once the solution has ended successfully, post-processing (in HYPERVIEW for contour plots) of the simulation results is next. Stresses, strains, and displacements are plotted and examined to see how the part responded to the various loading conditions. Based on the results, modifications may be made to the part and a new analysis may be run to examine how the modifications affected the part.

The effects of various regular type meshes on different elements on 2-d element , such as the CQUAD4, CTRIAS was solved with respect to static solution . Numerical examples are given in order to determine the effects of various mesh parameters on the response of the beam and the major results have been taken in this study.

1. Deflections

Find out whether the deflections predicted are very small comparative to the size of the structure. For small structures, a deflection that is less than the thickness would be considered a small deflection. The deflection involving two supports should be only a small percent of the distance among supports. This is especially true if the deflection causes a differential stiffness effect such as mid-plane simply supported beam.

2. Stress

In the majority cases, extreme fewer elements are necessary if only deflection or stiffness in sequence is required. Even fewer elements can be used if only the deflection under the load is to be considered. For example, one beam or one shell can approximate the deflection of a cantilever beam of rectangular section. It may, however, take large of shell or solid element to confine the stress at the boundary in a simple simply supported beam here is a high gradient of stress over a large area of SSB.

The new mesh model is generated by using both CQUAD and CTRIAS elements because from previous analysis we can find that different mesh type has given different value so modification of SSB can be done with both types of mesh are used for simply supported beam for the accurate result

The finite element model of SSB was used to explore the effectiveness of modifications to different types of meshes on the behavior of simply supported beam. The different types that were explored relied upon the input of the SSB manufacturer with regard to manufacturability and serviceability, as well as retrofit compatibility with existing simply supported beam installations. The modifications, in particular, proved feasible and favorably altered the simply supported beam response characteristics with regard to the subject failures.

VII. CONCLUSIONS

On the basis of FE analysis result, we can conclude that different types of meshes are showing different results in the simply supported beam so as to avoid the variation in the result then the CQUAD element is be a better choice than CTRIAS elements of meshing it gives accurate results . The way of achieving this goal by introducing different mesh types combination for the simply supported beam, then analytical results of the simply supported beam is near at its FEA results and the resulting analysis do not contribute to the variation problem. It must be close from analytical results and FEA analysis result of simply supported beam otherwise these will lead to very embracing situation to a designer.

Based on the above study we feel that the mesh quality parameters impact on the results can be one of the most challenging and demanding scope of further studies. These parameters can be jacobian, aspect ratio, max and min angles of triangle reflecting the shape of the elements. A simply supported beam is considered that way. The analytically found results correspond closely to FEA data. A beam with various number of meshes types arrangements is similarly analysed. According to those findings, spilt all side mesh type should be given a same result as analytical.

VIII. REFERENCE

[1]S. P. TIMOSHENKO "History of Strength of Materials" New York: Dover Publications,Inc.

[2] S. Ramamrutham "Strength of Materials" Dhanpat Rai Publishing Company (P) Limited, page384, 2007.

[3]ISMAIL Y.UKSEK "Natural frequencies of a flat viaduct road part simply supported on two ends" Sadhan a Vol. 31, Part 1, February 2006, pp. 9–20.

[4] Gokhale, Nitin S, and Sanjay S Deshpande." Practical Finite Element Analysis." FiniteTo Infinite, pune, 2008.

[5]You-Qi Tang Li-Qun Chen "Natural frequencies, modes and critical speeds of axially moving Timoshenko beams with different boundary conditions" Shanghai Institute of Applied Mathematics and Mechanics, Shanghai .

[6]Bert C W, Newberry A L "Improved finite element analysis of beam vibration".J. Sound Vib.105:179–183
Bhat R B 19.

[7]J.A. Loya "Natural frequencies for bending vibrations of Timoshenko cracked beams" Carlos III University of Madrid, spain.