

Design and simulation of flow through subsonic axial compressor

Mutahir Ahmed¹, Saeed Badshah², Rafi Ullah Khan², Muhammad sajjad², Sakhi Jan²

¹NESCOM, Islamabad Pakistan

²Department of Mechanical Engineering, International Islamic University, Islamabad Pakistan

¹mutahir.mir@hotmail.com

Abstract—This paper illustrates the theory behind computational fluid dynamics (CFD) of flow through a gas turbine engine low pressure compressor. The goal of study is to develop a tool to perform shape optimization of low speed airfoils. Mathematical model and computational tool are developed using programming software. Analytical solution is developed to create airfoil geometry. The NACA 4 digits library is used with design parameters that control camber and the thickness of the airfoil. Solver is capable of providing derivatives of the objective function and limiting constraints with the solution for each set of parameters.

Keywords: Panel method, point vortex, potential flow, compressor design, cascade flow,

I. INTRODUCTION

Axial compressor design has been a complex problem for the designers ever since, due to its complex flow. The designing of the compressors has well been established in the west since early 20th century. However, with the increasing need of highly efficient, high speed and long-range civil as well as military aircrafts, the designing of jet engine and axial compressor is still important for the designers [1-2]. This need keeps them researching for the efficient methods of designing the compressor, with concentrated effort in the designing of the blade profile due to its important role in deciding the effectiveness of the compressor. Usually algorithms are programmed for rapid solution of classical boundary value problems (Dirichletan Neumann) for the Laplace equation based on iteratively solving integral equation of potential theory [3]. Lu et al. [4] investigated fundamental mechanisms of axial skewed slot casing treatment and their effects on the subsonic axial-flow compressor flow field, the coupled unsteady flow through a subsonic compressor rotor and the axial skewed slot. They found that the axial skewed slot casing treatment can increase the stall margin of subsonic compressor by repositioning of the tip clearance flow trajectory further toward the trailing of the blade passage and retarding the movement of the incoming /tip clearance flow interface toward the rotor leading edge plane. Mc Glumphy et al. [5] investigated fluid mechanics of a tandem rotor in the rear stages of a core compressor. The literature reveals that the flow through axial compressors has been least investigated.

NOMENCLATURE

GTE = Gas turbine engine

C_p = Coefficient of pressure

LPCs = Low pressure compressors

M_b = Blade mach number

C_L = Coefficient of lift

k = Ratio of specific heats

P_c = Total pressure rise in compressor

II. GAS TURBINE ENGINE SELECTION

A. GTE Parameters

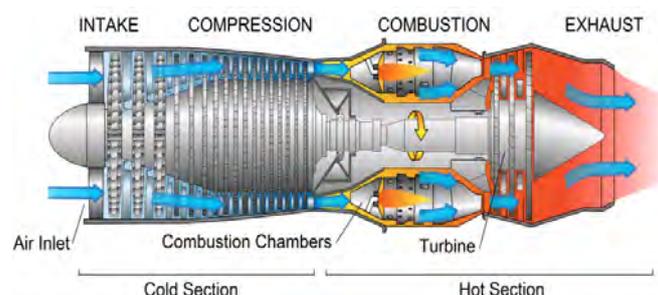


Figure: 1 Typical gas turbine engine assembly [20]

In newly designed GTE, performance calculations were carried to select pressure ratio across LPCs, number of compressor stages etc. which finally lead to velocity triangles and then airfoil geometry.

Table: 1 Engine specifications

Altitude & Mach No.	RPM	Compressor pressure ratio
1000m & 0.6	22,000	2.2
Number of stages	Pressure rise per stage	Turbine inlet temperature
3	$P_{ps} = P_c^{1/3} = 1.3$	1200 K
Air to fuel ratio	Fuel flow rate	Axial velocity ratio
65: 1	0.1 Kg/s	1.1

Isentropic efficiencies of compressor " η_c ", turbine, η_t , intake, η_i , propelling Nozzle " η_j " in percentage are 85, 80, 93, and 95 respectively.

B. Velocity Triangles

A cross section and a top view of a typical axial-flow compressor and flow through a cascade {an endless repeating array of airfoil that results from the "unwrapping" of the stationary (stators) and rotating (rotor) airfoil} is shown in Figure.2 [6].

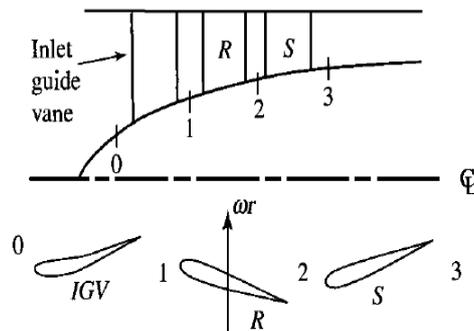


Figure: 2 Typical Axial Flow Compressor & Cascade flow [2]

Depending on the design, an inlet guide vane (IGV) may be used to deflect the incoming airflow to a pre-determined angle toward the direction of rotation of the rotor. The rotor increases the angular velocity of the fluid, resulting in increases in total temperature, total pressure, and static pressure. The following stator decreases the angular velocity of the fluid, resulting in an increase in the static pressure, and sets the flow up for the following rotor [6][7]. Each cascade passage acts as a small diffuser, and it is said to be well designed when it provides a static pressure rise without incurring total pressure losses and/or flow instabilities due to shock waves and/or boundary-layer separation. The changes in fluid velocity induced by the blade rows are related to changes in the fluid's thermodynamic properties in this section. The analysis is concerned with only the flow far upstream and far downstream of a cascade, the regions where the flow-fields are uniform. In this manner, the details about the flow-fields are not needed, and performance can be related to just the changes in fluid properties across a blade row [6] [7].

In the analysis that follows, two different coordinate systems are used: one fixed to the compressor housing (absolute) and the other fixed to the rotating blades (relative). The static (thermodynamic) properties *do not* depend on the reference frame. However, the *total properties do depend on the reference frame*. The velocity of a fluid in one reference frame is easily converted to the other frame by the following equation:

$$V = V_R + U, \text{ Where,}$$

V = Velocity in stationary coordinate system

V_R = Velocity in moving coordinate system

U = Velocity of moving coordinate system ($= \omega r$)

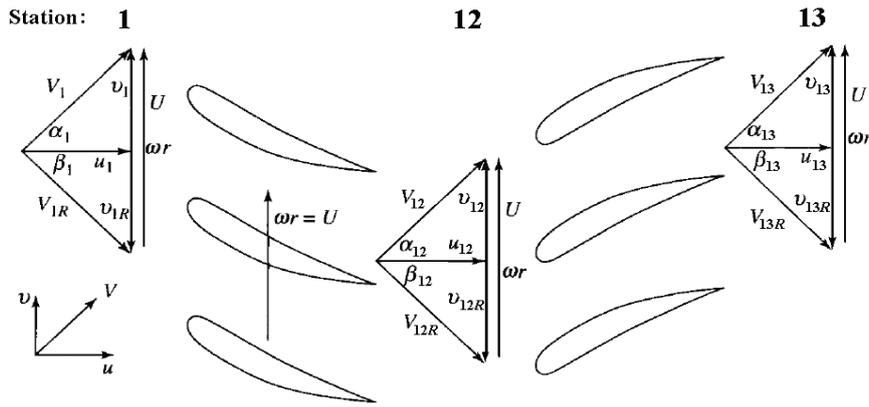


Figure: 3 Flow through single compressor stage [2]

If we consider that the flow angles and the velocity ratio \$u_2/u_1\$ are functions of the geometry, then the stage temperature rise is mainly a function of \$M_b\$ and the ratio \$M_1/M_b\$. From eq. (1), we can see that high \$M_b\$ is desirable to give higher stage temperature rise.

$$\frac{\Delta T_0}{T_{01}} = \frac{(k_{12} - 1)M_b^2}{1 + [(k_{12} - 1)/2]M_1^2} \left[1 - \cos \alpha_1 \frac{M_1}{M_b} \left(\frac{u_{12}}{u_1} \tan \beta_{12} + \tan \alpha_1 \right) \right] \quad (1)$$

C: 2D-Airfoil geometry

Mean camber line co-ordinates and thickness distribution above and below mean line is computed as follows [8]

$$y_c = \frac{m}{p^2} (2px - x^2) \quad \text{from } x = 0 \text{ to } p$$

$$y_c = \frac{m}{1 - p^2} ((1 - 2p) + 2px - x^2) \quad \text{from } x = p \text{ to } c$$

$$\pm y_t = \frac{t}{0.2} \{ 0.2969\sqrt{x} + 0.1260x - 0.351x^2 + 0.2843x^3 - 0.1015x^4 \}$$

$$X_u = x - y_t \sin \theta, Y_u = y_c + y_t \cos \theta \quad \text{Upper Surface}$$

$$X_L = x + y_t \sin \theta, Y_L = y_c - y_t \cos \theta \quad \text{Lower Surface}$$

Where,

$$\theta = \arctan \left(\frac{dy}{dx} \right)$$

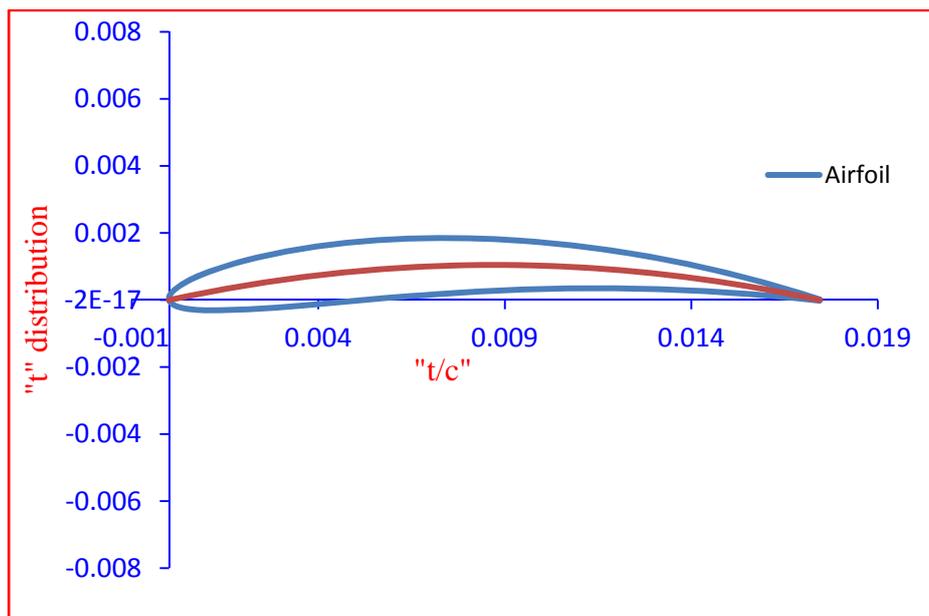


Figure: 4 2D airfoil of compressor blade at mean height (MATLAB code results)

Airfoil of compressor only depends upon the flow conditions, or flow conditions dictate the geometry of airfoil. But when choosing the blade angle, it must be remembered that air angles have been calculated for the design speed and pressure ratio, and under different operating conditions both the fluid velocities and blade speed may change with resulting changes in the air angles [6][9][2]. On the other hand, the blade angles, once chosen, are fixed. It follows that to obtain the best performance over a range of operating conditions it may not be the best policy to make the blade inlet angle equal to the design value of the relative air angle.

III. THEORY-INVISCID IN-COMPRESSIBLE FLOW

Vorticity in the high Reynolds number flow-fields is confined to the boundary layer and wake regions where the influence of viscosity is not negligible and so it is appropriate to assume an irrotational as well as inviscid flow outside these confine regions[8][9][10]. For many purposes, inviscid flow simulation by mean of two dimensional potential flow theory and point vortices provides valuable information about un-steady flow over high- aspect-ratio wings[10][11].

The continuity equation for an incompressible fluid reduces to

$$\nabla \cdot \mathbf{q} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (2)$$

Consider the following line integral in a simply connected region, along the line C (similar to Fig-5):

$$\int_C \mathbf{q} \cdot d\mathbf{l} = \int_C u dx + v dy + w dz \quad (3)$$

If the flow is irrotational in the region then $u dx + v dy + w dz$ is an exact differential of a potential ϕ that is independent of the integration path C and is a function of the location of a point P (x, y, z):

$$\phi_{(x,y,z)} = \int_{P_0}^P u dx + v dy + w dz \quad (4)$$

Where P_0 is an arbitrary reference point, ϕ is called the velocity potential and the velocity at each point can be obtained as its gradient

$$\mathbf{q} = \nabla \phi \quad (5)$$

And in Cartesian coordinates

$$u = \frac{\partial \phi}{\partial x} \quad v = \frac{\partial \phi}{\partial y} \quad w = \frac{\partial \phi}{\partial z}$$

The substitution of eq. (5) into the continuity eq. (2) leads to the following differential equation for the velocity potential.

$$\nabla \cdot \mathbf{q} = \nabla \cdot \nabla \phi = \nabla^2 \phi = 0 \quad (6)$$

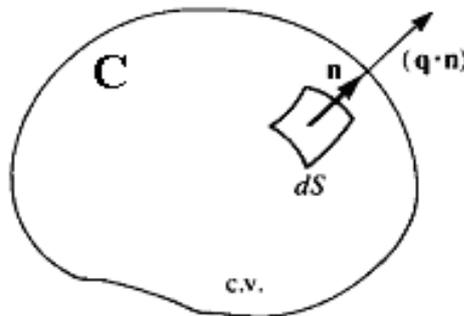


Figure: 5 Control volume in fluid [2]

Since the fluid's viscosity has been neglected, the no-slip boundary condition on a solid-fluid boundary cannot be enforced [12]. In a more general form, the boundary condition states that the normal component of the relative velocity between the fluid and the solid surface (which may have a velocity \mathbf{q}_B) is zero on the boundary:

$$\mathbf{n} \cdot (\mathbf{q} - \mathbf{q}_B) = 0 \quad (7)$$

A zero normal velocity component $\frac{\partial \phi}{\partial n} = 0$ on the surface is "direct" formulation called the Neumann problem [3]. This boundary condition is physically reasonable and is consistent with the proper mathematical formulation [13]. For an irrotational inviscid incompressible flow it now appears that the velocity field can be obtained from a solution of Laplace's equation (6) for the velocity potential. Note that we have not yet used the Euler equation, which connects the velocity to the pressure. Once the velocity field is obtained it is necessary to obtain the pressure distribution on the body surface to allow for a circulation of the aerodynamic forces and

moments [14].The Two-Dimensional Point and constant strength vortex method for an isolated and cascade airfoil are used, based on the level of approximation of the singularity distribution, surface geometry, and type of boundary conditions [10].Here it is important to give an idea about *Kutta-Joukowski condition*

$$\Gamma_1 + \Gamma_{NP} = 0 \quad (8)$$

To determine the circulation about the airfoil we need an additional condition on the flow field. If we think of the total flow as being composed of a uniform contribution (with no circulation) plus a circulatory contribution, then the circulation Γ will adjust itself until the total flow leaving the trailing edge of the airfoil is smooth. This is called the *Kutta-Joukowski condition*, and uniquely determines the circulation, and therefore the lift, on the airfoil[15].Initially flow is without circulation, with two stagnation points on the upper and lower surfaces of the airfoil. The fluid on the lower surface of the airfoil must accelerate around the sharp trailing edge in order to reach the rear stagnation point on the upper surface [2][14].

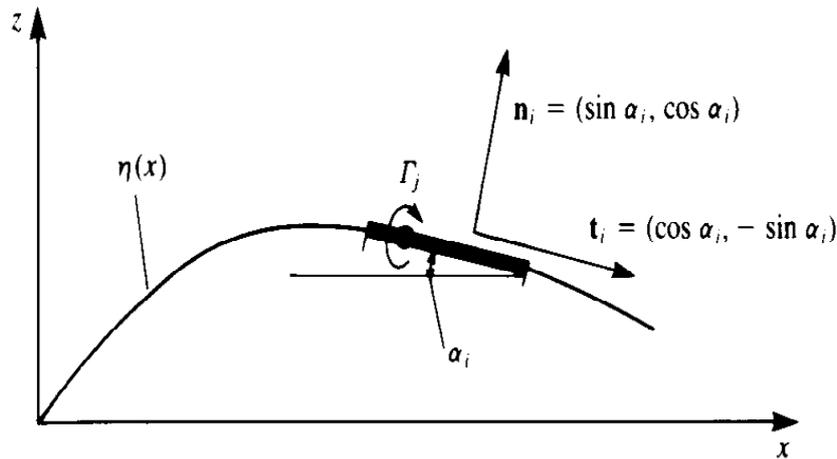


Figure: 6 Nomenclature of panel method [2]

This actually requires that the fluid velocity at the trailing edge be infinite--an unlikely circumstance in a viscous fluid. This flow configuration is unstable, and the rear stagnation point gets washed downstream, until it coincides with the trailing edge of the airfoil [2][14].

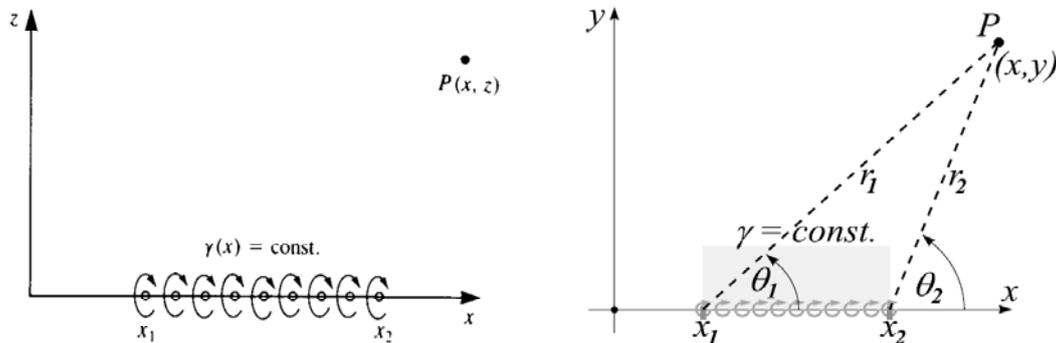


Figure: 7 Constant Strength Vortex distributions along x-axis [2]

At this point there is a net circulation about the airfoil, with the zero velocity stagnation point “cancelling” the infinite velocity at the trailing edge, resulting in smooth flow from the trailing edge. Now in a non-viscous fluid circulation is conserved, this is known as the Kelvin circulation theorem [2][14]. The constant strength vortex distribution is placed along the **x-axis** $\gamma(x) = \gamma = constant$ as shown in Fig-8.The influence of this distribution at a point "P" is an integral of the influences of the point elements between $x_1 \rightarrow x_2$.

Finally creating/solving mathematical model for point vortex method in a cascade, resulting in following [14][2][12]

$$u - iv = \frac{\Gamma}{2\pi i} \left[\cot \left(\frac{(z_c - z_v)\pi}{h^*} \right) X \frac{\pi}{h^*} \right] \quad (9)$$

Induced velocities in x and y direction (u and v respectively) can be calculated directly by using eq. (9). *Sign for “u” and “v” will depend upon the direction of circulation, also depending upon the directional calculation of airfoil (Clockwise or Anti-Clockwise)*

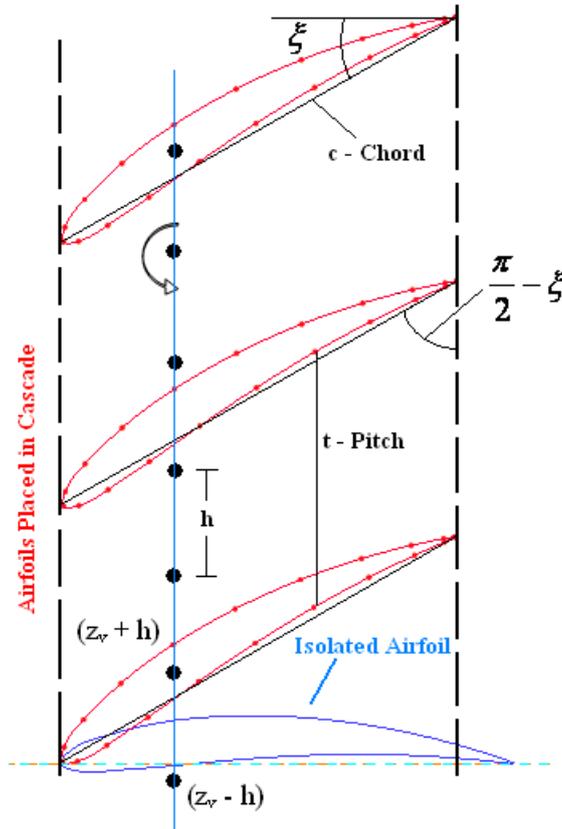


Figure: 8 Transformation of Isolated Airfoil to Cascade (MATLAB code results)

Note: To validate final equation, if higher value of “t” (Pitch) is used, it will give nearly the same results as Point Vortex Solution for an Isolated Airfoil[16][2][14].

$$u - iv = \frac{x_2 - x_1}{x} \sum \frac{\gamma}{2\pi i} \underset{\substack{\text{weighing} \\ \text{Function}}}{W} \left[\cot \left(\frac{(z_c - z_v)\pi}{h^*} \right) X \frac{\pi}{h^*} \right] - \sum \frac{\gamma(x_2 - x_1)}{x} \left(\frac{1}{z_c - z_v} \right) + \frac{\gamma}{2} - i \frac{\gamma}{4\pi} \ln \frac{(x - x_2)^2}{(x - x_1)^2} \quad (10)$$

Eq. (10) is the final equation after solving the analytical relations, defines the calculation of induced velocities for airfoil in cascade for constant strength vortex solution.

IV. RESULTS & VALIDATION

(MATLAB code results)

Theory session explains the methods to calculate flow around airfoil in isolated and cascade modes. Figure 9 clearly explains the effect of variation in inlet and exit flow angles of blades on velocity distribution from root to tip. Coefficient of pressure “C_p” at each location around isolated airfoil is calculated using point vortex method as well as constant strength method as shown in Figure 10 and Figure 12. Also coefficient of lift is calculated, whereas isolated airfoil approximation is good enough in case of airplane wings. But in compressor rotors or cascades, each blade feels the presence of adjacent blade. Therefore C_p distribution in cascade is calculated using equation 9 & 10 for point vortex and constant strength methods separately as shown in Figure 11 and Figure 13. MATLAB code for analytical calculations of isolated airfoil and airfoil in cascade must be validated. Therefore in order to verify the solution, simply pitch will be increased, so that influence of one airfoil will be negligible on other airfoils. So this can also be treated as an isolated airfoil, but the solution or calculations are carried out using cascade phenomenon. As result, C_p as shown in Figure 14 for point vortex and constant strength methods is same. Selection of “angle of attack” is a key factor for variation in flow angles, pressures as well as velocities around airfoil. Figure 15 shows variation in C_p distribution in cascade using point vortex theory.

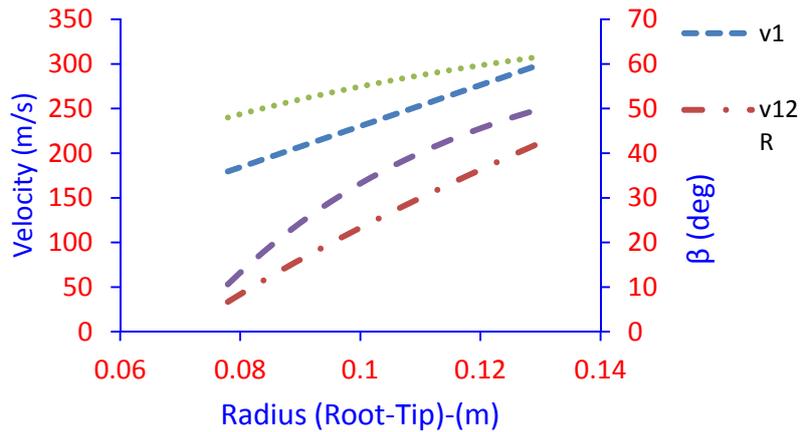


Figure: 9 Swirl velocity distribution and variation of rotor flow angles

C_p distribution is a prerequisite to calculate velocity distribution around airfoil. Again using equation 9 & 10, velocity distribution around airfoil using point vortex and constant strength methods is calculated as shown in Figure 16 & 17, respectively. C_p and velocity distribution due to various angle of attacks, helps in optimization of airfoil profile[17]. Effect of variation in angle of attack on velocity around airfoil for the two theories, is shown in Figure 18 & 19.

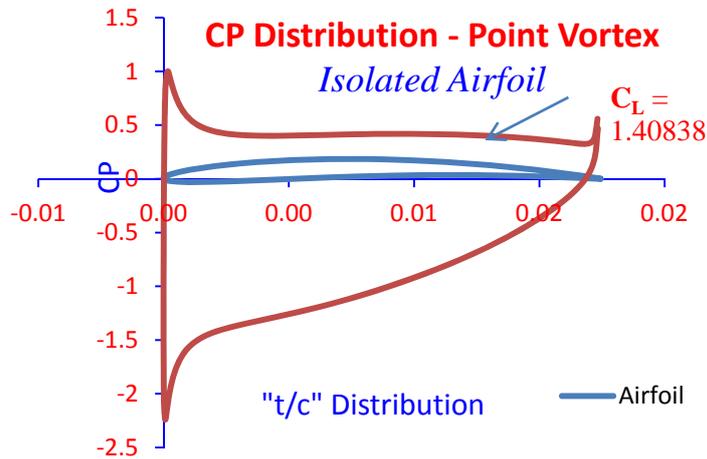


Figure: 10“ C_p ” Distribution around isolated airfoil (Point vortex theory)

Finally it is important to validate MATLAB code made on analytical calculations using point vortex and constant strength theories. Therefore NACA 4412 airfoil geometry is selected and solved in the MATLAB code. Results of NACA 4412 airfoil geometry solved using panel method shows promising results in comparison to the results extracted from MATLAB code as shown in Figure 20. Finally Comparison of theoretical results from analytical calculations with experimental data of NACA airfoil is shown in Figure 21.

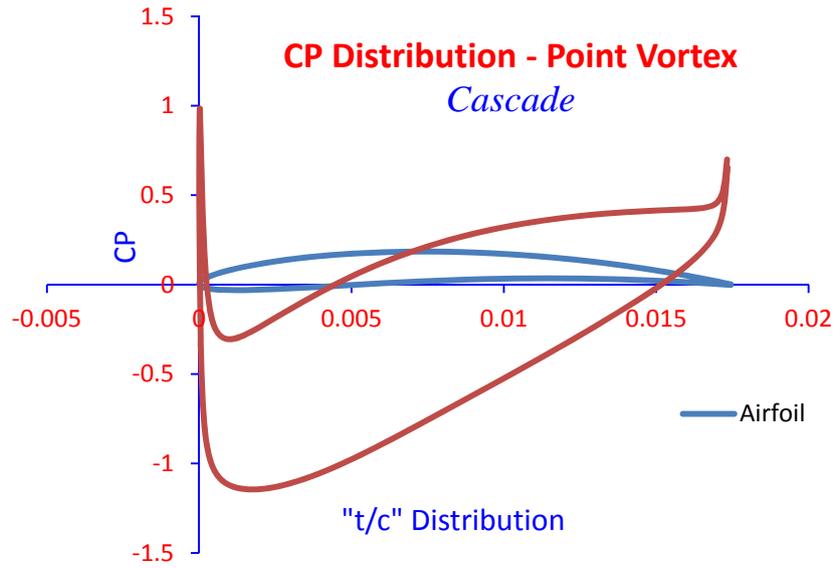


Figure: 11 "Cp" Distribution around airfoil in cascade (Point vortex theory)

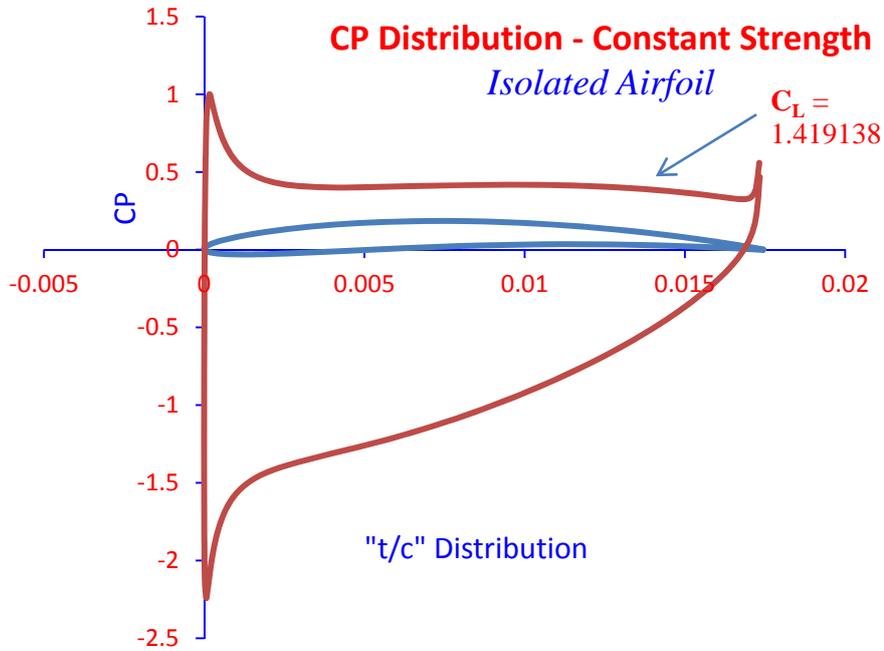


Figure: 12 "Cp" Distribution around isolated airfoil (Constant strength theory)

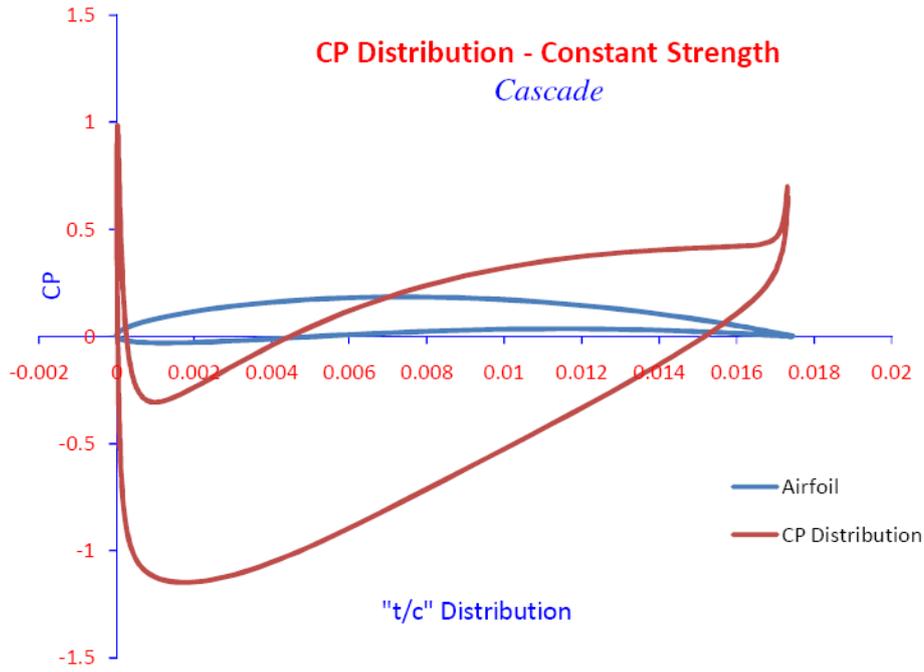


Figure: 13 "Cp" Distribution in cascade (Constant strength theory)

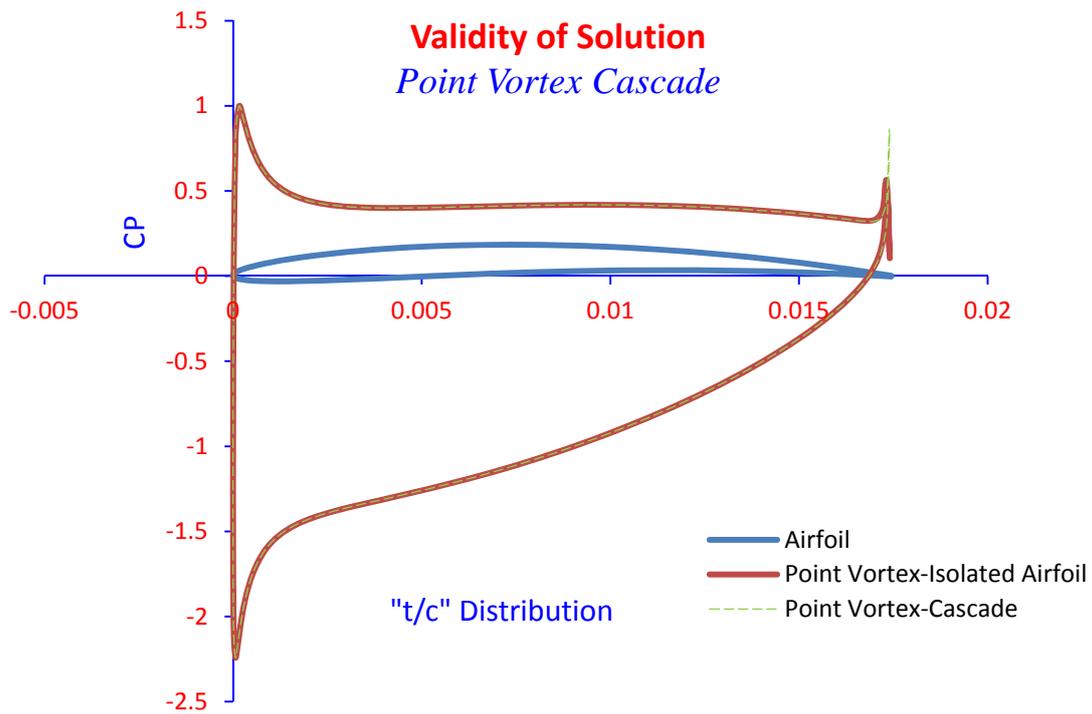


Figure: 14 Software validation for point vortex and constant strength theories

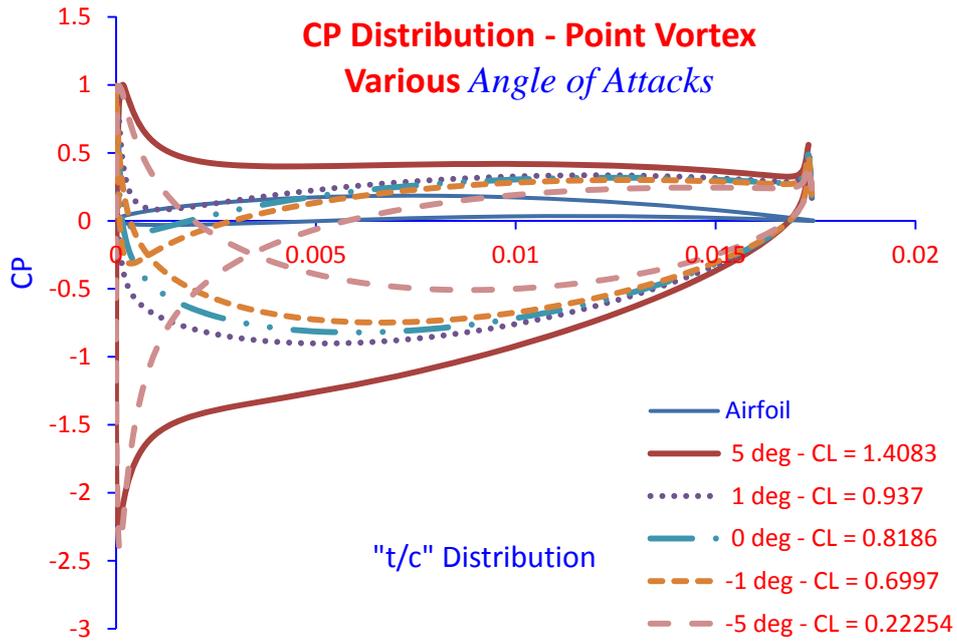


Figure: 15 "C_p" distribution at various angle of attacks (Point vortex theory)

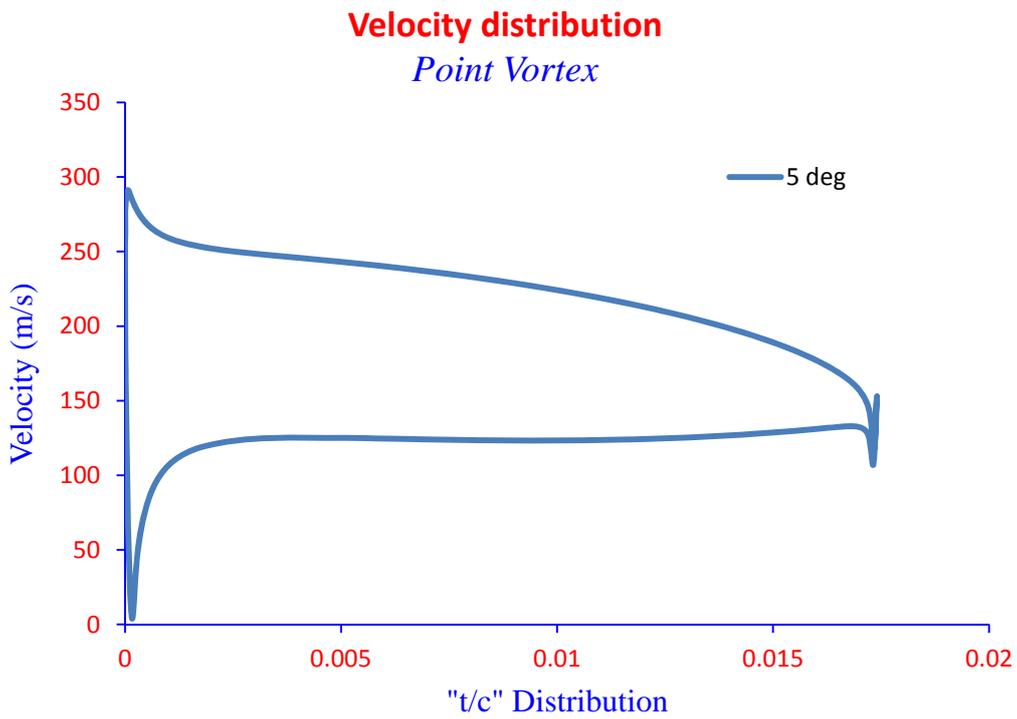


Figure: 16 Velocity distribution around airfoil (Point vortex theory)

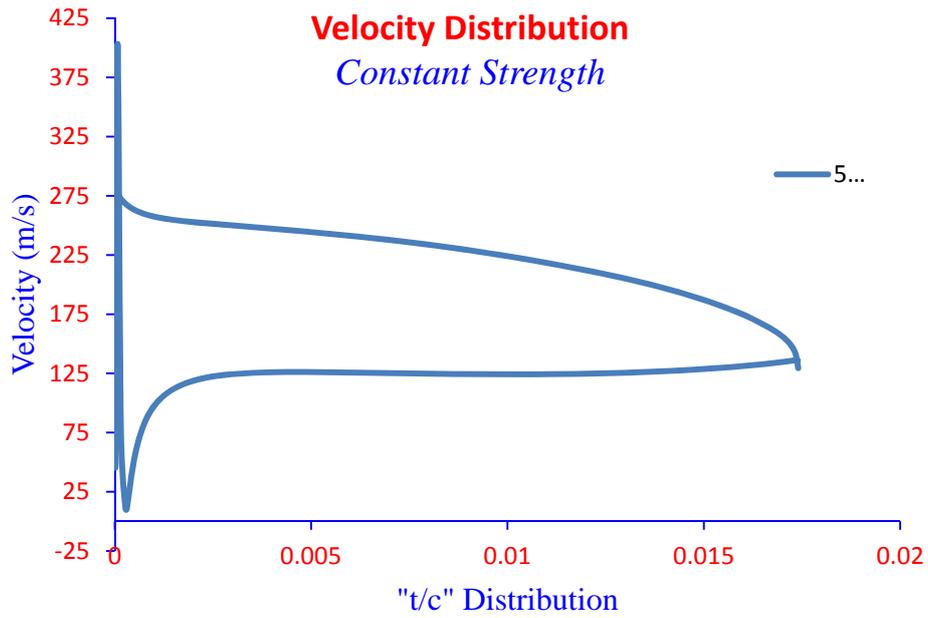


Figure: 17 Velocity distribution around airfoil (Constant strength theory)

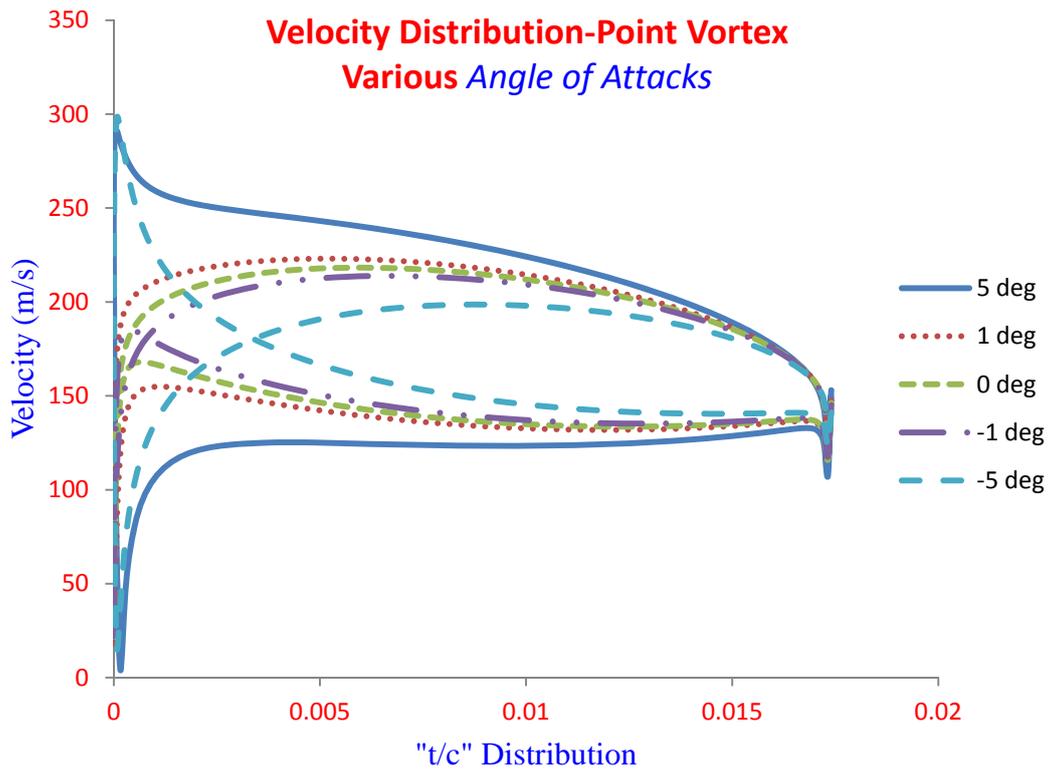


Figure: 18 Velocity distributions at various angles of attack (Point vortex theory)

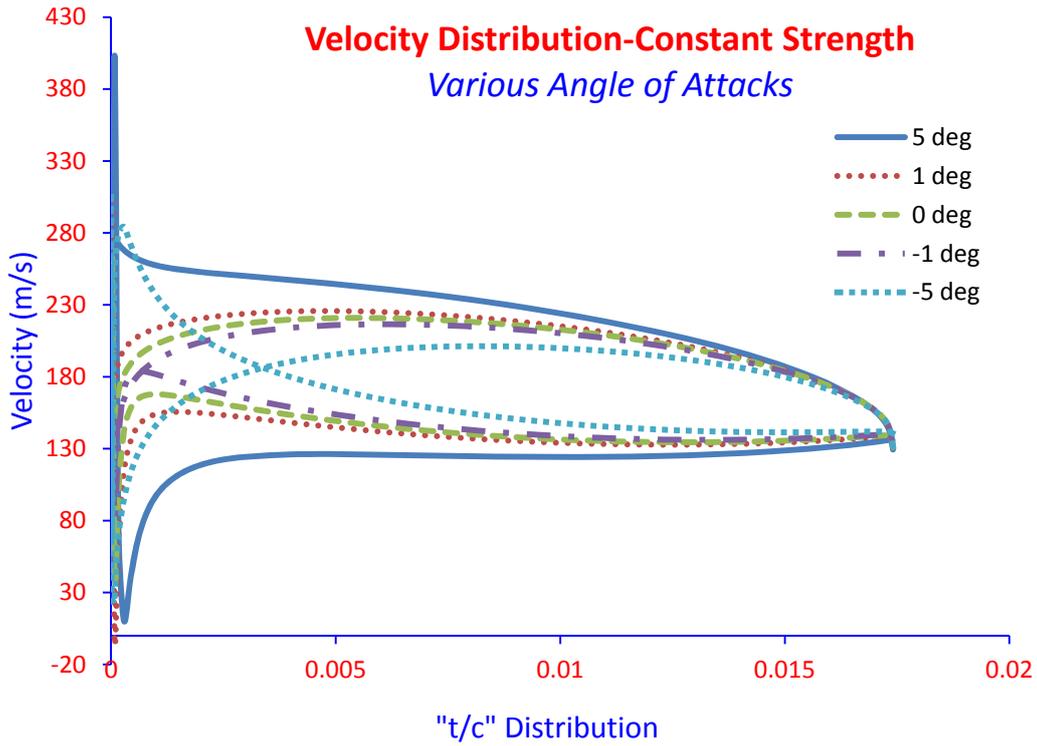


Figure: 19 Velocity distributions at various angles of attack (Constant strength theory)

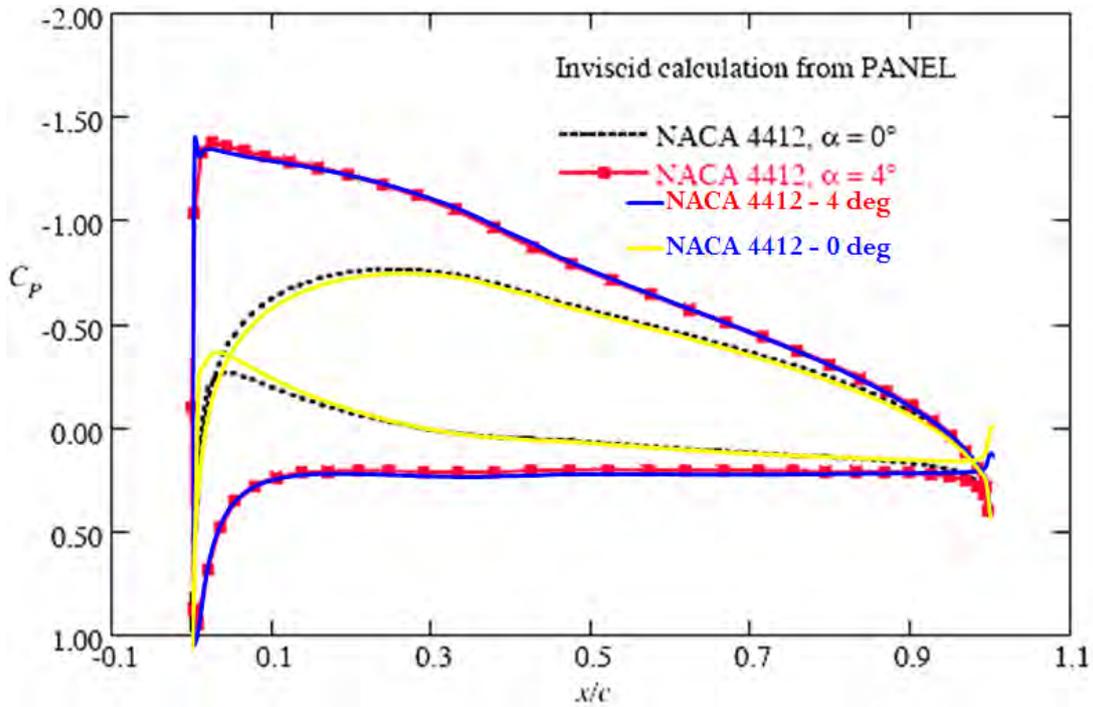


Figure: 20 MATLAB code validations [21]

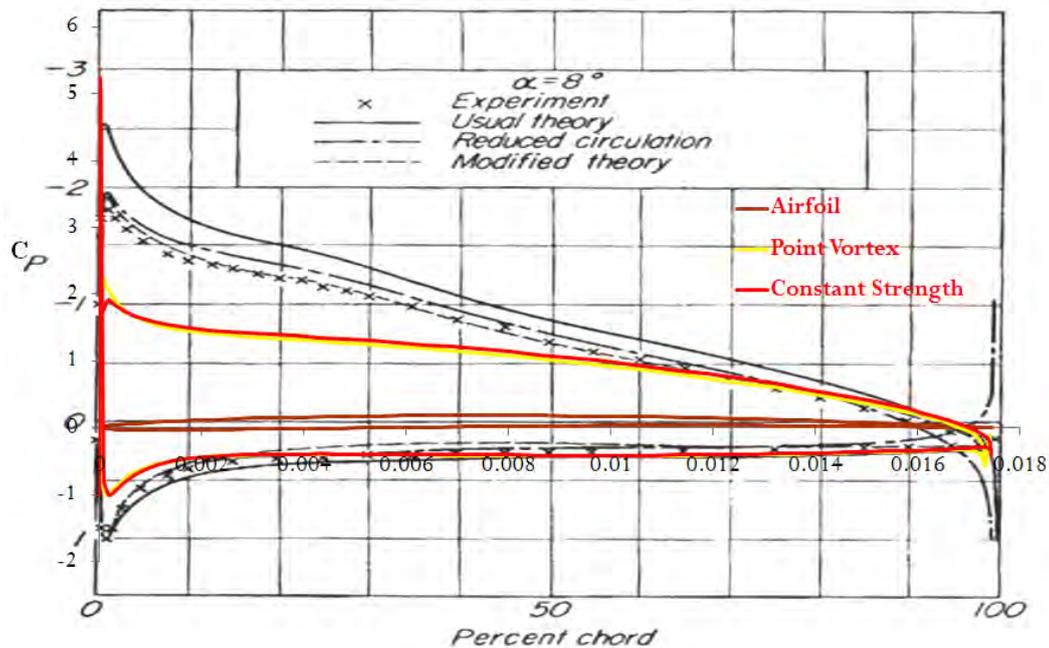


Figure: 21 Comparison of theoretical results with experimental data of NACA airfoil [21]

V. CONCLUSIONS & RECOMMENDATIONS

Flow around airfoil calculations has some limits, merits and demerits, discussed below.

- According to study conducted by Joseph R. Casper [18] geometrically cascades have stagger and spacing, which are meaningless for isolated airfoils, and cascade airfoil generally are thick and have more camber than isolated airfoils. Therefore it should be catered in detailed study.
- Design optimization of axial flow compressor blades can be done by 3-D Navier stock's equation [19].
- Singularity method is the alternate method to calculate flow around airfoil without solving Laplace's equation.
- Singularity method uses superposition; this limits the solution to be carried out only for linear equations.
- Trailing edge point is singular (have two tangents) so must be catered separately.
- Constant strength vortex solution can be improved by using linearly varying and then quadratic varying strength distribution.
- C_p must be same for trailing edge, this ensures the same velocity at leading edge, and there is no shift of flow on suction or pressure side of the airfoil.
- C_L depends upon the area between the C_p curves for upper and lower airfoils.
- Airfoil shape could be approximated with parabolic other higher order polynomials.
- In order to improve the CFD results, 1st step is to validate the results with some specialized turbo machinery software, and if found satisfactory results, proceed towards testing.
- The detailed study of time steady three dimensional boundary layer must be done followed by unsteady case.
- Study to minimize the secondary losses due to the tip leakage is a potential area for research.
- Using basic potential flows for generating blade profile can be a better way of minimizing losses.
- Designing of transonic compressor is the need of time.
- 3D simulation will provide better insight into flow.
- Turbulence modeling is necessary to improve solution.

VI. REFERENCES

- [1] Elements of Propulsion-Gas Turbine and Rockets J. Mattingly
- [2] Low-Speed Aerodynamics. From Wing Theory to Panel Method, By Joseph Katz & Allen Plotkin
- [3] V Rokhin, " Journal of computational physics, Volume 60, issue 2 (1985) Page 187-207
- [4] Xingen Lu, Wuli Chu, Junqiang Zhu and Yangfeng Zhang , Numerical Investigations of the Coupled Flow Through a Subsonic Compressor Rotor and Axial Skewed Slot, J. Turbomach.131(1), 011001 (2008) (8 pages)
- [5] Jonathan McGlumphy, Wing-Fai Ng, Steven R. Wellborn and Severin Kempf, 3D Numerical Investigation of Tandem Airfoils for a Core Compressor Rotor, J. Turbomach. 132(3), 031009 (Mar 25, 2010)
- [6] Cascade aerodynamics by JP Gostelow
- [7] Fletcher Gas Turbine Performance 2E.
- [8] Report No. 824, Summary of Airfoil Data, by IRA H. Abbott, Albert E. VON Doenhoff and Louis S. Stivers, Jr. Langley Memorial Aeronautical Laboratory Langley Field, Va.
- [9] HOWELL, A.R. The present basis of axial flow compressor design: Part 1- Cascade theory and performance. ARC R&M 2095 (1942).

- [10] K. Streitlien and M.S. Triantafyllou, "Force and moment on a Joukowski profile in the presence of point vortices", AIAA Journal Vol.33, No 4 (1995) pp 603-610
- [11] Hassan Aref, James B Kalke, Iranuszzawadzki, " Point vortex dynamics", Volume 3, No. 1-4(1988)
- [12] Luigi Morino, " Subsonic potential aerodynamics for a complex configuration: A general theory", Volume 12, No2 (1979)
- [13] Brian Maskew, "Prediction of subsonic aerodynamic characteristics: a case for low-order panel methods", AIAA 81-0252R Vol.19, No 2, February 1982, Journal of aircraft, page 157-163.
- [14] CFD one. (Computational Fluid Dynamics, Part one), by Zlatko Petrovic and Slobodan Stupar. University of Belgrade, Mechanical Engineering faculty Belgrade, 1996.
- [15] Brian Maskew, " Prediction of subsonic aerodynamic characteristics. Vol. 19, No2 (1982), Journal of aircraft, AIAA 81-0252R, pp156-163.
- [16] CARTER, ADS. The Axial Compressor. " Gas Turbine Principles and Practice." (ed. Sir H.Roxbee-Cox) G. Newnes, London (1955).
- [17] Raymond M.Hicks and Preston A henne, " Wing design by numerical optimization", Volume 15, No.
- [18] Joseph R Casper, David E. Hobbs and Roger L. Davis, "Calculation of two dimensional potential cascade flow using finite area methods" 7(1978), Journal of aircraft, pp-407-412
- [19] Sang-yun lee, Kwang-Yong Kim, "Design optimization of axial flow compressor blades with three dimensional navier stock's solver". KSME international journal, Volume 14, No. 9, pp1005-1012, 2000.
- [20] http://en.wikipedia.org/wiki/File:Jet_engine.svg
- [21] http://www.dept.aoe.vt.edu/~mason/Mason_f/SubsonicAirfoilsPres



First Author I have been working in the field of finite element modeling and analysis of various mechanical and aerospace structures for the last eleven years and so. I have carried out involved analyses of various complex structures in the fields of solid mechanics and turbo machinery. During MSc in mechanical field with specialization in aerospace, I worked on "Design and simulation of flow through sub-sonic axial flow compressor". In project, aerodynamic design of a subsonic single stage axial flow compressor was studied and subsequently validated by developing a computational fluid dynamics (CFD) based simulation code. Initially, thermodynamics and fluid dynamics involved in the flow through a compressor were studied.