Design and Analysis of Nose Landing Gear Supportive Bracket by Varying Stress Concentration Area, Based on ANSYS

Raviteja B[1] Srinivas Prasad S[2] P.S Subrahmanyam[3] N.V.S.Anil Babu Javaji[4] Chethana.K.Y[5]
[1,2,3,4]Department Of Mechanical Engineering
[1,2,3]NOVA College of Engineering and Technology, Department of mechanical Engineering,Jupudi(V), Ibrahimpatnam(M), Krishna(DT), Andhra Pradesh – India
[4]AMC Engineering College, Department of mechanical Engineering, Bangalore -560072, India

Abstract— Nose landing gear support bracket is one of the main parts in the nose landing gear assembly, whose function is to maintain the stability of the landing gear during the movements. This work is focused on the FEA analysis of support bracket by varying the stress concentration area, The objective of this work is to determine the static and modal analysis of support bracket with different stress concentration areas, then analysis is done using ANSYS WORKBENCH, These results could provide some useful suggestions for design and improvement for the better component

.***Key words:*** Nose landing gear support bracket, Stress concentration area, FEA, Static analysis, Modal analysis, ANSYS WORKBENCH.

I. INTRODUCTION

Engineering machines and structure in service, experiences stress, strain vibration, and their design generally requires consideration of their dynamic behavior. The type of input forces experienced by different components of aerospace in service is dynamic. The progressive, localized and permanent structure change that occurs in a material subjected to stress concentration area and fluctuating strains at nominal stresses that have maximum values less than the tensile strength of the material which may culminates in cracks and causes fracture after a sufficient number of fluctuations.

Durability design of any component of an aerospace structure plays an important role to ensure reliability. The total damage and life of a structure is due to a combination of static and dynamic loads arising from engine vibrations, service load. The basic aim is to enable stress strain and deformation calculations to be done at the design stage of a development process. It is necessary to clarify the modal analysis to obtain the natural frequency of the component or structure.

Ravi Kumar [1] investigates the fatigue analysis of nose landing gear support bracket with 4mm rectangular slot by taking into the x direction loading and z direction loading to. In their work they attempt to establish a comprehensive mathematical model for obtaining the static analysis and fatigue results.

Prasad kabade and Ravi ligannanavar [2] investigates design and analysis of landing gear lug attachment in an airframe by considering the standard passenger planes of 150-200 seating capacity the approximate specification and obtained the failure of the lug,failure of the pin by considering the axial and Transeverse loads,FEA is used to carryout and compare the design calculations and determine the stress concentration factor in order to estimate the maximum local tensile stress.

Norman s.Currey[3] shows the aircraft landing gear design.

Anil kumar.Matta[4] investigates the design optimization of landing gear's leg for unmanned aerial vehicle he obtained the optimized leg's weight and thickness results by kinematic analysis

II. COMPONENTS USED FOR TEST

For the present study, Four models are used for analysis.SAE1045 Steel is the material, Model with rectangular 4mm slot as the present working component[1],by changing the slots like circular, single Double V slots another three models are designed.

III. FINITE ELEMENT ANALYSIS OF COMPONENT

The component is modeled using CATIA V5 software. Manufacture of the component and geometric dimensions there on are used to model the component. Using the cross sectional area, the part is extruded. Displacement boundary conditions on the model with all degree of freedom constrained on the surfaces.

Also the evaluated engineering properties of the component are given in Table 1

Property	Unit	
Density	Kg/m^3	7870
Modulus of Elasticity	GPa	202
Poisson's ratio		0.3
Yield strength	MPa	380
Ultimate strength	MPa	724

Table 1: Engineering Properties of SAE 1045 Steel component

IV. FINITE ELEMENT ANALYSIS OF COMPONENT

The component is modeled using solid edge software. Manufacture of the component and geometric dimensions there on are used to model the component. Using the cross sectional area, the part is extruded. Displacement boundary conditions on the model with all degree of freedom constrained on the surfaces

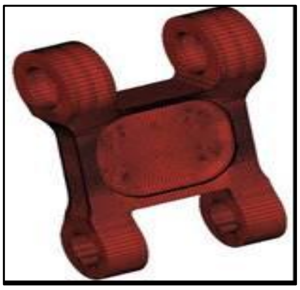

Fig. 1: Meshed model with rectangular slot component

Fig. 2: Model with rectangular slot component is subjected loading and boundary conditions

V. RESULTS AND DISCUSSIONS

The prepared CAD models are imported to ANSYS workbench and are meshed. Engineering properties of material like young's modulus, poisson's ratio and density are inputted as given in Table 2. Static analysis and Modal analysis is performed to determine the equivalent stress, strain and natural frequencies of the model. Natural frequencies were extracted for the first six modes. The deformations for each of the mode are recorded. The procedure is repeated for another three models. Fig. 1 shows the meshed model of the component, Fig. 2 shows the loading and boundary condition model

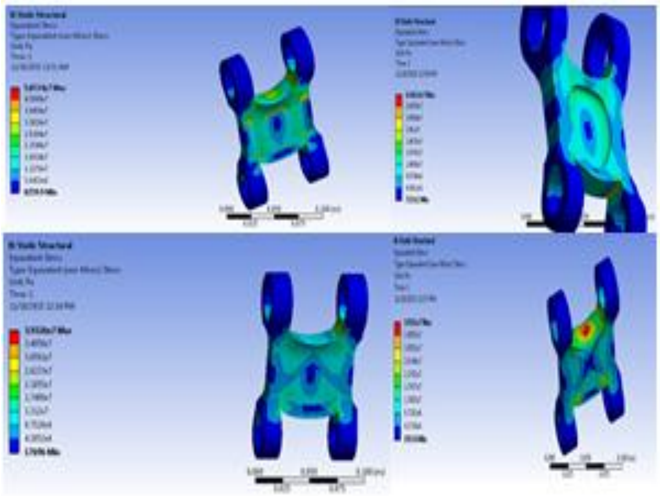

Fig. 3: Stress contour for load 12500N in X-direction on rectangular, circular, single V and Double V slot

Type of Stress Concentrations area	Deformation in m	Equivalent Elastic Strain in m/m	Normal Elastic Strain in m/m	Von mises Stress in Pa
Rectangular 4mm slot on either face of the bracket	8.5393e-6 Max	0.00025252 Max	0.00015184Max	5.0724e7 Max
	0 Min	1.2552e-7 Min	0.00011867 Min(-ve)	8259.9 Min
Circular 4mm slot on either face of the bracket	6.0063e-6 Max	0.0002205 Max	0.00013045 Max	4.4412e7 Max
	0 Min	1.2862e-7 Min	0.00016362 Min (-ve)	7224.2 Min
Sigle V 4mm slot on either face of the bracket	3.913e-9 Max	0.00019813 Max	0.00013276 Max	3.9326e7 Max
	0 Min	1.654e-7 Min	0.00014687 Min (-ve)	17696 Min
Double V 4mm slot on either face of the bracket	8.2791e-6 Max	0.00030497 Max	0.00012785 Max	3.921e7 Max
	0 Min	1.4428e-7 Min	0.00013658 Min (-ve)	19234 Min

Table 2: Shows static analysis results

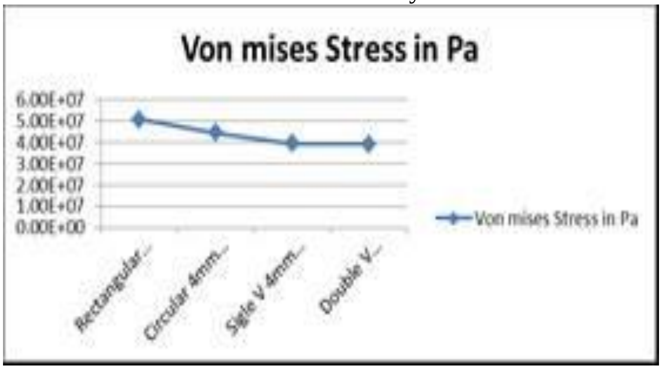

Fig. 4: Shows stress results

Above analysis observation shows that stress is goes on decreasing by changing the stress concentration area.

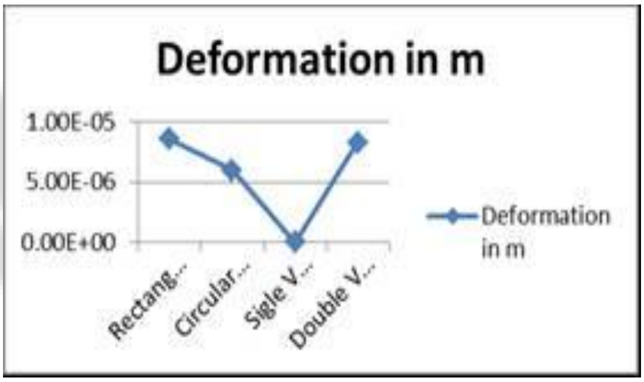

Fig. 5: Shows deformation results

Above analysis observation shows that deformation is goes on decreasing by changing the stress concentration area

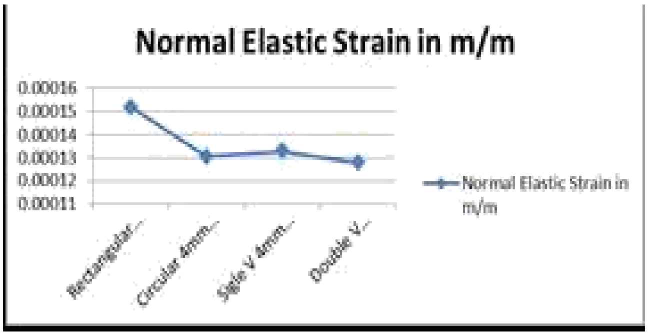

Fig. 6: Shows normal elastic strain results

Above analysis observation shows that normal elastic strain is goes on decreasing by changing the stress concentration area.

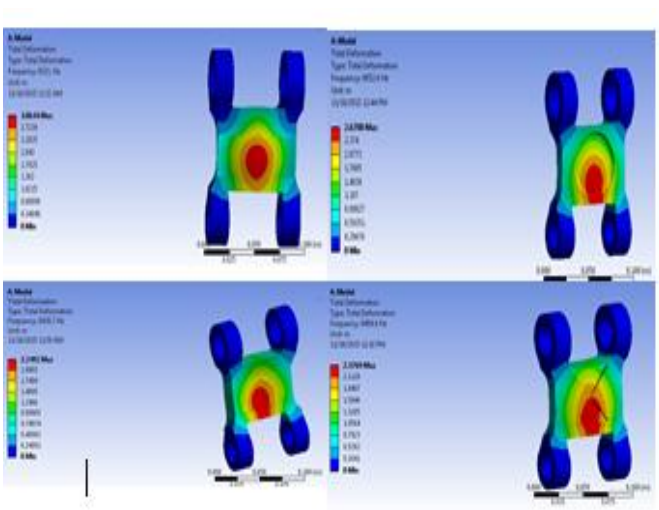

Fig. 7: Shows modal analysis results of rectangular, circular, single V, double V components

Type of Stress Concentration area	MODE 1	MODE 2	MODE 3	MODE 4	MODE 5	MODE 6
Rectangular 4mm slot on either face of the bracket	8321	13038	13728	14357	16591	18153
Circular 4mm slot on either face of the bracket	8652.6	13524	13581	13835	16626	19326
single V 4mm slot on either face of the bracket	8438.7	13207	13724	14049	17489	17989
Double V 4mm slot on either face of the bracket	8409.6	13072	13862	14791	17517	18772

Table 3: Shows Modal analysis results

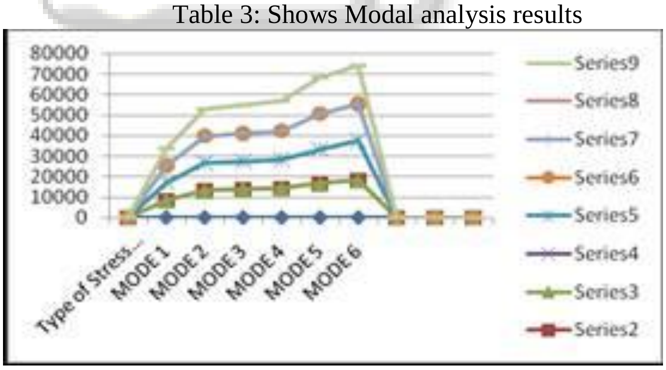

Fig. 8: Shows Modal analysis results

Above observation shows that by changing the stress concentration area natural frequency of the models will not much varies in these components

VI. Conclusions

Deformation goes on decreases by changing the stress concentration area in the order of circular, single v slot, double v slot. Stress and strains also decreases by changing the stress concentration area in the order of circular, single v slot, double v slot.

Changing of the stress concentration area is not much effect on the natural frequency of the component obtained frequencies are of same order

References

[1] Marcelino P, Nascimento, Herman J.C, Voorwald, Joao da C, Payao Filho, "Fatigue strength of tungsten inert gas repaired weld joints in airplane critical structures", Journal of Material processing Technology, Vol.211, 2011, pp 1126-1135.

[2] Ossa E.A, "Failure Analysis of civil aircraft landing gear", Engineering Failure Analysis, Vol.13, 2006, pp 1177-83.

[3] Joy Xiaoya Tao, Steve Smith, Andrew Duff "The effect of overloading sequences on landing gear fatigue damage", International Journal of Fatigue, Vol.31, 2009 pp 1837-1847.

[4] De Farias Azevedo CR, Hippert E, "Fracture of an aircrafts landing gear", Engineering Failure Analysis, Vol.9, 2002, pp 265-75.

