

# Design and Analysis of A Spar Beam For The Vertical Tail of A Transport Aircraft

Vinod S. Muchchandi<sup>1</sup>, S. C. Pilli<sup>2</sup>

P.G. Student, Department of Mechanical Engineering, KLE Dr.MSS CET, Belgaum, Karnataka, India<sup>1</sup>

Principal, Department of Mechanical Engineering, KLE Dr.MSS CET, Belgaum, Karnataka, India<sup>2</sup>

**Abstract:** Vertical tail and the rudder are important structural components of an aircraft. Movement of the rudder controls the yawing of an aircraft. Structurally speaking vertical tail is a typical mini-wing construction. A major difference could be absence of ribs and multiple spars (more than 2) in the vertical tail construction. Vertical tails have symmetrical airfoil cross sections. Therefore in the absence of rudder deflection there is no aerodynamic load acting on the fuselage. However significant side loads develop due to rudder deflection and this is the major design load for the vertical tail. For transport aircraft side gust load is also important from a design point of view. In this project a typical spar of a vertical tail of a transport aircraft will be analysed. Loads representative of a small transport aircraft will be considered in this study. An efficient tapered spar beam will be designed for this load. SOM approach will be used for preliminary sizing of the spar. This will be followed by FEA for a more accurate stress analysis that will be used to improve the design.

**Keywords:** Transport aircraft, Vertical tail, Spar, Symmetrical aerofoil, Side load, SOM, FEM

## I. INTRODUCTION

An aircraft is a complex structure, but a very efficient man-made flying machine. Air-craft structure is the most obvious example where functional requirements demand light weight and stronger structures. An efficient structural component must have three primary attributes, namely, 1) The ability to perform its intended function, 2) Adequate service life and 3) With the capability of being produced at reasonable cost.

The primary factors to consider in aircraft structures are strength, weight, and reliability. These factors determine the requirements to be met by any material used to construct or repair the aircraft. Airframes must be strong and light in weight. All materials used to construct an aircraft must be reliable. Reliability minimizes the possibility of unexpected failures. The major part of the aircraft structure consists of built-up panels of sheets and stringers, e.g. wing and fuselage skin panels, spar webs and stiffeners. Airframes are members and transverse frames to enable it to resist bending, compressive and vehicles which are able to fly by being supported by the air, or in general, the atmosphere of a planet. An aircraft counters the force of gravity by using either static lift or by using the dynamic lift of an airfoil, or in a few cases the downward thrust from jet engines. Aircrafts are generally built-up from the basic components of fuselage, wings, tail units and control surfaces. Each component has one or more specific functions and must be designed to ensure that it can carry out these functions reliably. Any small malfunction or failure of any of these components may lead to a catastrophic disaster causing huge destruction of lives and property. When designing an aircraft, it's all about ending with the optimal proportion of the weight of the vehicle and payload. It needs to be strong and stiff enough to withstand the exceptional environment in which it has to operate. Durability is also an important factor. Also, if a part fails, must not necessarily result in failure of the whole aircraft. It is still possible for the aircraft to glide over to a safe landing place only if the aerodynamic shape is retained and structural integrity is achieved.

## II. SOFTWARE DESCRIPTION

Software's used in the present work are

### 1. Finite element modeling (PATRAN)

A finite element pre and post processors (such as MSC/PATRAN) is a graphicbased software package primarily designed to aid in the development of Finite Element Model (Pre processing) and to aid the display and interpretation of analysis results (Post processing). MSC/ PATRAN software is a mechanical Computer Aided Engineering Design. Utilizing integrated automatic technologies. It enables design engineers to build and modify solid models of computer parts and predicts their behavior through design optimization. In addition preprocessing software helps the analyst modify the model if the result shows that changes and subsequent reanalysis are required.

### 2. Finite element solver [NASTRAN]

The finite element method has a seminal impact on the field of analysis and design. This technique has allowed the solution of a new class of problem that is practically difficult, costly or was not possible in the past. The acceptance of

FEM in the industry however did not enjoy wide popularity until the acceptance of general purpose programs finite element programs.

### III. DESIGN AND ANALYSIS OF SPAR BEAM

#### 1. Stress Comparison with Different Cross Sections

Considering the cantilever beam of 500 mm length and 350 kg load. The deformation and stress for different cross section considered and compared on Table 5.1.

Table 5.1: Stresses and deformation for different section

| Cross section    | Stress $kg/mm^2$ (fem) | Defomnation mm(fem) | Stress $kg/mm^2$ (som) | Deformn mm(som) |
|------------------|------------------------|---------------------|------------------------|-----------------|
| Circle           | 39.2                   | 26.18               | 39.243                 | 26.3            |
| Circle with hole | 12.5                   | 4.71                | 12.65                  | 4.64            |
| Rectangle        | 23.5                   | 12.61               | 23.48                  | 12.5            |
| Square           | 33.2                   | 25.1                | 33.212                 | 25              |
| I section        | 8.59                   | 2.87                | 7.899                  | 2.5239          |

The above values show that the I section experiences the least amount of stress for the given type of loading similar to the loads on the aircraft. This is because the I-section has two angles which take the drag load and a web supported by the angles which takes the sideward bending type of loads. Thus for a given strength to weight ratio criteria the I-section withstands more loads comparatively. Hence considering all these criteria spar of I section is used in design.

#### 2. Design Optimization of Spar Beam

- I-section with same cross section throughout.
- I-section with varying cross sections along the length.
- I-section of varying cross section with cutouts.

#### 3. Problem Definition of Spar Beam

Cantilever beam having length of 3250 mm and loads (62-620 Kg) as shown in Table and Fig. Its Ultimate strength is 343.35 MPa (From Westerman table)

Cantilever beam with the loads at each station ( 1-10) as shown in Fig.5.1.

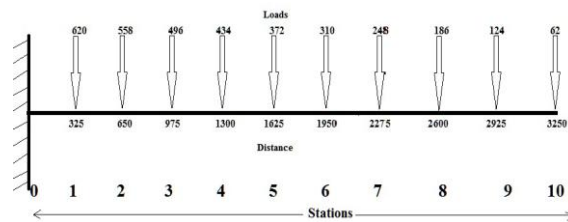


Figure 5.1: Loads on the spar beam

Table 5.2: Loads and distance from the root

| Station | Distance from root (mm) | Loads in Kg |
|---------|-------------------------|-------------|
| 0       | 0                       | 0           |
| 1       | 325                     | 620         |
| 2       | 650                     | 558         |
| 3       | 975                     | 496         |
| 4       | 1300                    | 434         |
| 5       | 1625                    | 372         |
| 6       | 1950                    | 310         |
| 7       | 2275                    | 248         |
| 8       | 2600                    | 186         |
| 9       | 2925                    | 124         |
| 10      | 3250                    | 62          |

The cantilever beam with loads indicated the bending moments and corresponding dimensions of I section at different stations are shown in Table 5.3

| Station | Bending moments N-m | D(mm)   | B(mm)  | t(mm)  | b(mm)  | d(mm)   |
|---------|---------------------|---------|--------|--------|--------|---------|
| 0       | 43487.7             | 164.106 | 82.053 | 8.2053 | 73.847 | 147.695 |
| 1       | 32615.7             | 149.1   | 74.55  | 7.455  | 67.095 | 134.19  |
| 2       | 23720.5             | 134.084 | 67.042 | 6.7042 | 60.338 | 120.676 |
| 3       | 16604.4             | 119.054 | 59.527 | 5.9527 | 53.574 | 107.148 |
| 4       | 1106.9              | 104.003 | 52.001 | 5.2001 | 46.801 | 93.603  |
| 5       | 6918.5              | 88.921  | 44.46  | 4.446  | 40.014 | 80.029  |
| 6       | 3953.4              | 73.789  | 36.894 | 3.6894 | 33.205 | 66.41   |
| 7       | 1976.7              | 58.566  | 29.283 | 2.9283 | 26.355 | 52.71   |
| 8       | 790.6               | 43.152  | 21.576 | 2.1576 | 19.418 | 38.837  |
| 9       | 197.6               | 27.184  | 13.592 | 1.3592 | 12.232 | 24.465  |
| 10      | 0                   | 13.592  | 6.795  | 0.6795 | 6.116  | 12.2325 |

I section is shown in Fig 5.2.  $B=0.5D$ ,  $t=0.05D$ ,  $b=0.45D$ ,  $d=0.9D$  (From Westermann table)

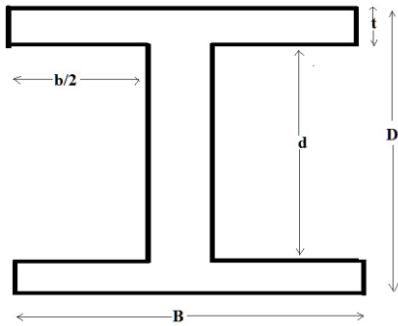


Figure 5.2: I section

5.3 I-Section with Same Cross Section Throughout the Length of Beam

I section with same dimension in all the stations of spar beam ( $D=164.106, B=82.053$  and  $t=8.205$ ) is shown in Fig.5.3. CATIA V5 software is used to draw this model. This model is used for further analysis in NASTRAN and PATRAN software.

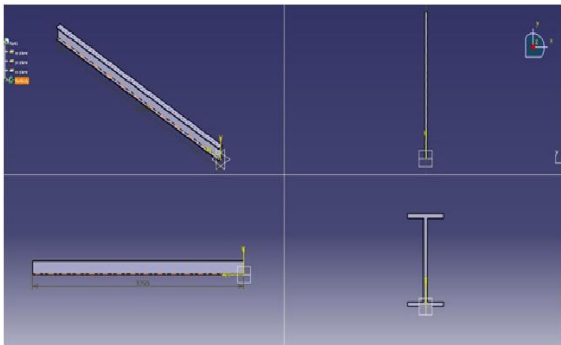


Figure 5.3: Geometric specification of spar beam (Uniform cross section)

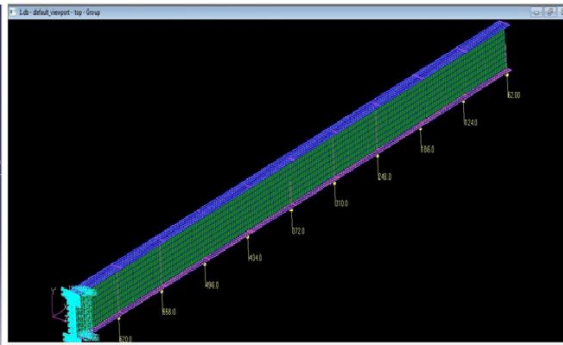


Figure 5.4: Mesh model (Uniform cross section)

Model is meshed using 2D mesh having quad elements. It contains 4400 elements and 4641 nodes. Software used for meshing is MSC/PATRAN. Mesh model is shown in Fig.5.4.

After applying loads (as shown in Fig.5.1) and boundary condition maximum stress occurs at bottom flange of spar beam near station 0-1 and minimum stress occurs at web of the spar beam near station 9-10. The stress distribution is shown in Fig.5.5.

Stress values at each station along the length of the spar beam are shown in Table 5.4

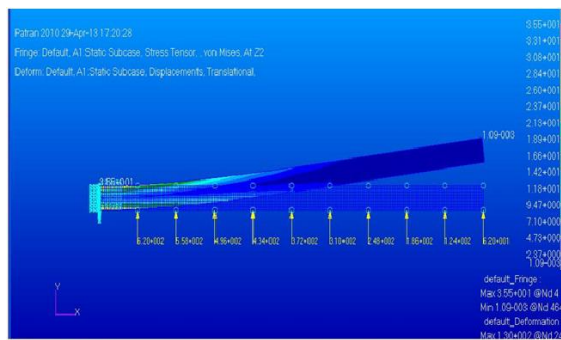


Figure 5.5: Stress distribution contour (Uniform cross section)

Table 5.4: Stress values along the length of spar beam

| Station | Stresses MPa |
|---------|--------------|
| 0-1     | 337.4        |
| 1-2     | 228.5        |
| 2-3     | 158.9        |
| 3-4     | 109.8        |
| 4-5     | 72.4         |
| 5-6     | 48.1         |
| 6-7     | 24.5         |
| 7-8     | 12.7         |
| 8-9     | 4.08         |
| 9-10    | 3.1          |

Maximum stress occurs near the station 0-1, its value is 337.4 MPa and minimum stress occurs near the station 9-10, its value is 3.1 MPa.

The plot of stresses Vs station is shown in Fig.5.6.

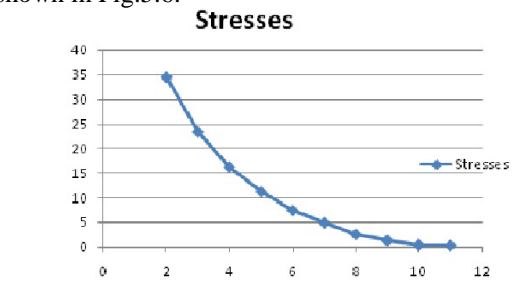


Figure 5.6: Plot stresses vs station

#### 5.4 I-section with Varying Cross Sections Along the Length of Spar Beam

The I section with different dimensions as shown in Table 5.3 in all the stations is shown in Fig.5.7. CATIA V5 software is used to draw this model. This model is used for further analysis in NASTRAN and PATRAN software.

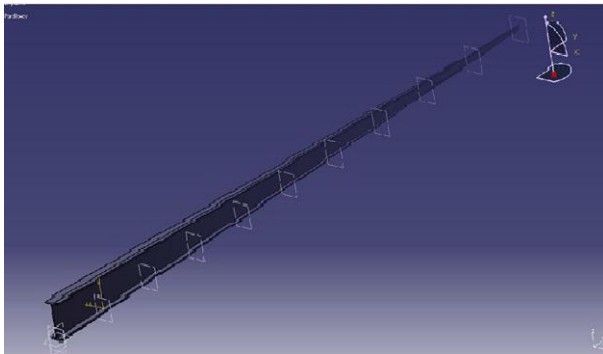


Figure 5.7: Geometrical specification of spar beam

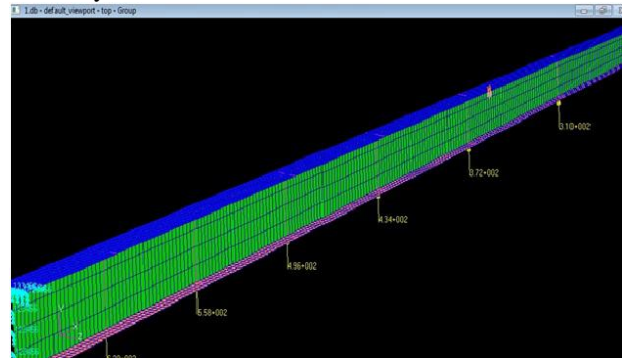


Figure 5.8: Mesh model (Varying cross section)

(varying cross section)

Model is meshed using 2D mesh having quad and tria elements. It contains 12364 elements and 17453 nodes. Software used for meshing is MSC/PATRAN. For this I section with different dimensions, mesh model is shown in Fig.5.8.

After applying loads (as shown in Fig 5.1) and boundary condition maximum stress occurs at top and bottom flange of spar beam near station 8-9 and minimum stress occurs at web of the spar beam near station 1-2. The stress distribution is shown in Fig.5.9.

Stress values at each station along the length of the spar beam are shown in Table 5.5. Maximum stress occurs near the station 8-9, its value is 336.4 MPa and minimum stress occurs near the station 1-2, its value is 86.4 MPa.

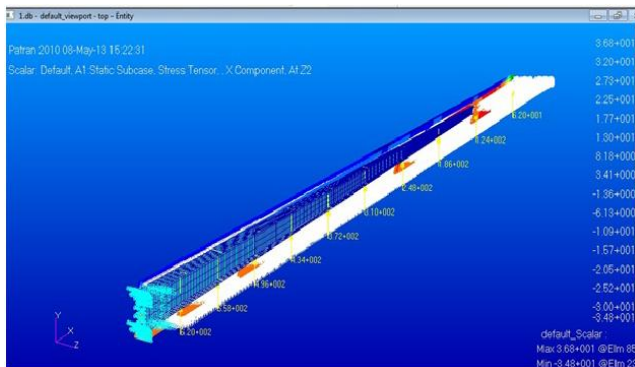


Figure 5.9: Stress distribution contour for varying cross section

Table 5.5: Stress values at different station

| Station | Top flange (stresses) MPa | Station      | Top flange (stresses) MPa |
|---------|---------------------------|--------------|---------------------------|
| 0-1     | 243.2 – 289.3             | 87.1 – 89.7  | 243.2 – 289.3             |
| 1-2     | 242.3 – 289.3             | 86.4 – 89.4  | 242.3 – 289.3             |
| 2-3     | 244.2 – 290.3             | 87.1 – 89.5  | 244.2 – 290.3             |
| 3-4     | 245.2 – 291.3             | 86.4 – 89.9  | 245.2 – 291.3             |
| 4-5     | 246.2 – 284.4             | 86.6 – 90.9  | 246.2 – 284.4             |
| 5-6     | 247.2 – 295.2             | 86.2 – 91.9  | 247.2 – 295.2             |
| 6-7     | 251.1 – 298.2             | 88.7 – 98.1  | 251.1 – 298.2             |
| 7-8     | 260.9 – 308.1             | 89.9 – 112.8 | 260.9 – 308.1             |
| 8-9     | 277.6 – 336.4             | 96.6 – 147.1 | 277.6 – 336.4             |
| 9-10    | 331.5 – 341.3             | 99.1 – 156.9 | 331.5 – 341.3             |

#### IV. SHAPE OPTIMISATION

The beam with varying section has been divided into six cases for the analysis, having varying cutout dimensions at each station. The schematic cutout dimensions are shown in Fig.6.1

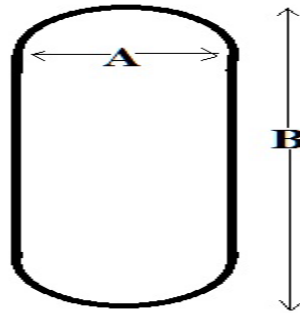


Figure 6.1: Cutout model

##### CSAE 1:

According to the Table 6.1 cutouts are introduced into the spar beam web, loads and boundary condition are applied to find out the stresses in each stations. If stresses are below the ultimate strength value then design is valid. The spar beam with the cutout dimensions are shown in Table 6.1.

The spar beam with the cutout dimensions are shown in Table 6.1 is geometrically represented in Fig.6.2. CATIA V5 software is used to draw this model. This model is used for further analysis in NASTRAN and PATRAN software.

Figure 6.2: Geometric model  
(B= length, A=breadth)

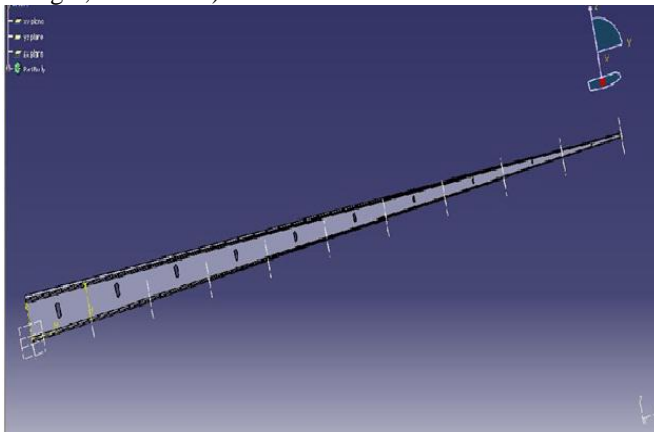


Table 6.1: Cutout dimensions

| Station | B (mm) | A (mm) |
|---------|--------|--------|
| 0-1     | 40     | 20     |
| 1-2     | 36     | 18     |
| 2-3     | 32     | 16     |
| 3-4     | 28     | 14     |
| 4-5     | 24     | 12     |
| 5-6     | 20     | 10     |
| 6-7     | 16     | 8      |
| 7-8     | 12     | 6      |
| 8-9     | 8      | 4      |
| 9-10    | 4      | 2      |

Model is meshed using 2D mesh having quad and tria elements. It contains 21325 elements and 26560 nodes. Software used for meshing is MSC/PATRAN. Mesh model for this case is shown in Fig.6.3

After applying loads (as shown in Fig 5.1) and boundary condition maximum stress occurs at top flange of spar beam near station 9-10 and minimum stress occurs at web of the spar beam near station 1-2. The stress distribution is shown in Fig.6.4

Figure 6.3: Mesh model

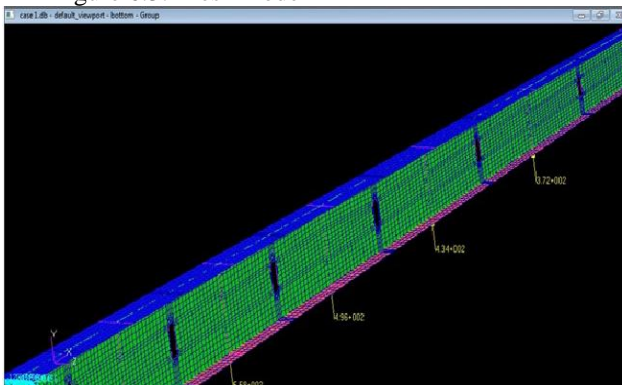
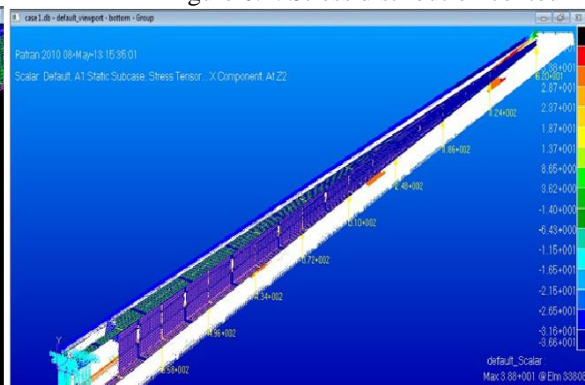


Figure 6.4: Stress distribution contour



Stress values at each station along the length of the spar beam are shown in Table 6.2

Table 6.2: Stress distribution along the length of spar beam

| Station | Top flange(stresses) MPa | Station     | Top flange(stresses) MPa |
|---------|--------------------------|-------------|--------------------------|
| 0-1     | 286.4-291.3              | 196.2-299.2 | 288.4-291.3              |
| 1-2     | 288.4-290.3              | 233.4-309.9 | 288.4-292.3              |
| 2-3     | 289.3-293.3              | 234.4-309.9 | 289.3-292.3              |
| 3-4     | 290.3-293.3              | 229.5-310.9 | 290.3-293.3              |
| 4-5     | 292.3-295.2              | 235.4-309.9 | 292.3-294.3              |
| 5-6     | 294.3-292.2              | 244.2-315.4 | 292.3-296.2              |
| 6-7     | 297.2-300.1              | 239.3-323.7 | 306.1-309.1              |
| 7-8     | 307.1-310.9              | 258.1-335.5 | 334.5-342.3              |
| 8-9     | 324.7-334.5              | 263.8-338.4 | 336.4-341.3              |
| 9-10    | 336.4-343.3              |             |                          |

Maximum stress occurs near the station 9-10, its value is 343.3 MPa and minimum stress occurs near the station 1-2, its value is 196.2 MPa. Observing stresses all are within the ultimate strength hence this case is valid for design of spar beam.

**CASE 2:**

The spar beam with the cutout dimensions are shown in Table 6.3 The spar beam with the cutout dimensions are shown in Table 6.3 is geometrically represented in Fig.6.5. CATIA V5 software is used to draw this model.

Figure 6.5: Geometric model

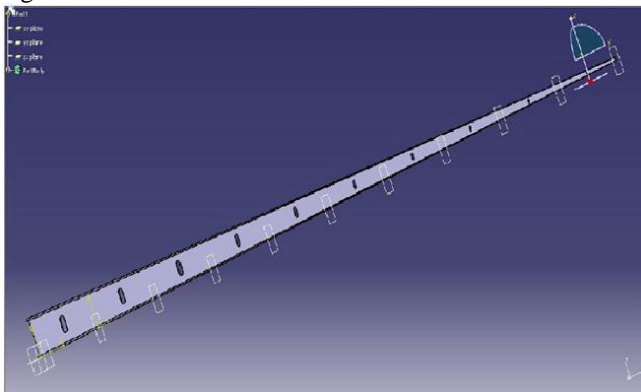


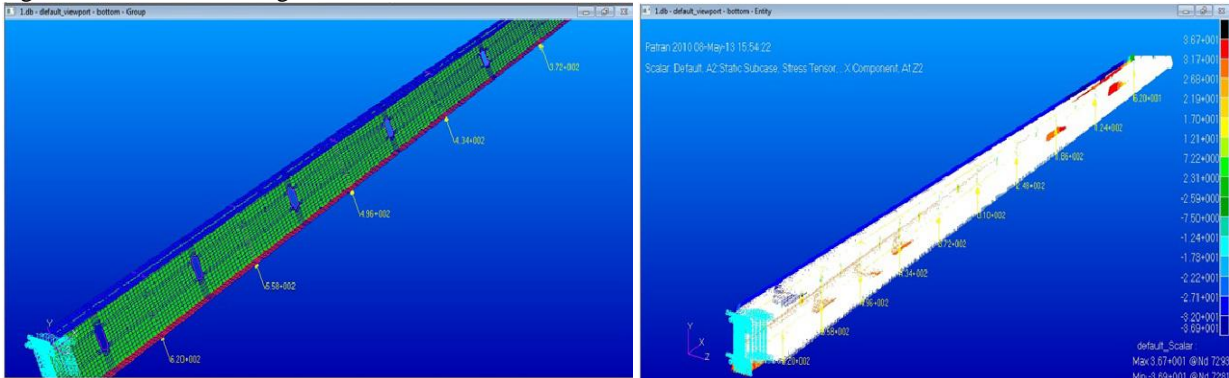
Table 6.3: Cutout dimensions (B= length, A=breadth)

| Station | B (mm) | A (mm) |
|---------|--------|--------|
| 0-1     | 44     | 25     |
| 1-2     | 40     | 22.5   |
| 2-3     | 35     | 20     |
| 3-4     | 31     | 17.5   |
| 4-5     | 26     | 15     |
| 5-6     | 22.4   | 12.5   |
| 6-7     | 18     | 10.5   |
| 7-8     | 13     | 8      |
| 8-9     | 9      | 6      |
| 9-10    | 4.5    | 3      |

Model is meshed using 2D mesh having quad and tria elements. It contains 20902 elements and 26139 nodes. Software used for meshing is MSC/PATRAN. Mesh model for this case is shown in Fig.6.6.

The stress distribution is shown in Fig.6.7. Maximum stress occurs at bottom flange of spar beam near station 9-10 and minimum stress occurs at web of the spar beam near station 1-2.

Figure 6.6: Mesh model Figure 6.7: Stress distribution contour



Stress values at each station along the length of the spar beam are shown in Table 6.4

Table 6.4: Stress distribution along the length of spar beam

| Station | Top flange(stresses) MPa | Web (stress) MPa | Bottom flange(stresses) MPa |
|---------|--------------------------|------------------|-----------------------------|
| 0-1     | 289.3-292.3              | 202.1-305.1      | 289.3-291.3                 |
| 1-2     | 290.3-293.3              | 208.9-307.1      | 288.4-292.2                 |
| 2-3     | 290.3-294.3              | 217.7-303.1      | 291.3-293.3                 |
| 3-4     | 291.3-295.2              | 216.8-303.8      | 290.3-294.3                 |
| 4-5     | 292.3-296.2              | 229.9-302.9      | 291.3-295.2                 |
| 5-6     | 295.2-298.2              | 236.4-312.9      | 293.3-296.2                 |
| 6-7     | 297.2-301.1              | 243.2-317.8      | 297.2-300.1                 |
| 7-8     | 306.1-310.9              | 257.1-326.6      | 307.1-309.9                 |
| 8-9     | 333.5-335.5              | 261.9-341.3      | 332.5-334.5                 |
| 9-10    | 336.4-342.3              | 266.8-344.3      | 338.4-345.3                 |

Maximum stress occurs near the station 9-10, its value is 345.3 MPa and minimum stress occurs near the station 1-2, its value is 202.1 MPa. Observing stresses all are within the ultimate strength hence this case is valid for design of spar beam.

#### V. CONCLUSIONS

The present work illustrates how stress analysis can be used in the design of aircraft components. The technology has been successfully used in an industrial environment within short span of time and it is proved to be able to provide efficient component designs as this eliminates actual prototypes for analysis. However for better results the detailed test has to be carried out. The objective of the present study is to investigate the stresses acting in the spar beam of the vertical tail with and without cutouts. The analysis shows that increase in the air drag load increases the maximum stress. The differential air drag load is applied between the range 62 to 620 Kg which varies non-linearly. In this study, the effects of stresses in aircraft spar beam structure with and without cutouts are determined. A typical spar beam with and without cutout of standard configurations is loaded and analysed. Parametric studies were done to examine the effects of different cutouts. Based on the results of finite element predictions and by the calculations of the stress analysis approach, it is apparent that the air drag load has more effect on the top and bottom flange. From the load cases the maximum stress is compared with yield stress and ultimate stress of 2024 T351 aluminum alloy. The obtained magnitudes of maximum stress are less than the yield stress and ultimate stress so we conclude that material is in elastic limit and not yet started yielding.

In the project work the following is carried out,

1. The standard dimensioned spar beam is considered for the stress analysis.
2. Finite element approach is used for the stress analysis.
3. Typical air drag load of non linear type is considered for loading.
4. The effects of the air drag load and the rudder loads have been studied by means of models of increasing geometrical complexity of vertical tail segment.
5. Higher mesh density with good quality of elements is used for finite element modeling.
6. Loads and boundary conditions are accurately simulated to obtain the realistic loading conditions.
7. Maximum stress is observed at bottom flange segment near cutout edges.

#### REFERENCES

- [1] Megson T.H.G., Aircraft structures for engineering students, 4th Edition. Publisher, Butterworth-Heinemann, 2007.
- [2] Best P.E., Analysis and comparison of I beam and solid wing spars. Publisher, Wolleat B., EAA 1953.
- [3] Hovell P.B., The stress distribution in a swept-back box-beam with perpendicular ribs, LONDON her majesty's stationery's office, Reports and memoranda No. 2837 December, 1950.
- [4] Arendsen P., Multi disciplinary wing optimisation, National Aerospace Laboratory NLR-TR-2001-557.
- [5] Garfinkle M., Pastore C., Aerospace: Smart spars intrinsically-smart composite structures. www.insensys.com/aerospace Published date 25.09.2009.
- [6] Grbovic A., and Rasuo B., FEM based fatigue crack growth predictions for spar of light aircraft under variable amplitude loading. <http://dx.doi.org/10.1016/j.engfailanal>. Accepted 25 July 2012.
- [7] Ajaj R.M., Saavedra Flores E.I., Friswell M.I., Allegri G., Woods B.K.S., Isikveren A.T., and Dettmer W.G., The Zigzag wing box for a span morphing wing, Aerospace Science and Technology, 28, 364-375, www.elsevier.com/locate/aescte 2013.
- [8] Swamy A.R.K., and Suresh Kumar R., Modelling and element analysis. Publisher, Subhas stores, avenue road, Bangalore 2011.