

A Project Report
On
**Aerodynamic flow field simulation of a commercial aircraft using CFD to
obtain drag polar**

BY
ARYAMAN BASANT (18XJ1A0310)
SUJAY PAMIDIKALVA (18XJ1A0341)
SIRISIPALLI SRI HARI (18XJ1A0353)

Under the supervision of
Dr. RAVIKIRAN BOMPELly

**SUBMITTED IN PARTIAL FULLFILLMENT OF THE REQUIREMENTS OF
PR 301: PROJECT TYPE COURSE**



**ÉCOLE CENTRALE SCHOOL OF ENGINEERING
HYDERABAD
(MAY 2021)**

ACKNOWLEDGMENTS

we would like to express our sincere gratitude to our project supervisor Dr Ravikiran Bompelly, Assistant professor, department of mechanical engineering, Mahindra University, Hyderabad, for providing invaluable guidance, resources, advice, help and opportunities throughout the project on **“Aerodynamic flow field simulation of a commercial aircraft using CFD to obtain drag polar”**, and introducing us to new topics, which helped us to learn something new and also to keep us motivated to successfully complete the project.

we would also like to thank our college “Mahindra ecole centrale”, for providing us resources that were utter crucial for the completion of the project.

we would also like to thank our friends for helping us in finding the solution for the problems and to provide their invaluable time and support.

Ecole Centrale School of Engineering

Hyderabad

Certificate

This is to certify that the project report entitled “**Aerodynamic flow field simulation of a commercial aircraft using CFD to obtain drag polar**” submitted by Mr. ARYAMAN BASANT (HT No. 18XJ1A0310), Mr. SUJAY PAMIDIKALVA (HT NO. 18XJ1A0341), Mr. SIRISIPALLI SRI HARI (HT No. 18XJ1A0353) in partial fulfillment of the requirements of the course PR 301, Project Course, embodies the work done by him/her under my supervision and guidance.

(SUPERVISOR NAME & Signature)

Ecole Centrale School of Engineering, Hyderabad.

Date:

ABSTRACT

This paper presents the CFD simulation on DLR F11 in landing configuration from the NASA 2nd AIAA CFD High Lift Prediction Workshop. Ansys Fluent R20.2, a cell-centered Reynolds-Averaged Navier-Stokes CFD solver, with $k-\omega$ (2nd equation) and SST turbulence model are used for the simulation and for obtaining the computational results.

The aerodynamic coefficients of an aircraft play an important role in the aircraft performance, trajectory prediction, fuel optimization and for estimating other parameters therefore it is very important to predict the drag polar for an aircraft.

The analysis was done for low **Re** = 1.35 million, **Ma** = 0.175 and velocity= 60 m/s.

The Mosaic Poly-Hexcore mesh is used in the analysis to obtain more accurate results. The Mosaic Poly-Hexcore mesh results in up 20-50% reduction in the element count and also speeds up the Fluent solver by 10-50% depending upon the application. The total number of elements in the mesh were 20 million. Finer mesh is used in the body of interest and coarse mesh is used in the flow domain. This simulation was used to obtain various aerodynamic coefficient such as- Lift Coefficient, Drag Coefficient and Moment Coefficient for different Angle of Attacks. And the Drag polar was plotted by accumulating the results.

The accuracy of the computational result is confirmed by comparing the CFD results with the results from the wind tunnel test from the NASA High Lift Prediction Workshop. The aerodynamic coefficient obtained from the simulation are also compared with the aerodynamic coefficients from the wind tunnel data from the NASA High Lift Prediction Workshop. The plots were also compared.

The CFD simulation computational results such as Polar Drag, C_l Vs α , C_l Vs C_m and C_d Vs α are discussed in this report. The C_p distribution over the wing is plotted along with the velocity profile.

Keywords- CFD, Polar Drag, Angle of Attack, Aerodynamic coefficient, Mosaic Poly-Hexcore.

NOMENCLATURE

CFD = Computational Fluid Dynamics

C_p = Pressure Coefficient

C_m = Moment Coefficient

C_l = Lift Coefficient

C_d = Drag Coefficient

α = Angle of Attack

Re = Reynolds Number

CAD = Computer Aided Design

Ma = Mach Number

C_{d0} = parasite drag

e = Oswald span efficiency factor

AR = Aspect ratio

BOI = Body of influence

L = Lift force

q = Dynamic pressure

S & A = wing Area

F_d = Drag force

ρ = Density of fluid

M = pitching Moment

P = static pressure at the point where pressure coefficient is been evaluated

P_α = static pressure in the freestream

P_0 = stagnation pressure in the freestream

CONTENTS

Title page.....	1
Acknowledgements.....	2
Certificate.....	3
Abstract.....	4
Nomenclature.....	5
1 Introduction.....	7-9
1.1 Aerodynamics.....	7
1.2 Application.....	8
1.3 Mesh generation and computational domain.....	8-9
2 Problem definition.....	10-11
2.1 Why CFD is better than conventional techniques.....	10
2.2 Predict polar drag and other Aerodynamic coefficients.....	10-11
2.3 Pressure profile.....	11
2.4 Mathematical Representation.....	11
3 Background And Related Work.....	12-16
3.1 Background.....	12
3.2 Geometric Configuration.....	13
3.3 Simulation Parameters.....	14
3.4 Paper and References.....	15-16
4 Implementation.....	17-19
5 Results.....	20-22
6 Conclusion.....	23
7 References.....	24

INTRODUCTION

1.1 AERODYNAMICS

Aerodynamics is the branch of physics which deals with the motion of air on impact with a solid object like a wing or an aircraft. The modern aircraft is often heralded as the pinnacle of 20th century innovation and a key discovery in improving the need for better transportation. To build a machine that is capable of flight, one must be proficient in the language of fluid dynamics. Whenever an object on Earth moves, it always faces a resistive force in the form of air resistance. It is akin to our hair blowing against the wind whilst travelling in a fast-moving car. Similarly, an Aircraft also experience the same resistive drag although the numbers involved are of a much larger magnitude. The very airflow which impedes the motion of an incoming object is also the reason for flight. Lift is created due to the pressure difference between the suction and pressure surfaces of the airfoil. So, to put it bluntly, the enigma of flight is intricately woven with the airflow around a solid object. Therefore, we can summarize that it is crucial to understand the flow patterns around an object in order to gauge its aerodynamic properties.

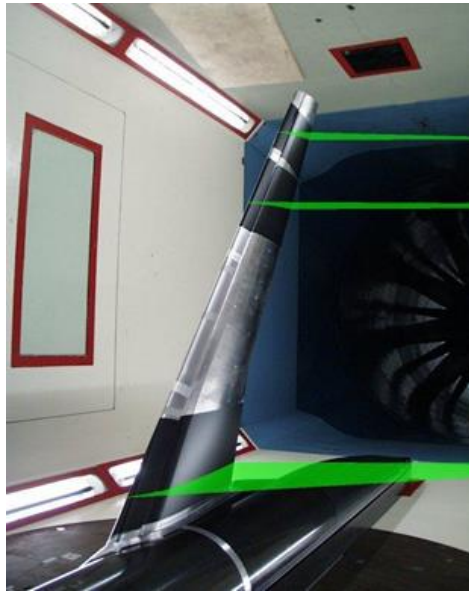


Figure 1. DLR F11 wind tunnel model

1.2 APPLICATION

Aircraft is basically a vehicle which uses the air in aiding its flight. It counters the force of gravity with the help of a dynamic lift on an airfoil. In a commercial airliner the cause of lift is due to the aircraft's forward airspeed and wing's shape. With the growing technological improvements in the field of computational physics software, the need for more economical aircrafts is a step closer to reality. The designs done in the software at the whim of our command mimic natural conditions to a fair amount of accuracy. So computational techniques have become the origin point for the design and analysis of any engineered object. The process of performing virtual fluid simulations using numerical analysis and data structures falls into branch of computational fluid dynamics (CFD). The use of CFD isn't limited to the aircraft industry. Just like fluid dynamics whose applications range from the working of water pump to the explosions in the sun (magneto hydro dynamics), same can be said about the scope of CFD. The conservation of energy, momentum and mass govern these CFD flows. Fluid dynamics is written in the language of partial differential equations. Complicated differential equations are converted to algebraic equations and then solved numerically. CFD analyses have a great potential to save time in the design process and are therefore cheaper and faster compared to conventional testing for data acquisition. Furthermore, in real life tests a limited number of quantities is measured at a time, while in a CFD analysis all desired quantities can be measured at once, and with a high resolution in space and time. Because CFD analyses approximate a real physical solution, it should be noted that these CFD analyses cannot fully exclude physical testing procedures. For verification purposes tests should still be performed.

Now back to the aircraft sector. Owing to the technical, logistical and environmental hurdles, the aircraft industry faces a huge challenge of designing efficient and better performing aircraft models. Its performance depends on the aerodynamic factors in consideration. Obtaining the dynamics of the flow around an aircraft model using a software gives the user a certain flexibility and freedom to explore. The results obtained must have to be validated with a reality check. They have to be close to the values obtained either in a wind tunnel test where a scaled down model is used and the aerodynamics of the model is thereby investigated.

Through our project we would like to perform CFD on a commercial airliner in order to obtain the flow field around it. The analysis was done using ANSYS FLUENT.

1.3 MESH GENERATION AND COMPUTATIONAL DOMAIN

The aircraft model our team selected was the DLR F-11 aircraft. A virtual wind tunnel analysis of the aircraft mode was performed by our team on the ANSYS FLUENT platform. Poly-Hexcore mesh was the preferred meshing approach that our team opted at the behest of our supervisor. This method of meshing has its own set of advantages. The Poly-Hexcore technology connects high-quality octree hexahedron in the bulk region, and isotropic poly-prisms in the boundary layer with the Mosaic polyhedral elements. When compared to the conventional hexcore mesh, this novel approach leads to a 20 to 50% reduction in the total element count. A direct consequence is the improvement in the speed of the ANSYS fluent solver by 10 – 50% based on the application. So, we have chosen a typical 100-passenger class regional jet airliner in a nominal landing configuration from the NASA 2nd AIAA CFD High-Lift Prediction Workshop. During nominal landing conditions of a high lift aircraft

we can expect several complexities like laminar-turbulent transition, strong adverse pressure gradient, wing tip vortices and the curvature of streamlines. They complicate the overall flow behaviour. Locating the transition between the different flows might be very difficult using the conventional CFD techniques. Calculation of the aerodynamic forces would be much more difficult in such a case. Therefore, understanding the physics of the flow is crucial. As of result we need better and efficient mesh preparation methods which can perform simulations in a smaller time frame than usual.

In our situation, we use the novel ANSYS Fluent Mosaic meshing technology to generate efficient, high-quality meshes using Poly-Hexcore topology. The new Poly-Hexcore feature in ANSYS Fluent uses this Mosaic technology to fill the bulk region with octree hexahedral elements. High quality isotropic poly-prims are formed in the boundary layer and these are conformally connected with the general polyhedral elements. These Poly-Hexcore meshes claim to give a reduced mesh count, higher mesh quality, and better solver performance when compared with other conventional meshing technologies. The surface meshing was accomplished on the system with 12 cores, 32 GB ram and Rx 570 gpu graphics.

Through this project we would like to perform aerodynamic flow field simulations in the form of a virtual wind tunnel test using the ANSYS fluent software. The dimensions of the wind tunnel form the flow domain of the aircraft. The dimensions of the flow domain are $4.3\text{m} \times 2.1\text{m} \times 2.1\text{m}$. The factors like the Reynolds number, aircraft dimensions, flow speed are determined from the experimental data from the AIAA 2nd workshop. After the completing of ANSYS mosaic Poly Hexcore meshing and simulation of the model we obtain the drag polar. It is the relationship between the lift and drag coefficient's. The simulation is performed at different angles of attack and the results are obtained. They are cross checked with the experimental data. The close proximity of the virtual and the experimental data would prove to be a testament to the robustness of this new meshing technique. Applying such CFD techniques would prove to vital in faster and efficient computational calculations related to large aircrafts like the BOEING 747, AIRBUS A380 and several others.

The flow domain was divided into 6 parts, 3 BOIs for the fuselage, 2 BOIs for the wings and 1 for the flow region for the fluid.

So let us dive into the world of CFD and flow field simulation around a DLR F-11 aircraft model.

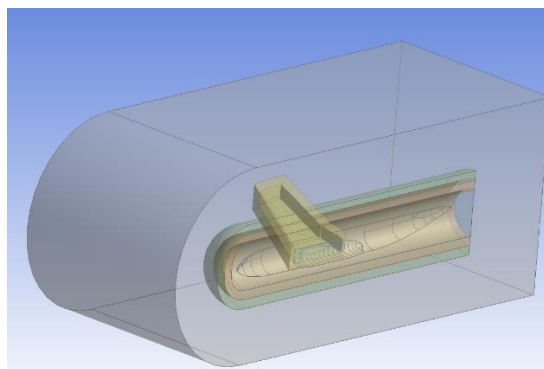


Figure 3. Body of influence and Flow Domain

PROBLEM DEFINITION

2.1 WHY CFD IS BETTER THAN CONVENTIONAL TECHNIQUES

It is not practical for any institute/organisation to test the aircraft in every stage of development or to test each and every model by making the prototypes of each one of them as it will cost the industry a lot of loss in terms of money, time, human resource and many other resources. Therefore, to counter this many players in the aircraft industry shifted to wind tunnel testing, where a scaled model of the actual aircraft is tested while maintain the same constants as that of the actual aircraft, constant such as **Re** and **Ma**. But even that was not easy job as-

- The scaled down model was also not very small, it would require a huge wind tunnel for testing purposes and building these wind tunnel would be very costly. Setting up wind tunnel for the analysis would also take a lot of time.
- Getting all the aerodynamic constant (such as **Re** and **Ma**) for the scaled down model at the same time is also difficult, there is some inaccuracy involved in it.
- Wind tunnel also can't be used on very frequent purpose.

That is where CFD software comes into the picture, this way we could save a lot of money, time and human resources. We could run simulation as many times as we want and it wouldn't require a large team to run it. Once we get the desired output from the simulation, we can then proceed to perform wind tunnel testing.

2.2 PREDICT POLAR DRAG AND OTHER AERODYNAMIC COEFFICIENTS

Prediction of C_l , C_d and C_m is very important in the view of designing an aircraft. There are mainly 2 kind of drag and together they form a drag polar. Many factors such as thrust required, power required, range, endurance, etc..., depends on the value of lift coefficient and drag coefficient.

The 2 kinds of drags are-

- i. Parasite drag- this is the drag at zero lift, it comprises of skin friction which is the drag produced due to the interaction between the air molecules and the solid body of an aircraft. and form drag, this source of drag depends on the shape of the aircraft.
- ii. Induced drag- this form of drag is produced due to the generation of lift, it occurs due to the non-uniform distribution of lift on the wings.

$$C_D = C_{D0} + K.(C_L - C_{L0})^2. \dots\dots\dots(1)$$

$$k = \frac{1}{\pi A Re} \dots\dots\dots(2)$$

We also found the the variation of Cl with AOAs, variation of Cm with Cl. The value of Cl increases with AOA until stall angle is reached, beyond which Cl reduces and Cd increase drastically.

2.3 PRESSURE PROFILE

When an aircraft flies there is a difference in velocity across the wings and fuselage, the lift contribution from the fuselage is almost 10% of the total lift produces and the rest comes from the wing. Due to the variation in velocity across the wings, there is a variation of pressure. Which in turn produces the lift.

Therefore it is important to know the pressure profile of the aircraft wings. The Cp values for AOA= 0 deg is plotted in the results.

2.4 MATHEMATICAL REPRESENTATION

$$C_L \equiv \frac{L}{q S} = \frac{L}{\frac{1}{2} \rho u^2 S} \dots\dots\dots(3)$$

$$c_d = \frac{2F_d}{\rho u^2 A} \dots\dots\dots(4)$$

$$C_m = \frac{M}{q S c} \dots\dots\dots(5)$$

$$C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2} = \frac{p - p_\infty}{p_0 - p_\infty} \dots\dots\dots(6)$$

BACKGROUND AND RELATED WORK

3.1 BACKGROUND

Physics of the problem: The fundamental physics behind any aircraft flow is that the air in the vicinity of an aircraft causes two forces. One is drag another is lift. These drag and lift forces have their corresponding coefficients which can plotted using computational simulation software like ANSYS. Experiments were done on a model aircraft in a wind tunnel and the required data was tabulated. The aim of this experiment is to emulate the same experiment however with the aid of a software like ANSYS. We need to design our aircraft bodies as aerodynamically efficient as possible to prevent effects like early stall and boundary layer separation. The general consensus is that turbulent flow greatly increases the drag effect thereby decreasing the efficient of the model. However, the very nature of designing an aircraft is to deal with the problem of turbulence via geometric manipulation. The velocity of the flow is 60m/s. Mach number is 0.175. Reynolds number was found to be 1.35 million which is relatively low when compared to real aircraft flow. So, the flow type is turbulent, which is much closer to reality.

Mosaic meshing is an innovative ANSYS Fluent meshing technology which automatically, conformally combines a boundary layer meshes using high-quality polyhedron to hexahedron elements in the bulk region. This mosaic method can be used in the future to connect any surface and volume elements.

3.2 GEOMETRY CONFIGURATION

The CAD model of DLR F11 is imported from the NASA 2nd AIAA CFD High Lift Prediction Workshop in IGES file format, for the analysis purpose. We have chosen a 100-passanger class jet airliner in nominal landing configuration with slats deployed at 26.5 deg and flap deployed at 32 deg.

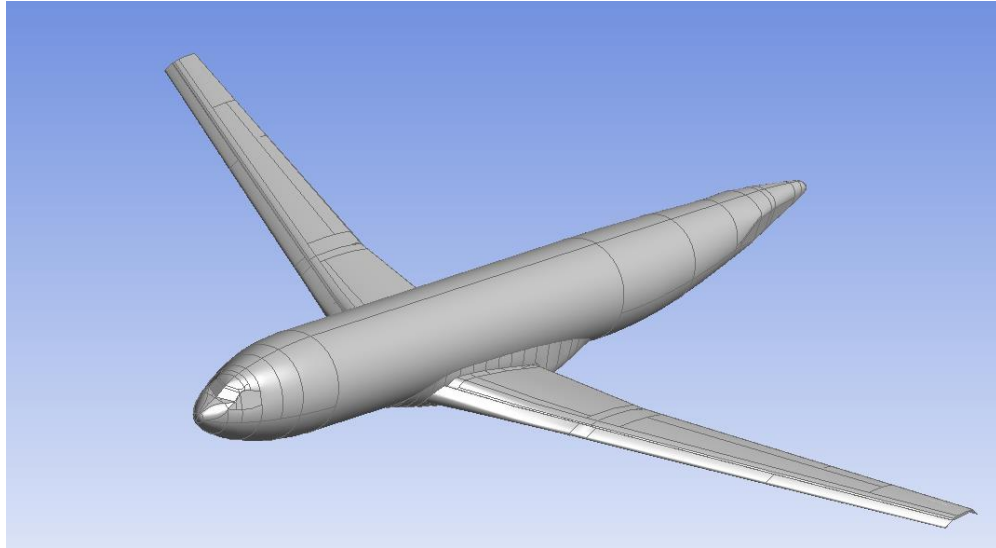


Figure 2. DLR F11 model

The geometric parameters are given in the table below:

Table 1. Model details

Mean aerodynamic Chord	0.34709 m (model scale)
Reference Area	0.419130 m ²
Wing semi span	1.4400 m
Moment reference centre	X = 1.42890 m, Y= 0 m, Z = -0.04161m
Fuselage length	3.077 m
¼ chord sweep	30 ⁰
Wing aspect ratio	9.353
Wing taper ratio	0.3
Wing dihedral	4 ⁰

3.3 SIMULATION PARAMETERS

This kind of a mosaic meshing technology is parallel scalable on high performance computing platforms which leads to a quicker mesh generation. For our project we have performed CFD with the help of a supercomputer in our college. The specifications are as follows:

Table 2. Computer specification

NAME	Nvidia DGX-1
CPU	2 X 20-Core Intel® Xeon® E5-2698 v4 2.2 GHz
SYSTEM MEMORY	512 GB DDR4 RDIMM

These supercomputers are a huge aid in executing millions of elements worth of meshing 6-10 times faster and reducing the runtime for the computation. The mosaic meshing also leads to 20-30 % reduction in the element count when compared to the conventional hexcore meshing. (The CAD geometry in the IGES-file format is taken from the NASA AIAA 2nd CFD High-Lift Prediction Workshop1. Minor CAD repairs, watertight box-shaped flow domain with a circular region which bears a close resemblance to the original wind tunnel and the Bodies of Influences (BOIs) were prepared using SOLDIWORKS CAD modeler. However, the local mesh refinement offset BOIs around the fuselage, wing, slat, flap, nacelle-pylon and support brackets are automatically prepared within ANSYS Fluent meshing tool, using an offset method, which smartly scales the selected surfaces according to the user input offset distance.

High-quality square surface mesh is prepared within ANSYS Fluent’s meshing tool. Scoped sizing functionality is used to define surface zone sizes, based on the curvature, proximity and soft size requirements. This surface mesh is used to generate Hexcore and Mosaic Poly-Hexcore volume meshes. However, for the Mosaic Poly-hexcore process, the surface mesh is converted to polys, by maintaining average edge length consistency with the triangular surface mesh.

Table 3. DLR F-11 Simulation Parameters

AOAs	0 ⁰ , 7 ⁰ , 16 ⁰ , 20 ⁰
Reynolds number	1.35 million
Reference static temperature	298.66 k
Reference static pressure	100700 Pa
Mean aerodynamic chord	0.34709 m
Volume of the flow domain	22.284 m ³

All the simulations are “free air”, no wind tunnel walls support systems.

The simulation accuracy, and the required processing time depend significantly on the generated grid quality, therefore, many considerations were taken into account for achieving an effective simulation.

3.4 PAPERS AND REFERENCES

Here we are going to discuss about the papers which we have used as references for the project.

3.2.1)

The paper which we have followed as a primary reference deals with the efficiency of the mosaic poly hexcore meshing for high lift aircraft configuration. The team members involved in this project have taken a typical 100-passenger class regional jet airliner in a nominal landing configuration from the NASA 3rd AIAA CFD High-Lift Prediction Workshop. ANSYS Fluent R19.2, a cell-centred Reynolds-Averaged Navier-Stokes CFD solver, with Shear Stress Transport (SST $k-\omega$) and Transition-SST ($\gamma-Re\theta$) turbulence model are used to obtain the computational results. When the Poly-hexcore meshing method was used a 48 % reduction in the total element count was observed when compared to the conventional hexcore mesh. A decrement of 41% in the computational time was also observed by the team. The computational processes were executed on a multi-core machine (CPU: 2x Intel(R) Xeon(R) Gold 6142 CPU @ 2.60GHz, RAM: 192GB (6GB/core)). Approximately 121 million element mesh is generated 6.6 times faster, on 32 cores compared to the serial core. Further, the accuracy of computational results obtained from both the meshes are confirmed by comparing aerodynamic force coefficients, wing spanwise pressure coefficient, surface oil flow and China-clay visualizations with the wind tunnel measurement data.

The primary difficult in dealing with the aircraft flow fields is the problem of complexities. Several parameters have to be taken into consideration like laminar-turbulent boundary layer separation, strong adverse pressure gradients, an increase in the wingtip vortices strength and wake boundary interactions just to name a few. Thus, the complex nature of the resulting flow fields and the uncertainties in locating onset transition makes it very difficult for conventional computational fluid dynamics (CFD) modelling approaches to accurately predict aerodynamic forces on high-lift configurations. Therefore, a solution to this problem by designing very refined and efficient meshes so that high fidelity simulations can be performed in a smaller timeframe. The model selected by the team was a JAXA standard model (JSM) with engine nacelle pylon which is a scaled down model of a commercial airliner which can house 100 members in it. The model was tested in a 6.5m x 5m low speed wind tunnel. Although the object was imported from the 3rd AIAA NASA workshop several refinements for the flow domain along with the modelling of the BOI's was performed using ANSYS space claim CAD modeller. High quality triangular meshes were used as per the report however our team has performed square surface meshes within the ANSYS fluent meshing tools.

After comparing the results of the two meshes the team has summarized that mosaic Poly-hexcore provides a distinct advantage over the conventional hexcore meshing pedagogy. On Comparing the averaged C_L over the last 1000-iterations, for $\alpha=18.580$ and the Transition-SST $a1=1$ model, shows a difference of only 1.1% between both the meshes. However, Poly-Hexcore and Hexcore predict ~1.5% and ~0.4% higher lift respectively ($C_{L-exp} = 2.75$), which is well within the acceptable limits considering the wind tunnel measurement uncertainties.

3.4.2)

The second paper pertained to the aerodynamic flow field simulation around a commercial aircraft model which was performed by the students of Zagazig university in Egypt.

The aircraft model was designed with the help of SOLIDWORKS CAD software. Dimensions were obtained from the technical specifications provided by Boeing Commercial Airplane Company and all associated details were modelled almost similar to those of the actual vehicle.

ANSYS Fluent 19.0 was used for studying the flow characteristics over Boeing 737-700 NG aircraft model at cruising conditions of the 10,000 m altitude above sea level and a speed of 0.5 Mach number. Drag coefficient ($C_d = 0.056$) and lift coefficient ($C_l = 0.245$) were obtained. The wing being utilized in the design of this aircraft (BOEING 737 MIDSPAN AIRFOIL) was studied at 1.8° and 0° AOA at the same cruising conditions. At both AOA, drag coefficient (C_d), lift coefficient (C_l), and pitching moment coefficient (C_m) were obtained. In addition, Mach contours, velocity vectors, pressure coefficient contours, total pressure contours were illustrated for the flow field corresponding to the airplane and air foil. The team have used an asymmetric air foil as a result of which we can observe significant lift at zero angle of attack (this is due to Bernoulli's principle). Since the model was to scale, the plots and inferences obtained from this analysis can be useful for future aircraft design purposes.

These are the inferences our team has gathered by studying the two reports. Although we have used the poly-hexcore method mentioned in the 1st report, the dimensions of our aircraft and the simulation parameters were entirely different. As a result, we would like to conclude that these reference papers served as an excellent jumping-off point in executing our project.

IMPLEMENTATION

We used geometry files based on deployed coordinates of the DLR F11 in landing configuration with Slat 26.5, Flap 32 (Wing/Body/HL system + SOB Flap seal+ Slat Tracks and Flap Track Fairings). We created a CAD model using solid works with 3 BOIs around fuselage and 2 BOIs around wing. After importing this CAD model into ANSYS fluent (with fluent meshing), created a mesh.

Table 4. element size of the BOIs

SL.NO	LABEL	ELEMENT SIZE (mm)
1	Fuselage BOI-1	6
2	Fuselage BOI-2	12
3	Fuselage BOI-3	24
4	Wing BOI-1	2
5	Wing BOI-2	6

First the surface mesh was generated with minimum size of 2mm and maximum size of 70mm .

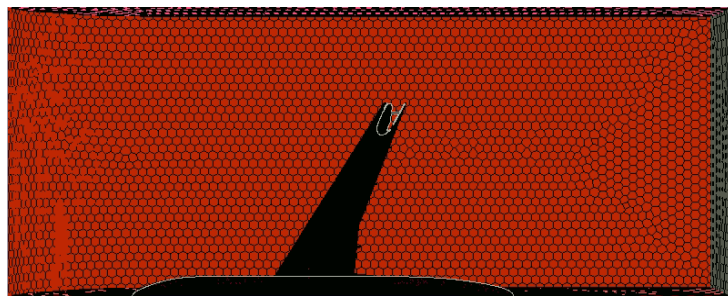


Figure 3. Surface Mesh

Updated boundaries- bc_exit- pressure outlet, bc_inlet- velocity inlet, pl_right_wall-wall, pl_symmetry -symmetry, surf_fuselage-wall, surf_wing-wall. Flow domain contained the fluid. In Boundary layers, we used smooth transition with layers-3 that was present only on walls. And created a volume mesh using poly-hexcore. One of the faces was a symmetry plane, since our model was a half aircraft, therefore by using symmetry plane we could obtain result for an aircraft.

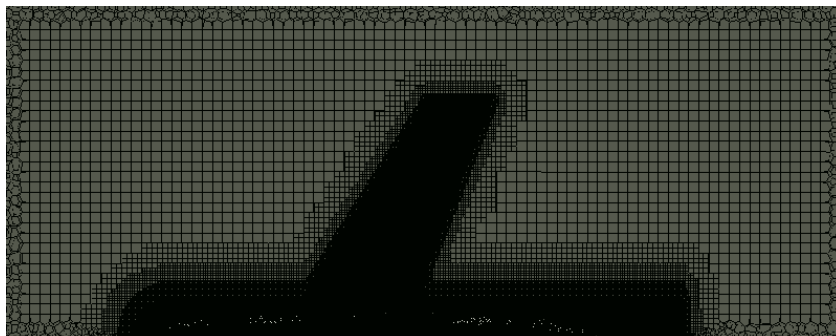


Figure 4(a). Volumetric Mesh

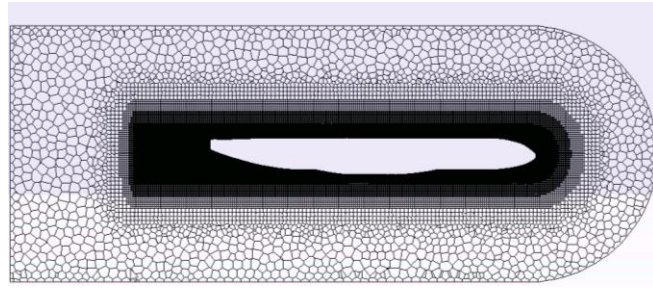


Figure 4(b). view from plane $y = 0.07$ m

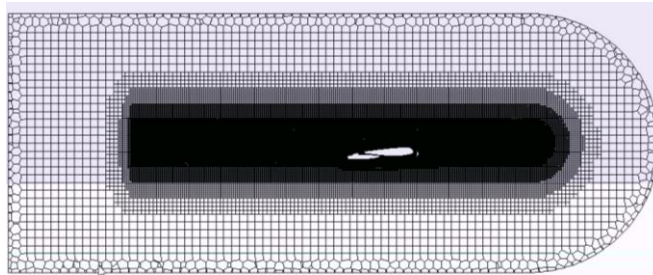


Figure 4(c). view from plane $y = 0.16$ m

The simulation was conducted in ANSYS fluent with the following Model and Material Properties-

Table 5. Model and Material properties

SL.NO	PARAMETERS	VALUE/OPTION
1	Model	Viscous (SST-k omega)
2	Fluid	Air
3	Solid	Aluminum
4	Air density	1.225 kg/m^3
5	Air viscosity	$1.7894\text{e-}05 \text{ kg/m-s}$

Table 6. Reference Values

SL.NO	PARAMETERS	VALUE
1	Area	0.419 m^2
2	Density	1.225 kg/m^3
3	Enthalpy	0
4	Length	0.347 m
5	Pressure	100700 Pa
6	Temperature	298.8 K
7	Velocity	60 m/s
8	Viscosity	$1.7894\text{e-}05 \text{ kg/m-s}$
9	Ratio of specific heats	1.4
10	Yplus for heat transfer coefficient	300

Report definition was created to get the value of Lift coefficient, Drag coefficient and Moment coefficient. The moment centre for calculating the moment was (1.428m,0m,-0.0416m) and the moment axis was Y-axis. The free stream velocity was = 60 m/s

We performed the analysis for multiple AOA, for each AOA we needed to change the components of velocity and force vectors for Cl and Cd in the report definition. In order to that we used following table-

Table 7. Directions

Velocity		Lift Coefficient		Drag Coefficient	
X-component	Z-component	X-component	Z-component	X-component	Z-component
Cos(θ)	Sin(θ)	-sin(θ)	Cos(θ)	Cos(θ)	Sin(θ)

In order to verify the results from the simulation we have the aerodynamic data set from the '2nd AIAA CFD HIGH LIFT Prediction Workshop'.

Table 8. Data from wind tunnel test by NASA

AOA	Cl	Cd	Cm/4
0.04	1.165	0.103	-0.532
6.99	1.871	0.163	-0.486
11.98	2.27	0.222	-0.383
14	2.419	0.248	-0.333
16	2.523	0.273	-0.268
17	2.552	0.286	-0.223
18	2.581	0.299	-0.179
18.49	2.603	0.307	-0.157
19	2.622	0.311	-0.137
20	2.602	0.366	-0.089
20.99	2.526	0.450	-0.093

RESULTS

In this section we will compare the results obtained from the CFD to the experimental data.

We ran simulation for 500 iterations, to get a stable value of C_l , C_d and C_m . we noted down the values of C_l , C_d and C_m their plot become constant or when they converged to a specific value. We monitored the residual plot and the convergence criterion was set to none, we used the aerodynamic coefficients variation as a measure of convergence.

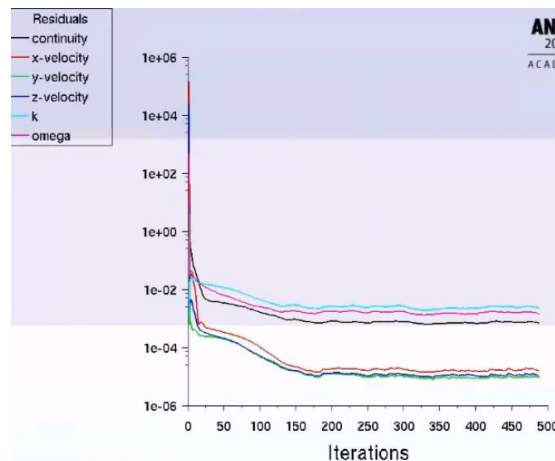


Figure 5. Residuals curve

Figure 6 shows C_l Vs AOA, C_d Vs AOA, C_l Vs C_d and C_l Vs C_m , comparison between the CFD results with experimental data from the wind tunnel.

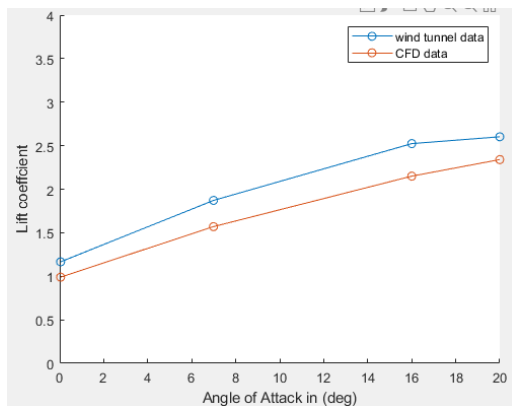


Figure 6(a). C_l Vs AOA

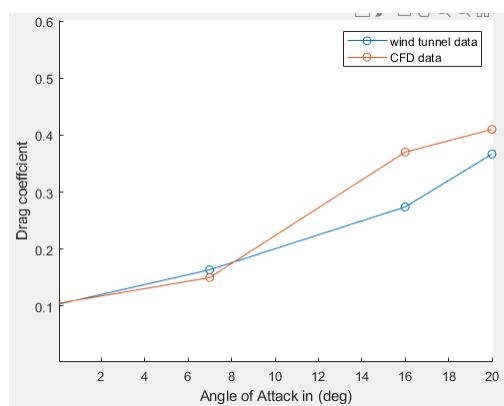


Figure 6(b). C_d Vs AOA

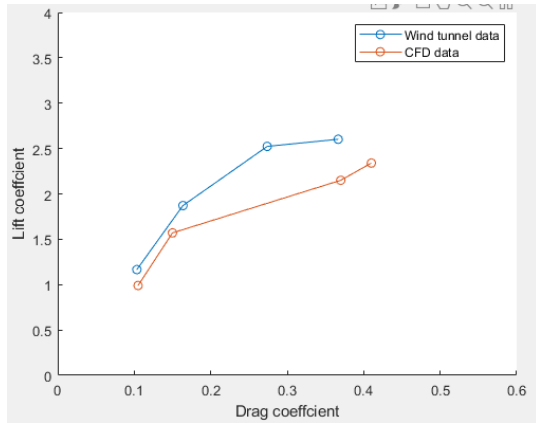


Figure 6(c). Drag Polar

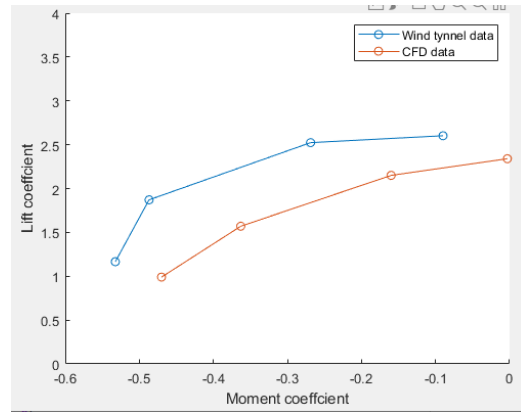


Figure 6(d). Cl Vs Cm

We could observe from figure 6(a) that the value of C_l obtained from the CFD is less than the experimental data value, the overall error in predicting C_l Vs AOA was 13%. We could observe from figure 6(b) that C_d predicted by CFD was almost equal to the experimental data for the smaller AOA's but deviation was observed for the higher AOA's. The overall error in predicting C_d was 7% (for AOA's = 0, 7, 20), at AOA = 16 deg the deviation was higher than other AOA's. Figure 6(c) C_l Vs C_d there was observed very small differences between CFD results and Experimental data for small AOA but the differences were higher for large AOA. Figure 6(d) deviation can be observed in the CFD values and Experimental data. There is deviation between the CFD results and Experimental data due to many factors such as-

- i. At higher AOA's, the flow separation is high, due to this behavior of the flow it's difficult to calculate the accurate values of aerodynamic coefficients. Higher the flow separation, higher the unpredictability in flow.
- ii. The mesh used for the simulation was a coarse mesh, we could use a finer mesh for a better result.
- iii. Drag coefficient predicted by CFD is usually higher than the Experimental data. This could be due to the type of model used for the analysis. At higher AOA the flow is complex.
- iv. Grid convergence.
- v. Mesh quality

Table 9. Aerodynamic coefficients from CFD simulation

AOA	Drag coefficient	Moment coefficient	Lift coefficient
0	0.105	-0.47	0.99
7	0.15	-0.363	1.57
16	0.37	-0.15	2.15
20	0.41	-0.003	2.34

The C_p value for $AOA = 0$ deg, obtained from the CFD is plotted –

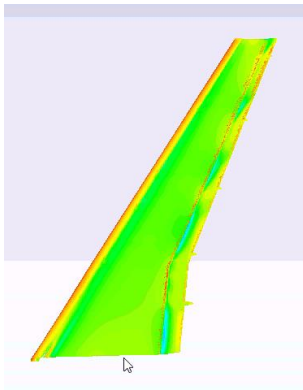


Figure 7(a) C_p distribution at the top surface of the wing

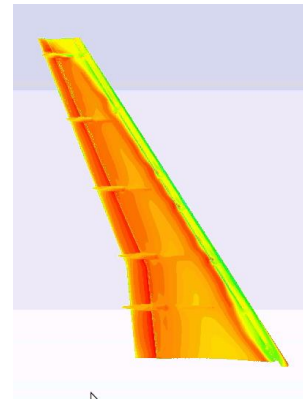


Figure 7(b) C_p distribution at the bottom surface of the wing

The velocity profile vector for $AOA = 0$ deg, obtained from CFD is -

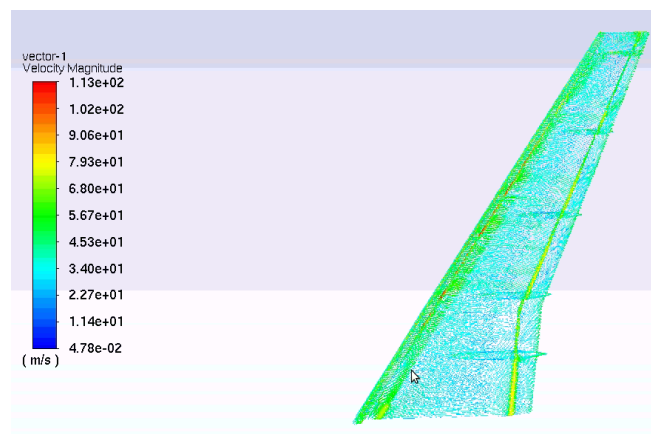


Figure 8. velocity vector for the wing of the aircraft

CONCLUSION

ANSYS fluent was used for calculating the aerodynamic coefficients- C_l , C_d and C_m for DLR F11 in landing configuration with slats deployed at 26.5 deg and flaps deployed at 32 deg from the 2nd AIAA CFD High Lift Prediction Workshop. The free stream velocity was 60 m/s, Reynolds number for MAC is 1.35 million and Mach number is 0.175. The results of the CFD were compared with Experimental data provided in the 2nd AIAA CFD High Lift Prediction Workshop.

Aerodynamic coefficients play an important role in aircraft performance so it is very important to predict it. CFD and wind tunnel testing is conducted to find the value of aerodynamic coefficients. Improvement in CFD methods and precise analysis both form a wind tunnel test and CFD is desirable for better understanding of the flow physics.

REFERENCES

1. 2nd AIAA CFD High Lift Prediction Workshop, url- <https://hiliftpw.larc.nasa.gov/index-workshop2.html>
2. Aerodynamic coefficients dataset, url- https://hiliftpw.larc.nasa.gov/Workshop2/DataForceMoment/for-mom_RUN29293-Re=1.35e6.dat
3. Pressure coefficient dataset, url- https://hiliftpw.larc.nasa.gov/Workshop2/DataPressure/cp_distr_alpha=7_RUN29293_Re=1.35e6.dat
4. <https://hiliftpw.larc.nasa.gov/Workshop2/AIAA-2012-2924-Rudnik-HiLiftPW2.pdf>
5. <https://hiliftpw.larc.nasa.gov/Workshop2/ParticipantTalks/HLPW2-rumsey.pdf>
6. Abdallah mosaad, Ahmed Abdel Gawad, “ Aerodynamic CFD simulation study of a commercial aircraft model”.
7. Adnan Munshi , Erwin Sulaeman , Norfazzila Omar , Mohammad Yeakub Ali, “CFD analysis on the effect of winglet cant angle on aerodynamics of ONERA M6 wings”.
8. Yuzuru Yokokawa, Mitsuhiro Murayama, Takeshi Ito and Kazuomi Yamamoto, “Experiment and CFD of a High Lift configuration civil transport aircraft model”, Japan Aerospace Exploration Agency, Chofu, Tokyo, 182-8522, JAPAN.
9. Falk Göttén, D. Felix Finger, Matthew Marino and Cees Bil, “A review of guidelines and best practices for subsonic aerodynamic simulation using RANS and CFD”. Asia Pacific international symposium on aerospace technology.
10. Krishna Zore, Balasubramanyam Sasanapuri, Alan Varghese, “ Ansys Mosiac Poly-Hexcore mesh for high lift aircraft configuration”- conference paper, 21st annual CFD symposium.
11. Klaus Schindler, Daniel Reckzeh and Ulrich Scholz, “Aerodynamic design of High-Lift device for civil transport aircraft using RANS CFD”, Airbus Operations GmbH, 28199 Bremen, Germany.
12. Junzi Sun, Jacco M. Hoekstra, Joost Ellerbroek, “Aircraft drag polar estimation based on a stochastic hierarchical model”, Delft University of Technology, the Netherlands.