OPEN ACCESS

International Journal Of Modern Engineering Research (IJMER)

Computational Analysis of Multi Objective Conceptual Design of an Load Carry Aerial Vehicle

M. Gokulraj[1], V. Sankar[2]

[1, 2] (Department of Aeronautical Engineering, Nehru Institute of Engineering and Technology /India)

ABSTRACT: *Technology has the enemy of nature in one way .But sometimes technologies do come out as an exception to the above rule. Technological advancements have improvised them over time. A Load Carry Aerial vehicle (LCAV) helps overcome the labor problems. This project is based on Load Carry aerial vehicle which used to carry the various types of load (light load, heavy load and water bottle). The LCAV hover using Coanda effect principle. The LCAV works using Coanda effect principle with multi rotor and also its works on vertical takeoff and landing technique. Vertical Takeoff and Landing (VTOL) flight has been one of the main interests of both the civilian and military aviation communities throughout the history of flight. Today, several aircraft capable of achieving V/STOL flight exist. But it is entirely different from VTOL aircraft or helicopter or multi copters (tri copter, quarter copter or hexa copter). In LCAV there is no big rotor like on a helicopter and the flight is very stable and safe for the surrounding. In this project we shall carryout the designing of LCAV using CATIA. Then flow characteristic can be take using Computational fluid dynamics software ANSYS.*

Keywords: *Coanda effect, helicopter, LCAV, Technology stable*

I. INTRODUCTION

Henri Coanda was a Romanian inventor, aerodynamics pioneer and the designer and the builder of the world's first jet powered aircraft, the Coanda-1910 an evolutionary plane of the 20th century beginning. The effect presently named after Coanda Henri Coanda described 20 years later, when he made public his discoveries. In aeronautics, this effect is used today primarily in helicopters that have no tail rotors. The first design of a Coanda UAV was created in 1932, by the Romanian inventor Henri Marie Coanda. In his first patents related to Coanda effect applications, in order to generate the jet of fluid over the upper surface of the fuselage, he was using mainly other means than a rotor, i.e. a burner or a combustion chamber. But in a patent he obtained in 1935, he was enumerating the possibility to use also a centrifugal fan for supplying the necessary air flow. The agricultural aviation and military aviation are based on piloted aircraft. However, agricultural missions and military missions are quite dangerous, and require substantial piloting activities. In addition, ground markers are also exposed to great danger, since the aircrafts fly very close to ground with speeds close to stall speed. There is not only a need to carry out agricultural aviation or military aviation tasks without jeopardizing the lives of the human pilots, the current technology is mature enough to realize and operate an Remote control flying Load Carry Aerial Vehicle(LCAV).

II. MODEL DESIGN

The way geometry is represented strongly affects the optimization process. Axially symmetric bodies, and other more complex three dimensional shapes (wings, fuselages, etc.) can be represented by means of class function, shape function representation. The class function is used to define general class of geometries, while shape function is used to define specific shapes within the geometry class. The main idea is to decompose the basic shape into scalable elements corresponding to discrete components, by representing the shape function with a Bernstein polynomial. The Bernstein polynomial of order n is composed of the n +1 terms of the form.

$$S_{r,n}(\Upsilon) = K_{r,n}\Upsilon^{r}(1-\Upsilon)^{n-r} \qquad (1)$$

Usually geometry of cross section of typical Coanda-effect UAV is “elliptic-arc like”. Therefore the class function will be the function defining an elliptic arc. By adjusting the coefficient K in the class function representation one can choose the aspect ratio of an elliptic arc. Taking K to be equal to two, gives a circle. By scaling the coefficients in the component representation of a shape function, we can derive appropriate variations of the basic shape. We need to stress that the order of Bernstein polynomial, used for the function decomposition, is chosen freely. The higher order renders more localized variations of the basic shape possible.

Upon construction in CATIA it was evident this concept solution may not accommodate all of the components. A Balsa wood sub carriage was considered to hold some of the bulky components such as batteries, however it was not used as the hardware was attached to both the top and bottom of the frame, alleviating the need for an undercarriage. The final frame design is estimated to weigh the same as the first concept designed in CATIA, however through the use of larger materials its strength and hence robustness against damage was greatly increased.

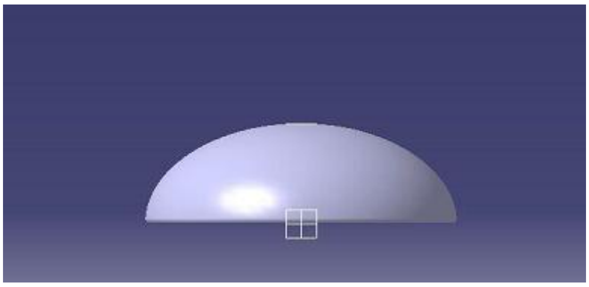

Fig 1. CATIA Three dimensional model

III. COMPUTATIONAL METHOD

Any CFD analysis can be divided into three broad categories namely, preprocessing, solver settings and post processing. ANSA AND ICEM is an aid for preprocessing of the problem in FLUENT. It is a software package designed to help analysts and designers build a mesh around the geometry under consideration. ANSA AND ICEM receives user input by means of its graphical user interface (GUI). The ANSA AND ICEM GUI makes the basic steps of building, meshing, and assigning zone types to amodel in a simpler and intuitive manner, and it is versatile enough to accommodate a wide range of modeling applications. The GUI of ANSA AND ICEM includes four major divisions which are geometry creation pad, meshing pad, specifying zone types and tools pad. Each of these is subdivided into various parts. Geometry creation in ANSA AND ICEM is done with the help of required commands from the geometry creation tool pad. The geometry creation tool pad contains command buttons that allows performing operations which include creating vertices, lines, faces, volumes etc. Meshing in ANSA AND ICEM can be done in various forms namely edge meshing, face meshing and volume meshing. Meshed edges, faces and volumes can be copied, moved, linked or disconnected from one another. To perform grading and meshing operation on the edges the following parameters can be specified edge(s) to which the grading specifications apply, grading schemes, mesh node spacing (number of intervals), and edge meshing options. Grading schemes include successive ratio, first length, last length, first last ratio, last first ratio, exponent, bi – exponent and bell shaped. Double sided grading can also be performed. The interval size, count or the shortest length percentage can be specified for starting the mesh.

The grid for the model was generated using the ANSA AND ICEM software. Unstructured grid was used for the analysis of the flow field around the model. The details of the grids are discussed in the following sections.

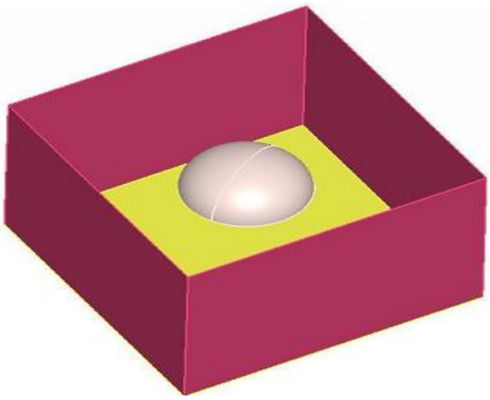

Fig 2 Domain in isometric view

The solution method in FLUENT can be broadly divided into three parts namely: Pre – processing, Solver and Post processing. Pre – processing of the problem was done in ANSA and ICEM as discussed in detail in the preceding sections. Once the problem is meshed and the boundary conditions are specified the meshed geometry is then imported as a 'mesh file' into FLUENT.

CFD uses a series of cells (previously referred to as control volumes), elements and nodes that combined form the so called mesh. It is at each of these node locations, that CFD calculates the fundamental equations of fluid dynamics, as mentioned in the previous section, the shape of the cells greatly impacts the accuracy of the solution due to discretisation errors, and therefore the meshing stage is one of the most crucial stages in the problem simulation.

CFD uses a series of cells (previously referred to as control volumes), elements and nodes that combined form the so called mesh. It is at each of these node locations, that CFD calculates the fundamental equations of fluid dynamics, as mentioned in the previous section, the shape of the cells greatly impacts the accuracy of the solution due to discretisation errors, and therefore the meshing stage is one of the most crucial stages in the problem simulation.

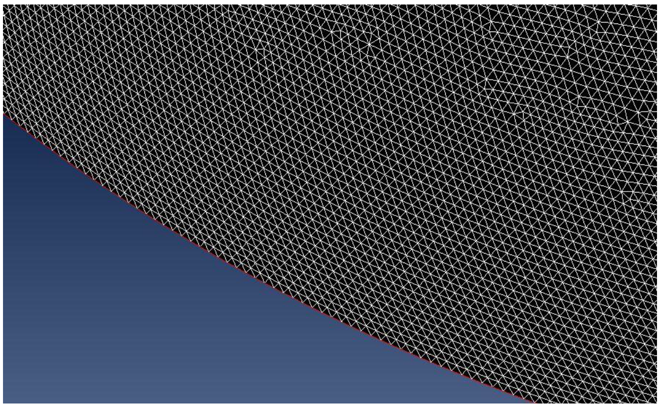

Fig 3 Grid cell distributions at the bottom of model

The tetra mesh can use different meshing algorithms to fill the volume with tetrahedral element and to generate a triangle surface mesh on the object surface. This can define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved element quality. The Tetra mesh incorporates a powerful smoothing algorithm. As well as tools for local manual mesh refinement and fine.

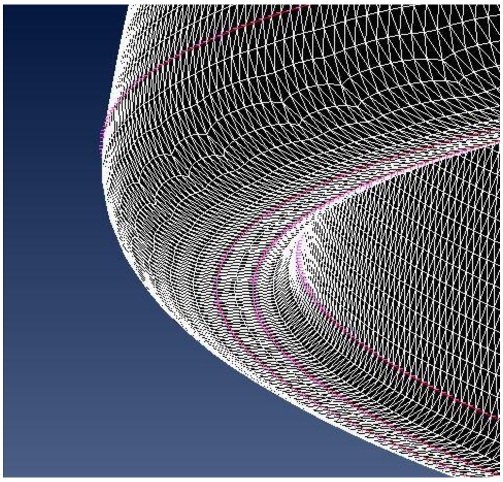

Fig 4 Surface mesh containing tetrahedral mesh only

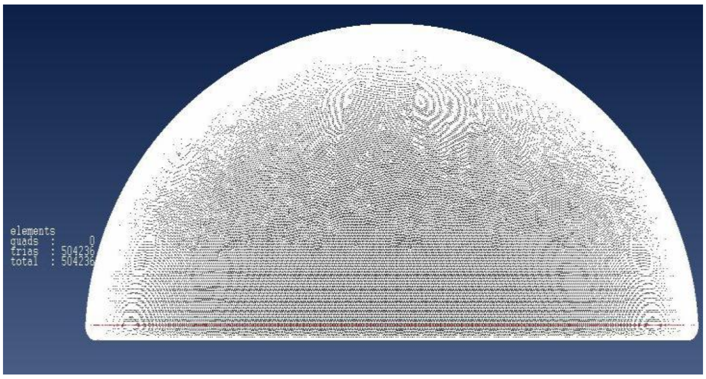

Fig 5 Grid cell distributions in the vicinity of body

Existing triangle surface mesh for all or part of the geometry can be specified as input to the tetra mesh. The final mesh will then be consistent with and connected to the existing mesh nodes

The simulation setup can be stored as a case and data file. The auto save option can be used to save the results of the iterations from step to step. The case file includes the information for the grid, the boundary conditions and the solver settings. The data file stores information about the data in each node of the cells. The contour plots, vector plots and the surface data plots etc. of pressure, velocity and density etc. can be checked during the solution process and at convergence. These plots can be saved as image files and the data from surface plots can be written on to a file and plotted. Points, lines, rakes and planes can be created in the flow domain to analyze the properties at the desired locations. FLUENT offers a very good range of post processing options which can be used to analyze the computational data, compare computational results with experimental results as desired. For the present analysis points, lines and planes were used to analyze the properties at the desired locations in the flow domain. Multi grid option was also used to reduce the computational time.Computations were performed to understand the flow field around a scaled down model of a LCAV. Computations using the commercially available software FLUENT was carried out for three dimensional fully developed flows. A validation test was performed by referring to the model at 15 m/s velocity. The details of the computations are discussed in this chapter

IV. COMPUTATIONAL RESULTS

Three dimensional computational simulations was carried out for studying the flow field around a LCAV at 15 m/s. The results of computations performed are discussed in detail in the following sections. The computations were performed on a work station Core 2 Duo processor with a bus speed of 4.1GHz and a RAM of 8GB. Typical time taken for inviscid problem was 10 iterations per hour and for viscous three dimensional problems were 15 iterations per hour. For all the cases, an average 3000 iterations were performed until desired convergence is obtained. The inviscid analysis was performed for comparing the results obtained for viscous flows.

For the given configuration and inlet conditions, air jet is able to stay attached on upper surface, and the presence of Coanda effect is clearly visible. We are able to observe entrainment of the surrounding air into the jet which causes gradual increase of air jet velocity as the jet is approaching the upper's surface edge. At the edge point, velocity has an 25% increase compared to the inlet and reaches 15 m/s. This is followed by pressure decreasing on the upper surface. Suction force is present, and is gradually increasing towards the edge of the upper surface. This produces upward net force, which is able to keep the UAV with described geometry and flow conditions in hovering flight even with the mass around 1.5 kg. Varying the inlet velocity around the circumference produced asymmetric flow picture. Asymmetry is most clearly visible by observing streamline pattern. Less surrounding air is entrained; velocity at the upper surface edge reaches smaller value. Pressure is unevenly distributed along the upper surface and lateral force is present which enables forward motion.

Due to strong curvature at the edge of the upper surface

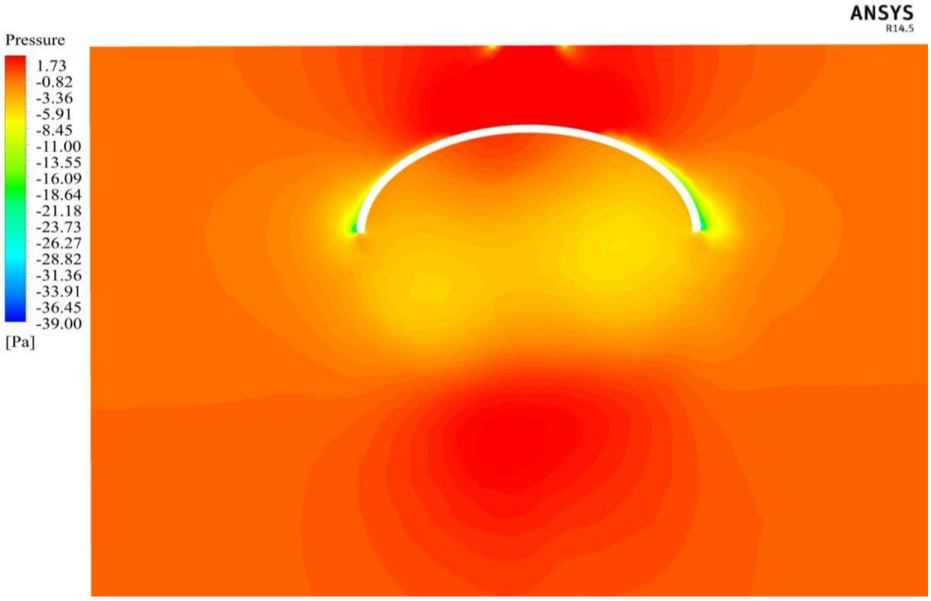

Fig 6 Mach contour vicinity of the model

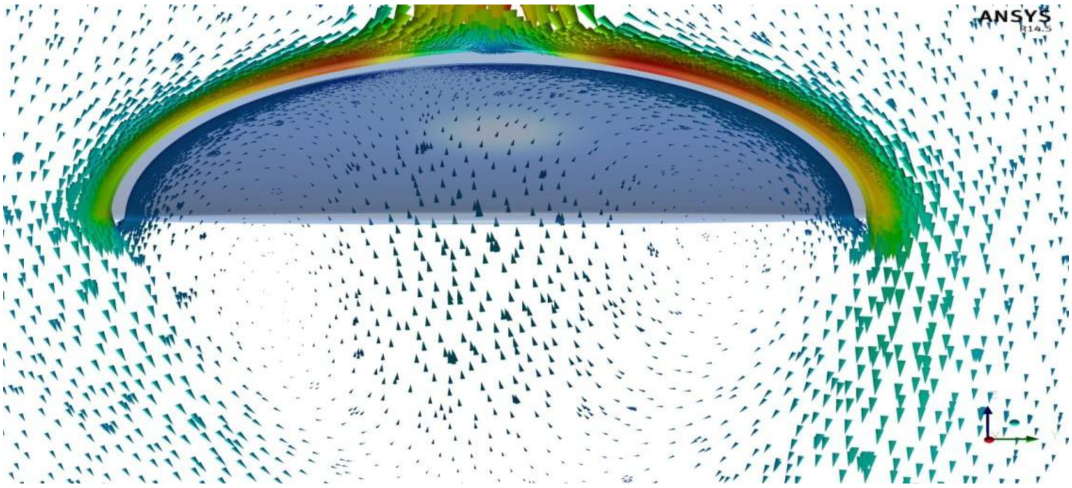

Fig 7 Velocity vector 3D grid for a jet flow over model

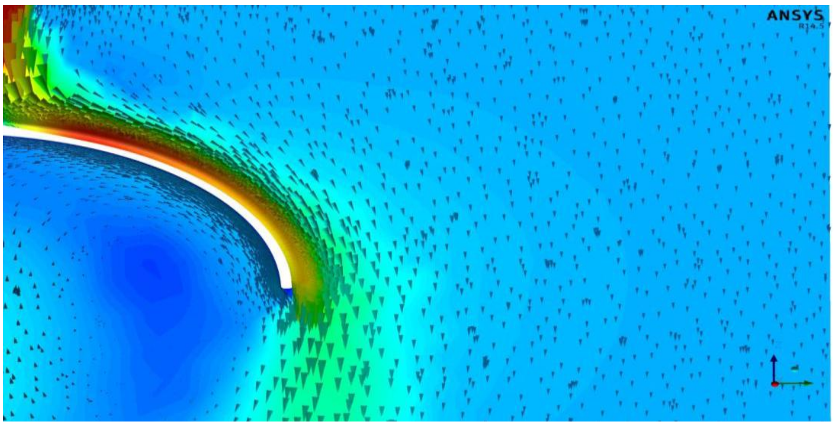

Fig 8 Pressure contours the streamlines flow, and wake

V. CONCLUSION

Computational studies were carried out to get an understanding of the flow field LCAV. Three dimensional simulations of the flow field using FLUENT were performed. SA turbulent model was adopted to capture the flow field. Computations were validated through a simulation of flow field around the similar geometry at 15 m/s. After a good agreement with reported results, simulation of the present case was carried out

ACKNOWLEDGEMENTS

We first thank our 'GOD', the supreme power for giving us a good knowledge and our parents for making us study in a renowned college. We owe a great many thanks to my colleagues and friends for their help and encouragement.

REFERENCES

[1]. Coanda, H. (1936). Device for Deflecting a Stream of Elastic Fluid Projected into an Elastic Fluid. US Patent Office, US Patent # 2,052,869.
[2]. Collins, R. (2003). Aerial Flying Device. UK Patent Office, Patent # GB 2,387,158.
[3]. Geoffrey Hatton (2010), Thrust generating apparatus, US Patent Office, Patent # US 7857256 B2
[4]. Arvin Shmilovich, Yoram Yadlin, Robert D. Gregg, III, Roger W. Clark (2010), Systems `and methods for control of engine exhaust flow, US Patent Office, US # 7823840 B2.