Asian Engineering Review
ISSN: 2409-6725
Vol. 2, No. 2, 23-29, 2015
http://asianonlinejournals.com/index.php/AER

Numerical Investigation of Aerofoil Cascade and Tandem Cascade Using Vortex Panel Method

J. Bruce Ralphin Rose[1*] --- M. Raguraman[2]

[1]Assistant Professor, Department of Aeronautical Engineering, Regional Centre of Anna University Tirunelveli, Tamilnadu, India
[2]PG Scholar, Department of Aeronautical Engineering, Regional Centre of Anna University Tirunelveli, Tamilnadu, India

Abstract

Panel methods are the numerical schemes for solving linear, inviscid, irrotational flow fields about arbitrary bodies at subsonic free-stream Mach numbers. The basic procedure is to discretize the body in terms of singularity distribution on the body surface then satisfy the necessary boundary conditions. It helps to determine the resulting distribution of singularity on the surface, and there by obtaining fluid dynamic properties of the flow. This project work describes a method for simulating, the potential flow field about the arbitrary two-dimensional bodies using MATLAB program. Even though singularities can be used as sources, doublets, or vortices, at this point the panel method uses the vortex element because it is talented to model both lifting forces and pressures. The numerical codes developed for this purpose computes the circulation, flow velocities, coefficient of lift and coefficient of pressure distribution over various geometries along with the streamline of corresponding bodies. Similarly the flow analysis is done for the same two dimensional bodies using FLUENT flow simulation tool and the results have been compared. The advantages of this numerical scheme over the conventional flow analysis are also presented in terms of reliable flow field data.

Keywords: Potential flow, Coefficient of pressure, Panel Method, vortex distribution, MATLAB, FLUENT

Contents

** Corresponding Author*

1. Introduction

The potential flow past a body placed in a uniform stream can be modeled equally well by replacing the body surface with either a source or a vortex sheet of appropriate strength [1-3]. Integral equations can then be written expressing the Neumann boundary condition of zero normal surface velocity for the source model or the Dirichlet condition of zero parallel surface velocity for the vorticity model [4]. Any type of singularity is chosen the final outcome is one and the same, namely a prediction of the potential flow velocity close to the body profile [5].

1.1. Steps toward Constructing a Numerical Solution

When establishing a numerical solution for potential flow a sequence similar to the following is recommended.

a) Selection of singularity element
b) Discretization of geometry
c) Influence Coefficients
d) Establish RHS
e) Solve linear set of equations
f) Secondary computation

2. Flow over an Aerofoil

Using Aerofoil coordinate developer MATLAB script the NACA four series airfoil is generated for this potential flow analysis. Similarly we can use NACA five series airfoil too. The following steps are scripted in MATALB for the potential flow analysis of airfoil [6].

2.1. Coupling Coefficient

From Biot- Savart law,

$$dq_{mn} = \frac{\gamma(s_n)ds_n}{2\pi r_{mn}} \qquad (2.1)$$

Resolve dq_{mn} parallel to the body surface at m where the profile slope is defined as β_m. It can be expressed in terms of coordinate locations through

$$dU_{mn} = \frac{\gamma(s_n)ds_n}{2\pi r_{mn}}\sin\phi_{mn} = \left(\frac{y_m - y_n}{2\pi r_{mn}^2}\right)\gamma(s_n)ds_n \qquad (2.2)$$

$$dV_{mn} = \frac{\gamma(s_n)ds_n}{2\pi r_{mn}}\cos\phi_{mn} = \left(\frac{x_m - x_n}{2\pi r_{mn}^2}\right)\gamma(s_n)ds_n \qquad (2.3)$$

Coupling Coefficient Matrix

$$K(s_m, s_n) = \frac{\Delta s_n}{2\pi}\left\{\frac{(y_m - y_n)\cos\beta_m - (x_m - x_n)\sin\beta_m}{(x_m - x_n)^2 + (y_m - y_n)^2}\right\} \qquad (2.4)$$

As earlier, take the control points at the midpoint of the panel. Eq. (2.4) should be satisfied at all the points on the body surface. This can be achieved most simply if the surface is broken down into finite number M of straight line elements of length Δs_n, which can be expressed as

$$\sum_{n=1}^{M} K(s_m, s_n)\gamma(s_n) = -U_\infty\cos\beta_m - V_\infty\sin\beta_m \qquad (2.5)$$

U_∞ and V_∞ are the components of W_∞ parallel to the x and y axes. It will be observed that Eq. (2.5) is finite but indeterminate as written for the special case n = m since both numerator and denominator are then zero, which may be written then

$$K(s_m, s_m)\Delta - \frac{1}{2} \qquad (2.6)$$

With simplified notation K_{mn}= K (s_m,s_n) and right hand side is

$$rhs_m = -U_\infty\cos\beta_m - V_\infty\sin\beta_m \qquad (2.7)$$

The circulation induced around the profile interior due to a unit vortex located at s_m, namely

$$\Delta\Gamma_m = \oint K(s_n, s_m)ds_n \qquad (2.8)$$

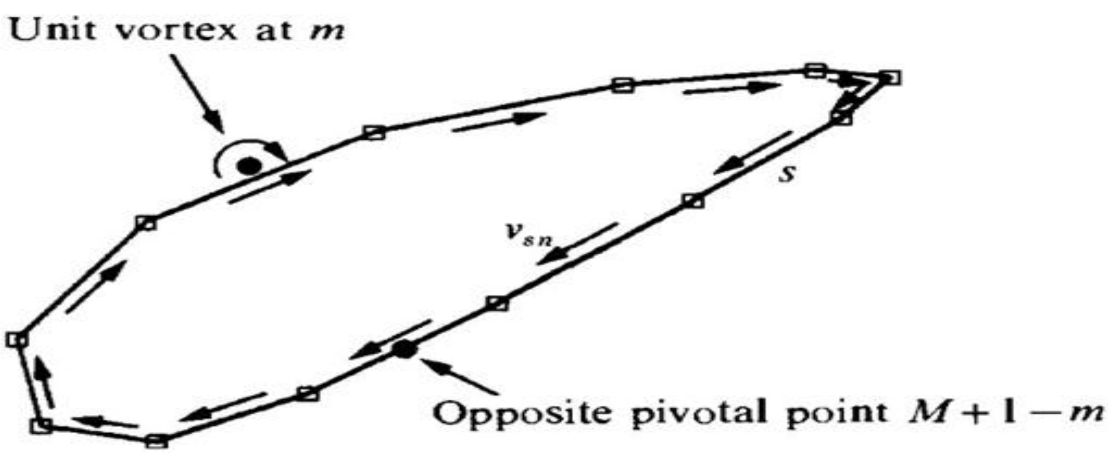

Fig-2.1. Circulation induced around profile interior due to a unit vortex just outside the profile at element m

2.2. Back Diagonal Correction

The net circulation $\Delta\Gamma_m$ around the profile interior induced by a surface vorticity element such as $\gamma(s_m)\Delta s_m$ should be zero [7, 8]. If this condition is enforced upon matrix coefficients, Eq. (2.8) becomes

$$K(s_{opp}, s_m) = \frac{1}{\Delta s_{opp}} \sum_{\substack{n=1 \\ n \neq opp}}^{M} K(s_n, s_m)\Delta s_n \qquad (2.9)$$

opp = M+1-m

Introducing m=4 into Eq. (2.9) for example gives, in the matrix notation adopted here,

$$K_{24} = -\frac{1}{\Delta s_2}(K_{14}\Delta s_1 + K_{34}\Delta s_3 + K_{44}\Delta s_4 + K_{54}\Delta s_5) \qquad (2.10)$$

which involves matrix coupling coefficients in column 4 only. Back diagonal element K_{24} is to be replaced by minus the sum of all other column 4 coefficients scaled by their element lengths Δs_n and finally divided by $-\Delta s_2$.

2.3. Wilkinson`S Kutta Condition

$$\gamma(s_{te}) = -\gamma(s_{te+1}) \qquad (2.11)$$

Remembering that for smooth flow leaving the trailing edge $\gamma(s_{te})$ must be clockwise and $\gamma(s_{te+1})$ must be anticlockwise and therefore negative

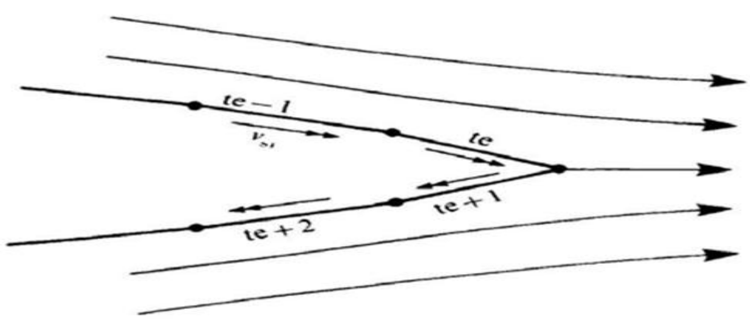

Fig-2.2. Aerofoil trailing edge flow

2.4. Lift and Pressure Coefficients

The pressure distribution and lift coefficients are obtained from the panel velocities. The velocity at each panel is the summation of the induced velocity contributions of the other panels. These velocity contributions are obtained from the solved vortex strengths [9, 10].

Pressure Coefficient can be obtained from the equation

$$C_p = 1 - \left(\frac{V_s}{V_\infty}\right)^2 \qquad (2.12)$$

Lift coefficient can be calculated from the equation,

$$C_L = \frac{2\Gamma}{SQ_\infty} \qquad (2.13)$$

2.5. Mesh File

Dimension of the control surface is width of 20c, height of 25c and diameter of 12.5c. Totally 25712 mesh faces in the control surface in which each cell of size 0.705. Unstructured triangular mesh is used for defining the flow angle and it avoids the frequent repeated meshing with respect to the different angles of attack [10]. Appropriate Boundary condition for flow analysis is shown in the Fig 2.3.

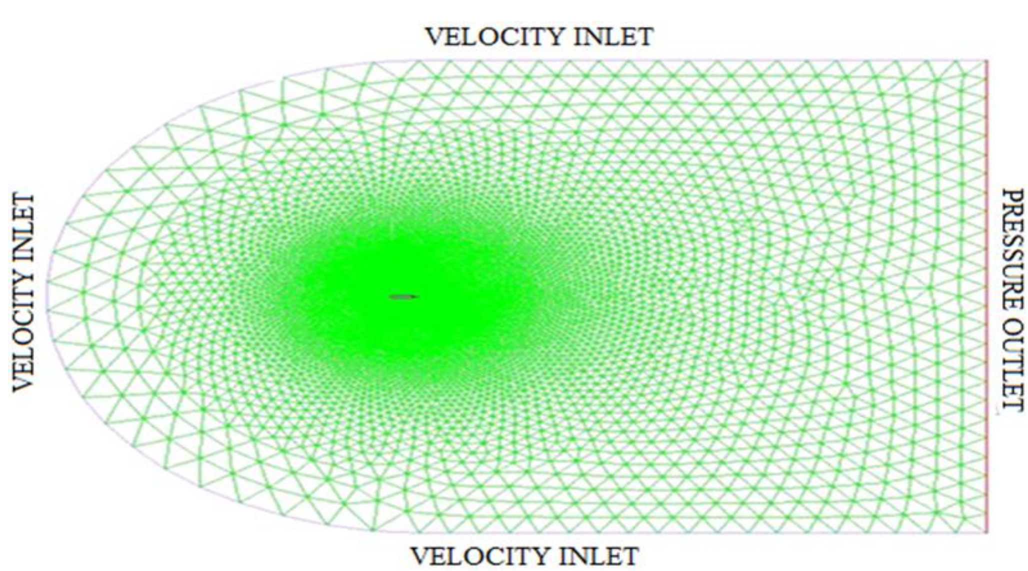

Fig-2.3. Mesh file of NACA 0012 Airfoil

2.6. CFD Results

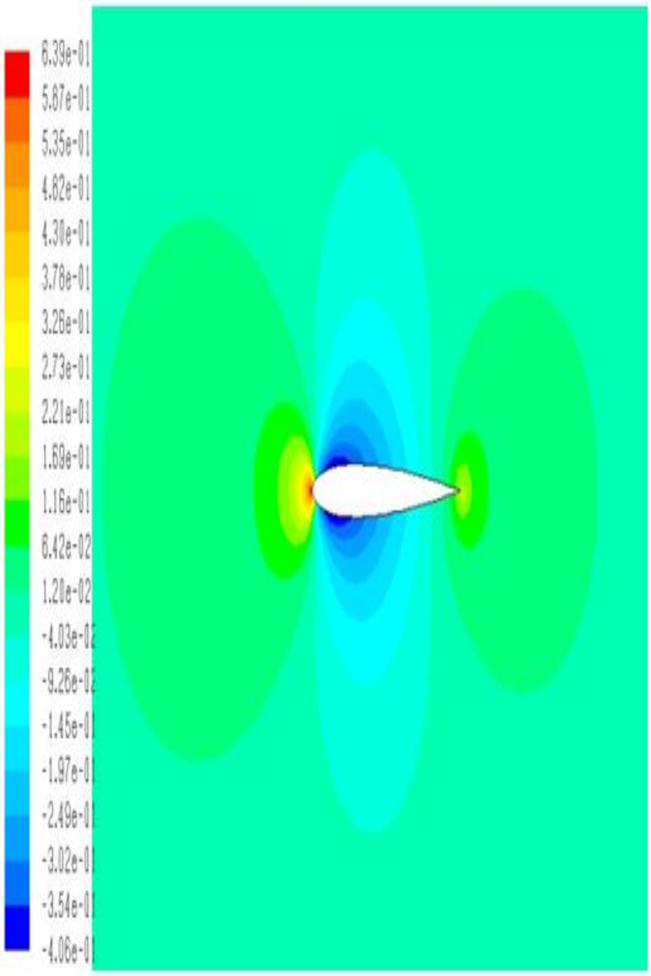

Fig-2.4. C_p distribution for NACA 0012 at 0^0

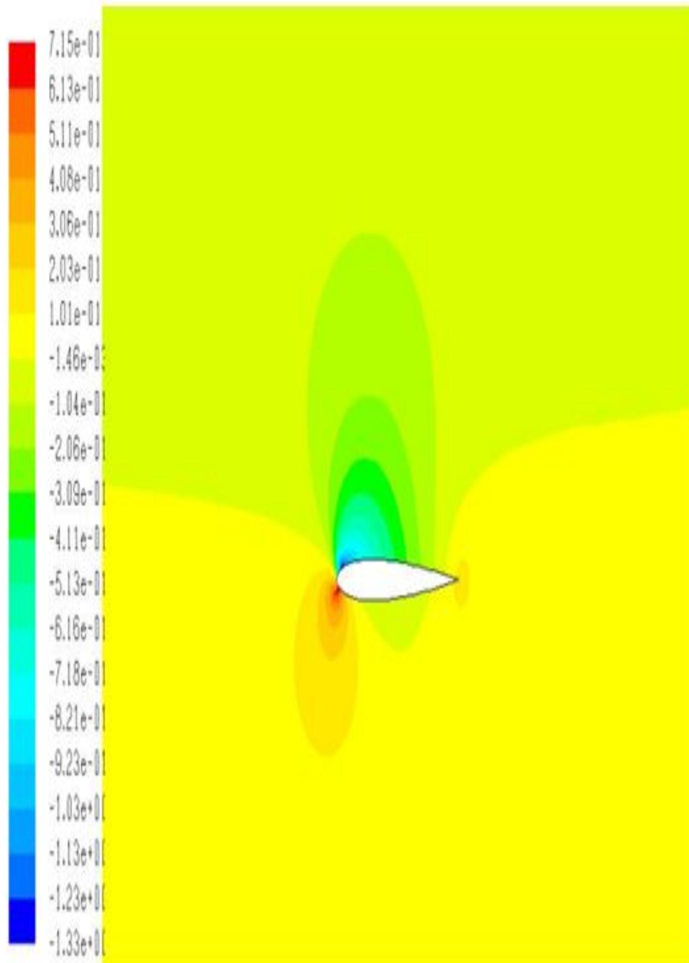

Fig-2.5. C_p distribution for NACA 0012 at 5^0

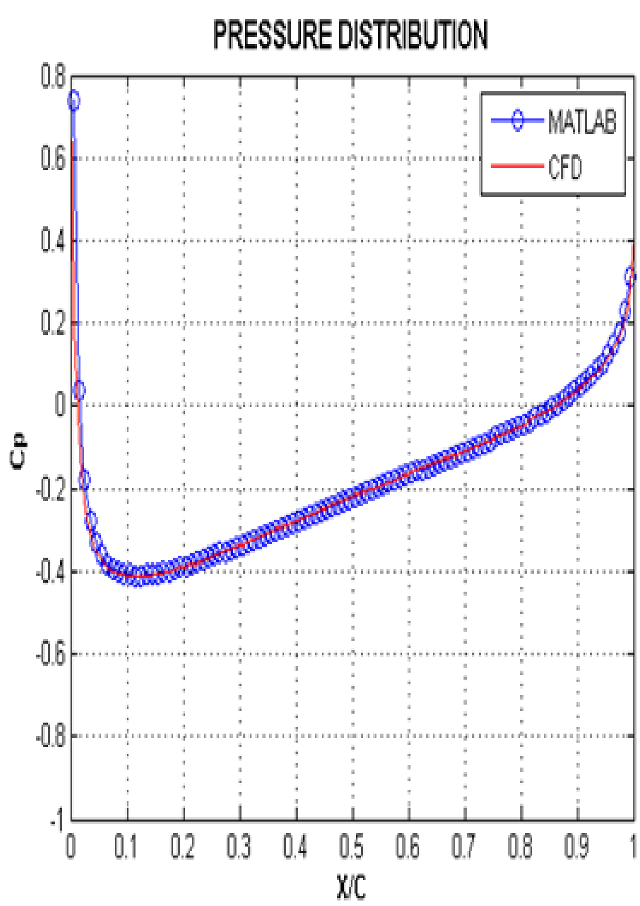

Fig-2.6. Pressure Distribution of NACA 0012 at 0^o angle of attack

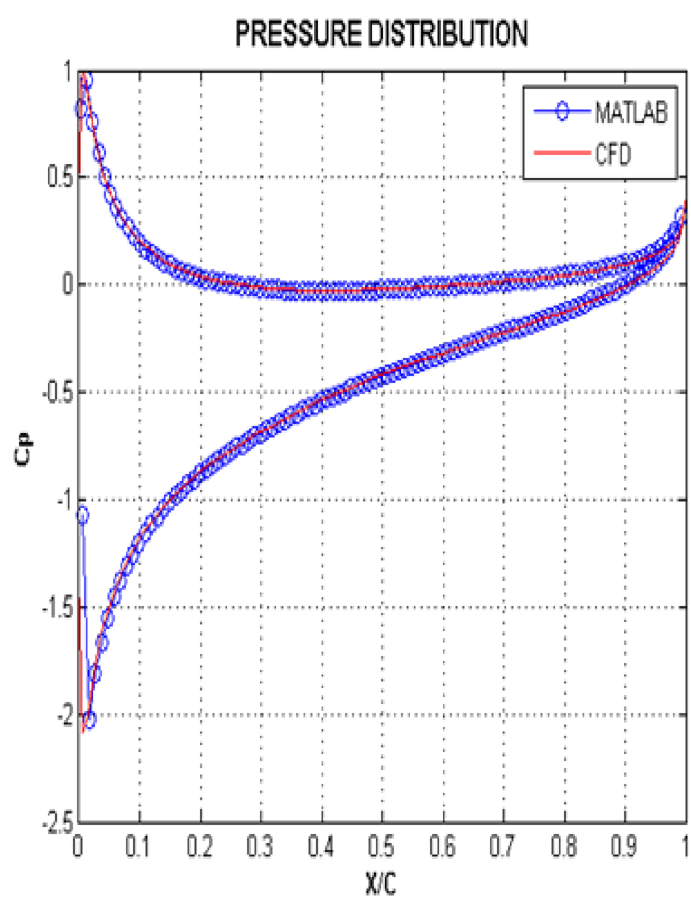

Fig-2.7. Pressure Distribution of NACA 0012 at 5^o angle of attack

Meshed airfoil is analyzed using inviscid flow solver and the resulting pressure distributions over the profile are compared with the potential flow method. Pressure distribution over the NACA 0012 aerofoil at 0^0 and 5^0 angles of attack is presented at this point and the results almost match with one another as shown in Fig 2.6 and Fig 2.7.

3. Flow over Compressor Cascade

Consider a compressor blade profile of C4/70/C50 with inlet angles β_1 and of stagger angle λ. The fluid approaches the cascade from far upstream with velocity W_1 at an angle β_1 and leaves far downstream of the cascade with velocity W_2 at an angle β_2.

The modified cascade coupling coefficient is as follows,

$$K(s_m,s_n)=u_{mn}\cos\beta_m+v_{mn}\sin\beta_m=\frac{\Delta s_n}{2t}\left\{\frac{\sin\frac{2\pi}{t}(y_m-y_n)\cos\beta_m-\sinh\frac{2\pi}{t}(x_m-x_n)\sin\beta_m}{\cosh\frac{2\pi}{t}(x_m-x_n)-\cos\frac{2\pi}{t}(y_m-y_n)}\right\} \quad (3.1)$$

The self-inducing coupling coefficients for a cascade are identical to those for a single aerofoil, namely

$$K(s_m,s_n)=-\frac{1}{2}-\frac{\Delta\beta_m}{4\pi} \quad (3.2)$$

The corresponding unit bound circulations are then given by,

$$\Gamma_u=\sum_{n=1}^{M}\gamma_u(s_n)\Delta s_n \quad (3.3)$$

$$\Gamma_v=\sum_{n=1}^{M}\gamma_v(s_n)\Delta s_n \quad (3.4)$$

The outlet flow angle can be found from the equation,

$$\beta_2=\arctan\left\{\left(\frac{1-\Gamma_v/2t}{1+\Gamma_v/2t}\right)\tan\beta_1-\left(\frac{2}{1+\Gamma_v/2t}\right)\frac{\Gamma_u}{2t}\right\} \quad (3.5)$$

By taking the circulation about path *abcd* for one blade pitch, Γ may be related to *t*, W_∞ and the flow angles through

$$\Gamma=t(V_1-V_2)=t(\tan\beta_1-\tan\beta_2)W_\infty\cos\beta_\infty \quad (3.6)$$

From the velocity triangles, an additional important relationship can be obtained, linking β_∞ to β_1 and β_2, namely

$$\tan\beta_\infty=\frac{1}{2}(\tan\beta_1+\tan\beta_2) \quad (3.7)$$

For the cascade, W_∞ is the vector mean of W_1 and W_2. The mean velocity W_∞ is defined as,

$$W_\infty=\frac{W_x}{\cos\beta_\infty} \quad (3.8)$$

$$W_\infty=W_1\left(\frac{\cos\beta_1}{\cos\beta_\infty}\right) \quad (3.9)$$

Then the pressure coefficient can be obtained from the equation,

$$C_p=\frac{P-P_1}{\frac{1}{2}\rho W_1^2}=C_{p\infty}\left(\frac{\cos\beta_1}{\cos\beta_\infty}\right) \quad (3.10)$$

3.1. Mesh File

Dimension of the control surface is width of 2c and height of 1.5c. Totally 13125 mesh faces are created in the control surface in which each cell size is about 0.835.

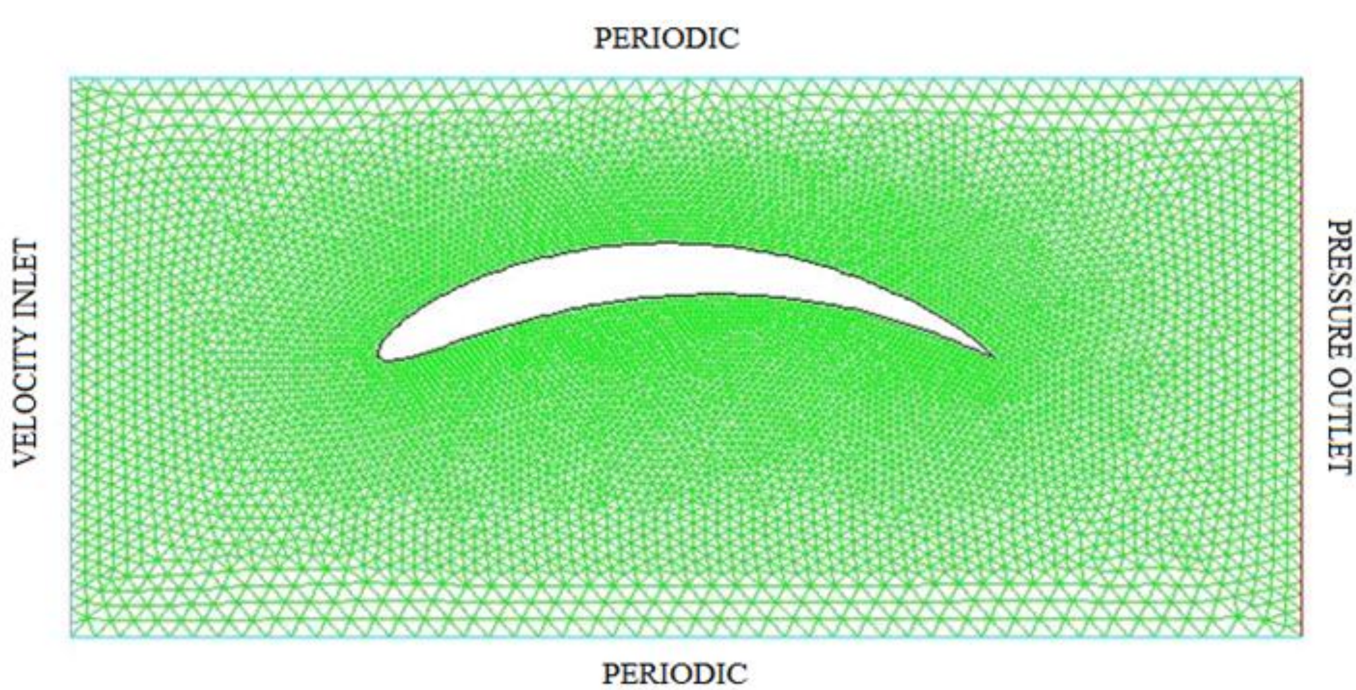

Fig-3.1. Mesh file of compressor C4/70/50 blade

3.2. CFD Result

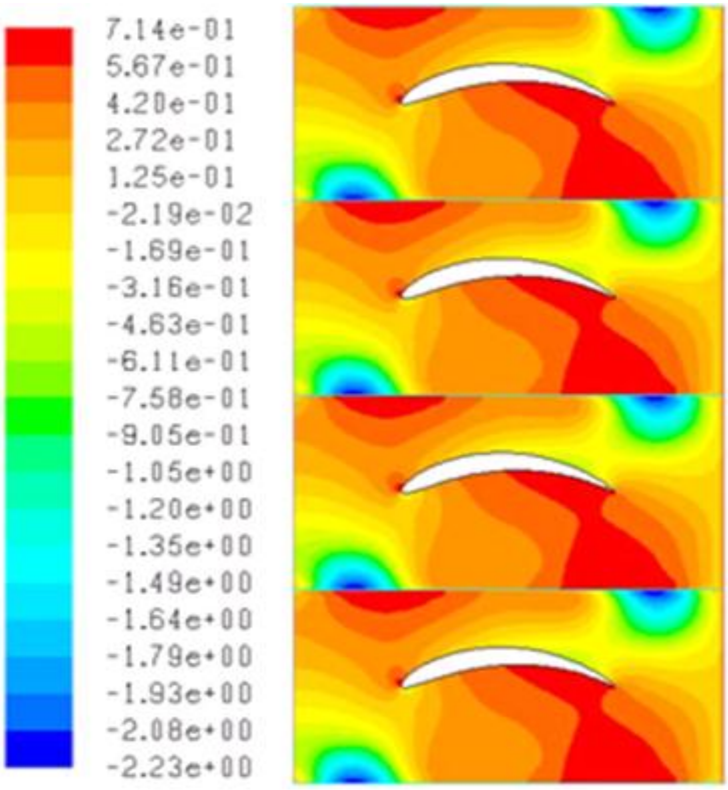

Fig-3.2. Coefficient of Pressure distribution over the compressor cascade

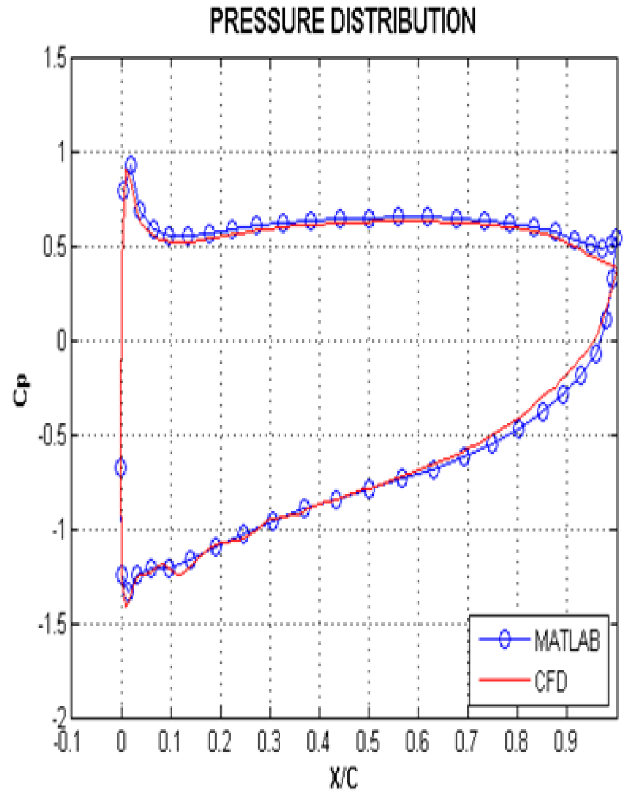

Fig-3.3. Pressure Distribution for C4/70/50 compressor blade

3.3. Output from MATLAB Program

In potential flow method it depends on stagger angle, Pitch/chord ratio and Inlet flow angle whereas flow analysis just requires inlet flow angle. The inlet flow angle is resolved into two components such as cosine and sine for analyzing the blade in normal position. These components are multiplied with velocity and given to the respective axis. Pressure distribution obtained from the potential flow method having slightly greater magnitude when compared to the conventional flow analysis method. This variation does affect the other parameter such circulation and coefficient of lift.

4. Flow over a Tandem Cascade

Consider a tandem blade having solidity=0.736, Inlet Flow Angle=45.46^0 and Total chord=3.45cm. It includes two airfoil elements with well optimized nozzle gap. First airfoil having leading edge radius of 0.0555 inch and trailing edge radius of 0.0150 inch. Then second airfoil having leading edge radius of 0.0542 inch and trailing edge radius of 0.0150 inch.

The coupling coefficient representing the induced velocity at pivotal point ‘m’ of body ‘p’ due to element ‘n’ of body ‘q’ is then given by

$$K_{mn}^{pq} = \frac{\Delta s_{qn}}{2\pi}\left\{\frac{\left(y_{pm}-y_{qn}\right)\cos\beta_{pm}-\left(x_{pm}-x_{qn}\right)\sin\beta_{pm}}{(x_{pm}-x_{qn})^2+\left(y_{pm}-y_{qn}\right)^2}\right\} \quad (4.1)$$

In this case, the self-inducing coupling coefficients (when p = q and m = n) are solved as same as that for airfoils.

$$\begin{pmatrix} A_{11} & A_{12} \\ A_{21} & A_{22} \end{pmatrix}(\Gamma(s)) = (rhs) \quad (4.2)$$

4.1. Mesh File

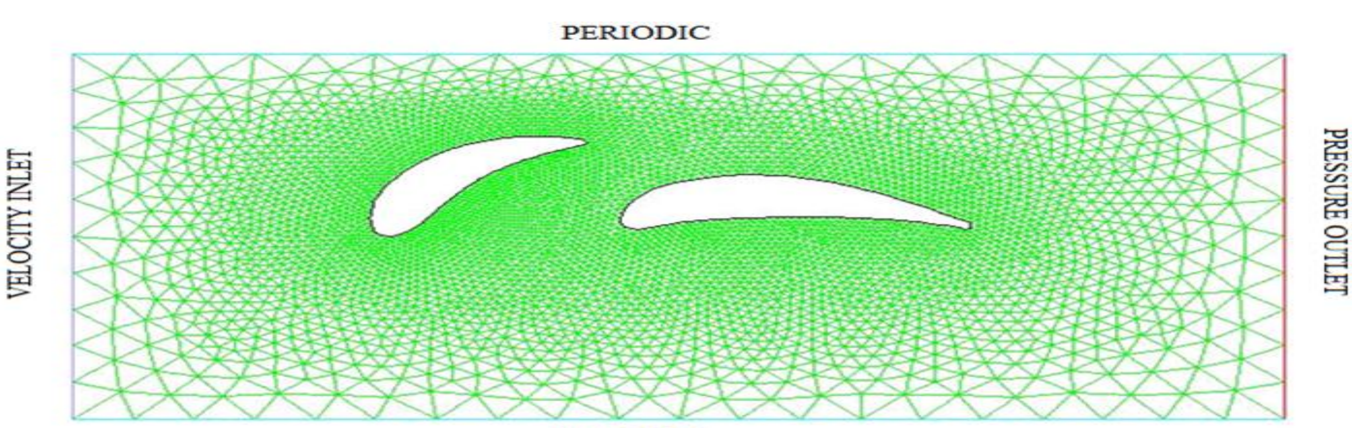

Fig-4.1. Mesh file of low solidity tandem blade

For defining the inlet flow angle rectangular control surface is implied with dimension of width 2c and height 1.5c. Totally 15432 mesh face are present in the control surface having each element of size 0.896. Boundary condition almost same to the aerofoil mesh but the side walls of velocity inlet is replaced by periodic to provide the cascade arrangement. Boundary condition such as periodic should be edge meshed.

4.2. CFD Results Output from MATLAB Program

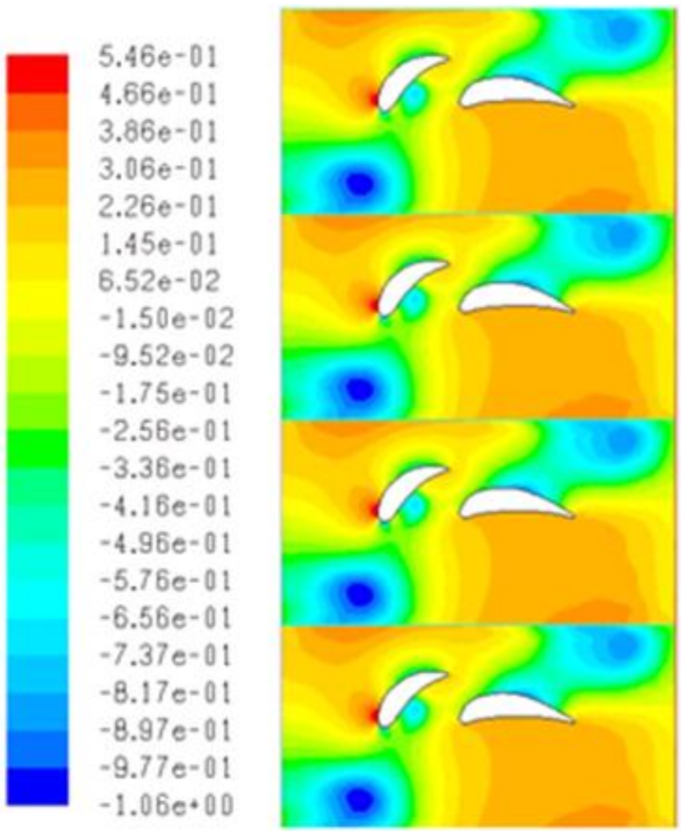

Fig-4.2. C_p distribution over tandem cascade

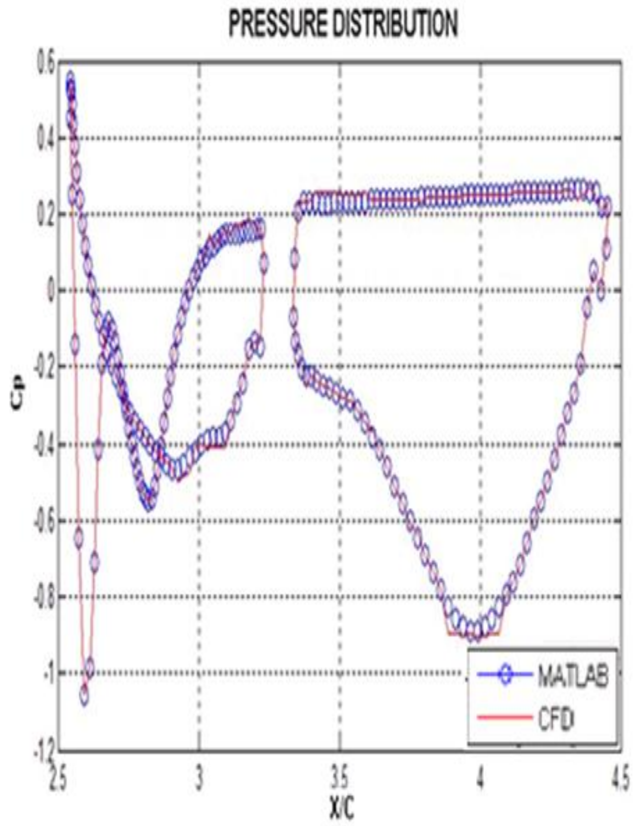

Fig-4.3. Pressure Distribution in Low solidity Tandem Blade

Pressure distribution for tandem blade obtained from the potential flow method having slightly greater magnitude when compared to the conventional flow analysis method.

5. Conclusions

In conclusion, the MATLAB programs for analyzing the flow over various geometries have been written using the vortex panel method. These codes require less time to compute the flow properties when compared to other finite volume solvers and could be adapted to any single and multi-element aerofoil geometries. Hence, it can be utilized as an alternative approach to fully coupled analysis procedures that requires huge computing resources. The optimization of geometric and flow variables is also allowed in the computer program developed. Thus, the best combination of flow and geometric dimensions can be finalized through the proposed algorithm. Further, the achieved results are validated against CFD results and good agreement is achieved.

References

[1] M. Hoeger, R. D. Baier, S. Fischer, and J. Neudorfer, "High turning compressor tandem cascade for high subsonic flows, part 1: Aerodynamic design," presented at the 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, California, AIAA-5601, 2011.

[2] M. Lars, K. Dragon, W. Detlev, F. Susanne, and S. Udo, "High turning compressor tandem cascade for high subsonic flow, part 2: Numerical and experimental investigations," presented at the 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Ohio, AIAA-5602, 2011.

[3] H. Hivoaki, M. Akinori, and S. Shinya, "Development of highly loaded fan wih tandem cascade," presented at the 41st Aerospace Sciences Meeting and Exhibit, Nevada, AIAA-1065, 2003.

[4] T. Sumeet, S. S. Jaskirat, and R. Bhaskar, "Cascade studies of tandem blades for axial flow compressors/fans," presented at the 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Hartford, AIAA-4698, 2008.

[5] R. Bhasker and V. Srivastasava Puranam, "Aerodynamic design of a part-span tandem bladed rotor for low speed axial compressor," presented at the 27th AIAA Applied Aerodynamics Conference, Texas, AIAA-3964, 2009.

[6] L. Fearn Richard, "Airfoil aerodynamics using panel methods," *The Mathematica Journal*, vol. 10, pp. 725-739, 2008.

[7] R. I. Lewis, *Vortex element methods for fluid dynamical analysis of engineering systems*. New York: Cambridge University Press, 1991.

[8] E. Albert Von Deonhoff and H. Ira Abbott, *Theory of wing sections*. USA: Dovers Publication, 1959.

[9] J. Katz and A. Plotkin, *Low-speed aerodynamics*, 2nd ed. Cambridge, U.K: Cambridge University Press, 2001.

[10] M. Arnold Kuethe and C. Cheun-Yen, *Foundations of aerodynamics: Bases of aerodynamic design*. New York: John Wiley & Sons, Inc, 1998.