DESIGN AND ANALYSIS OF A TYPICAL WING RIB FOR PASSENGER AIRCRAFT

Bindu H.C[1],Muhammad Muhsin Ali.H[2]

P.G. Student, Department of Mechanical Engineering, Ghousia college of Engineering, Ramanagaram,

Karnataka, India[1]

Associate Professor, Department of Mechanical Engineering, Ghousia college of Engineering, Ramanagaram,

Karnataka,India[2]

Abstract: Ribs maintain the shape of the wing and also support the bending and compressive loads which acts on the wing. The objective was to increase the critical buckling strength and reduce the weight of the rib. Linear static and buckling analysis are performed on the idealized configuration using FEM packages. Plate rib was used in the analysis. When the detail design is subjected for optimization and it found safe. Finally we concluded that initially the design is of 4.592 Kg & optimize up to 2.964 Kg so that we reduce the weight by 1.628 Kg.

Keywords: Aluminium alloy, wing rib, modeling in Catia V5,FEA analysis in Nastran and Patran.

I INTRODUCTION

When designing an aircraft, it's all about finding the optimal proportion of the weight of the vehicle and payload. The main components of a passenger aircraft are Fuselage, Engines, Wings, Spoilers, and Control Surfaces such as Flaps and Slats, Elevators, Stabilators and Rudder. The material taken is an aluminum alloy. Design processes follows with Plate-type ribs consist of sheet-metal and weight-saving holes cut out into it. These ribs are used in conditions of light to medium loading. A compressive load produces buckling stress in the plate. Using this concept the design is optimized and variation of buckling load with respect to number of holes in the plate is studied.

II METHODOLOGY

Due to increasing cost on conducting real-time crash simulations, CAE tools are very widely used in auto industry. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulations rather than physical prototype testing. Tools used in this study are briefly explained below.

1.CATIA V5

It is an integrated site of CAD, CAM and CAE applications for digital product definition and simulation. Catia v5 provides advanced three - dimensional product life cycle management (PLM) solution for collaborated product development. Catiav5 products are organized on three different platforms like P1, P2, and P3. We have used the platform P3 to do the modeling.

CatiaV5 platform maximizes the productivity of specific advanced process using focused solutions.

Key benefits of Catia V5 package

- ➢ Improved enter price competitiveness
- ➢ Ease of use
- ➢ Latest technologies
- ➢ Leverage of your existing environment

2.HYPERMESH

Altair HYPERMESH is a high-performance finite element pre-and postprocessor for major finite element solvers, allowing engineering to analyze design condition in highly interactive and visual environment. Hypermesh's user-interface is easy to learn and supports the direct use of cad geometry and existing finite element models, providing robust interoperability and efficiency. Hyper mesh can check meshes for duplicate elements, facet Warpage, and aspect ratio, skewness, and can display and modify element normal.

3.MSC 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.

4.MSC Patran

It is the widely used pre/post-processing software for Finite Element Analysis (FEA). Patran provides a rich set of tools 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.

5 Static Analysis

A static analysis calculates the effects of *steady* loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include *steady* inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. The model considered in the project has to go through a series of iterations. The static analyses performed just know whether model is well within the limit of yield strength of material.

6 Buckling Analysis

Buckling analysis is a technique used to determine,

> ➢ Buckling loads - critical loads at which a structure becomes unstable, and

> ➢ Buckled mode shapes - the characteristic shape associated with a structure's buckled response.

There are two types of Buckling analysis,

 A. Eigen value Buckling Analysis and

 B. Nonlinear Buckling Analysis

A. Eigen value Buckling Analysis

Eigen value buckling analysis predicts the theoretical buckling strength (the bifurcation point) of an ideal linear elastic structure. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength.

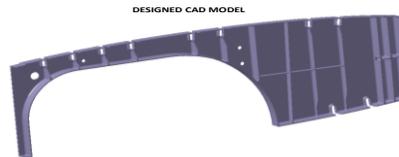### B. Nonlinear Buckling Analysis

Nonlinear buckling analysis is usually the more accurate approach and is therefore recommended for design or evaluation of actual structures. This technique employs a nonlinear static analysis with gradually increasing loads to seek the load level at which our structure becomes unstable. In this type we can include features such as initial imperfections, plastic behaviour, gaps, and large-deflection response.

III RESULTS AND DISCUSSION

Here we taken inputs from preliminary design and the same taken as bench mark for the further detail design for the modelling of aircraft wing rib, we considered Numerical master geometry (N.M.G). The same procedure followed also for the detail design model and optimized detail design model of the aircraft wing rib and analysis is done using Nastran and Patran Analysis tools. We carried out static and buckling Analysis for detail design model and optimized detail design model of aircraft wing rib.

1 Rib thickness without any cut outs.

1.1 CAD Model of Wing Rib: The figure shows three dimensional aircraft wing rib. The geometry is built in sketcher and layer extruded to three dimensional parts in the part modeller to get three dimensional representations. Model is built using CATIA V5 Software.

DESIGNED CAD MODEL

Figure 1.1: Design Model

1.2 FE Model of Wing Rib: The figure shows the FE model of Wing Rib. It is built using HYPERMESH software. Hypermesh is a good meshing software to control the mesh at irregular geometries.

Figure 1.2: FE Model

Boundary conditions for the part is applied which is shown in figure 1.3. Boundary condition is a method of applying load and constraining the part. Wing rib is constrained in all 6 DOF on either sides and a pressure of 2.75e-2 is applied on the

elements which is shown in the below figure. Meshed model is imported to Nastran software in bdf format to calcute displacement.
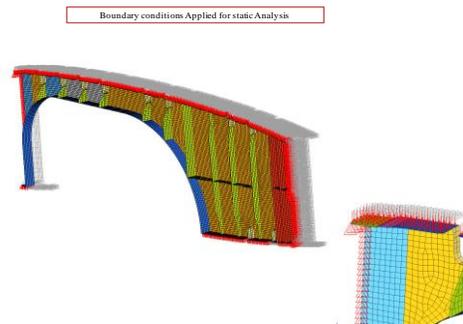

Figure 1.3: Boundary Conditions for Static Analysis

1.3 Displacement plot of Wing Rib: Figure1.4 shows the displacement of the rib in z-direction when a pressure 2.75e-2 is applied on the rib. we observe maximum diplacement is 3.21mm on node 18743e.
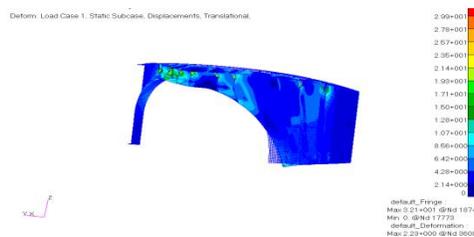

Figure 1.4: Displacements for Detail Design

1.4 Stress plot of Wing Rib: Figure1.5 shows the stresses developed on the rib due to a pressure of 2.75e-2. Maximum stress observed is 2.23 at node 36082.
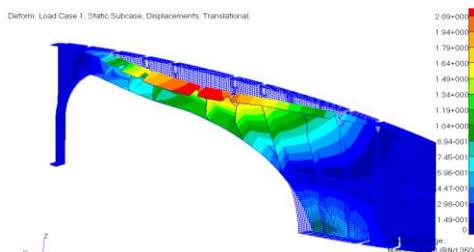

Figure 1.5: Stresses for Detail Design.

1.5 FE Model and Boundary conditions for Buckling analysis: For doing buckling analysis we constrained the boundary nodes of the wing rib with 3 DOF and concentrated load is applied. The boundary condition of the part is similar to simply

supported beam. To know critical loads we have taken 10 iteration for buckling analysis. We got the results in 5th iteration. Eigen value results are tabulated.

Table 1 : Eigen value results

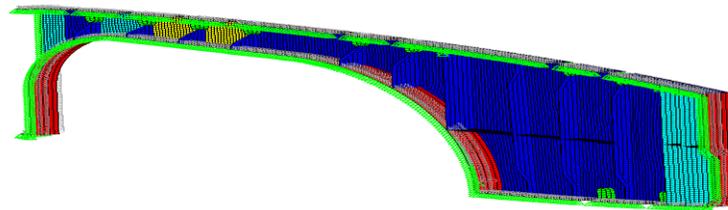| BLF Value	Buckling Status	Remarks
>1	Buckling not predicted	The applied loads are less than the estimated critical loads.
= 1	Buckling predicted	The applied loads are exactly equal to the critical loads. Buckling is expected.
< 1	Buckling predicted	The applied loads exceed the estimated critical loads. Buckling will occur.
-1 < BLF < 0	Bucklin possible	Buckling is predicted if you reverse the load directions.
-1	Buckling possible	Buckling is expected if you reverse the load directions.
< -1	Buckling not predicted	The applied loads are less than the estimated critical loads, even if you reverse their directions.

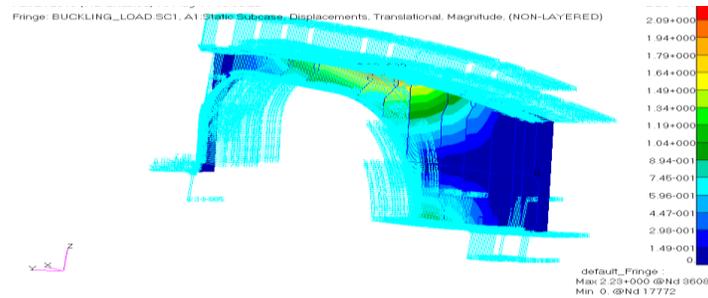**Figure 1.6: FE Model for Buckling Analysis**

Figure1.7: Boundary Conditions for Buckling Analysis

1.6 Buckling Analysis Iteration : In this iteration we observed the value obtained is greater than 1, we can predict that buckling occured in the part. The load factor considerd for this iteration is 417.53N/mm2. we should go for more iterations by increasing the Load factors.
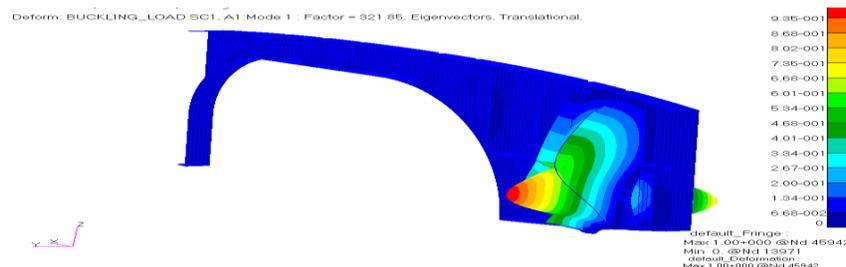
Figure 1.8: Mode 1 of Buckling Analysis

1.7 Real Eigen values: Eigen values are more than 1 or equal to 1 hence the design is safe.

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	3.232981E+02	1.798049E+01	2.861684E+00	2.654963E+01	8.583442E+03
2	2	3.818020E+02	1.953975E+01	3.109848E+00	5.462089E+01	2.085437E+04
3	3	3.992471E+02	1.998117E+01	3.180102E+00	2.316388E+01	9.248111E+03
4	4	4.072634E+02	2.018077E+01	3.211869E+00	1.455906E+01	5.929372E+03
5	5	4.633732E+02	2.152611E+01	3.425986E+00	5.494464E+01	2.545988E+04
6	6	4.893752E+02	2.212183E+01	3.520798E+00	7.709487E+01	3.772832E+04
7	7	4.933511E+02	2.221151E+01	3.535071E+00	2.455755E+01	1.211549E+04
8	8	4.958209E+02	2.226703E+01	3.543909E+00	2.333410E+01	1.156954E+04
9	9	5.360605E+02	2.315298E+01	3.684911E+00	1.340452E+01	7.185636E+03
10	10	5.398686E+02	2.323507E+01	3.697977E+00	6.144969E+01	3.317476E+04

Table 2: Eigen value results

2. Rib thicknesses with two cut-outs.

2.1 Optimized FE Model of Wing Rib: We did 3-4 iterations to get the optimized model for rib. we added few holes to the model to get optimized. FE model and Boundary conditions for the same is shown in below figures. Total No of elements for the below model is 1212 .
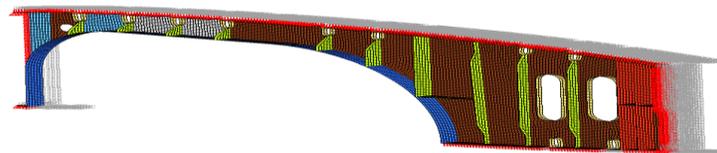

Figure2.1: Optimized model

Figure 2.2: Boundary conditions for static Analysis of optimized model

2.2 Displacement plot of Wing Rib: Figure2.3 shows the displacement of the rib in z-direction when a pressure 2.75e-2 is applied on the rib. we observe maximum diplacement is 3.26mm on node 36082.
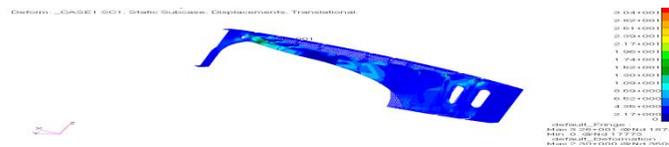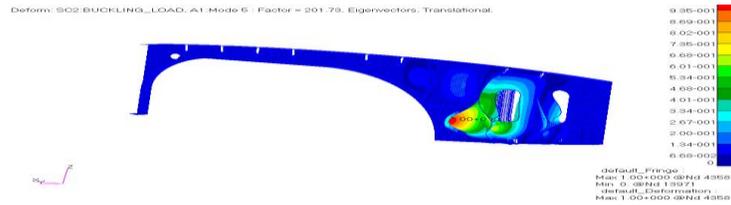

Figure 2.3: Displacements for optimized model of detail design

2.3 Buckling analysis for optimized Wing Rib: For doing buckling analysis we constrained the boundary nodes of the wing rib with 3 DOF and concentrated load is applied. The boundary condition of the part is similar to simply supported beam. To know critical loads we have taken 10 iteration for buckling analysis.

Figure 2.4: Mode 5 of Buckling Analysis for optimized model

2.4 Real Eigen values:

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
			R E A L E I G E N V A L U E S			
1	1	1.001152E+02	1.000576E+01	1.592466E+00	1.546616E+01	1.548398E+03
2	2	1.324631E+02	1.150926E+01	1.831756E+00	3.640903E+01	4.822853E+03
3	3	1.503302E+02	1.226092E+01	1.951386E+00	2.655852E+01	3.992549E+03
4	4	1.564613E+02	1.250845E+01	1.990782E+00	1.639509E+01	2.565197E+03
5	5	2.017283E+02	1.420311E+01	2.260495E+00	1.848520E+01	3.728988E+03
6	6	2.197874E+02	1.482523E+01	2.359508E+00	8.301899E+00	1.824653E+03
7	7	2.427117E+02	1.557921E+01	2.479508E+00	5.118188E+01	1.242244E+04
8	8	2.487501E+02	1.577181E+01	2.510162E+00	3.512366E+01	8.737014E+03
9	9	2.840625E+02	1.685415E+01	2.682422E+00	4.062589E+01	1.154029E+04
10	10	2.922813E+02	1.709624E+01	2.720951E+00	1.198986E+01	3.504413E+03

Table 2: Eigen value results for optimized model

IV CONCLUSION

From the studies it was found that inserting the circular hole in the plate enhance the buckling strength of the plate.The buckling strength of the plate is increased as the number of holes increased. Mean while the von misses stress in the component keeps on increased as the number of holes increased. Inserting holes in the rib found out to be effective with weight reduction compared to the initial geometry. The maximum stress occurs around the holes and it is to be considered as the critical region in the later stages.

V SCOPE OF FUTURE WORK

 Using composite materials to reduce the weight. The buckling strength of the plate is increased as the number of   holes increased. Mean while the von misses stress in the component keeps on increased as the number of holes increased.

REFERENCES

 [1] William L. KO, Dryden Flight Research Center, Edwards, California "Mechanical- and Thermal-Buckling Behaviour of Rectangular Plates with **Different** Central Cutouts", NASA/TM-1998-206542, March 1998.
[2] R.C. Batra, Z. Wei, "Dynamic buckling of a thin thermoviscoplastic rectangular plate", Department of Engineering Science and Mechanics, Virginia Polytechnic Institute and State University,MC 0219, Blacksburg, VA 24061, USA, November 2004.
 [3]J. Purbolaksono, M.H. Aliabadi."Application of DRBEM for Evaluating Buckling Problems of Perforated Thin Plates" European Journal of Scientific Research ISSN 1450-216X Vol.31 No.3 (2009), pp.398-408
[4] Altair Hypermesh, version 9, user's guide – Altair Engineering Inc., USA