

Computational Investigation of Flow Separation Over Naca 23024 Airfoil at 3 Million free Stream Reynolds Number

B. S. Anil Kumar¹,

¹ Department of Mechanical Engineering,
BNMIT, Bangalore

Dr. Ramalingaiah²,

²Department of Mechanical Engineering,
PESCE, Mandya

Dr. S. Manjunath³

³Department of Mathematics,
BNMIT, Bangalore

Rudresh Ganganna⁴

⁴Founder Director,
Niharika Institute of Computational Engineering (NICECFD),
Bangalore.

Abstract— The following work is the CFD analysis of NACA 23024 airfoil. The analysis is carried out for a free stream Reynolds number of 3 million for which the wind tunnel results are available. The CFD analysis is carried out using Ansys Fluent Solver. The analysis is carried out using Spalart Allmaras turbulence model, K-omega SST turbulence model with flow transition capabilities, Standard K-Epsilon Turbulence model and K-omega SST turbulence model. The analysis results are then compared with the wind tunnel results and the performance of the turbulence models are discussed.

Index Terms—NACA 23024, Turbulence models, Lift curve, Drag curve, stalling.

The CFD analysis on the chosen airfoil NACA 23024, where first digit when multiplied by 3/2 yields the design lift coefficient in tenths of chord, the next two digits when divided by 2 gives the position of the camber in tenths of chord and the final two digits indicate the maximum thickness in percentage of chord that is NACA 23024 airfoil has maximum thickness of 24%, a design lift coefficient of (2 X 3/2) 3 in tenths and maximum camber located (30/2) 15% back from the leading edge is carried out using the ANSYS package. ICEM CFD is used as a meshing tool where a mesh required for the analysis of the airfoils are developed. The mesh is adjusted to obtain a y-plus close to 30 so that the best approximation is achieved. In operating conditions, the wing section that is the airfoil of the aeroplane moves through the wind. However, for simulation and experimental purposes the airfoil is kept stationary and the working fluid is assumed to have motion over the airfoil. Hence, a circular far field is created to obtain the best approximation.

The actual solving of the problem is done using the Fluent package which is a part of the ANSYS package. The case is analysed as an incompressible low regime problem. In the physics setup, the atmospheric pressure is input as absolute pressure. There are no temperature inputs as it is an incompressible flow. The input boundary condition is the velocity inlet boundary condition and the output boundary condition is the outflow type boundary condition. A good convergence criteria is set to make the results more accurate.

Geometry creation and mesh generation:

1. The coordinates of the airfoil is obtained and is imported into ANSYS ICEM-CFD software, which is essentially a meshing tool. The curves option is checked while importing the points into CFD which draws splines between the points for the upper and lower surface.
2. The coordinates obtained are for an open trailing edge type airfoil and therefore, the two end points have to be joined using the curve option.
3. An exact circular far field is created around the airfoil equidistant from its chord in all the four directions.
4. After the far field has been generated, blocking is done. Blocking is essentially done to capture the shape of the far field of the airfoil and further splitting of the block is done to achieve the capturing of the airfoil
5. Once the splitting of the block is complete, point to point association of the main block is done along with the edge to curve association.
6. An O-grid is a tool which is available in the blocking option itself. This is created around the airfoil in order to obtain the mesh exactly in the shape of the airfoil surface around it.
7. An O-grid creates a localized block in an O shape around the airfoil. Edge to curve association is carried out in order to associate the O-grid with the airfoil. The link edge command is made use of to make the lines of the O-grid take the shape of the airfoil surface.
8. After all the associations are done, the pre-mesh parameters are applied to the edges. The selection of the meshing law and the number of nodes along with the ratios and spacing options play the most important part in the meshing process as they decide the quality of the mesh around the airfoil.
9. It is desirable to have more grid elements near the airfoil surface to increase computational accuracy and to have comparatively lesser number of grid elements away from the airfoil in the far field in order to reduce computational time

and reduce the size of the output file. To achieve this requirement, the exponential 2 mesh law is used.

10. The y-plus is checked for after the meshing is complete and modified if necessary through the spacing option in the pre-mesh parameters.
11. After the entire mesh generation and obtaining a quality mesh, it is converted to unstructured mesh to make it possible to be read in Fluent. The file is saved in the output option to obtain a file which is compatible with Fluent. It is saved with the extension.msh

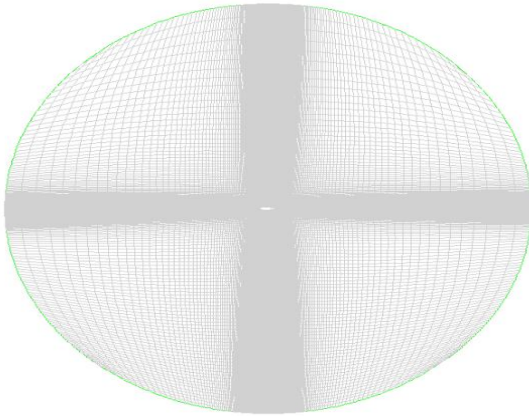


Fig 1: Mesh generation of NACA 23024 airfoil

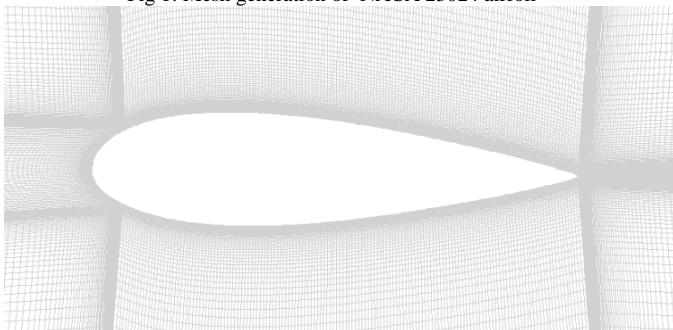


Fig 2: Mesh resolution in the boundary layer region
CFD analysis in FLUENT software:

1. The ANSYS fluent is opened and the option is set to 2D and double precision.
2. Import the mesh by selecting read mesh option. Scale the mesh to the suitable unit. The check command may be used to check for the quality of the mesh.
3. Fluent software used not only for aerodynamic studies but also used in heat transfer problems, acoustic problems etc. therefore, it is important to select the correct set of equation for the required analysis.
4. After the turbulence model is selected for the analysis, air is selected as the material and the corresponding material properties are input and saved.

5. For the cell zone boundary condition, the far field is selected to be a fluid and the material properties which were previously input are associated with the far field cell zones.
6. Input of the boundary condition is the most important part in the physics setup. Since it is an incompressible flow, the input boundary condition is the velocity type boundary condition where the magnitude of the free stream velocity is directly input. The outlet boundary condition is the outflow type boundary condition as the airfoil functions in atmosphere and there is no external pressurization.
7. The far field is set to the pressure far field boundary condition where the pressure is input as the atmospheric pressure at the location.
8. The solution method is set to simple. The monitors are activated by selecting the coefficient of lift, coefficient of drag.
9. The solution initialization is done by setting it to hybrid initialization and the residual monitors are edited to have convergence criteria to obtain more accurate results.
10. The run calculation command is selected and 2000 iterations are allowed. The calculation is run and it terminates automatically depending on the input convergence criteria or it may be manually interrupted in case the coefficient of lift and coefficient of drag become constant all the remaining iterations.
11. The required plots such as the Cp plot and required contours such as pressure and velocity vectors are obtained as the post processor results.

RESULTS AND DISCUSSIONS

Figure 3 and 4 shows the lift and drag coefficients respectively for 3 million Reynolds number. CFD analysis results are plotted along with wind tunnel measured results to understand the accuracy of results captured with CFD analysis with different turbulence models employing different strategy for boundary layer modeling.

Figure 9 and 10 shows the velocity vectors and pressure coefficient plots respectively for angle of attack 15 degrees with K-epsilon turbulence model with standard wall function. It is observed from pressure coefficient plot that even though angle of attack is in the stalling region, there are no signs of flow separation observed in the pressure coefficient plot. Similar trend is repeated in angle of attack 16 degrees.

The k-epsilon models with standard wall function starts capturing flow separation from angle of attack 17 degrees as seen in figure 16. Hence K-epsilon turbulence model with standard wall functions over predicts the lift even after stalling. K-epsilon model with standard wall function starts boundary layer computation from third region of the turbulent boundary layer and relatively coarser mesh in the boundary layer is sufficient.

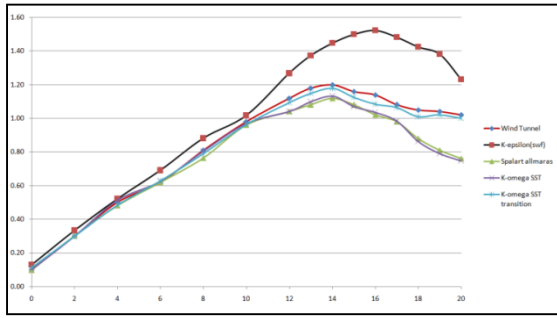


Fig 3: Lift curve

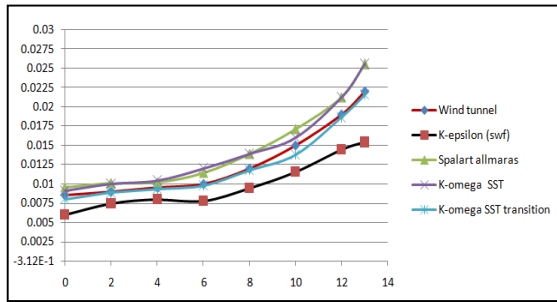


Fig 4: Drag curve

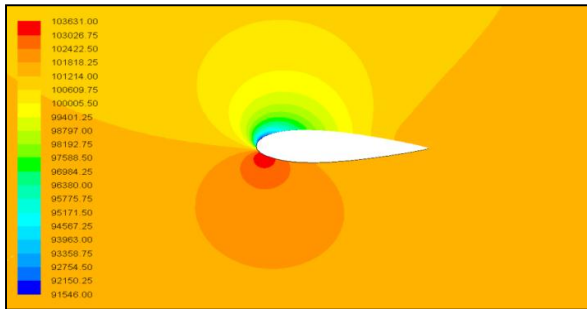


Fig 5: Pressure Contour at $\alpha = 14^\circ$

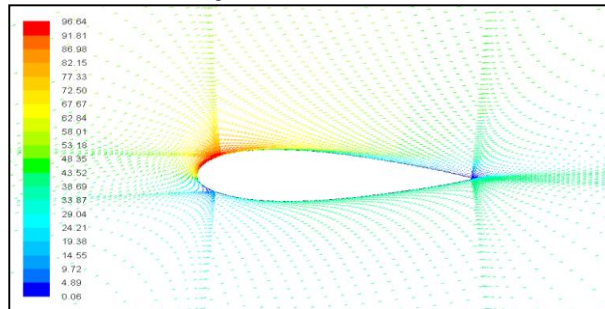


Fig 6: Velocity Vector at $\alpha = 14^\circ$

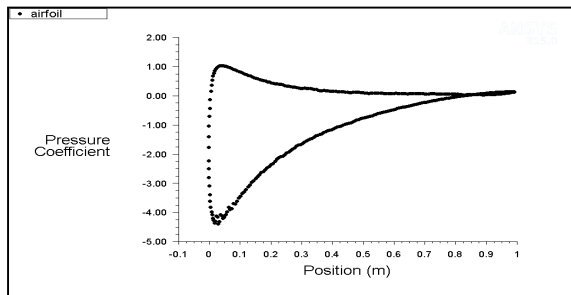


Fig 7: Pressure coefficient at $\alpha = 15^\circ$

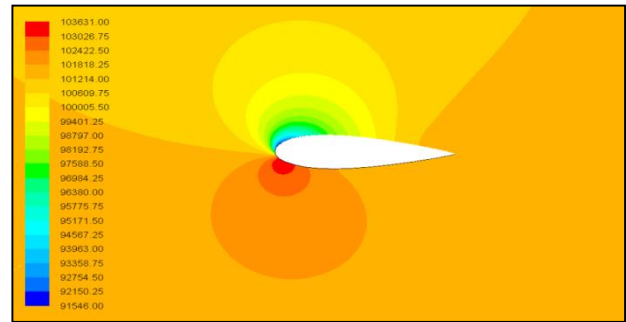


Fig 8: Pressure Contour at $\alpha = 15^\circ$

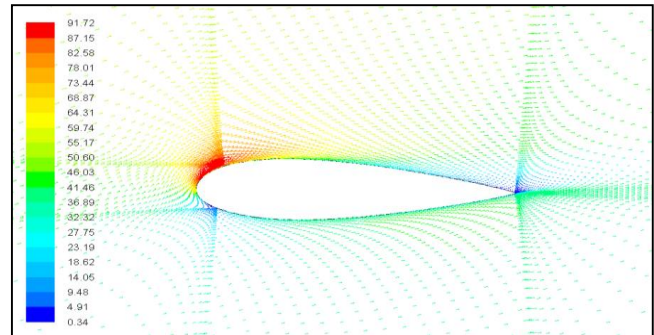


Fig 9: Velocity Vector at $\alpha = 15^\circ$

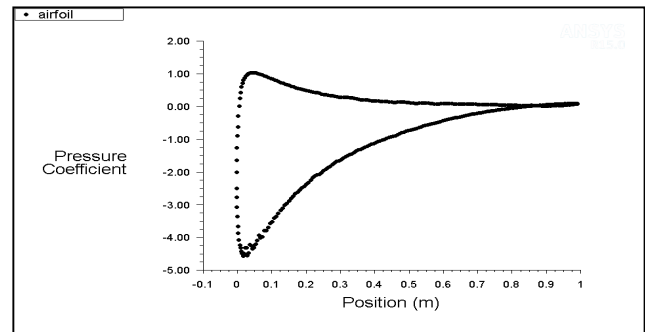


Fig 10: Pressure coefficient at $\alpha = 16^\circ$

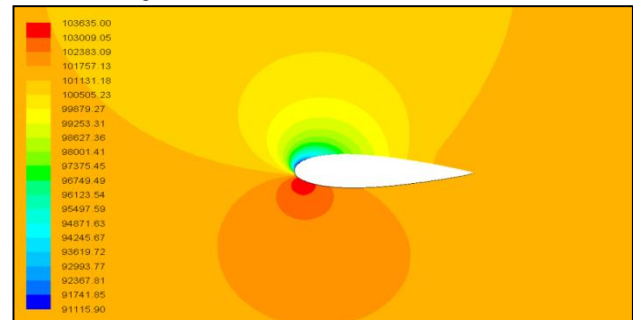


Fig 11: Pressure Contour at $\alpha = 16^\circ$

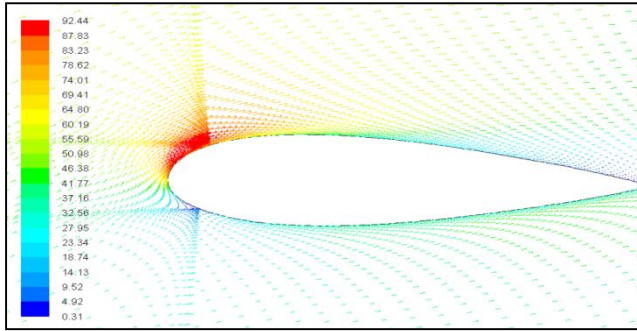


Fig 12: Velocity Vector at $\alpha = 16^\circ$

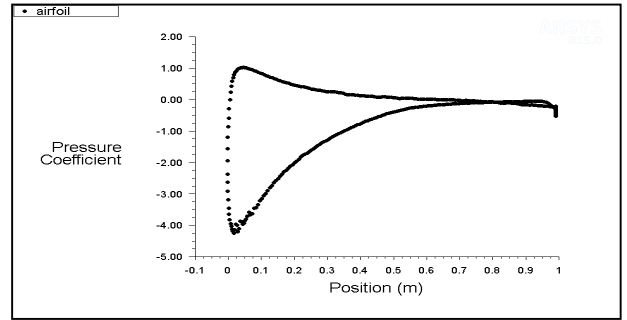


Fig 16: Pressure coefficient at $\alpha = 18^\circ$

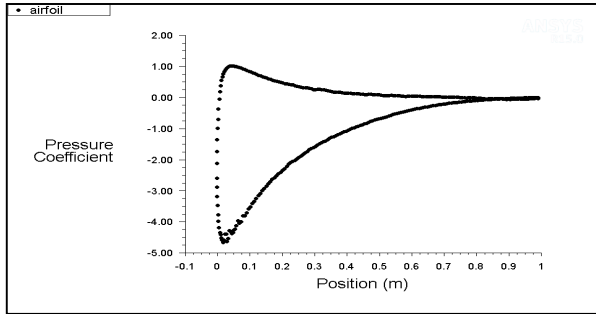


Fig 13: Pressure coefficient at $\alpha = 17^\circ$

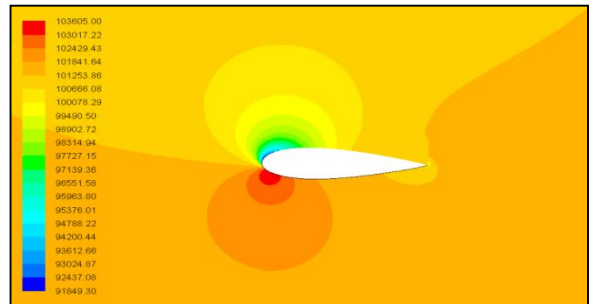


Fig 17: Pressure Contour at $\alpha = 18^\circ$

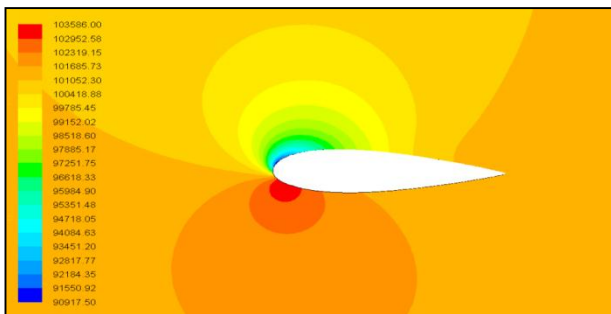


Fig 14: Pressure Contour at $\alpha = 17^\circ$

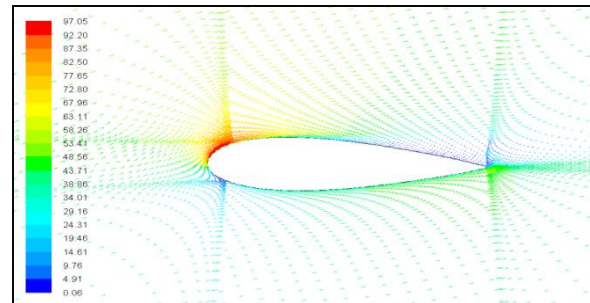


Fig 18: Velocity Vector at $\alpha = 18^\circ$

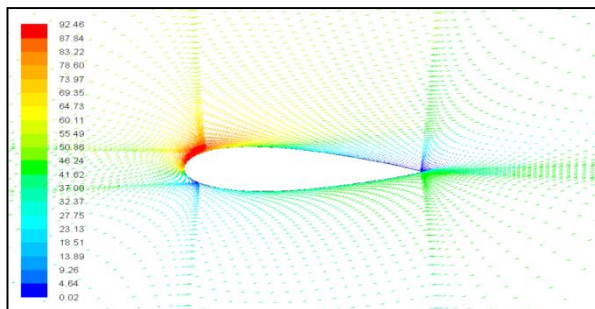


Fig 15: Velocity Vector at $\alpha = 17^\circ$

CONCLUSIONS

- ✓ All turbulence models yield similar results in the pre stall region and the CFD analysis results are providing a good match with wind tunnel test results.
- ✓ In the post stall region approximation of boundary layer regions with wall functions as in the case of K-epsilon turbulence model with standard wall functions, leads to delayed prediction of stalling and hence lift forces are over approximated.
- ✓ Capturing of all the three regions of the turbulent boundary layer with a very fine mesh layers will provide accurate CFD analysis results and trend same as that of wind tunnel test results.
- ✓ Prediction of both laminar and turbulent boundary layer regions with transition turbulence model provides most accurate CFD analysis results, and these results are very close to wind tunnel measurements.

ACKNOWLEDGEMENTS:

Authors would like to express sincere gratitude to “Niharika Institute of Computational Engineering (NICECFD), Bangalore. (www.nicecfcd.com) for all the technical guidance and facilities providing us to use licensed version of ANSYS ICEMCFD meshing software and ANSYS FLUENT CFD analysis software

REFERENCES

- [1] 1. David Hartwanger et.al “3 D modeling of a Wind Turbine using CFD” NAFEMS Conference, United Kingdom, 2008.
- [2] 2. Frank Bertagnolio et.al “Wind Turbine airfoil catalogue” RISOE National Laboratories, Denmark, 2001.
- [3] 3. H.Gao et.al “Computational study of unsteady flows around dragonfly and smooth airfoils at low Reynolds number” 46th AIAA Aerospace sciences meeting and exhibit, Reno, Nevada, 2008.
- [4] 4. Vance Dippold, III, “Investigation of Wall Function and Turbulence Model Performance within the Wind Code”, 43rd AIAA Aerospace Sciences Meeting and Exhibit, 10 - 13 January 2005, Reno, Nevada
- [5] S.Sarada, M.Shiva Shankar and Rudresh Ganganna “ Numerical simulation of Viscous, incompressible flow around NACA 64618 subsonic airfoil using Computational Fluid Dynamics”, Proceedings of National conference on advances in Mechanical Engineering, Allied Publishers Pvt Ltd, 2012
- [6] 6. Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications” AIAA Journal, Vol. 32, No. 8, August 1994, pp. 1598-1605 (<http://turbmodels.larc.nasa.gov/sst.html>).
- [7] Abbott.I.H, “Theory of wing section, including a summary of airfoil data”, Dover book on Physics, 1995.