

Modification of A57-Strategic Bomber Aircraft

Hariramakrishnan.L^{#1}, David.V^{*2}, Rishi Kumar.S^{*2}

¹Assistant Professor, ²UG Scholar

Department of Aeronautical Engineering

SNS College of Technology, Coimbatore, India

¹harikrishnan304@gmail.com, ²davidraj9970@gmail.com, ²rishikumar96.rk@gmail.com

Abstract-A57–Strategic bomber aircraft combines of unsymmetrical airfoil in cross section as well as with NACA 23012 1935 inverted airfoil to improve the performance of the vehicle .Compound delta wing is embedded to decrease shockwaves and reach transonic to supersonic (Mach no: 2.5). Five afterburning turbojet engines were installed between twin inward-tilted fins, providing up to 130,000kg of thrust. After a brief background on the development of the , A57 – strategic bomber aircraft And previous research ventures, we including forward swept wing (30degrees) to improve additional lift forces and also to reduce drag forces. In addition to that new install of forward swept wing to reduce wing tip vortices. Future work involves new inclusion of flaps in certain angle to permit additional lift force and also to easy take off at shorter distance itself. To attain specified work is designed by “CATIA” and analyzed by “CFD”.

Keywords- Forward swept wing, Take off, Unsymmetrical Airfoil, Twin inward-tilted fins.

I. INTRODUCTION

The A-57 strategic bomber concept has landing on the water (or ice) retractable ski was planned. For take-off from land bases special droppable 'cart' was designed. All services of the aircraft were routed to the upper surface, allowing service and refuelling at open sea. The flying boat concept allowed omitting huge, expensive concrete-paved airfield. Five afterburning turbojet engines were installed between twin inward-tilted fins, providing up to 130,000kg of trust. Wheel gear was missing completely.

II. DESIGN TOOL

Aerodynamic analysis of theoretical calculation of shock wave angle and interaction of shock wave are considered because the shock wave interaction between isolator inlets gives more separation and flow disturbance and shock wave creates more temperature. The positions of cowl and ramp length and ramp angle are determined and sketched. This sketch was made 3D model by using CATIA V5.

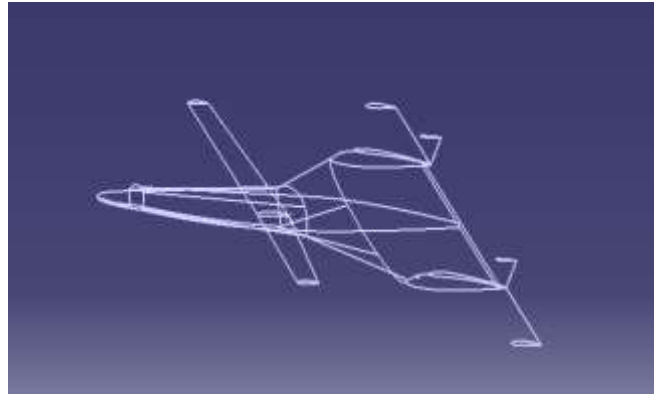


Fig. 1 2D View

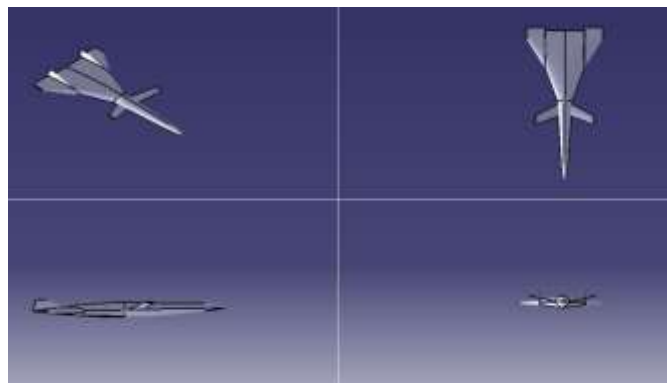


Fig.2 All Views of A -57

III. ANALYZING TOOL

The simulation process of the model scramjet inlet begins with creation of the geometry in the computational domain and generating the grid for the created computational domain geometry. The process of computational geometry creation and the grid generation is called as the pre-processing. The grid generated geometry is the imported in to the solver for the numerical simulation by utilizing various solver setting parameters and the simulated results are visualized using the setting in the solver itself or separate postprocessor is used for these purpose. The method of visualizing the results is called as the post processing.

IV. PROBLEM DEFINITIONS

The main objective of this study is to determine the optimum position of the forward swept wing at 30 degree. So that flow over the forward swept wing interacts with the flow over the main wing. Then at this most suitable position using NASTRAN analysis, the

maximum value of Coefficient of lift is obtained is easily found out.

For this purpose the entire work is divided into 3 different phases.

1. Selection of airfoils with respect to NACA series.
2. Selection of forward swept wing at an angle of 30 degree.
3. Determination of Coefficient of lift (CL) values at various angles of attack.

V.METHODOLOGY

The A-57 designed with catia-v5 design geometry was taken from the existing analysis. That final designed nozzle is shown in below with geometry.

TABLE I
DIMENSION OF A-57

| | |
|----------------|--------------------|
| Length | 71295m |
| Wing Span | 37000 m |
| Overall height | 7.5 m |
| Wing area | 755 m ² |
| Sweep angle | 30(degree) |

In this work we are using to design A-57 with two types. At first regular flow simulation in A 57 without plenum, plenum is a device which is used to stored high pressure or velocity of fluid in that typical drawing is drawn with CATIA v5 as shown in below sketch

The CATIA work is simple to generate part body sketch/model by importing pointed geometry from excel formatted data. First we have importing pointed data from excel sheet and connect that points by using lines and arc commands, complete the 2D sketch. After revolving the sketch to create a full part body by using revolving command.

In plenum model is also done by CATIA at same methodology as importing pointed data from excel sheet and connect that points by using lines and arc commands, complete the 2D sketch. Finally the plenum area is defined by our model to choose divergence cone. It will be created by using whole command.

VI. GRID GENERATION

The un-structured grid generation processes have been taken for this project with an ICEM CFD Tool. The geometry of inserted can be read into ICEM CFD using IGES format. ICEM-CFD is mesh generation software developed by ANSYS. ICEM-CFD is used to create and mesh the model. The main advantage of this software than any other meshing software packages is that, it can be used for any type of complex geometries. Tetrahedral, hexahedral, hybrid and Prism type of cells can be used in meshing the geometry.

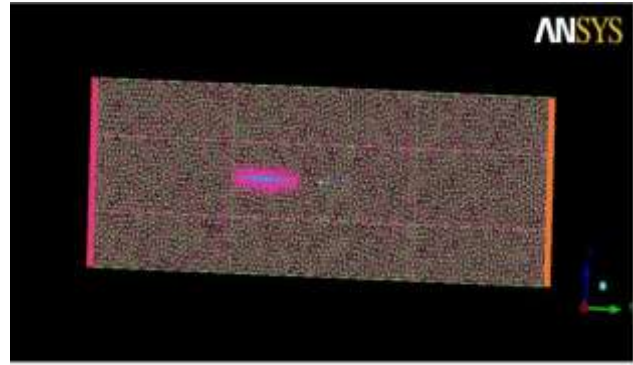


Fig.3 A 57 Mesh File

Grid generation is a discretization of part by converting equations. The most used equations are depends upon the problem descriptions. In our problem we are using TETRA meshing, the no of nodes and elements are 110878 and 651413 respectively. The above figure is tetra meshed view of A 57, in these tetra hedra grid nodes and elements are done normally.

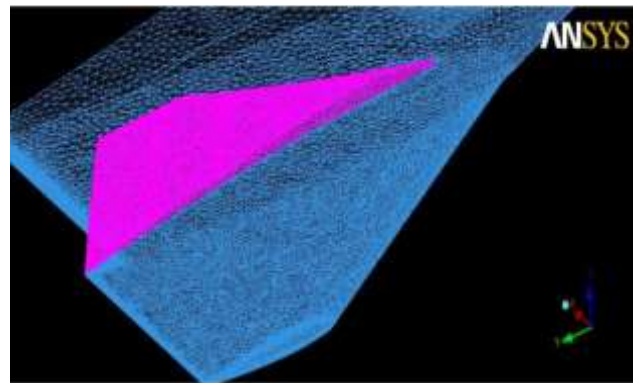


Fig.4 Meshing Over Fins

VII. PRE-PROCESS

The CFD method used in this project as well-known CFX solver v13.0. Its reliability has been demonstrated by a great number of aerospace and some other industrial applications.

The CFD methodology including

1. Specify the boundary condition
2. Specify the initial condition
3. Setup the CFD simulation
4. Perform and monitor the CFD simulation
5. Examine and process the CFD simulation
6. Report the findings

VIII. THE BOUNDARY CONDITION

The boundary conditions produce different solution for a given geometry and set of physical models. Hence, boundary condition determine to a large extent the characteristics of the solution obtain. Therefore, it is important to set boundary conditions that accurately reflect the real situation to allow obtained accurate result.

For the internal steady flow simulation, velocity inlet boundary condition is used for the nozzle inlet, which includes the flight Mach number, velocity and pressure. This value as initial condition of the flow simulations, the inlet flow speed set as supersonic with Pressure, Velocity, and Temperature are 100 kpa, 600 m/s and 120 k respectively. Pressure outlet boundary condition is set as the flow domain outlet with supersonic speed. We have received the nozzle outlet values are captured after taken this simulation. And the nozzle wall set as free slip wall boundary condition. These includes- Purpose of boundary condition, Inlet, Outlet, Wall

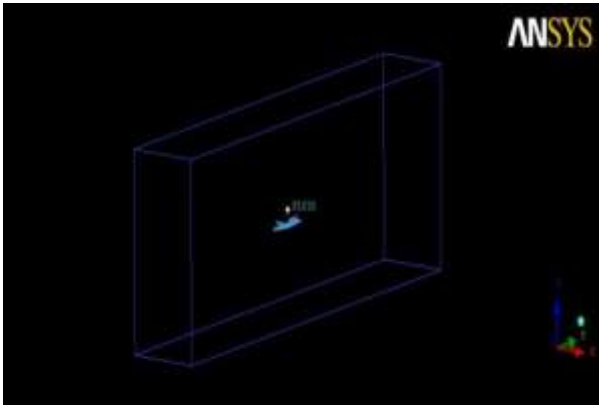


Fig.5 A-57 Domain with Boundary

IX. CFX-POST PROCESSING

We have to take two types of nozzle conditions are nozzle without plenum and nozzle with plenum for controlling thrust vectoring. The thrust vectoring simulations are taken to adding extra flow inlet at the divergence cone portion, which will affect the normal inlet flow to deflect opposite side from the plenum. The plenum introduces the flow to affect normal inlet flow, is a small circular section on the divergence cone.

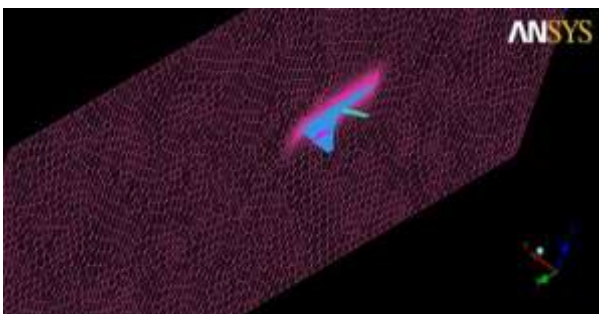


Fig.6 Flow over Divergence Cone

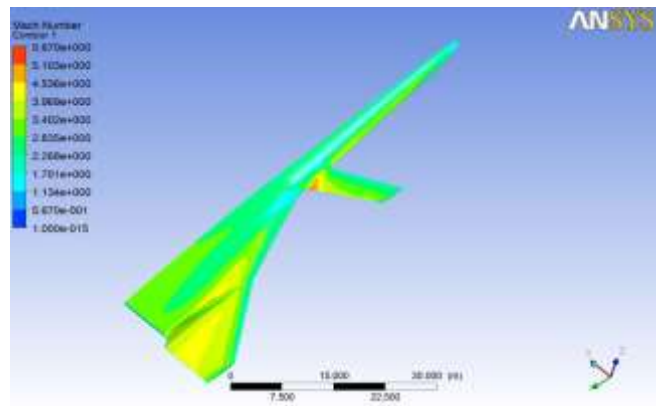


Fig.7 Analysis over Mach No.

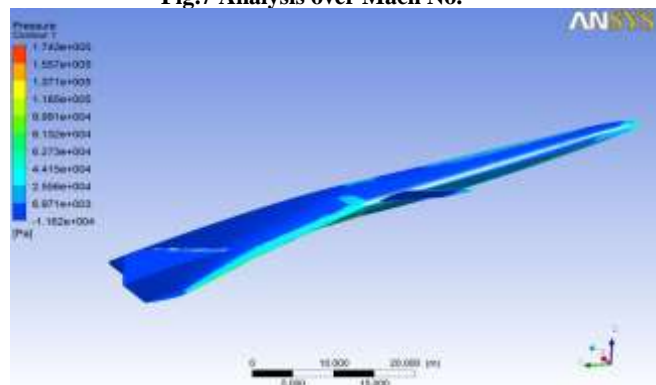


Fig.8 Analysis over Pressure

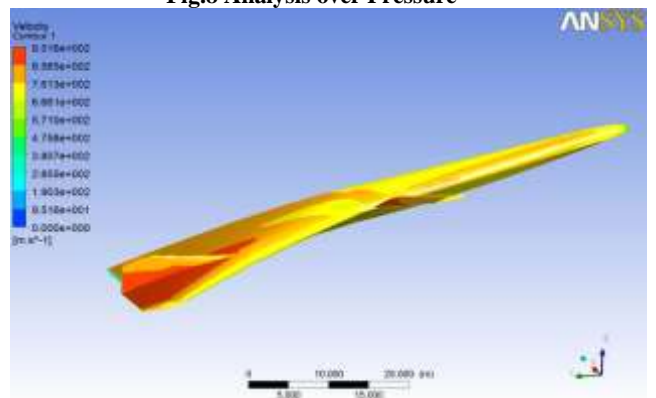


Fig.9 Analysis over Velocity

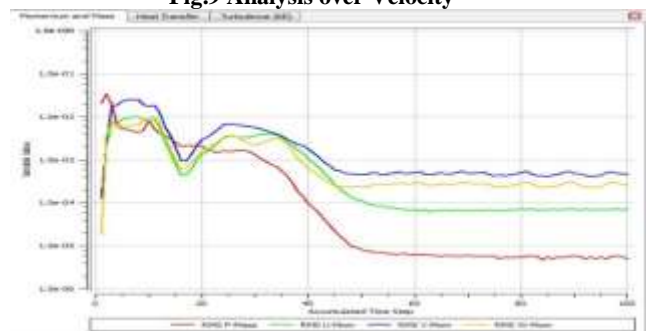


Fig.10 Momentum and Mass

| Without Using Sweep Wing | A.O.A | ACH *10 ⁶ | DRAG *10 ⁶ | IFT *10 ⁶ | L/D |
|--------------------------|-------|----------------------|-----------------------|----------------------|------|
| | 0 | 4.628 | 1.310 | 1.042 | 0.79 |
| | 4 | 4.569 | 1.702 | 4.699 | 2.76 |
| | 8 | 4.196 | 2.504 | 8.492 | 3.39 |
| | 12 | 4.087 | 3.949 | 1.253 | 3.17 |
| With Using Sweep Wing | 0 | 4.462 | 1.571 | 847.5 | 5.39 |
| | 4 | 4.503 | 1.922 | 5.233 | 2.75 |
| | 8 | 4.414 | 2.88 | 9.75 | 3.38 |
| | 12 | 4.47 | 4.55 | 1.46 | 3.21 |



Fig.13 Drag vs Lift



Fig.14 Drag vs Lift

X. RESULT

We studied the datas of A57 – strategic bomber aircraft and included forward swept wing for attaining maximum co efficient of lift. We experimented the data of forward swept wing and model of aircraft structure. We designed this model by using CATIA and analysed by CFD. The effect of forward swept wing at certain angle of attack such as 0,4,8,&12.we had achieved the required lift and reduced drag in our project.

REFERENCES

- R.L.BARTINI , "Bombardirovshiki" (The Bombers) by V.II'in and M.Levin, Moscow, 1997 [1]
- N. C. LAMBOURNE, D. W. BRYER and J. F. M. MAYBREY , "The Behaviour of the Leading-Edge Vortices over a Delta Wing Following a Sudden Change of Incidence", Aeronautical research council and memoranda, 1963. [2]
- A A PASHILKAR, "Flight Mechanics & Control Division", National Aerospace Laboratories. [3]
- Dr.SOFIRI JOAB-PETERSIDE, "Research Fellow Center for Advanced Social Science (CASS) Port Harcourt", Nigeria. [4]
- EVERT G.M. GEURTS, "Theoretical and Experimental Study of a Forward Swept Wing", Anbar journal for engineering sciences AJES-2010, Vol.3, No.2. [5]

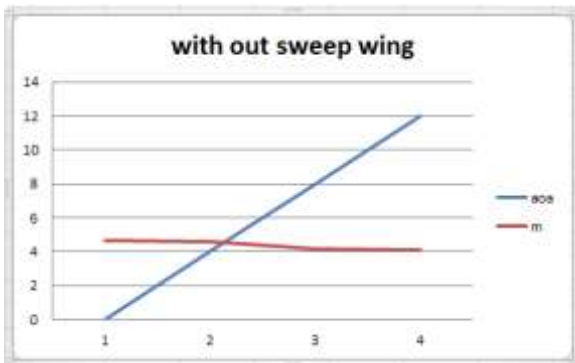


Fig.11 Angle Of Attack vs Mach No:



Fig.12 Angle of attack vs Mach no: