International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in; editorijoer@gmail.com

Vol.6., Issue.2, 2018
March-April

RESEARCH ARTICLE

ISSN: 2321-7758

DESIGN AND ANALYSISON BIONIC BRACKETOF AN AIRCRAFT STRUCTURE

K. SHIVA SHANKAR[1], SAI PRANEETH NARISETTY[2], SESHANK HARSHA CHAGANTI[3], V.S.K. MANIKANTA[4], NEELAKANTAM PRADEEP KUMAR[5], NAGAM MAHESH[6], MANNEPALLY SUDHEER[7], NADEEM AHMED MOHAMMED[8]

[1]Assistant Professor, Aeronautical Engineering Department, MLR Institute of Technology, Hyderabad, India.
[2]Master's Student, Mechanical Engineering Department, University of Bridgeport, USA.
[3,8]Student in Master of Science in Engineering Management, Kent State University, USA.
[4, 5, 7]Trainee, TATA Technologies Limited, Pune, India.
[6]B.Tech Student, Mechanical Engineering Department, MLR Institute of Technology, Hyderabad, India.

ABSTRACT

The paper deals with bionic aircraft structure of an aircraft to identify stress levels on entire bracket. Stress analysis is determined at variant pressure and by choosing aluminium alloy material to extrapolate stress levels on the joint structure. The bracket is miniatured using CATIA V5 and imported to ANSYS tool to carry stress analysis by utilizing FEM technique. Maximum stress& deformation is observed at variant pressure conditions.

Key words: Airbus A320, Bionic aircraft, aluminum alloy, FEM technique

Introduction

Bionic Airplane is a creative approach of assembling layer by layer. The main objective is to enhance the expedient productiveness of layer by layer craft. With the layer by layer craft which is a tool less procedure, remarkably finite configuration and interior edifice can be composed which can be achieved with classical craft method besides with enormously material, time and expenditure processing steps.

Airbus has team up with 3D design and engineering company Autodesk to configure printed airplane cabin elements. Airbus has almost minimized the weight of the constituents it uses to separate sections inside its A320 aircraft, by using a combination of "generative" design techniques and 3D printing. The industry is clearly at front for high quality metal 3D printing innovation where large number of pint size parts often with complex shapes, are already being 3D printed for use on Airbus aircrafts.

Some of the elements in the cabin are configured using additive layer manufacturing. The process repeatedly prints very thin layers of material on top of each other until the layers form a solid object ranging from high-grade titanium alloys to glass and concrete, thus making it simpler to produce very complex shapes.

The new Bionic separation uses Scalmalloy, a second-generation aluminum-magnesium-scandium alloy created by AP Works.

BASIS OF STRUCTURAL STRENGTH CALCULATIONS

Material Properties

Material used for bionic bracket aircraft design is aluminum alloy in all places while bearing majorly titanium alloys are using for high strength and to withstand heavy loads we used aluminum alloy those properties are described below

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in; editorijoer@gmail.com
Vol.6., Issue.2, 2018
March-April

Aluminum alloy 7075 are used for high strength those mechanical properties are

Table: 1 Physical Properties of Aluminum alloy

SI No	Physical Properties	Metric value
1.	Density	2.81 g/cc
2.	Ultimate Tensile Strength	572 MPa
3.	Tensile Yield Strength	503 Mpa
4.	Modulus of Elasticity	717 Gpa
5.	Poisson's Ratio	0.3
6.	Fatigue Strength	159 Mpa
7.	Shear Modulus	26.9 Gpa
8.	Shear Strength	331 Mpa

Finite Element modeling and stress analysis in bionic bracket

The stress values at different pressure values are 90 Pa and 120 Pa on the bionic bracket structure and the displacement contours are shown in the below figures. A maximum stress of 1.052e5 N/mm2 is observed at the middle of the bracket joint section. A maximum displacement of 1.56mm at the open end of the other end structure can be observed from the displacement contour in the below figure. The maximum stress value obtained from the analysis is used as the input for the fatigue calculations.

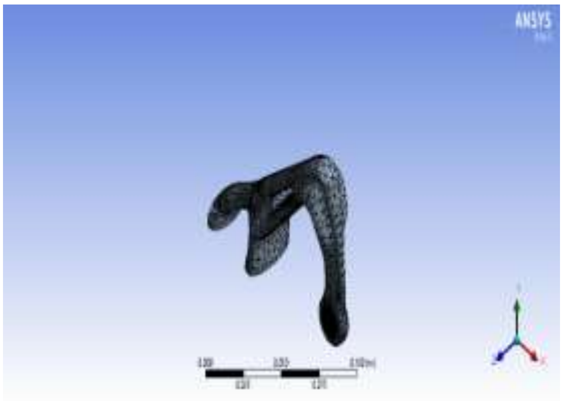

Fig:1FEA Modal

Modal Statistics

Nodes	Elements
72112	41273

The following plots obtained the pressure at 90 Pa

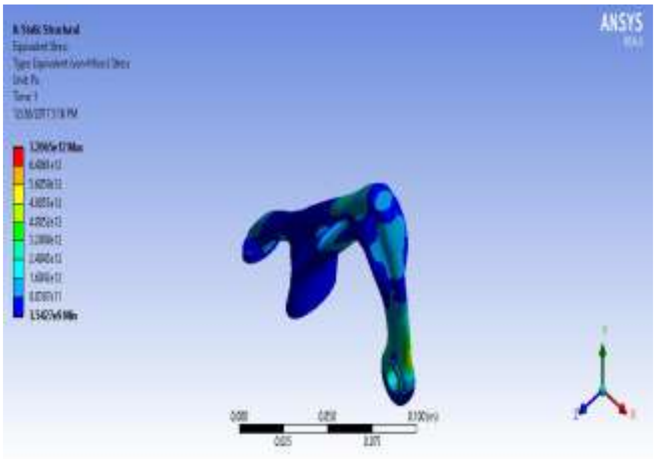

Fig: 2 Equivalent von mises stress at 90 Pa

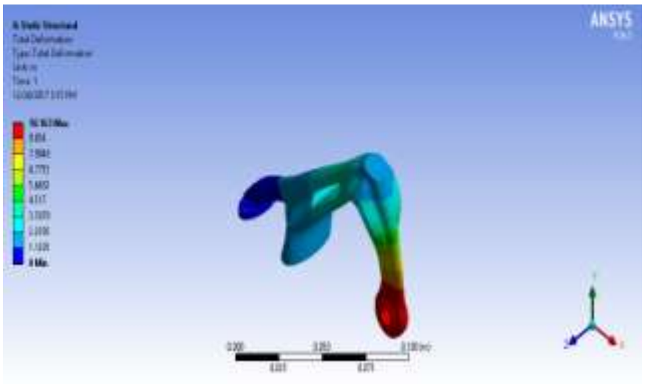

Fig: 3 Total Deformations at 90Pa

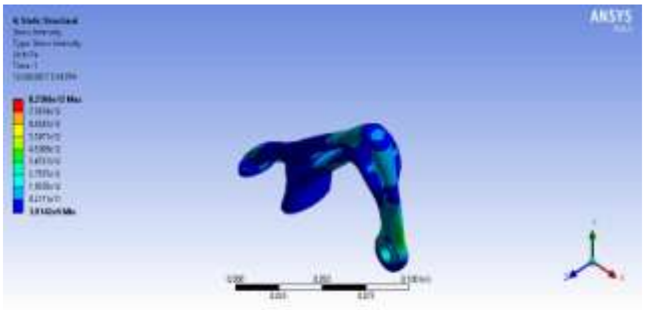

Fig: 4 Stress Intensity at 90Pa

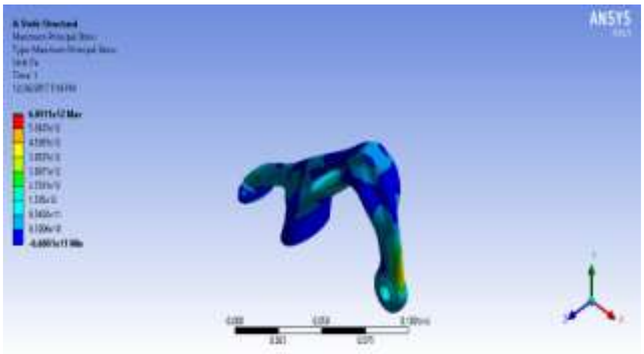

Fig:5 Maximum Principal stress at 90Pa

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in; editorijoer@gmail.com

Vol.6., Issue.2, 2018
March-April

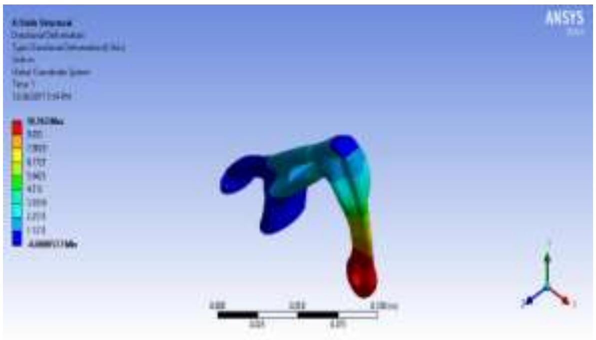

Fig: 6 Directional Deformations at 90Pa

The following contours obtained at pressure value at 120 Pa

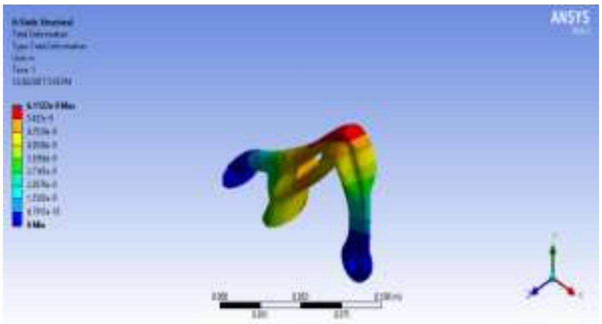

Fig: 7 Total deformations at 120 Pa

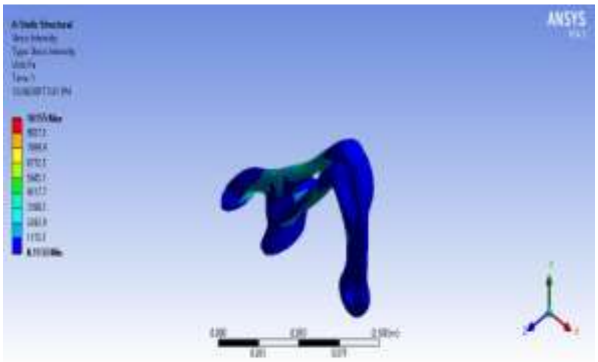

Fig: 8 Stress intensity at 120 Pa

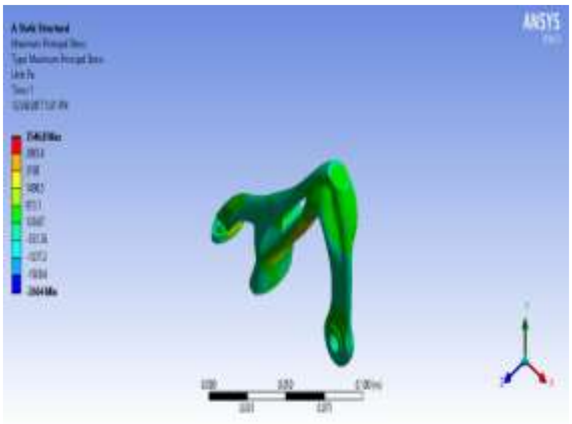

Fig: 9 Maximum principal stresses at 120 Pa

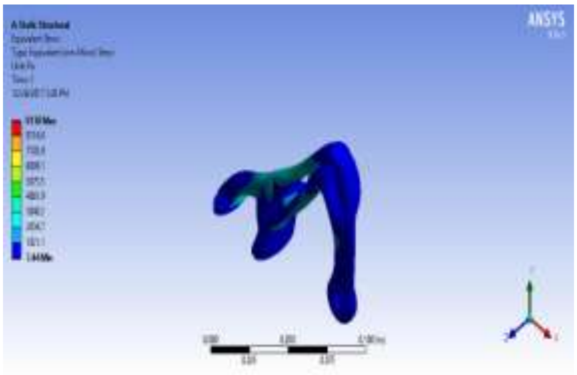

Fig: 10 Equivalent Von mises stress at 120 pa

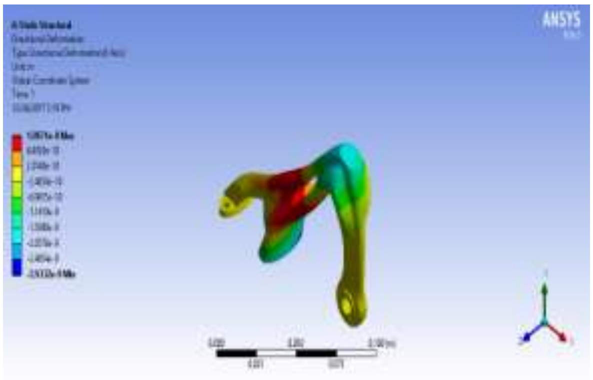

Fig: 11 Directional Deformations at 120 Pa

Results and Discussion

Table.2 Stress results on bracket structure at different pressure values 90Pa and 120Pa

SI No	Contour Plots	Pressure load at 90 Pa		Pressure load at 120 Pa	
		Maximum (N/m^2)		Maximum (N/m^2)	
		Maximum	Minimum	Maximum	Minimum
1.	Total Deformation	10.163	0	6.11E-09	
2.	Directional Deformation	10.163	-0.00896	1.10E-09	-2.93E-09
3.	Equivalent von mises stress	7.21E+12	3.54E+09	9130	7.44
4.	Maximum Principal stress	6.09E+12	-6.68E+11	3546.8	-2604
5.	Stress Intensity	8.26E+12	3.91E+09	10155	8.1133

Conclusion

The stress analysis of the bionic aircraft bracket is carried out at different pressure condition at 90 Pa and 120 Pa taking the aluminium alloy material and determine the maximum stress levels on joint structure. The FEM approach used for stress analysis of the bionic bracket. The overall maximum stress observed at 7.205e12 N/m2 on applied pressure at 90 Pa and the maximum stress value at 9130 N/m2 on applied pressure at 120 Pa. Several iterations done using Ansys to obtain the maximum stress values.

References

[1]. R. Rigby and M. H. Aliabadi,"Stress intensity factors for cracks at attachment lugs". British Aerospace, Filton,

[2]. Bristol BS99 7AR, U.K., Wessex Institute of Technology, Ashurst Lodge, Ashurst, Southampton S040 7AA,U.K

[3]. O. Gencoz, U.G. Goranson and R.R. Merrill," Application of finite element analysis techniques for predictingcrack propagation in lugs", Boeing Commercial Airplane Company, Seattle, Washington, 98124, USA.

[4]. Chandrapatla, "introduction to FEA, 3rd edition, 2008.