Aerodynamic Analysis of Forward Swept Wing Using Prandtl-D Wing Concept

Srinath R [1], Sahana D S[2]

[1]*Assistant Professor, Mangalore Institute of Technology and Engineering, Moodabidri-574225, India*
[2]*Assistant Professor, Mangalore Institute of Technology and Engineering, Moodabidri-574225, India*

ABSTRACT

This project is about the implementation of PRANDTL-D WING to the forward swept wing and Aero dynamical analysis of the new wing configuration obtained. Since the forward swept wings have lesser aerodynamic performance characteristics and greater manoeuvrability at transonic and supersonic Mach ranges than that of the backward swept wings, if the new configuration can boost up the performance characteristics of forward swept wings then the current scenario will be changed. A series of forward swept wing configurations (of varying forward swept angle, twist angle, twist angle position along the wing span and direction of twist) will be will be modelled in the CATIA-V5 and supersonic flow simulations will be run in the ANSYS software and the configuration with best aerodynamic & structural performance characteristics will be made into physical model . The results obtained will be compared with that of existing forward swept wing configurations and backward swept wing configuration

Keywords — *CATIA V-5, ANSYS, PRANDTL-D WING, Forward sweep*

I. INTRODUCTION

A. Forward swept wings

The persistent problems with the backward swept wings (tip stall conditions and lesser maneuverability) seemed to be eliminated by reverse sweeping the wings i.e., by sweeping the wings forward. By doing so, the flow is made to take place from wing tips towards the root. And hence tip stall problem is alleviated.

Since the flow takes place from tips to root, stall tends to occur in the region after the root toward the stabilizers. This keeps the separated air well away from the ailerons and pilots have full authority over them even beyond the stall. This tendency of stall tending to happen at rear part induces a high lift leading to higher pitching moment co-efficient. Higher the pitching moment, lesser is the stability and hence high manoeuvrability.

B. Prandtl-D Wing

NASA's Armstrong Flight Research Center engineers in Edwards, California, are working on an increasingly complex wing called the Preliminary Research Aerodynamic Design to Lower Drag, or Prandtl-D wing.

This features a new method for determining the shape of the wing with a twist that could lead to an 11-percent reduction in drag. The concept may also lead to significantly enhanced controllability that could eliminate the need for a vertical tail and potentially to new aircraft designs.

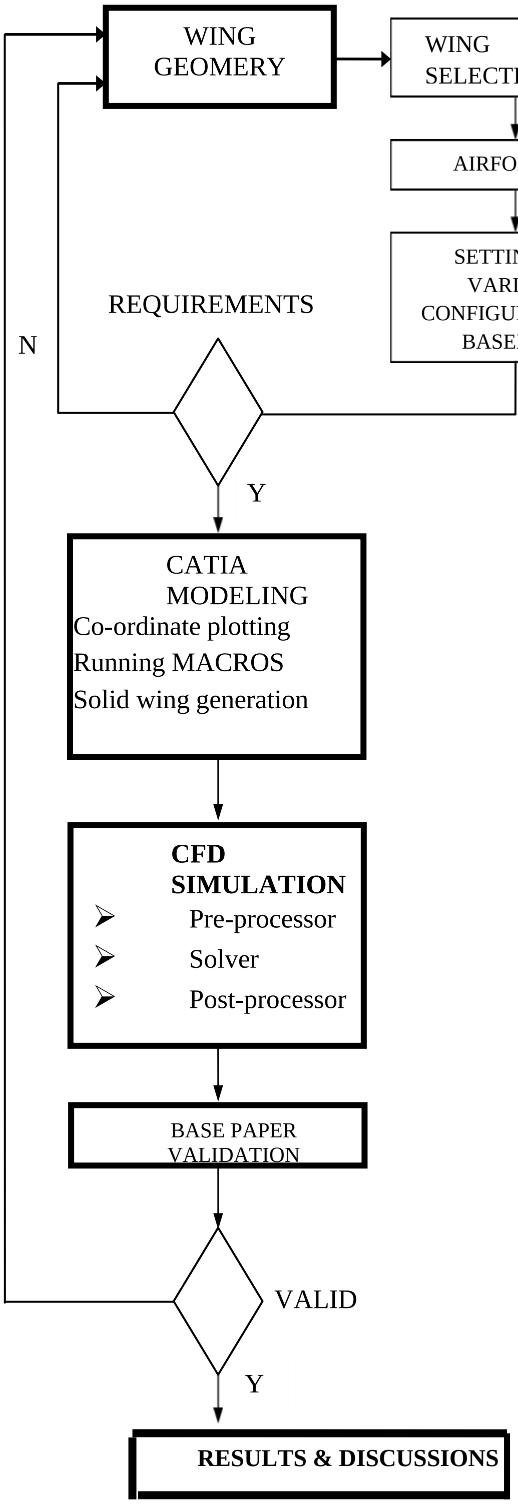
Flight data from the first two Prandtl-D vehicles validated the use of twist to tailor the lift distribution across the aircraft's wing – bell shaped rather than the traditional elliptical shape – leads to more efficient flight. In fact, engineers estimate future aircraft could see more than a 30 percent increase in fuel economy by using the new methods of wing design and eliminating the weight of the modern aircraft tail and its flight control surfaces.

In that regard, Prandtl-D research also borrows from how birds fly. Birds turn and bank without vertical tails that are required for such maneuvers on traditional aircraft, but not on the Prandtl aircraft

II. IMPLEMENTATION OF THE CONCEPT

- ➢ Employing a span-wise twist in the forward swept wing of a referral configuration with supercritical airfoil section and analyzing the results in a CFD tool.
- ➢ To analyse whether the aerodynamic characteristics of the forward swept wing are enhanced or not, on employing the span-wise twist as in the PRANDTL –D wing concept.

III. METHODOLOGY FLOW CHART



IV. WING GEOMETRY FORMULATION

The wing geometry formulation is a critical step in the process as the testing of the wrong wing geometry will not yield the required results. The steps followed in the formulation of the wing geometry are given below step by step.

B. Base wing configuration selection

The base wing configuration was taken as in the reference paper as it gave us the opportunity to compare a normal forward swept wing with the one that had a span wise geometric twist (PRANDTL-D WING CONCEPT) and validate our design and simulation process at the same time.

The specifications of the base wing configuration along with the image of the model are given below.

Table I. Model Geometry Specifications

Wing Plan form Area (sq. in.)	25,725
Root Chord (in.)	6,125
Tip Chord (in.)	2.45
Taper Ratio	0.40
Wing Semi-Span (in.)	7.58
Sweep Angle of Quarter Chord (degrees)	+45, -45
Aspect Ratio	3.34
Mean Aerodynamic Chord (in.)	5.62

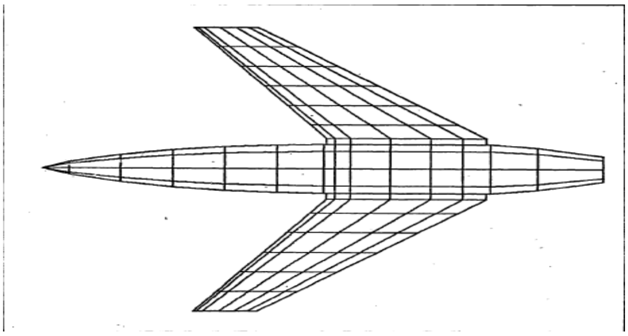


Fig 1. Reference forward swept wing geometry

TABLE II. Operating conditions

Operational Altitude	9000 meters (30000ft)
Ambient Temperature	229.59 K
Ambient Pressure	3.08E4 Pa
Airspeed	340.3 m/s
Dynamic viscosity	1.493E-5 Ns/m
Density	0.4671 Kg/m^3
Acceleration due to gravity	9.779 m/s^2

The application of the PRANDTL-D wing concept along with anhedral concept to a wing involves giving it a span wise geometric twist.

To design various configurations of the FSW, the position of the twist was fixed at the tip chord. The only parameter that was varied was the twist angle.

The twist was given at the tip chord and twist angles of +2, +5, -2 and -5 were given to the base wing configuration to obtain various models for testing. Along with this the base wing configuration, with zero twist was modelled for testing.

TABLE III. Configured Wing Models

WING MODEL	**TWIST ANGLE**
MODEL 1	0 DEGREES
MODEL 2	+2 DEGREES (counter-clockwise)
MODEL 3	+5 DEGREES (counter-clockwise)
MODEL 4	-2 DEGREES (clockwise)
MODEL 5	-5 DEGREES (clockwise)

V. MODELLING IN CATIA

The CAD models of the wing configurations were done in CATIA V5. The coordinates for the selected airfoil, or the required chord length were obtained from online airfoil database and airfoil plotter.

These coordinates were initialized into CATIA using Microsoft Excel and by running macros. This imported the coordinates of the selected airfoil onto the plane of our choice in CATIA. The process was repeated for root chord as well

Fig 2. Twist angle = 0 degrees

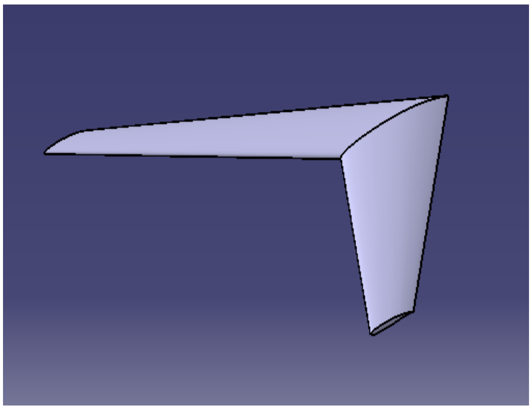


Fig 3. Twist angle = +2 degrees

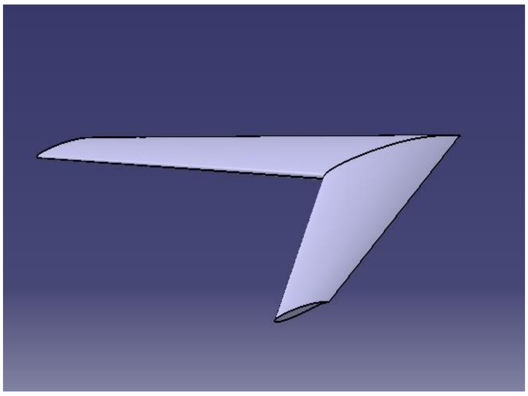


Fig 4. Twist angle = +5 degrees

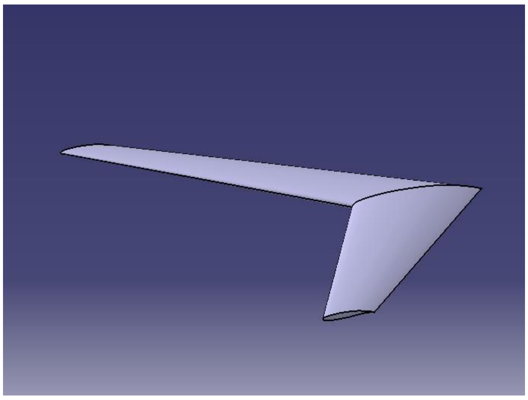
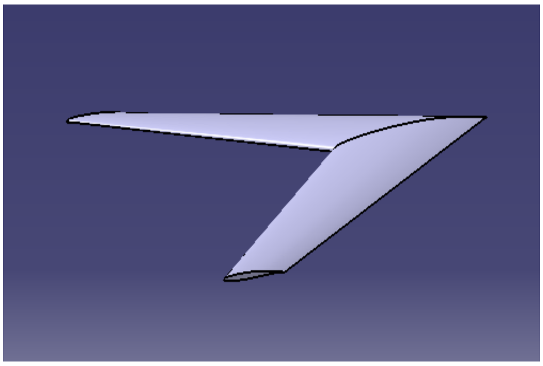


Fig 5. Twist angle= -2 degrees

Fig 6. Twist angle = -5 degrees



VI. CFD SIMULATIONS

ANSYS ICEM- CFX was used for the pre-processing stage of the CFD simulations. The pre-processor stage includes:

1) Initialization and importing of the geometry.
2) Definition of the flow domain.
3) Meshing of the flow domain along with the imported geometry
4) Definition of the flow conditions and boundary conditions.
5) Selection of the relevant flow models to simulate the flow.
6) Initialization of values and solving until convergence of values is obtained

A. Initialization of geometry and Meshing

The IGES file of the wing geometry modelled on CATIA was imported to ANSYS ICEM-CFX for pre-processing flow domain, which is a cube of dimension 1m was defined around the imported wing geometry and the two bodies were combined using a Boolean subtract operation using the wing geometry as the tool body and the domain as the solid. Then the resultant geometry was meshed with unstructured free mesh of relevance centre as fine. The meshed files in the wireframe configuration are shown below

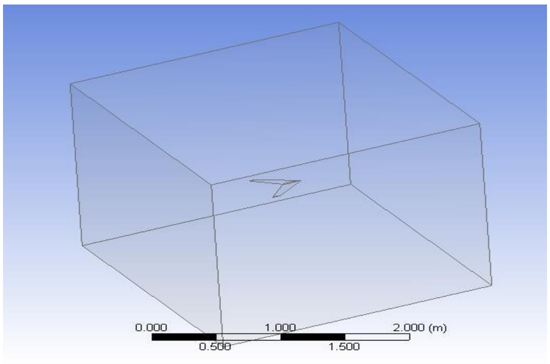


Fig 7. Combined geometry after Boolean subtract operation

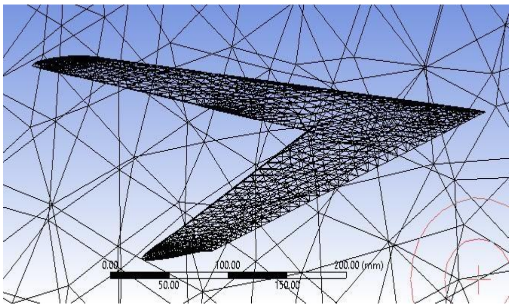


Fig 8. Meshing of the domain and wing geometry

B. Selection of Flow Models

The flow models used in CFD are the mathematical models that used to simulate various types of flows. These flow models are usually in the form of governing equation that is Naiver Stokes equation, Euler equations and energy equations.

There are various flow models available in ANSYS FLUENT to simulate wide range of flows from laminar to turbulent.

The wing design proposed in this report is supposed to operate in the supersonic regime with the minimum free stream Mach number being 1.In the supersonic regime, the formation of shockwaves creates and dissipates a lot of energy. So the Energy Equation is selected to calculate the flow parameters.

There are many turbulent flow models available in FLUENT, which are used in different cases for accuracy. The selection of the turbulent flow model k-ω-SST was done because

- ✓ The basic k-ω flow model gives very accurate results close to the surface of the wing, thus helping in the study of boundary layer.
- ✓ The k-ω SST model, which is a modification of the standard k-ω flow model, is known for its good behaviour in regions of adverse pressure gradients and separating flow.

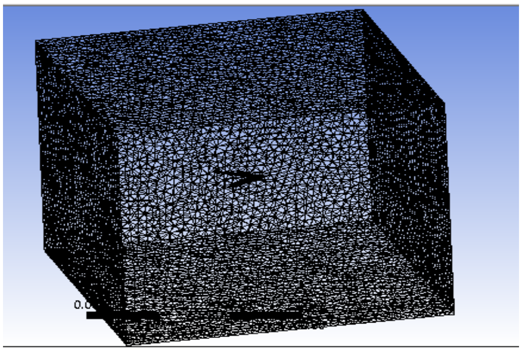
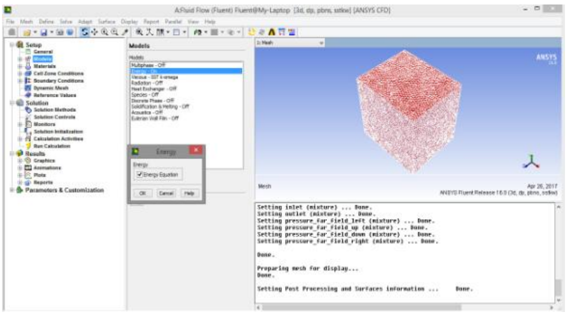


Fig 9. Meshing of the wing geometry

Fig 10. Energy Equation Flow Model



```
Setting inlet (mixture) ... Done.
Setting outlet (mixture) ... Done.
Setting pressure_far_field_left (mixture) ... Done.
Setting pressure_far_field_up (mixture) ... Done.
Setting pressure_far_field_down (mixture) ... Done.
Setting pressure_far_field_right (mixture) ... Done.

Done.

Preparing mesh for display...
Done.

Setting Post Processing and Surfaces information ...    Done.
```

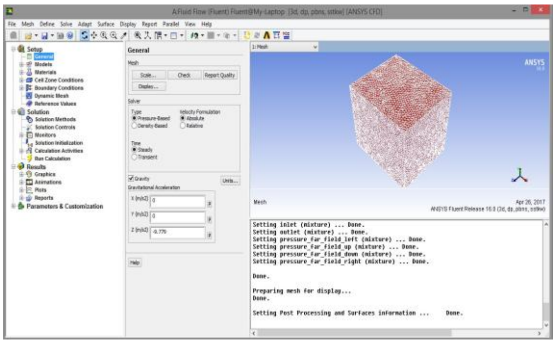
C. Solver

In ANSYS FLUENT solver, there are many solver setting to choose from. The solver settings chosen determine the accuracy of your solution.

The main choice between solvers for supersonic flows is between pressure based solver and density based solver.

Pressure based solver was historically developed and used for low speed, incompressible laminar flows and the density based solver was developed with

high speed compressible flows in mind. But now both solving techniques have evolved and can be used for a wide range of flows from low speed incompressible flows to high speed compressible flows.



```
Setting inlet (mixture) ... Done.
Setting outlet (mixture) ... Done.
Setting pressure_far_field_left (mixture) ... Done.
Setting pressure_far_field_up (mixture) ... Done.
Setting pressure_far_field_down (mixture) ... Done.
Setting pressure_far_field_right (mixture) ... Done.

Done.

Preparing mesh for display...
Done.

Setting Post Processing and Surfaces information ...    Done.
```

Fig 11. Solver Setting

After setting up the boundary conditions and the solver, then the number of iterations is given. The number of iterations may change from one case to another as the inlet velocity conditions vary as a function of the angle of attack. But the calculation is continued until convergence of value is achieved in all the important parameters.

Convergence criteria are the criteria that must be met for the solution to be accurate. In FLUENT, the variation of values of various variables such as x velocity, y velocity etc. are tracked over each iteration and plotted. If the graph shows too much variation of the value of the parameter, then additional iterations are done until convergence is reached.

The number of iterations varies from one simulation to another because of the change in geometry and change in the inlet velocity conditions

VII. Results and Discussion

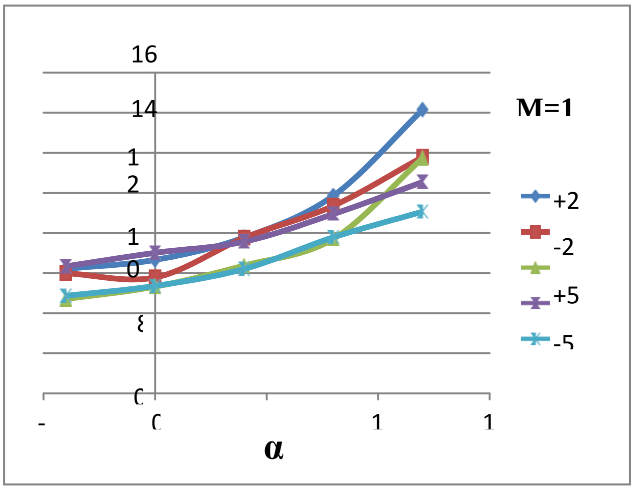


Fig 12. Lift co-efficient v/s Angle of attack

The above plot of Lift to drag ratio v/s Angle of attack for forward swept wings with a range of twist angles (-4° to +12°) implies that the performance is obtained by the forward swept wing with a geometric span-wise twist of +2 degree. It can also be seen that it provides a higher lift at higher angle of than that of the conventional forward swept wing configuration. The below table provides data sheet of various parameters obtained for forward swept wing with +2 degrees twist.

TABLE IV.Aerodynamic data sheet for forward swept wing with +2-degree twist

Angle of Attack					
	-4	0	4	8	12
Lift	687.6	784.4	863.9	956	1028
Drag	114.8	135.7	111.3	102.1	87.1
C_L	0.012	0.028	0.015	0.017	0.018
C_D	0.002	0.005	0.002	0.001	0.001

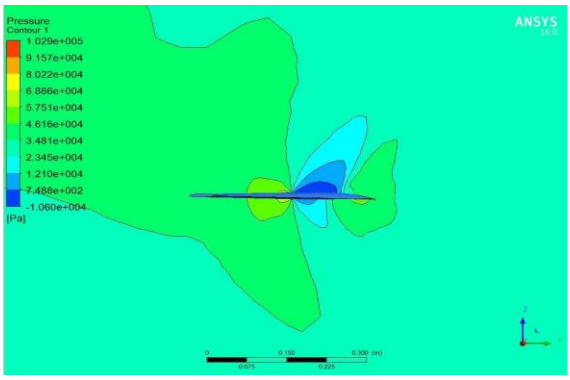


Fig 13 Pressure contours for forward swept wing with +2° twist at -4° AOA

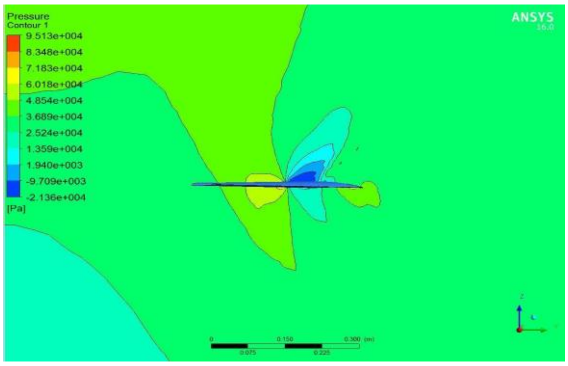


Fig 14 Pressure contours for forward swept wing with +2° twist at -4° AOA

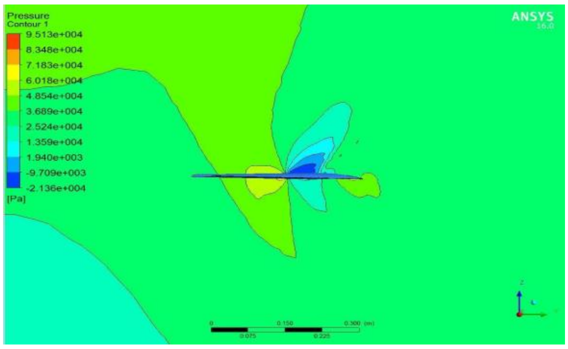


Fig 15 Pressure contours for forward swept wing with +2° twist at -4° AOA

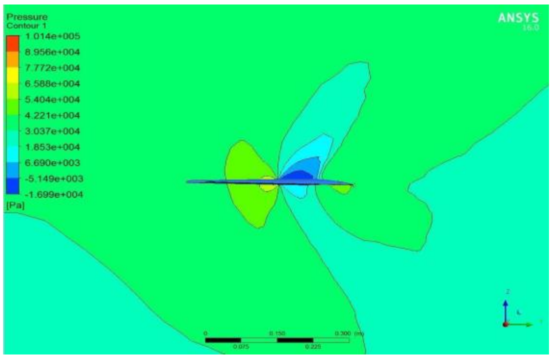


Fig 16. Pressure contours for forward swept wing with +2° twist at -4° AOA

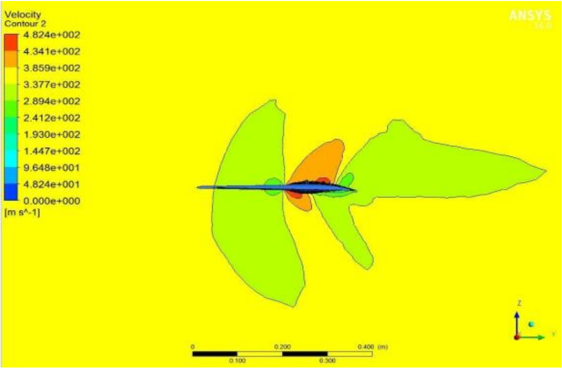


Fig 17 Velocity contours for forward swept wing with +2° twist at -4° AOA**.**

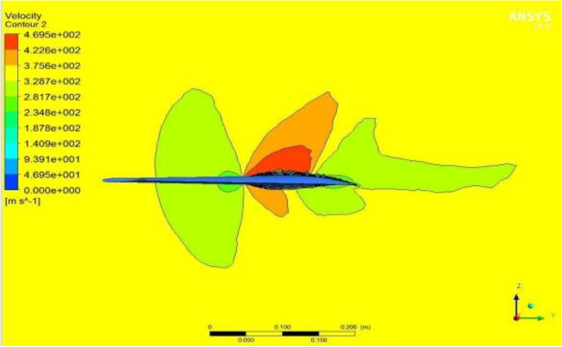


Fig 18 Velocity contours for forward swept wing with +2° twist at -4° AOA

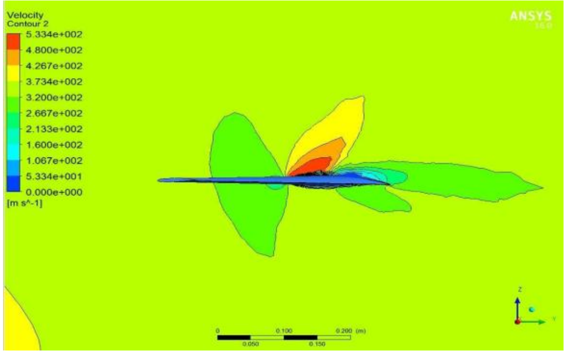


Fig 19. Velocity contours for forward swept wing with +2° twist at -4° AOA**.**

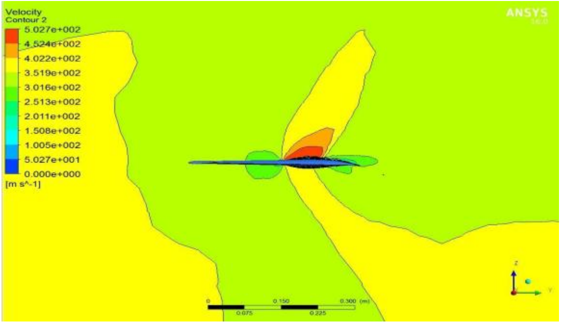


Fig 20 Velocity contours for forward swept wing with +2° twist at -4° AOA**.**

VIII. Conclusion

With respect to the above discussed results, it can be concluded that the positive span- wise twist provides an increment in Lift to drag ratio. But when the co-efficient of lift and drag are being compared, forward swept wing with the +2 degree twist came out as a configuration with better aerodynamic performance out of various possible set configurations. It seen by comparing of the data sheet of the normal wing and wing with the +2 degree twist, that there is an increase of average (over angle of attacks of range -4° to +12°) Lift to drag ratio by 32.059%.

Thus employing a small positive twist in a forward swept wing implies better aerodynamic performance and hence the better Range, Endurance and other essential aerodynamic characteristics.

IX. Future scope of the work

Only the CFD analysis is being carried out for the forward swept wing with span-wise twist and a prototype is fabricated. Further studies on the concept can be made by fabricating a wind tunnel testable model with pressure ports and testing in a transonic- supersonic wind tunnel and calculations can be carried out. The obtained experimental results can be compared with this simulated data. Research can be brought about to reach the optimized level of stability and control

X. References

[1] Edwin J. Saltzman and John W. Hicks, "In-flight lift-drag characteristics for a forward-swept wing aircraft", NASA Technical paper 3414.

[2] Kenneth L. Sims, B.S, "An aerodynamic investigation of a forward swept wing", Air University, United States Air Force.

[3] Charles D Harris, "NASA Supercritical airfoils", NASA Technical paper 2969.

[4] W. F. Lindsey, Bernard N. Daley, and Milton D. Humphreys, "The flow and force characteristics of supersonic airfoils at high subsonic speeds", NACA Technical note No.1211.

[5] Richard, "Review of nasa supercritical airfoils", Whitcomb, National Aeronautics and Space Administration Langley Research Center Hampton, Virginia.

[6] Franklin W Diederich and Barnard Budiansky, "Divergence of swept wings" NACA TN.1680.

[7] John.D Anderson, "Fundamentals of aerodynamics", McGraw-Hill Inc.

[8] John.D Anderson, "Introduction to flight", McGraw-Hill Inc.

[9] John.D Anderson, Jr, "Computational Fluid Dynamics", McGraw-Hill Inc.