International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013

Numerical Simulation of Turbulent Flow past Stationary NACA 0012 Aerofoil Using FLUENT

Deepa M. S[1], Nithya S. N.[2], Karthik Prabhu[2]
[1]*Associate Professor, Department of Aeronautical Engineering, MVJCE, Bangalore*
[2]*Graduate Students, Department of Aeronautical Engineering, MVJCE, Bangalore*

Abstract

The prediction of dynamic stall on aerofoil or any lifting surfaces in unsteady flow environment plays an important role in the aircraft design process. Both computational and experimental investigations have established that a predominant feature of dynamic stall is the vortex shedding phenomenon from the leading edge of the aerofoil which produces transient forces and moments which are significantly different from their static-stall counterparts. One of the major challenges in the computation of unsteady aerodynamic flows is the accurate prediction of the dynamic stall (DS) phenomenon.

In the present work, the phenomenon of dynamic stall is studied by considering NACA0012 aerofoil beyond its static-stall incidence angle. The commercial CFD tool FLUENT will be used to simulate the stationary aerofoil with a sinusoidal incidence motion. The aerodynamic load and transient pressure distribution obtained by the present simulation using FLUENT will be compared with the available measurement data and the results obtained using flow solution code.

1. Introduction

In the field of unsteady aerodynamics the term dynamic stall is frequently used to describe the complex fluid mechanical phenomena that occur during extreme incidental movements of an aerofoil beyond the angle of static stall. These processes are characterized by unsteady boundary layer separation, where a multitude of dynamic and viscous factors influence the flow field evolution.

The unsteady Reynolds-averaged Navier Stokes (RANS) equation constitutes the basic mathematical model for numerical simulation of turbulent viscous flows. The Approximate solution of the Navier-Stokes equation enables us to analyse many of the complex flow phenomenon observed in wing aerodynamics such as shock/boundary layer interaction, unsteady flow separation and wake, periodic vortex shedding, hysteresis in loading during pitching motion, wing stall and flutter, buffeting, gust response etc. Unsteady flow past NACA 0012 aerofoil pitching in a sinusoidal manner about an average angle of incidence using FLUENT solver forms the main objective of the present work.

2. Problem Formulation:

The present computations have been carried out using FLUENT (commercial solver), simulation over stationary airfoil is done. User Defined Function (UDF) and results obtained are compared with suitable measurement data and in-house data. The approach towards solving the problem is briefed as follows:

2.1 Numerical simulation of turbulent flow past stationary NACA0012 aerofoil:

Data points for a basic NACA 0012 airfoil is generated and imported in POINTWISE (Pre-processor). Structured 'O' grid of size 478 X 110 is generated around the airfoil, suitable boundary condition is specified and mesh is exported to the solver i.e. FLUENT. It is then solved using suitable viscous model and scheme for flow at various angles of attack, keeping airfoil stationary. The aerodynamic coefficient and pressure coefficient for each angle of attack is obtained up to the static stall and validated against in-house and measurement data. The exercise is repeated by keeping flow horizontal at zero angle of attack and changing the orientation of airfoil. The results obtained from both the cases are compared.

3. Objective of the present work

- To generate 2D structured 'O' type grid using an appropriate pre-processor
- Using FLUENT to analyze flow past stationary NACA0012 at Re= 3×10^6 using the generated O-grid and validation the computed aerodynamic coefficient *viz*., lift, drag and moment and pressure distribution against available measurement data
- To create a UDF (User Defined Function) by writing a code in C which includes the effect of moving grid in an inertial frame of reference
- To couple the UDF with FLUENT in order to generate the movement of the aerofoil based on the sinusoidal pitching function

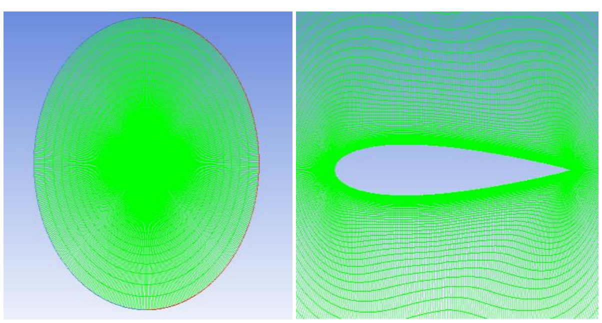
4. Computational details of stationary aerofoil

A single block O-grid (Fig.1) with grid size of 478×110 = 52580 control volumes is generated using POINTWISE. Initially the aerofoil coordinates are loaded into POINTWISE and connectors are specified. The aerofoil is then dimensioned as per the required number of grid points and extruded (by selecting the "Extrusion" option provided in POINTWISE) with a step size of 110. The radius of the resulting O-grid is measured and found to be about 15C,Where C is the chord of the aerofoil ,with origin at centre of the aerofoil (0.5,0).

The minimum wall normal distance is maintained to be 0.0001.In order to resolve the boundary layer; the grids are stretched so that it is fine near the aerofoil boundary and comparatively coarser as we move in the perpendicular direction. The parameter which gives an idea of how fine or coarse the grid is the y^+. It is a function of Reynolds number and the minimum wall normal distance. It is important in turbulence modelling to determine the proper size of the cells near domain walls. The turbulence model wall laws have restrictions on the y^+ value at the wall. A faster flow near the wall will produce higher values of y^+, so the grid size near the wall must be reduced. For simulation with k – ω SST model using wall functions the value of y^+ should be around 1 whereas in the absence of wall function, as is the case in FLUENT, the y^+ value can range between $30 \leq y^+ \leq 300$.Hence an average y^+ value of 10.35 is employed at present.

Second order upwind scheme for convective flux discretization coupled to k-ω SST turbulence model has been used for the present computations since the in-house data is also available for the same. The results obtained are compared for mean aerodynamic coefficient and mean surface pressure distributions with available measurement [1] and in-house data .The boundary conditions used in FLUENT are as follows

- On the 1st half of the curved surfaces i.e. till 0.5C of the aerofoil, velocity inlet condition is specified
- On the other half i.e. from 0.5C of the aerofoil to 15C, outflow condition is given
- On the aerofoil no-slip condition was specified



a) Single block O-grid (478 ✕ 110) b) Zoomed view near the aerofoil

Fig.1 Grid around stationary NACA 0012 generated by extrusion method

4.1 Problem definition and method of simulating the flow using FLUENT

- **Reference Parameters:** as shown in Table.1
- **Solver :** pressure based
- **Material :** air
- **Convergence criteria :** Absolute 10^{-5}
- **Time :** Steady

Reference Parameters	Values
Reynolds Number	3×10^6
Velocity	1m/s
Density	$1 kg/m^3$
Viscosity	$0.3333 \times 10^{-6} Nm/s^2$
Operating pressure	101325 Pa

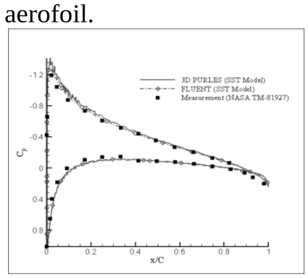
Table 1. Values of the reference parameters used

4.2 Surface pressure distribution and flow pattern

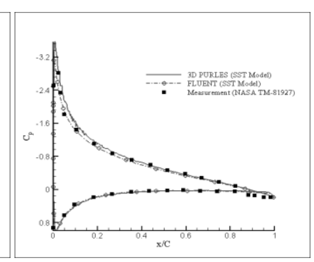
The data of velocity field, surface pressure coefficient and the aerodynamic coefficient obtained through FLUENT is plotted against the required parameters (i.e. Streamlines and aerodynamic coefficient v/s angle of attack and pressure coefficient against v/s thickness to chord ratio ‘x/C’) as shown in Fig.2.

Fig.2 compares the computed surface pressure distribution ($C_p = (p - p_\infty)/0.5 \rho U_\infty^2$) over the aerofoil at four different angles of attack (α) using FLUENT with the available measurement data and computational data obtained using the in-house flow code. The present FLUENT simulations using O-grid shows a reasonably good agreement with the measurement data and in-house at all the angle of attack. The suction peak is properly predicted by present simulations except at α = 8˚ where it is slightly over predicted.

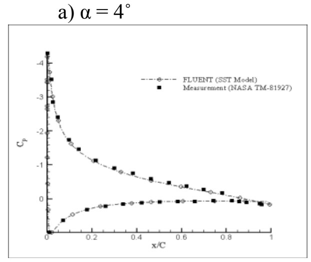
The streamlines computed from time integration of velocity field at four different angle of attack are shown in Fig.3.These plots clearly show the gradual bending of the streamlines near the aerofoil surface as the angle of attack increases. A small separation bubble is observed on the upper surface near the trailing edge at α = 17˚. Beyond this separation bubble gradually grows in size and spreads over the upper surface finally leading to the stalling of the aerofoil.



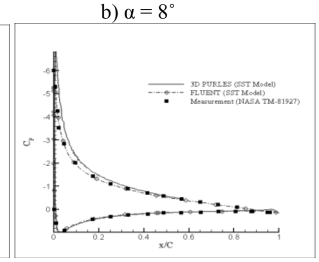
a) α = 4˚



b) α = 8˚



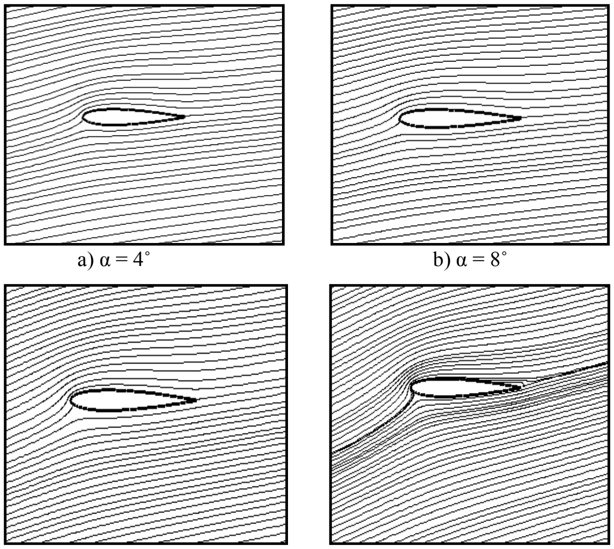
c) α = 10˚



d) α = 12˚

Fig.2 Variation of surface pressure coefficient for flow past stationary NACA 0012 aerofoil (Re = 3 ✕10^6, SST Turbulence Model, 2nd order Upwind scheme)

ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013



a) α = 4˚ b) α = 8˚

c) α = 12˚ d) α = 17˚

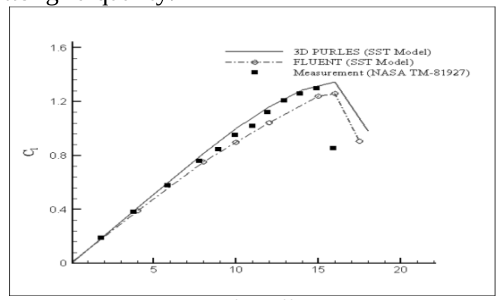
Fig.3 Computed streamlines for flow past stationary NACA 0012 aerofoil (Re = 3 $\times$10[6], SST Turbulence Model)

4.3 Aerodynamic coefficient

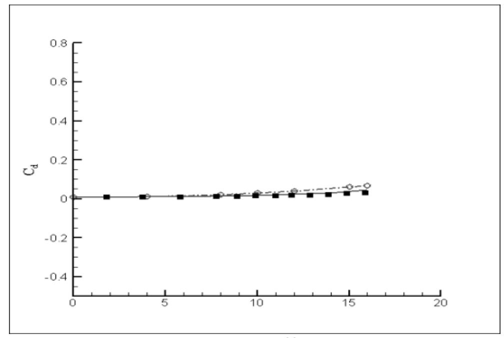
The aerodynamic coefficient like the lift (C_l) and drag (C_d) coefficient can be easily computed from the numerical integration of the surface forces *viz.*, the pressure acting normal to the surface and the shear stress acting along the surface. The drag and the lift coefficient represent the resultant forces on the aerofoil along the flow and normal to the flow direction respectively, non-dimensionalised by the product of the dynamic head and the aerofoil chord length which is projected area of the curved aerofoil on which the surface forces act. The variation of aerodynamic coefficient with angle of attack obtained by FLUENT using SST turbulence models are shown in Fig.3.

The variation of aerodynamic coefficient with α is observed to follow the expected trend and matches well with the measurement data and computational data especially at lower angle of attacks (α ≤10˚).The maximum lift (Fig.5.5a) for FLUENT and in-house is observed at 16˚ as compared to the measurement data of 15˚with the value of maximum lift for RANS3D being closer to the measurement data. Beyond α = 16˚ there is a sudden drop in lift coefficient, thus by definition it is the static stall angle of attack.

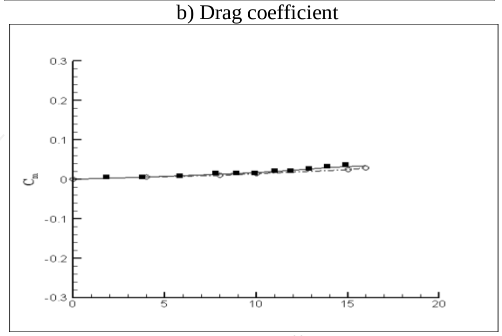
The variation of drag coefficient and moment coefficient about the quarter chord (0.25C) is shown in Fig.3b and Fig.3c respectively. The plot clearly shows that RANS3D has closer agreement with measurement than FLUENT. Even though both FLUENT and RANS3D predict the stall later by 1˚ as compared to measurement data the overall prediction of RANS3D has better agreement with the measurement data. The better prediction obtained using the in-house code 3D-PURLES may be mainly attributed to the fact that the grid used for this simulation was finer (y+<1) and having a better grid quality.



a) Lift coefficient



b) Drag coefficient



c) Moment coefficient

Fig.3 Variation of aerodynamic coefficient for flow past stationary NACA 0012 aerofoil (Re = 3$\times$10[6], SST Turbulence Model)

5. Rotated grid

The aerodynamic coefficient and surface pressure distributions discussed in sections 4.2 & 4.3 are the results obtained by giving the flow (i.e. velocity) at an angle of attack. In this case the lift and drag are also resolved for their x & y components which are respectively as follows:

$$L = -\sin\alpha + \cos\alpha$$
$$D = \cos\alpha + \sin\alpha$$

These values are manually calculated and entered each time for the required angle of attack of the flow. Our area of interest now, was to check whether FLUENT gives the same results when the aerofoil along with entire grid is rotated about the quarter chord (as solid body rotation)to the corresponding angle of attack as that of the flow. This can be achieved using "Rotate" option provide in FLUENT. For example to rotate the gird to an angle of attack of 4˚ clockwise (i.e. upward)

ISSN: 2278-0181
Vol. 2 Issue 12, December - 2013

the rotation angle is specified as 'α = - 4°' with the rotation origin at (0.25, 0).

The values of the aerodynamic coefficient obtained at five different angles of attack for both the cases of flow being rotated and the grid rotated are tabulated as shown in Table.2.From the table is it clear that all the aerodynamic coefficient are same for both the cases at corresponding angles of attack hence, FLUENT gives same results irrespective of the method by which rotation is achieved.

Angle of attack (α°)	**Flow**			**Aerofoil**		
	C_l	C_d	C_m	C_l	C_d	C_m
4	0.3899	0.01148	0.0055	0.3937	0.01143	0.0053
8	0.7521	0.02089	0.0097	0.7474	0.02116	0.0111
12	1.0416	0.04045	0.0186	1.0544	0.04051	0.0187
14	1.1926	0.05221	0.0236	1.1513	0.05522	0.0231
16	1.2590	0.06410	0.0294	1.2511	0.07098	0.0278

Table.2 Comparison of the aerodynamic co-efficient for the flow at an angle of attack and grid rotated for corresponding angle of attack

6. Conclusion and Future work

Performance of widely used CFD commercial code, namely FLUENT, is reported for turbulent flow past stationary NACA 0012 aerofoil and the results obtained are encouraging and shows a good agreement with the measurement data and in-house computational data.

As a future work the generated O- grid can be resolved further or regenerated to maintain a y+ value less than 1 which may give a closer agreement of FLUENT results with measurement data.

Computations may also be repeated with different models and schemes and can be compared with the present simulation with k-ω SST second order upwind scheme in an attempt to understand the advantage/disadvantage of their usage.

7. References:

[1] K.W.McAlsiter, S.L.Pucci, W.J.McCorskey, and L.W.Carr, *An Experimental study of dynamic stall on advanced aerofoil sections, pressure and force data*, NASA TM 84245, 2, 1982

[2] Ismail H.Tunker, James C.Wu and C.M.Wang, *Theoritical and numerical studies of oscillating airfoils*, AIAA Journal, Vol.28, No.9, 1990

[3] Weixing Yuan, Mahmood Khalid, *Computation of Unsteady Flows Past Aircraft Wings at Low Reynolds Numbers*, Canadian Aeronautics and Space Journal, 2004, 50(4): 261-271, 10.5589/q04-020

[4] Zifeng Yang, Hirfumi Igarashi, Mathew Martin and Hui Hu, *An experimental investigation on aerodynamic hysteresis of a low-Reynolds number airfoil.* AIAA-2008-0315.

[5] D.H.Charles (1981), *Two-dimensional aerodynamic characteristics of the NACA0012 airfoil in Langley 8-foot transonic pressure tunnel*, NASA TM 81927