Simulation of Aerofoil for Linear Cascade

[1] Bhavik N. Solanki, [2] Dr. Umang J. Patdiwala,
[1]Student of Master of Technology (CAD/CAM), [2]Assitant Professor
[1]Department of Mechanical Engineering,
[1]Indus University, Ahmedabad, India.

Abstract- Analysis of Effect of flow over NACA 6409 aerofoil linear cascade was done using the available Computational Fluid Dynamics methods. Popular CAE software ANSYS CFX was used for analyzation and simulation. The coordinates were imported to the geometry modeller of ANSYS. CFX (Fluid Flow) was used to generate mesh and conduct the experimentation. At the end of modelling and meshing, simulation was done to observe the effect of flow at a high Reynolds number, on the said aerofoil model with respect to pressure distribution and various aerodynamic forces while maintaining variation in its angle of attack.

Keywords: Aerofoil, Linear cascade, Wind Tunnel, Computational Fluid Dynamics, ANSYS.

I. INTRODUCTION

Linear Cascade

A row of blades representing the blade ring of an actual turbo machine is called cascade, grid, lattice or a mesh of blades. In a straight or "rectilinear cascade" the blade is arranged in a straight line. The blade can also be arranged in an annulus, thus representing an actual blade row. This arrangement is known as an "annular cascade" and is closer to the real-life situation. The aforementioned arrangements are employed for the cascade of axial-flow turbo machines.
When the flow through the ring of the blade is in the radial direction, the arrangement is known as a radical cascade.
Before studying a particular turbine, compressor or fan stage, it will be useful to study the behaviour of flow through the blade rows of such a machine. As far as such a study is concerned, it is immaterial whether the flow occurs over a row of stationary blades (absolute flow) or moving blades (relative flow). The quantities that matter are flow parameters and blades rows. Models of flow in an actual machine can be constructed in stationary rows of blades (cascade) by maintaining the geometric, kinematic and dynamic similarities as far as possible.

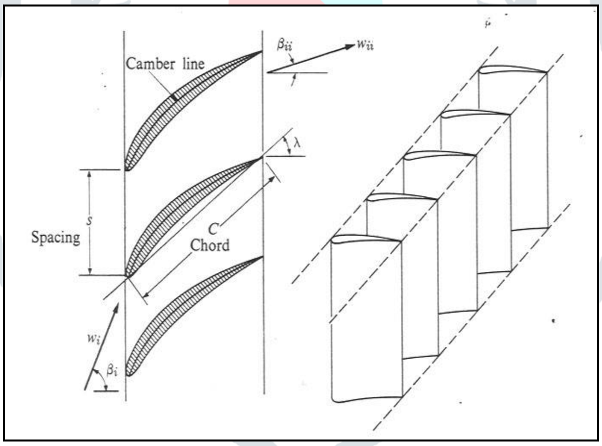

Fig.1 The geometry of a linear Cascade

II. OBJECTIVE

The objective of this project is to understand the phenomena of the uniqueness of aerofoil shape. Aerofoil shapes are employed in aircraft sectors as well as in automobile and energy production sectors e.g. steam turbines, gas turbines, wind turbines, etc. It can generate lift as well as a down force when used in a specific manner. So it is quite important to decode the phenomena behind its shape and the process by which it produces necessary to lift and down force.

Following are the essential objectives of the work:
Pressure distribution study of suction surface & pressure surface of blades, lift force & drag force of aerofoil profile using ANSYS at various AOA. Variation of AOA like -10, -5, 5, 10 degree.

III. RESULTS AND DISCUSSION

Fluent outcomes

This figure highlighting the mesh of an aerofoil with a c mesh domain. The mapped meshing is created on the entire c-domain. The cross-section is developed to be fine at areas near to the aerofoil and coarser more remote far placed irrespective of the position of the aerofoil. For this particular aerofoil, a quadratic formation of an element was utilized. The mesh has to be smooth and fine also in some regions away from the aerofoil. Various edge sizing has been adopted to accomplish the task.

Pressure Distribution

The above diagrams have their significance and also the colour tone has its meaning. The Above diagram is a contour diagram for an angle of attack -10, -5, 5 and 10 degree which says that how lift is generated and how Bernoulli's law comes to the play. exactly when flow occurs over an aerofoil after due course of time as flow keep on cross through an initial starting vortex get formed behind the aerofoil and to counteract that developed anticlockwise vortex a velocity vector get into the process due to conservation of angular momentum and subsequently that velocity get added with the mainstream velocity and give rise to higher velocity over the upper surface of an aerofoil and lower velocity under the lower surface which consequently as per Bernoulli's results in higher pressure in the lower side and higher pressure over the top surface.

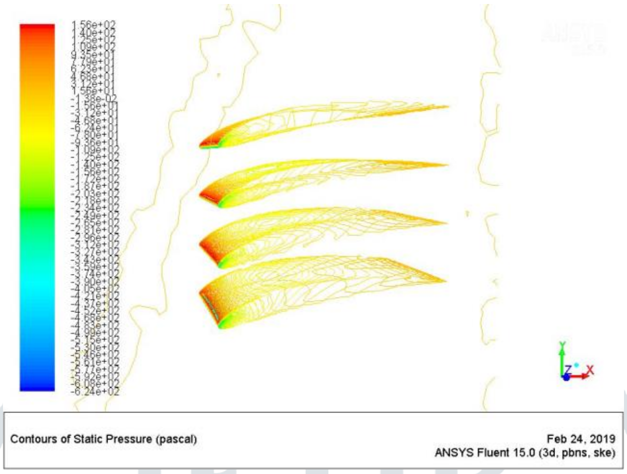

Fig.2 Pressure difference at Angle Of Attack -10^0

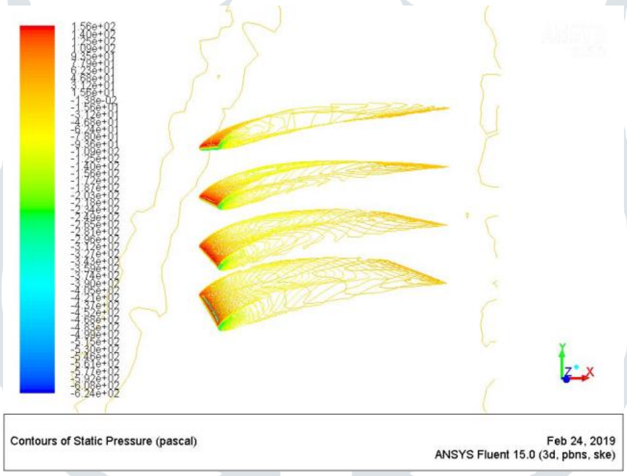

Fig.2 Pressure difference at Angle Of Attack -5^0

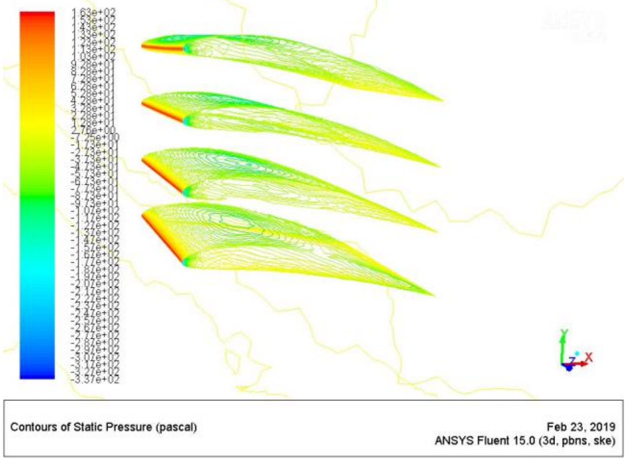

Fig.3 Pressure difference at Angle Of Attack 5^0

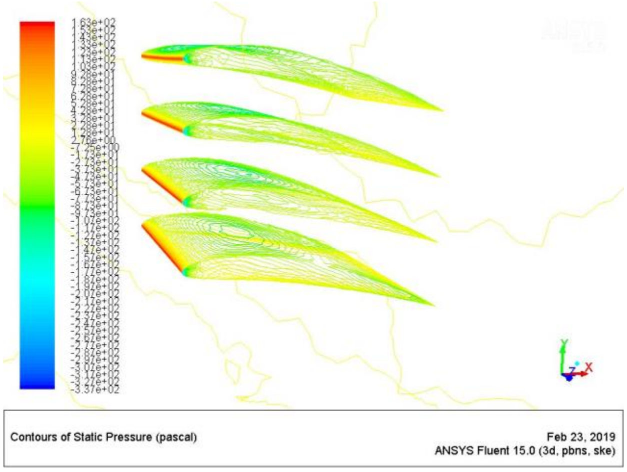

Fig.4 Pressure difference at Angle Of Attack 10^0

Tabel-1 Different Angle Of Attack Drag & Lift Force

Angle of attack	Drag force	Lift force
-10^0	0.10411225	0.11862871
-5^0	0.15695418	0.039974764
5^0	0.31657001	0.27307215
10^0	0.31609893	0.27439806

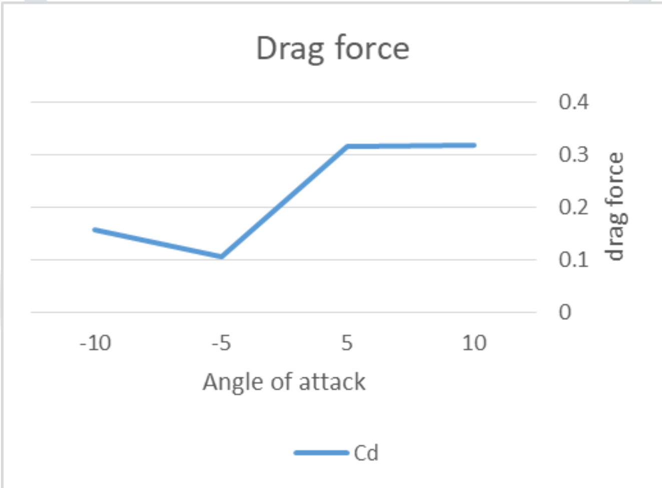

Fig.5 Coefficient of drag

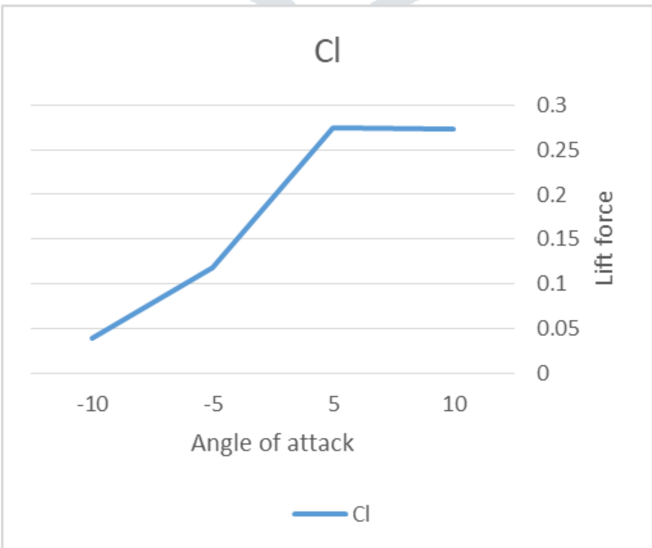

Fig.6 Coefficient of lift

The above reflects how the coefficient of lift and coefficient of drag varies concerning the angle of attack. The above graph depicts that the coefficient of lift increase as per the increment of the angle of attack whereas the coefficient of drag shows some gradual response. Coefficient of drag increases slowly till the angle of attack 10 degrees and after that, it shows spontaneous and upwards increments.

IV. CONCLUSIONS

This research work presents the simulated flow over a NACA 6409 aerofoil linear cascade model with a low Reynolds number at various angles of attack.

1) In all cases, it has been observed that the lift force and drag force increase respectively as the angle of attack increases until it reaches stall.
2) 10º angle of attack is the optimum angle. At this specific angle, the said aerofoil model produces the maximum ratio of Lift & Drag, which is ideal for efficient performance compared to any other angles of attack.
3) It has been found that aerofoil experiences low pressure at its top surface, whereas high pressure at its bottom. This occurs because when a plane moves forward, the air pushes on the bottom of the aerofoil increasing pressure. Meanwhile, air moves faster over the top of a wing, which results in an area of lower pressure. The difference between high and low pressure generates lift. This phenomenon justifies Bernoulli's Principle which states that pressure decreases when air moves faster.

V. ACKNOWLEDGMENT20

I wish to express my sincere thanks to the project guide Dr. Umang J. Patdiwala and all Faculties of Mechanical Department for his cordial support in every step and constant encouragement for this paper, help, and guidance given by him time to time shall carry me to publish thispaper.

VI. REFERENCES

[1] Askan, A, Tangoz, S., The effect of aspect ratio on the aerodynamic performances and flow separation behaviour were investigated on a model rectangular wing composed four different profile, 6th Europe Conferences Renewable Energy System,2018
[2] B Antony Samue and Rinku Mukherjee, A study of the unsteady aerodynamics of a wing at high angles of attack using decambering to model separated flow, Indian Institute of Technology, Madras, Chennai 600036, India, MS received 16 May 2017; revised 9 December 2017; accepted 18 January 2018; published online 27 June 2018
[3] M r M. Boer, Wind Tunnel Testing of a NACA0012 Aerofoil, University of the Witwatersrand, Johannesburg School of Mechanical, Industrial & Aeronautical Engineering Johannesburg, June 2017
[4] Shivananda Sarkar, Shaheen Beg Mughal, CFD analysis of the effect of flow over NACA 2412 aerofoil through the shear stress transport turbulence model, Proceedings of ASAR International Conference, 07th May 2017, New Delhi, India
[5] Dr. John E Matsson, John A. Voth, Mr. Connor A. McCain, Mr. Connor, Aerodynamic Performance of the NACA 2412 Aerofoil at Low Reynolds Number, McGraw, ASEE's 123rd Annual Conference & Exposition, New Orleans, LA, June 26-29,2016
[6] Er. Shivam Saxena1, Mr. Rahul Kumar, Design of NACA 2412 and its Analysis at Different Angle of Attacks, Reynolds Numbers, and a wind tunnel test, International Journal of Engineering Research and General Science Volume 3 Issue 2, March-April, 2015
[7] Kumar, V., Tomar, V., Kumar, N., and Jain, S., Flow Simulation and Theoretical Investigation on Aerodynamics of NACA-2415 Aerofoil at Low Reynolds Number, Article in SAE Technical Papers, 2015
[8] Chen Fen-fen, Gui Xing-min, Jin Dong-hai, Qiu Dao-bin, Influence of Geometric Scaling on Linear Cascade Aerodynamic Performance, Asia-Pacific International Symposium on Aerospace Technology, 2014
[9] Prasad Chougule, Søren R.K. Nielsen, Simulation of flow over double-element aerofoil and wind tunnel test for use in vertical axis wind turbine, Journal of Physics: Conference Series 524 (2014) 012009
[10] K. Mulleners, P. Gilge, and S. Hohenstein, Impact of Surface Roughness on the Turbulent Wake Flow of a Turbine Blade, Hindawi Publishing Corporation Journal of Aerodynamics, 2014
[11] Kondapalli Siva Prasad, Vommi Krishna, B.B. Ashok Kumar, Aerofoil Profile Analysis and Design Optimization, Journal of Aerospace Engineering and Technology, 2013
[12] Brent W. Pomeroy, Gregory A. Williamson, and Michael S.Selig., Experimental Study of a Multi-element Aerofoil for Large Wind Turbines,30th AIAA Applied Aerodynamics Conference, 2012
[13] W. B. Tay • K. B. Lim, Analysis of non-symmetrical flapping aerofoils, Chinese Society of Theoretical and Applied Mechanics and Springer-Verlag GmbH, 2009
[14] Paul M. Kodzwa Jr., Amanda Vicharelli, Gorazd Medic, Christopher J. Elkins, John K. Eaton, Evaluation of alternatives for two-dimensional linear cascade facilities, Aerospace Engineering Publications, 2009
[15] Dan M. Somers, James L. Tangler, Wind-Tunnel Tests of Two Aerofoils for Wind Turbines Operating at High Reynolds Numbers, Presented at the ASME Wind Energy Symposium Reno, Nevada, January 10.13, 2000