ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 1 – No.2, November 2014

FE Analysis of Rib of Commercial Aircraft

Deepak kumar
Student
B.tech(aeronautical)
Gurukul vidyapeeth
Deepakgujjar45@gmail.com

Loveleen
Student
B.tech(aeronautical)
Gurukul vidyapeeth
Loveleenk62@gmail.com

ABSTRACT

The paper is about the FE analysis of rib of commercial aircraft. Ribs maintain the shape of wing and also support the bending and compressive load which act on the wing. Rib are attached to the spars, with stressed skin. It act as a load distributor . CATIA V5 software is used for modeling. The stress analysis of rib is carried out to compute the stresses at rib structure. FE analysis is used to find out the saftey factor of the structure. It show the maximum stress location. Linear static and buckling analysis has been analyzed in Patran and Nastran.

.Keywords

FE analysis , rib , load ,Patran , Nastran.

1.INTRODUCTION

For aerodynamic reasons the wing contour in the chord direction must be maintained without appreciable distortion. Ribs are used to hold the cover panel to contour shape. The other major purposes of Ribs are to act as a transfer or distribute the load. Ribs are composed of caps, stiffener and webs. It also contains lightening holes and beads in its structure. The stress analysis of the rib structure is carried out to compute the stresses at rib structure. It shows the maximum stress location.

1.1 FE Analysis

Finite element analysis is a type of computer program that uses the finite element method to analyze a material or object and find how applied stresses will affect the material or design. FEA can help determine any point of weakness in a design before it manufactured. The analysis is done by create a mesh of points in shape of object that contains information about the material and the object at each point for analysis. FEA can also analyze the effect of vibrations, fatigue, and heat transfer.

1.2 PATRAN

Patran is the world's widely used pre and post processing software for finite element analysis providing solid modeling, meshing, analysis setup and post processing for multiple solvers including MSC Nastran. Patran provides a rich set of tool that streamline the creation of analysis ready models for linear, nonlinear, explicit dynamics ,thermal, and other finite element solutions. Patran's comprehensive and industry tested capabilities ensure that your virtual prototyping efforts provide results fast so that you can evaluate product performance against requirements and optimize your designs.

1.3 NASTRAN

Nastran offers a complete set of linear static and dynamic analysis capabilities along with unparalleled support for super elements enabling users to solve large, complex assemblies more efficiently. MSC Nastran also offers a complete set of implicit and explicit nonlinear analysis capabilities, thermal and interior/exterior acoustics, and coupling between various disciplines such as thermal, structural, and fluid interaction.

2. CATIA MODEL

Rib is modelled in CATIA-V5 has been shown in figure 1.

Component	Material	Thickness	Young's modulus	Poisson's ratio
Rib	AL - 7075	1.2 mm	71700 MPa	0.33

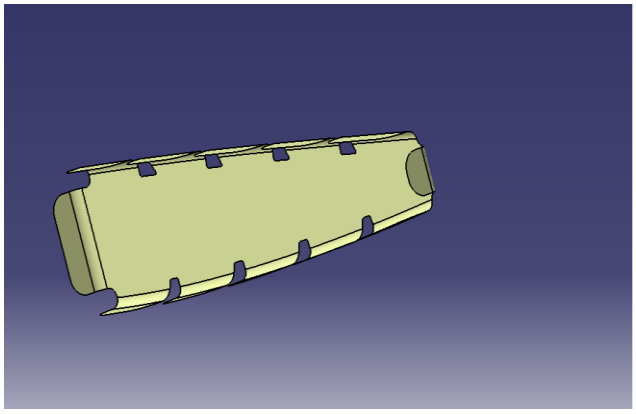
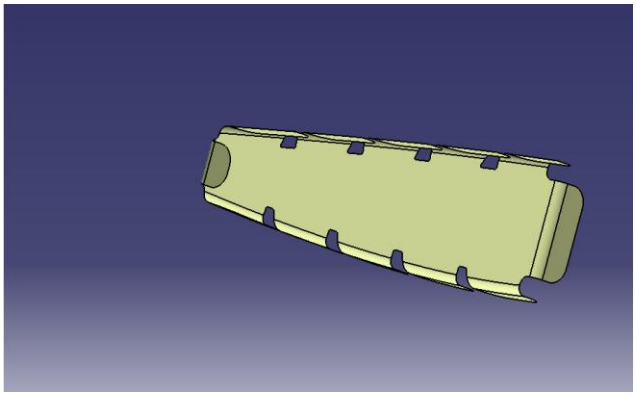


Fig 1: Rib 3D model

ISSN (O): 2393-8609

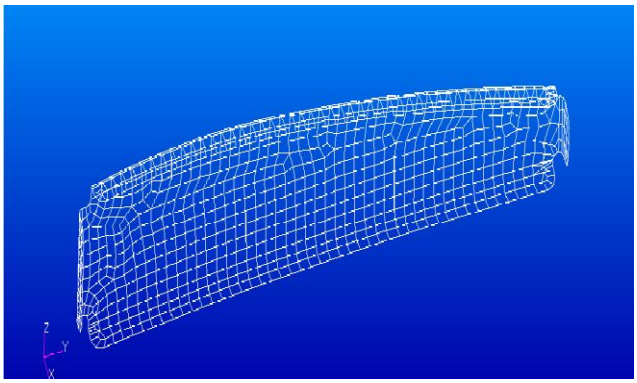
International Journal of Aerospace and Mechanical Engineering

Volume 1 – No.2, November 2014



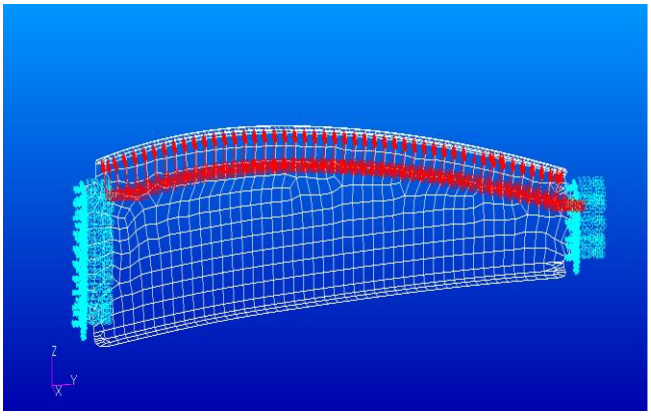
3. MESHING

The model is meshed using quad4 element for better results as shown below.



4. BOUNDRY CONDITIONS

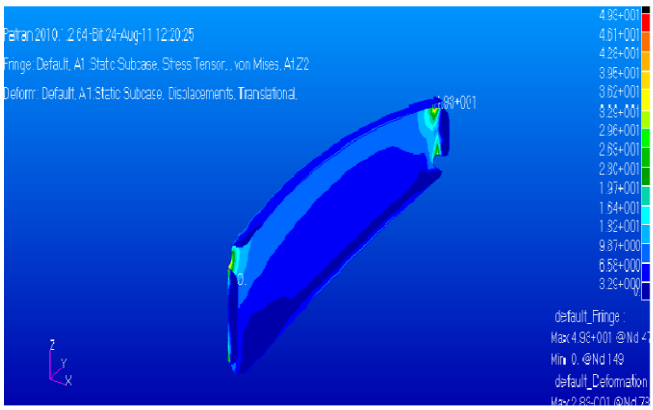
- A pressure of 0.03 MPa is applied on the upper flange of the rib that is equivalent to the aerodynamic pressure exerted during the flight.
- Side flanges of the rib are constraint in all degrees of freedom as it is in contact with the front and rear spar.



5. RESULT

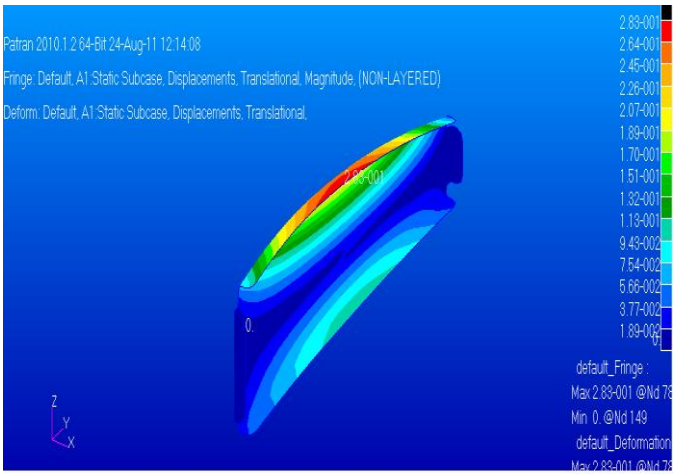
5.1 Stress

- 49.3 MPa stress is produced at the relief corner which is far below the allowable yield value of the material.



5.2 Displacement

- Displacement produced is 0.28 mm which is quite low as shown below



6. CONCLUSION

Stress analysis of the rib structure is carried out and maximum stress is identified at relief corner which is found out to be lower than yield strength of the material. It also concluded that there is minimum presence of stress at the mid-section of rib. Weight can be reduced by forming the lighting holes at the sections of rib where stress is minimum.

7. FUTURE SCOPE

Weight optimization can be carried out by creating lightning holes in the rib and further configurations can be studied.

8. REFRENCES

[1] Brown, L. D., Hua, H., and Gao, C. 2003. A widget framework for augmented interaction in SCAPE.

[2] "PRELIMINARY SIZING PROCEDURE OF RIB OF AN AIRCRAFT" international journal of aerospace and mechanical engineering Vol no -1 September 2014