

“DESIGN AND ANALYSIS OF SLAT OPERATED BRAKING SYSTEM”

C Ganesh¹, Dr. Chikanna N², Sanjeev G Palekar³ Prashanth Radhakrishnan⁴

¹PG Student,²Associate Professor & Chairman,³Assistant Professor,⁴Director-Aerospace R&D,
¹⁻³Department of Aerospace Propulsion Technology, Visvesvaraya Technological University -Centre for Post Graduate Studies, Bengaluru Region, VIAT, Muddenhalli, Chikkaballapura, Karnataka, India.

⁴Dautya Aerospace Pvt Ltd, Bengaluru, Karnataka, India.

Abstract - Cross wind effect is one of the biggest challenges faced by aviation industries this is due to poor braking efficiency as one of the biggest factors. Today in aircraft operation of breaking especially during landing we are using the support of spoiler, thrust reversal, flap to create lift After having different braking systems still this problem is existing where the aircraft encounters runway excursion, landing gear failure and crush As a solution for this problem slat diploid braking system can tremendously reduce the lift produce by increasing the drag in order to improve the braking efficiency

Key Words: Slat, crosswind effect, Drag.

1. INTRODUCTION

The main objective of the project is to reduce the crosswind effect that impacts the aircraft at the time of landing and in special cases during take-off also present days aircraft have thrust reversal, spoiler, and flap to create drag in spite of all the other controlling system we still facing the failure of safe landing during the time of crosswind and also delaying the landing time which causes a lot of loses to the aviation industries due to poor landing or braking efficiency the more fuel it consumes.

Another major drawback is that in the domestic plane the passenger feels less comfortable during the time of landing due to poor landing or braking efficiency. Sometimes the aircraft get damaged due to the failure of the braking efficiency so we come up with a solution through the design and analysis of a slat that can be used to increase the braking efficiency in the aircraft during the landing.

Another problem was aircraft usually take a larger space to land there was no efficient system for short-distance landing. our project will also be the solution to this problem. the short-distance landing saves time and fuel consumption and the area of land that can be used for other purposes.

2. METHODOLOGY

The project carried out would undergo the following steps of work all the simulation and the analysis.

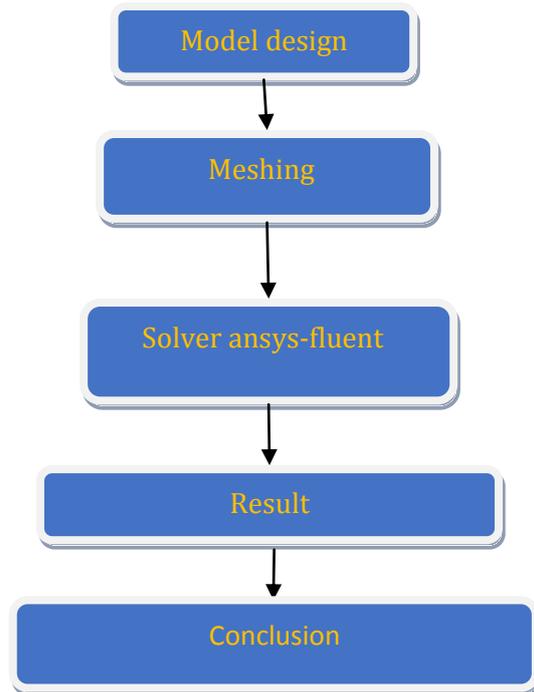


Fig -1: flowchart of the steps of the project

3. PROBLEM SETUP

In designing the airfoil we used the NACA2412 series slat placed before the airfoil As shown in fig.2 as its looks after designing the model in CATIA imported the model to Ansys software to further process

CATIA is a well-known software for design it is more convenient to use this software first step is from the NACA tool we have to download the coordinate of the airfoil import to CATIA readable form then by the coordinate we have to design the airfoil through the same design as to use for both slat fig 3 and without slat model after finishing the design save the file in isg format

Ansys

ansys is the most popular software for the analysis of fluent, thermal, and also some other conditions we are using Ansys for our project to verify the modified model to check the behaviour of the forces that act on the airfoil

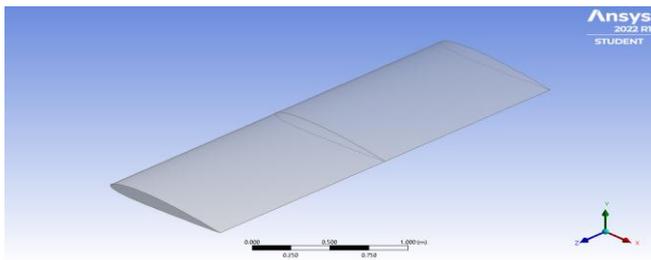


Fig -2: NACA2412 series Airfoil without slat

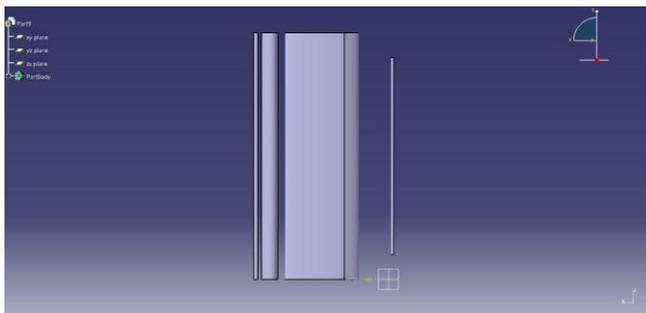


Fig -3: NACA2412 series Airfoil with slat



Fig -4: NACA2412 series Airfoil with slat

Details of "Mesh"	
[-] Display	
Display Style	Use Geometry Setting
[-] Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
<input type="checkbox"/> Element Size	Default (0.71015 m)
Export Format	Standard
Export Preview Surface Mesh	No

Fig -4: Details of Mesh

In the meshing select the element size 2m to set select face mesh then select all the planes of the model to uniform meshing on the surface of the model then select the growth rate as 1.1 in the mesh setup click the generate. below fig 5 is the meshing of the without slat model and fig 6 denotes the with slat mode

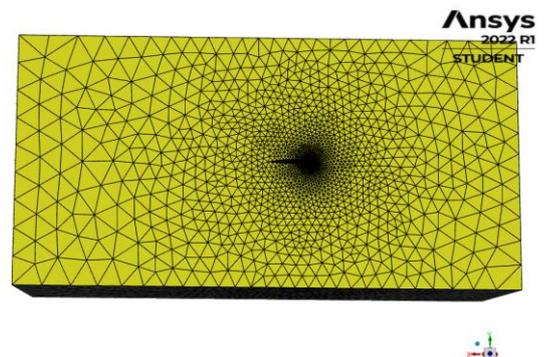


Fig -5: Meshing of Airfoil without slat

4. MESHING

Meshing is a very important part of CFD simulation. Without a proper mesh, will not be possible to get a converged solution in this project meshing has been done using ANSYS meshing feature.

Initially, the mesh was checked for any mappable feasibility. This initial overview gives us a rough idea of how our mesh would be. According to this necessary mesh, control can be done according to the surface or body type. A control to the extent of managing the number of elements to be present at the specific edge or face can also be managed through mesh control.

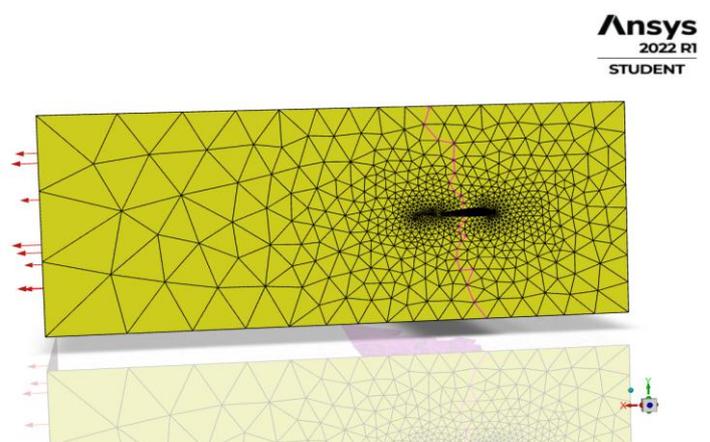


Fig -6: Meshing of Airfoil with slat

5. FLUENT SETUP

On starting the FLUENT setup, the FLUENT launcher appears providing options with 2D/3D model based on the geometry model and options like double precision, display mesh, etc. Additionally, the launcher also enables the user to select a number of processors that are being dedicated for the FLUENT program. The user can also set up as shown in fig 7 additional cores of the Dedicated core based on the model complexity.

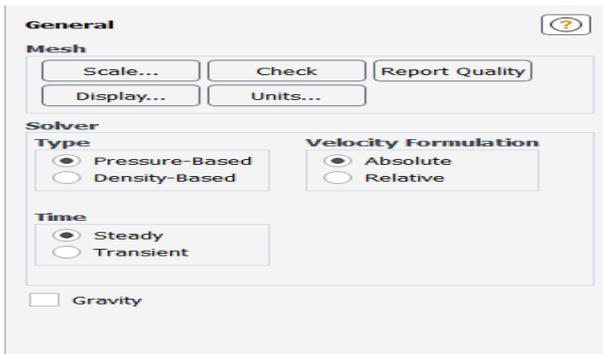


Fig -7: FLUENT SETUP

The mesh will be automatically loaded into the setup and creates automated partitions, zone, and necessary boundary condition which will include our named selection that we have created in the meshing stage. The first step to being done upon loading the mesh into the FLUENT setup is to check the mesh integrity loaded onto the setup. This is a preliminary check that has to be done in order to avoid errors (in case) in the late stage.

Click the check button in the general settings of the task plane and wait till it message appears on the console of the client which states the mesh has no error and the setup is carried forward.

Then we will configure the model that is to be considered for the setups i.e., Energy, Viscous, and species model. viscous SST k omega then click the komega 2eq then click ok is shown in fig8

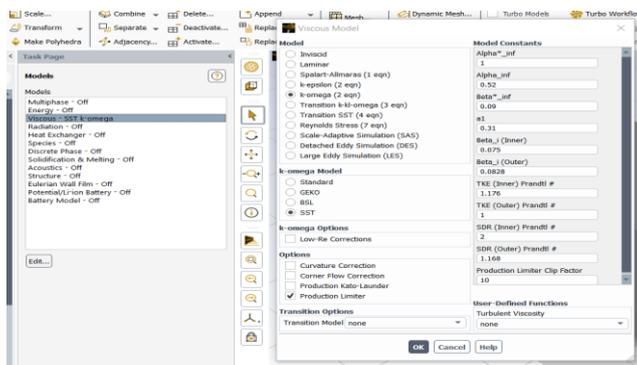


Fig -8 Solution Setup Of Model

After the setup of the solution after selecting the report definition in that select, the force of lift and drag coefficient then create the report definition. After the report, the setup computes the solution from the inlet of the model selects initialization Then clicks on the standard initialization afterward clicks on compute from the inlet .reference frame as to be absolute.

Click on initialization after the initialization clicks on the run calculation in this module set the 500 iterations and then set the interval 10 after clicking on run. after clicking the run system take time to compute the graph of lift, drag, and residual graph after that go to the result and select the plane to create a plane on the XY plane, create a counter select the plane, and the wing region select the static pressure as to show click on save and plot as shown in fig

After the model set up move and select the material in the material select air set the viscosity and density of the material as te constant and density is 1.225 than click change as shown in fig-8

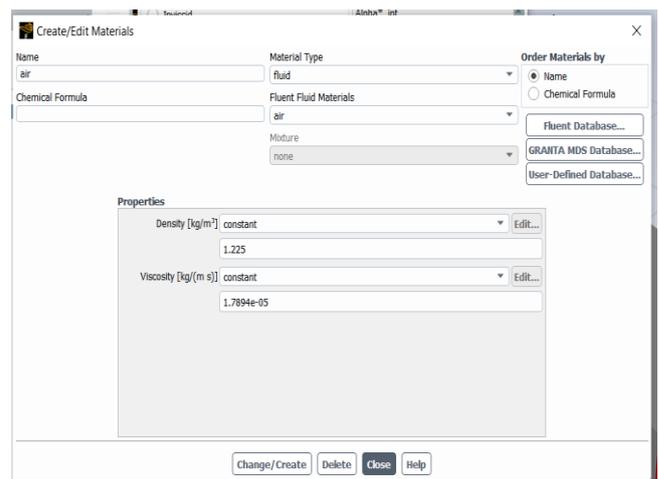


Fig -9 Create material setup taskbar

After the material setup go to the solution method in the sostepwiseselect theshownhostepwisese is as shown in the figure select the gradient as least square node based and the pressure as second-order momentum as second-order upwind turbulent kinetic energy as second-order upwind scheme as simple

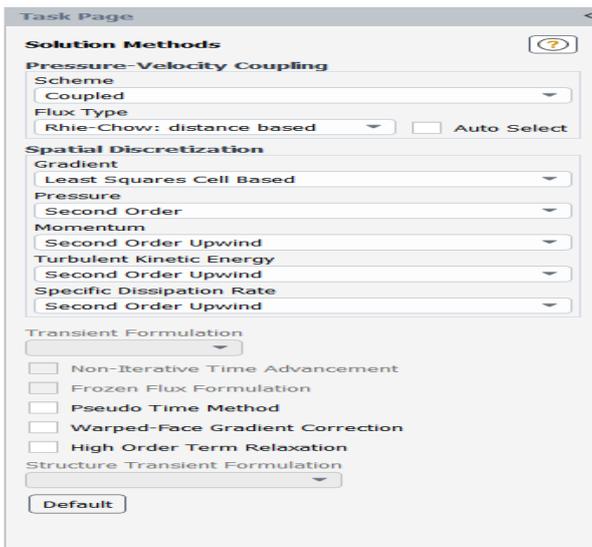


Fig -8 Solution method set up taskbar

After the setup of the solution after select report definition in that select, the force of lift and drag coefficient then create the report definition. After the report, the setup computes the solution from the inlet of the model selects initialization Then clicks on the standard initialization afterward clicks on compute from the inlet .reference frame as to be absolute.

Click on initialization after the initialization clicks on the run calculation in this module set the 500 iterations and then set the interval 10 after clicking on run. after clicking the run system take time to compute the graph of lift, drag, and residual graph after that go to the result and select the plane to create a plane on the XY plane, create a counter select the plane, and the wing region select the static pressure as to show click on save and plot.

Between the solution and initializing the solution, there is another set is report pots we have to define the cl and cd to be posted as a result .

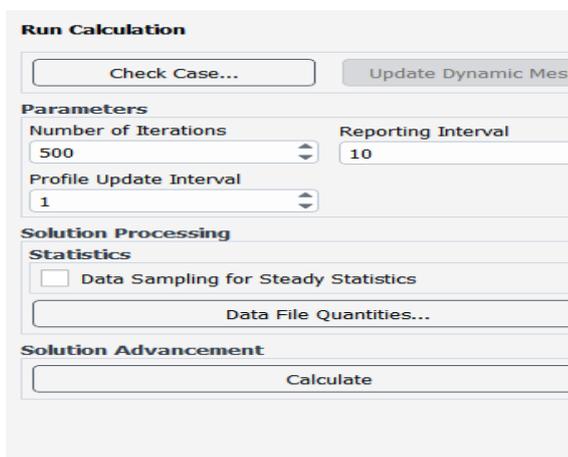


Fig -10 Run calculation set up taskbar

Run calculation taskbar we have to enter the number of iteration as 500 and report interval as 10 as shown in fig9

6. RESULT AND DISCUSSION

After the computation by comparing our model airfoil with slat against the model in which normal airfoil fig10 cl graph is showing that the lift is increasing in without slat or normal airfoil but in our model fig11 lift is going decreases as we computed the graph. From this graph, we can say that our model has achieved the objective of the project

In the below graph fig7.1 of the model without slat model shows that the lift coefficient increases for the iteration of 500 readings x-axis is the lift coefficient and y is the number of iterations.

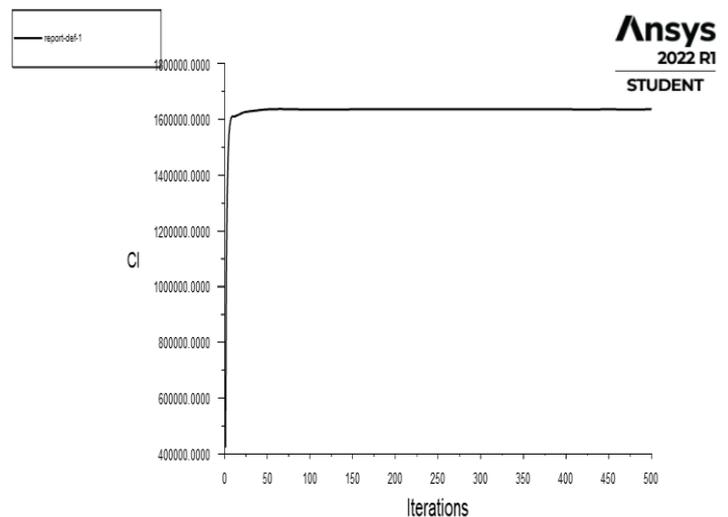


Fig -11 coefficient of lift without slat model

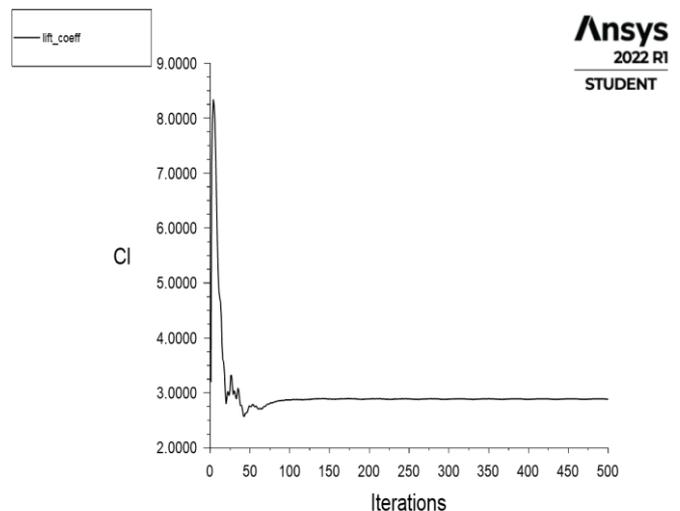


Fig -12 coefficient of lift with slat model

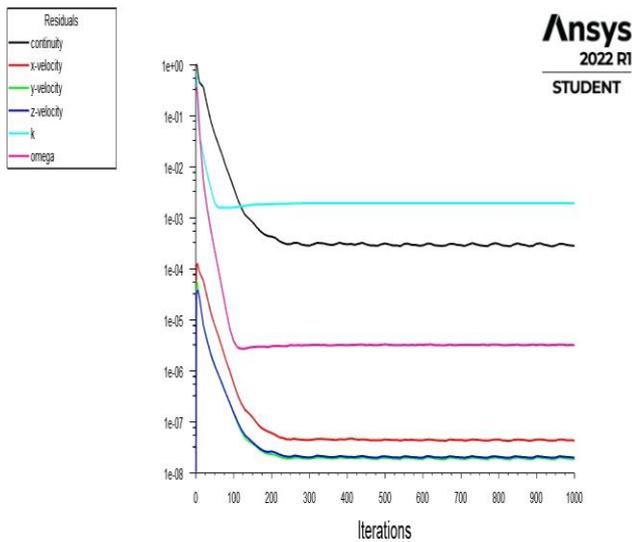


Fig -13 Residuals graph of without slat model

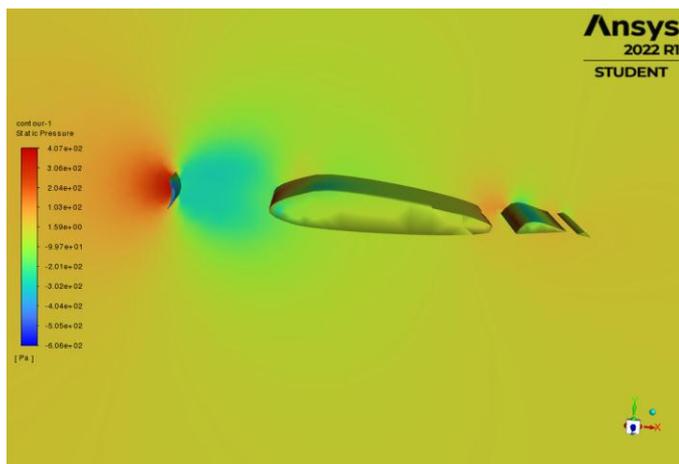


Fig -14 Residuals graph of without slat model

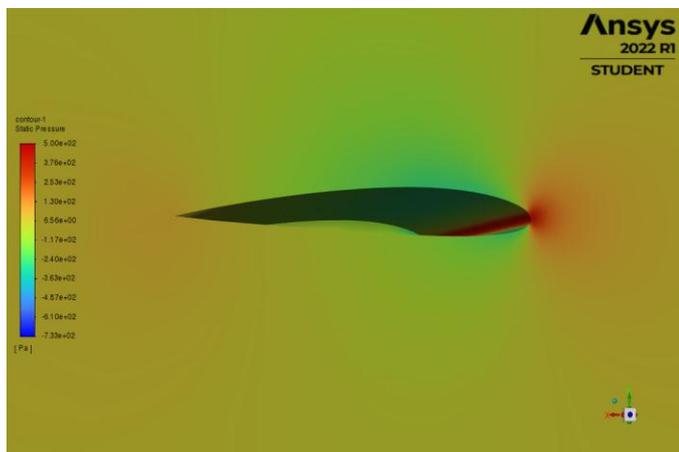


Fig -15 Residuals graph of without slat model

From the above pressure distribution of both, our model fig14 versus the normal airfoil model pressure distribution around the normal airfoil fig 15 is more compared it with our slat model due to the direct contact of air force pressure is more at the normal airfoil but in our model

Pressure is cut off by the slat which increases the efficiency in the braking system of the aircraft and also helps with the safer landing and also short distance landing of aircraft.

8.CONCLUSION

This project’s main objective is to reduce the lift force of the aircraft during the time of crosswind effect on the aircraft landing movement. Reduce the crosswind effect and allows the aircraft to smooth and safe landing as per our objective the lift is considerably decreased. while the without slat model has generated the lift by this result we can conclude that the experiment has shown a positive result towards the objective of the project this result can be not only applicable during the crosswind effect but also in the time of failure of another control unit.

This slat model can also be used for the short-distance landing of aircraft thereby reducing the time of landing and also the fuel consumption of the aircraft. And the space required during the landing. After the project outcome further, we have test the model on WindTunnel for more confirmation and also rectifies the problem what are faced during the landing of the aircraft

The project as gives a positive result towards the objective . satisfied the condition has now

REFERENCES.

- [1] "Drag Assessment of a High Performance Aircraft using System Identification Techniques" Khadeeja Nusrath, Jatinder Singh and Basappa National Aerospace Laboratories, CSIR, Bangalore-560017, India
- [2] Crosswind Landings in General Aviation: A Modified Method of Reporting Wind Information to the Pilot Matt Ebbatson, Don Harris, and Steve Jarvis Department of Human Factors Cranfield University, Bedford, United Kingdom
- [3] The effect of flap-end additions on aircraft trailing vortices W. R. Graham and T. Berteny Department of Engineering, University of Cambridge, UK L. David Laboratoire d'Etudes Aérodynamiques, Université de Poitiers, France
- [4] Effect of Leading-Edge Slats at Low Reynolds Numbers Lance W. Traub and Mashaan P. Kaula †Aerospace Engineering Department, Embry Riddle Aeronautical University, Prescott, AZ 86301, USA;

- [5] Aircraft Braking System 1Shruti Nair, 2Shreya Nair
1,2Dept. of Mechanical and Automation Engineering,
Indira Gandhi Delhi Technical University for Women,
Delhi, India
- [6] Wind-tunnel and CFD investigations of UAV landing
gears and turrets – Improvements in empirical drag
estimation Falk Götten a,b,*, Marc Havermann a, Carsten
Braun a, Matthew Marino b, Cees Bil b a Department of
Aerospace Engineering, FH Aachen University of Applied
Sciences, Hohenstaufenallee 6, Aachen, 52064, Germany
b School of Engineering, RMIT University, 264 Plenty Rd,
Bundoora 3083, Australia
- [7] Wind-tunnel and CFD investigations of UAV landing
gears and turrets – Improvements in empirical drag
estimation Falk Götten a,b,*, Marc Havermann a, Carsten
Braun a, Matthew Marino b, Cees Bil b a Department of
Aerospace Engineering, FH Aachen University of Applied
Sciences, Hohenstaufenallee 6, Aachen, 52064, Germany
b School of Engineering, RMIT University, 264 Plenty Rd,
Bundoora 3083, Australia