RESEARCH ARTICLE OPEN ACCESS

Modeling and Analysis of Bracket Assembly in Aerospace Industry

P. Thameem Banu[1], Dr. R. Satya Meher[2]
[1]M.Tech, Student, Department of Mechanical Engineering, QIS College of Engineering & Technology, Ongole
[2]Professor, Department of Mechanical Engineering, QIS College of Engineering & Technology, Ongole

ABSTRACT
This project deals with the Model and static Analysis of Bracket assembly used in Aerospace Industry. This bracket assembly is used for placing components for various purposes like carrying fuel and air. The mounting bracket assembly consists of a circular plate with nine lugs for which three different tanks are mounted.
The individual components i.e., circular plate, lugs etc are modeled and assembled through NX-CAD Software.The loads are transferred by the tanks to the bracket are considered as pressure loads. To reveal the stresses induced due to pressure loads, Finite Element (FE) Analysis is performed with the help of ANSYS. Then the occurrence of max stress is found and factor of safety is calculated. This project provides a methodology for analysis of an assembly consisting of components made of composite materials and metal components.
Keywords: NX-CAD, ANSYS, metal and composite materials.

I. INTRODUCTION:

In this current project the mounting bracket is a bracket assembly which consist of a circular plate with nine lugs for mounting three different sizes tanks. Tanks are filled with fuel and oxygen. These gases are actually used during propulsion time to mount these tanks. For that we use composite circular plate which is made up of carbon composite material.

This assembly usually consists of a circular plate, lugs and tanks. It is used to mount three tanks on a composite plate. The circular plate is considered to be thin (of thickness 11.7mm) and consists of ribs to increase the strength. On the circular plate, nine lugs are placed with varied dimensions. On every set of three lugs, one tank is being inserted. Tanks are used to carry air and fuel. This whole assembly is then fixed on a circular ring of the conical section, which is one of the peripherals of the aerospace vehicle. As the tanks are fixed on the lugs, the lugs have to bear the weight of tanks which further acts on the circular plate and finally on the circular ring, which is a part of the conical section, also known as bracket. The load transferred by the tanks to the conical section is considered as the pressure load. In order to reveal the stresses induced due to these pressure loads, static and modal analysis are carried out to a very high acceleration. Then, the occurrence of maximum stress is found and factor of safety is calculated to check whether it is safe or not.

The modeling of the individual components and assembly is done in the modeling software NX-CAD and analysis is carried out by using ANSYS 10.0 software.

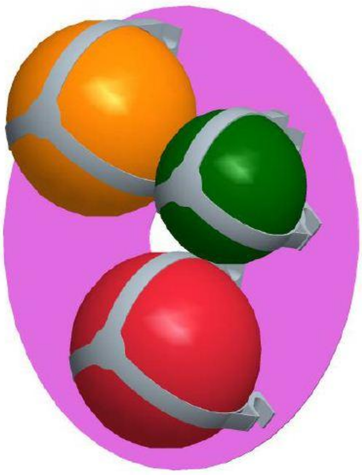

Fig. 1 Total Bracket Assembly with Tanks

II. PROBLEM DEFINITION

This bracket assembly is used for placing c omponents for various purposes like carrying fuel and air. The mounting bracket assembly consists of a circular plate with nine lugs for which three different tanks are mounted. The loads are transferred by the tanks to the bracket are considered as pressure loads. To reveal the stresses induced due to pressure loads, Finite Element (FE) Analysis is performed with the help of ANSYS.

III. METHODOLOGY

The methodology followed in my project is as follows:

- Perform the Design calculations of the mounting bracket assembly.

- Create a 3D model of the mounting bracket assembly using NX-CAD software.
- Perform steady state analysis using ANSYS software and obtain the deflections and von mises stresses for aluminum alloy.
- Perform steady state analysis using ANSYS software and obtain the deflections and stresses for composite materials like HM carbon /epoxy materials.
- Perform steady state analysis using ANSYS software and obtain the deflections and stresses for composite materials like HS carbon /epoxy materials.
- Perform steady state analysis using ANSYS software and obtain the deflections and stresses for composite materials like BMI carbon /epoxy materials.

IV. MODELLING OF BRACKET ASSEMBLY: CIRCULAR PLATE

Initially the circular plate was modeled by using commands revolve ,extrude, cut of dimensions 516mm as radius and thickness of 11.6 mm. This circular plate have equal spaced holes of pitch circle diameter as 1060mm and hole diameter as 8.5mm. The circular plate has ribs on both sides for strengthening of plate was extruded by 11.7mm. Now the total thickness of the plate becomes 35mm. The length of ribs on top same as plate diameter and length of ribs on bottom of plate is 512.5mm from center of plate.

V. SMALL LUG

Then we model the lugs, which are used to place the tanks. There are two different sizes of tanks. The smaller one has dimensions of 100*120*50mm.by using extrudes command. In this lug has a round corner by using fillet command as per the given dimension. The lug have holes of same dimensions as that of holes on the circular plate diameter is 8.6mm for fixing bolt and nut assembly.

BIG LUG:

The bigger one has dimensions of 198*175*70 mm by using extrudes command. In this lug has a round corner by using fillet command as per the given dimension. The lug has holes of same dimensions as that of holes on the circular plate diameter is 8.6mm for fixing bolt and nut assembly.

ASSEMBLY:

The components circular plate, big lug and small lug are assembled by the assembly options with fully constrains. The following assemble is obtained.

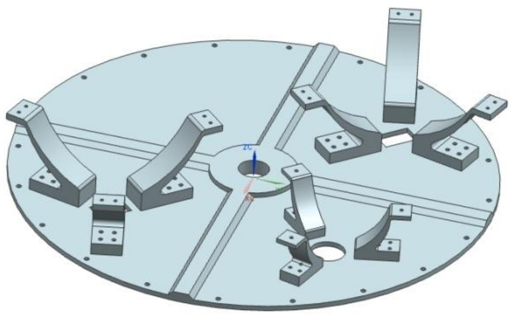

Fig. 2 A 3-D View Total assembly

VI. STRUCTURAL ANALYSIS OF BRACKET ASSEMBLY

Perform Finite element analysis of a bracket assembly under pressure loading conditions. Characteristics of stress distribution and high stress locations are determined from the analysis.

The 3d model of the bracket assembly is created in UNIGRAPHICS and converted into parasolid. The parasolid file is imported into ANSYS and finite element analysis is carried out using ANSYS software to perform the structural analysis of the bracket assembly.

Element Types used:
Name of the Element: SOLID 92
Number of Nodes: 10
DOF: UX, UY & U

Fig. 3 The meshed model of bracket assembly

Boundary Conditions:

- All Bolting locations are constrained in all Dof.
- Pressure is applied on outer area of lugs of bracket assembly.

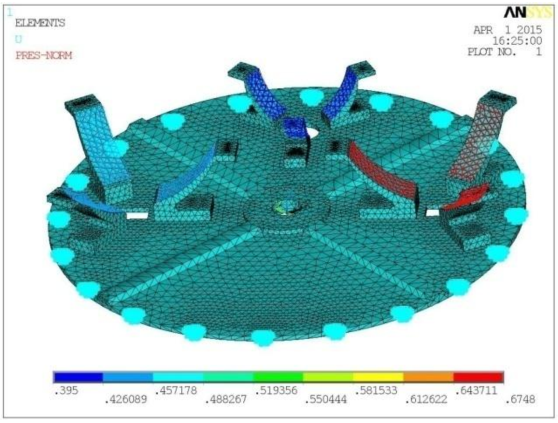

Fig. 4 Shows the Boundary conditions applied on the bracket assembly

CASE-1: Static Analysis of Bracket Assembly with aluminum alloy material:

Material used for bracket assembly is Aluminum Alloy 24345:
Young's Modulus: 72GPa
Poisson's Ratio: 0.3
Density: 2800kg/m3
Yield strength: 420Mpa

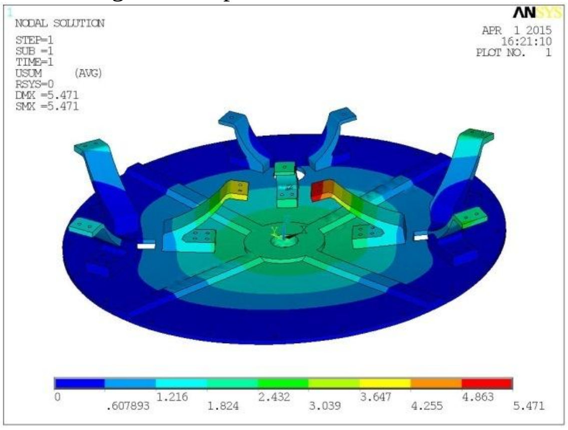

Fig. 5 Shows avg deflection of bracket assembly

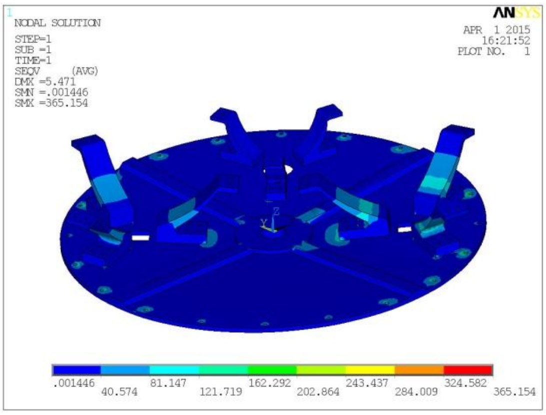

Fig. 6 Shows max Von Mises stress of bracket assembly

Table.1 Shows the Deflections and stresses for AL Alloy 24345

Sl.NO	DEFLECTIONS		STRESSES	
1	X-def	1.639	1st Principal stress	293
2	Y-def	2.14	2nd Principal stress	113
3	Z-def	0.142	3rd Principal stress	90
4	Avg. def	5.47	Von Mises Stress	365

From the above results it is observed that the Von Mises stress (365.15MPa) is less than the yield strength of the material (420MPa). Hence according to the Maximum VonMises Stress Theory, the Bracket Assembly model is safe for the above operating loads.

CASE-2: Static Analysis of Bracket Assembly with HM Carbon / Epoxy composite material.

Based on the advantages, the HM Carbon/Epoxy, materials is selected for composite bracket assembly.

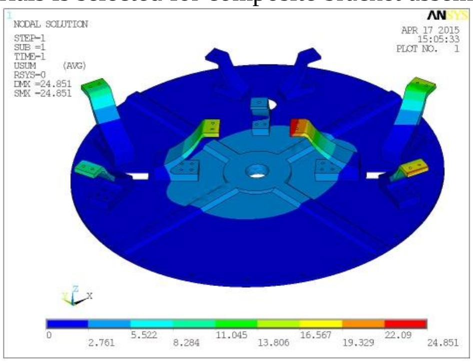

Fig. 7 Shows avg deflection of bracket assembly

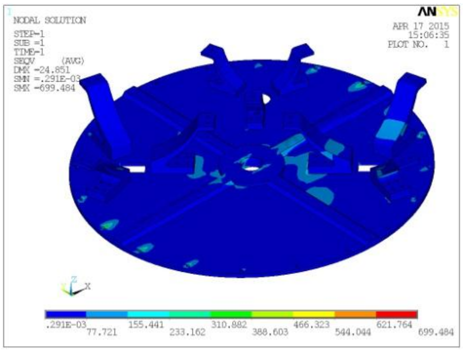

Fig. 8 Shows max Von Mises stress of bracket assembly

Table.2 Shows the Deflections and stresses for HM Carbon

Sl.NO	DEFLECTIONS		STRESSES	
1	X-def	7.9	1st Principal stress	656
2	Y-def	11.7	2nd Principal stress	116
3	Z-def	0.29	3rd Principal stress	11
4	Avg def	24.8	Von Mises stress	699

From the above results it is observed that the principal stresses values 656MPa, 116MPa, and 11MPa are less than the principal stresses values of the material 870MPa, 154MPa, and 30MPa with respectively 1st, 2nd and 3rd principal stresses. Hence according to the Maximum Stress Theory, the Bracket Assembly model is safe for the above operating loads.

CASE-3: Static Analysis of Bracket Assembly with HS Carbon / Epoxy composite material.

Based on the advantages, the HS Carbon/Epoxy, materials is selected for composite bracket assembly.

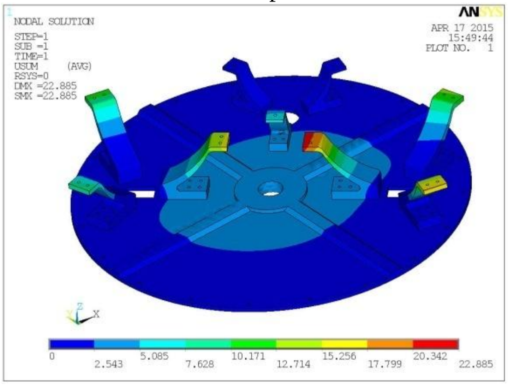

Fig. 9 Shows avg deflection of bracket assembly

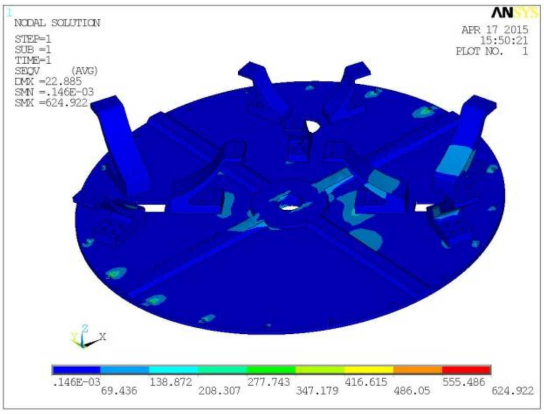

Fig. 10 shows max von mises stress of bracket assembly

Table.3 Shows the Deflections and stresses for HS Carbon

Sl. NO	DEFLECTIONS		STRESSES	
1	X-def	3.8	1st Principal stress	549
2	Y-def	6.5	2nd Principal stress	107
3	Z-def	0.2	3rd Principal stress	19
4	Avg def	14.2	Von Mises stress	580

From the above results it is observed that the principal stresses values 584MPa, 102MPa, and 13MPa are less than the principal stresses values of the material 880MPa, 160MPa, and 97MPa with respectively 1st, 2nd and 3rd principal stresses. Hence according to the Maximum Stress Theory, the Bracket Assembly model is safe for the above operating loads.

CASE-4: Static Analysis of Bracket Assembly with BMI Carbon / Epoxy composite material.

Based on the advantages, the BMI composite material is selected for composite bracket assembly.

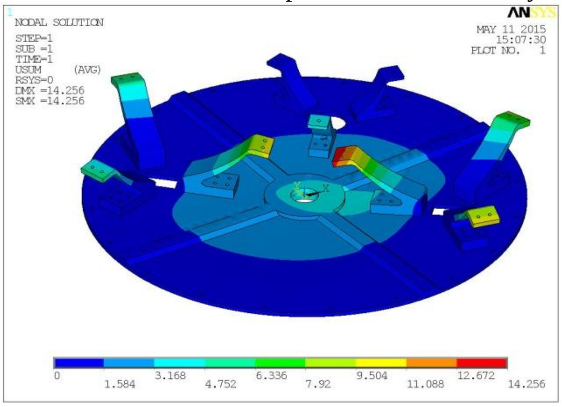

Fig. 11 Shows avg deflection of bracket assembly

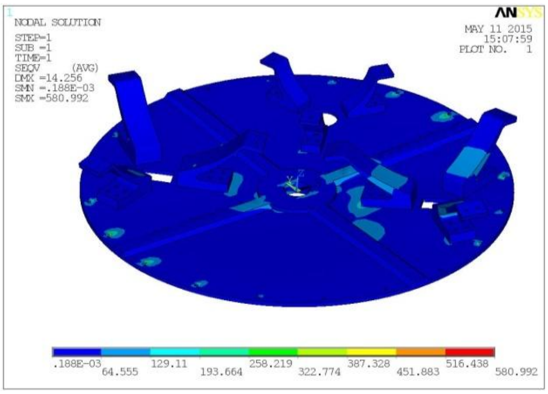

Fig. 12 Shows max von mises stress of bracket assembly

Table.4 Shows the Deflections and stresses for BMI

Sl.NO	DEFLECTIONS		STRESSES	
1	X-def	6.5	1st Principal stress	584
2	Y-def	10.7	2nd Principal stress	102
3	Z-def	0.28	3rd Principal stress	13
4	Avg def	22.8	Von Mises stress	624

From the above results it is observed that the principal stresses values 549MPa, 107MPa, and 19MPa are less than the principal stresses values of the material 765MPa, 168MPa, and 32MPa with respectively 1st, 2nd and 3rd principal stresses. Hence according to the Maximum Stress Theory, the Bracket Assembly model is safe for the above operating loads.

VII. CONCLUSIONS

- In the present project a Bracket Assembly has been modeled for static loading conditions with different material properties.
- From the above analysis it is concluded that the Bracket Assembly has stresses and deflections within the design limits of the aluminum alloy material and composite materials (HM Carbon /epoxy, HS Carbon /epoxy and BMI Carbon /epoxy,).
- From the above analysis it is also concluded that the HS Carbon /epoxy composite Bracket Assembly model having high factor of safety (1.4) than the other materials and also have less weight than the aluminum alloy model, because of the HS Carbon /epoxy density is less than the aluminum alloy.

REFERNECES:

[1] Materials Science And Engineering – Vol. III – Aerospace and Space Materials – M. Peters and C. Leyens
[2] Dieter G. E. (1997). ASM Handbook Materials Selection and Design, Vol. 20, pp.900.Materials Park, OH: ASM International. [Comprehensive overview on design concepts and criteria, design tools, materials properties and selection, and manufacturing aspects of design.]
[3] Vasudevan A. K. and Doherty, R. D. (1989).Aluminium Alloys– Contemporary Research and Applications, pp.702. San Diego, CA: Academic Press. [Research and technology on processing, structure and properties of present and future Al alloys.]
[4] E.L. Wilson, "*Automation of the Finite Element Method—A personal Historical View*," Finite Element in Analysis and Design, vol.13, Nos. 2&3, 1993, pp- 91-104
[5] HarrisS,B. A perspective view of composite materials. Mat & Design, vol 12, no. 5.1991. pp. 259-271.
[6] Carter ,F.C., and Paul,D.E. Material science and engineering. Ohio, ASM International,1991 351 pp
[7] Bodily B, Heinimann M, Bray G, Colvin E, Witters J. Advanced aluminum and aluminum lithium solutions for derivative and next generation aerospace structures. SAE paper no 2012-01-1874