Load Calculation through CFD Examination and Basic Investigation of Unmanned Test Rocket

Dhilip Kumar U C¹, Mohammed Imran², Dr. Mohamed Haneef³
¹M. Tech Student, Department of Mechanical Engineering, Gousia College of Engineering, Ramanagara, INDIA
²Assistant Professor, Department of Mechanical Engineering, Gousia College of Engineering, Ramanagara, INDIA
³Principal & Professor, Department of Mechanical Engineering Gousia College of Engineering, Ramanagara, INDIA

ABSTRACT
In this work an unmanned test rocket structure will be developed for the required dimensions using modelling software using solid works. The each section will be theoretically checked for structural strength after analysing the aerodynamic loads. Later the modelling and meshing will be carried out using ANSYS. The modelled structure will be optimized using Finite element software's and suggestions for the proper geometry will be carried out based on the results.

Keywords--- F E Modelling, aerodynamics load, meshing

I. INTRODUCTION

A Manned space exploration has had a large impact on our society. Space has always been a mysterious unknown to man, and being a curious species, man has always wanted to explore space. The first Satellite ever sent into space was launched on Oct.4, 1957, named Sputnik. A cabin fire broke out in 1967 during the launch test of the Apollo 1 mission claiming the first lives of astronauts, and raising alarm for safety issues. After the incident there was a jump in unmanned spacecraft such as satellites, orbiters, and probes, soon realizing the potential and the impact that they could have on the scientific community. Unmanned space flight costs around $420 Million per launch, almost half that of the traditional manned space exploration.

II. METHODOLOGY

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations Conservation of mass, Conservation of mass, Conservation of momentum, and Conservation of energy, Conservation of species and Effects of body forces. CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition. ANSYS CFD solvers are based on the finite volume method –Domain is discretised into a finite set of control volumes –General conservation (transport) equations for mass, momentum, energy, species. Modelling of the individual components is drawn by using solid edge software. In Solid edge software 3D modelling can be done easily. By using hyper mesh software meshing is done to increase accuracy of the results. The components are checked for structural safety using two methods by Analytical method using ANSYS and the Theatrical method using design data handbook.

III. PRIOR APPROACH

Rockets are the present day terminology for superfast data transportation. Since rockets move at high speeds, it is always better to check the structural safety of the problem. Since they move against aerodynamic pressure loads, computational fluid dynamic analysis is important to find the pressure loads acting on the structure. Unmanned vehicle design and checking the safety through finite element structural analysis is the main definition of the problem. The software system used for creating the unmanned craft by victimization ANSYS, Hyper mesh, Solid edge etc. you may have a much better recognition with expertise in ANSYS. COSMOS is additionally FEA software system and restricted analysis will be performed. But again, for lots of routine analysis between ANSYS by meshing with HyperMesh and Solid works Cosmos. The ANSYS/HyperMesh answer tends to be favoured quite Cosmos, particularly for a lot of substantial and rigorous analysis. ANSYS has more sorts of solutions and then will handle a larger number of sorts of issues, therefore its larger attractiveness. This may provide larger
potential for employment in rigorous FEA work victimization ANSYS than would Cosmos. Model will be foreign and exported among software system’s accurately with software interface. In ProE, surface modelling and model change will cut back modify and pure mathematics clean adds HyperMesh. In HyperMesh, first we have a tendency to classify and manage model surface, then check surface edges, and so mesh mechanically. We have a tendency to should check part quality to make sure undefeated calculation in ANSYS. Finally solve by ANSYS. Operating method the issues and answer is planned.

IV. OUR APPROACH

Unmanned vehicle design and checking the safety through finite element structural analysis is the main definition of the problem. The objectives include Aerodynamic load estimation through CFD and the Theoretical check of all the members for structural safety and Finite element analysis for final assembly.

Analyses and Results

Fig-1 Assembled view the Rocket Structure

Fig-1 shows the Assembled view of rocket structure has been built using Solid Edge version 19 for the given dimensions. The rocket structure has 5 parts (nose, nose cone, body, tail end and blades). 3 blades are arranged in circumferential fashion.

Fig-2 Meshed view of the rocket structure

Fig-3 Overall Stress in the assembly model

Fig 3 shows the maximum stress distribution is 148.587N/mm².

Fig: 4 Pressure Plots

The figure shows pressure plot using CFD analysis. The red region shows highest pressure built up region (Maximum pressure: 0.26Mpa). Since the pressure is gauge pressure, for final pressure atmospheric pressure will be added.

Fig:-5 Vector plot

Fig:-6 Velocity at the nose part from CFD analysis

Figure 6 shows velocity at the nose region is around 179.658m/sec shown with red colour arrows. The flow is observed to be laminar as no interference of vectors.
The results show closeness of finite element solution and theoretical solutions. Small variations of results can be attributed to stress concentration effect, mesh quality and load distribution etc.

V. CONCLUSIONS

The rocket model has been built as per the specifications. The geometry model has been analyzed using Computational fluid software Ansys/Flotran to find the pressure acting on the outer surface due to velocity of 350 KMPH. The aerodynamic loads are estimated from the cfd analysis. The pressure value has been taken to find the structural safety of the rocket structure for structural safety as the thrust force will not create much problem in the structure.

REFERENCES

[8] Luiza Favato, " Linear Static and Dynamic analysis of Rocket Engine testing Bench Structure using the Finite Element Method", IJSN: 2248-9622, iSSUE4., VOL5, APRIL 2015 PP. 70-77

<table>
<thead>
<tr>
<th>Component</th>
<th>Theoretical</th>
<th>Finite element Solution</th>
<th>Percentage of Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nose</td>
<td>110.52</td>
<td>112.2</td>
<td>1.5</td>
</tr>
<tr>
<td>Nose Cone</td>
<td>12.71</td>
<td>11.25</td>
<td>11.5</td>
</tr>
<tr>
<td>Body</td>
<td>42.46</td>
<td>44.86</td>
<td>5.3</td>
</tr>
<tr>
<td>Tail</td>
<td>6.6</td>
<td>8</td>
<td>17.5</td>
</tr>
<tr>
<td>Blade or Fin</td>
<td>188.7</td>
<td>208.462</td>
<td>9.5</td>
</tr>
</tbody>
</table>

Table: 1 Comparison of the results