Abstract

Overset Grid method is a popular technique used to solve flow around complex geometries in CFD. In this study, a two-dimensional viscous flow around NACA0018 airfoil has been modeled using OpenFOAM® v1706’s new overset mesh functionality. Discretization has been performed such that the domain grids overlap the single blocked airfoil grids with an inter-grid connectivity. During simulations, these blocked and overlapping grids communicate with each other and transfer relative information. Computational simulation has been carried out for Reynolds numbers 3x10^5, 5x10^5 and 7x10^5 and angle of attack, α = 0° to α = 30° with an interval of 5° using k-ε turbulence model. Finally, aerodynamic performance characteristics such as lift, drag and moment were observed and compared with experimental study.

Keywords: Mesh generation, overset, OpenFOAM®, NACA 0018, validation study.

1. Introduction

The major concern in any numerical simulation is generation of a proper refined mesh around the area of interest in any numerical structures such as an airfoil. Appropriate mesh elements in a computational domain produce accurate results and reduce computation time. Early 70’s saw a boom in the field of computational fluid dynamics with the introduction of 2D structured meshes to solve Reynolds-averaged Navier–Stokes (RANS) equations for aircraft components, which cut down cost and time involved in wind tunnel simulations [1]. Two-dimensional Cartesian C-type grids on the surface of airfoil were algebraically generated by solving elliptic partial differential equations using computer codes and were incorporated into the numerical simulations [2]. Further development in the field of mesh generation led to the introduction of 3D meshes for complete flow visualization. In 3D mesh generation, the entire fluid domain is divided into rectangular blocks or multi-blocks, which are later discretized by perimeter discretization producing two-dimensional grids over the rectangular block. On applying suitable boundary conditions, these rectangular blocks form a volume grid [3]. With the passage of time, unstructured grid generation technique was introduced and adopted widely in numerical simulation as it required less computation time with its simplified mesh creation around complex geometry. In order to solve Euler equations on complex geometries, unstructured triangular meshes are constructed using multigrid scheme. Using the common O-mesh technique, the mesh nodes are generated for regular triangular grids. Later, by shifting these mesh nodes by half the width of the mesh and connecting them with a straight line, symmetrical triangular meshes around the complex geometry are produced [4]. A mixture of triangular and quadrilateral meshes on a three-component body are produced by developing Cartesian grids over an individual body, then eliminating the unused grids and finally reconnecting all the remaining grid points allowing only the quadrilateral mesh over the surface of the geometry [5]. Despite the ease to generate mesh around composite structures, the computational resource consumed by the unstructured grids is high. The unstructured tetrahedral grids are large cells and their inefficient geometrical shape makes it difficult to model viscous layers. Therefore, an algorithm had to be developed in such a way that the grid points are not connected with each other. This led to the formulation of the meshless technique [6]. Grid less approach to treat the interface or boundary problems was developed with a structured grid on curved boundaries. The least square method is used to figure out fluxes of the cut cells. However, this method does not play well for three-dimensional mesh study [7]. Hence, an approach in which all the individual components of the complex geometry are meshed independently and made
overset with the fluid domain was adopted. This particular mesh generation approach is more efficient for multibody, dynamic simulations and near wall treatment cases with no mesh deformation [8]. The overset method treats the individual component meshes as single mesh component. Hence, the hole grid points are ignored, the interpolation is performed between the overlapping points and the boundary conditions are discretized using the discretization points [9]. A number of overset mesh solvers are available in commercial softwares as well as in third-party add-ons [10] to open source softwares and have been explored, implemented and compared by different groups [11-13]. Though principle behind their implementation remains the same, there are minor variations in the results produced by each of them when used for unsteady flow simulations due to the differences in interpolation of cell values from the background mesh to the overset mesh [11]. In the present work, a flow past an airfoil is validated using the open source software OpenFOAM®'s newly released overset mesh functionality. OpenFOAM® v1706 version allows cell-to-cell mapping between the fluid domain cell and the structure cell through proper interpolation. These overlapping grids or overset mesh formulation helps in visualizing the aerodynamic and viscous flow phenomena around any complex geometry or moving body accurately without any cell deformation. If a need to change the geometry or grid arises, it can be done locally without changing the complete mesh. The geometrical design can be incorporated into the existing domain by adding the geometry and creating a mesh around it to become one with the existing grid system. This specific functionality allows inclusion of any number of bodies while simulating dynamic motion without any grid deformation. Generation of overset mesh around the geometry and its integration with the solver was cumbersome in previous versions of OpenFOAM® and other commercial CFD softwares. The new overset mesh functionality in OpenFOAM® version v1706 provides a straightforward method of creating an overset mesh for challenging topological changes where the simulations can be run directly without the need for much modification to the files or mesh [14].

2. Theory

2.1 Governing equations

The steady, incompressible and turbulent flow past an airfoil is governed by incompressible Navier–Stokes equations described as:

\[
\frac{\partial \bar{u}}{\partial t} + \nabla \cdot (\bar{u} \bar{u}) = -\nabla p + \nu \Delta \bar{u}
\]  \hspace{1cm} (1)

The incompressibility of the flow is defined by the continuity equation:

\[
\nabla \cdot \bar{u} = 0
\]  \hspace{1cm} (2)

Both pressure and velocity fields fluctuate in space and time for turbulent flow, therefore equation (2) & equation (3) are time averaged to form Reynolds Averaged Navier-Stokes (RANS) equation:

\[
\frac{\partial \bar{u}}{\partial t} + \nabla \cdot (\bar{u} \bar{u}) - \nabla \cdot (\nu \nabla \bar{u}) = -\nabla \bar{p} + \Delta \cdot (\bar{u} \bar{u})
\]  \hspace{1cm} (3)

\[
\nabla \cdot \bar{u} = 0
\]  \hspace{1cm} (4)

2.2 Turbulence modeling

The unknown term \( \bar{u} \bar{u} \) in equation (3) is modeled using the standard two-equation k-\( \varepsilon \) turbulence model that includes two extra transport equations. The k-\( \varepsilon \) turbulence model solves for two variables, namely the turbulence kinetic energy (k) and rate of dissipation (\( \varepsilon \)) of turbulence kinetic energy, The equations for turbulence kinetic energy (k) and rate of dissipation (\( \varepsilon \)) are as follows:

Turbulence kinetic energy

\[
\frac{\partial k}{\partial t} + \nabla \cdot (\bar{u} k) - \nabla \cdot [(\nu) \nabla k] = \nu \left( \frac{1}{2} \left( \nabla \bar{u} + \nabla \bar{u}^T \right) \right)^2 - \varepsilon
\]  \hspace{1cm} (5)

Dissipation rate

\[
\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\bar{u} \varepsilon) - \nabla \cdot [(\nu) \nabla \varepsilon] = C_1 \frac{\varepsilon}{k} \nu \left( \frac{1}{2} \left( \nabla \bar{u} + \nabla \bar{u}^T \right) \right)^2 - C_2 \frac{\varepsilon^2}{k}\]

2
2.3 Aerodynamics Coefficients

The non-dimensional aerodynamic force coefficients that act on the airfoil are:

Lateral lift coefficient

\[ C_L = \left( \frac{L}{0.5\rho A U_{\infty}^2} \right) \]  

(7)

Longitudinal drag Coefficient (side force)

\[ C_D = \left( \frac{D}{0.5\rho A U_{\infty}^2} \right) \]  

(8)

Moment Coefficient (side force)

\[ C_M = \left( \frac{M}{0.5\rho A U_{\infty}^2} \right) \]  

(9)

The term \( 0.5\rho U_{\infty}^2 \) denotes the dynamic pressure of air (Pa), \( A \) is the area of the airfoil (m²), \( L, D, M \) are the lift (N), drag (N), moment (N) force on the airfoil surface respectively and \( U_{\infty} \) freestream wind speed (m/s).

3. Methodology

3.1 Computational domain

The geometry has been modeled and meshed with a structured grid using open source utility Gmsh 3.0.1 [15]. NACA 0018 airfoil profile has been selected for the study with a chord length of 0.25m and the fluid domain adopted is the same as that of the experimental wind tunnel test section used by W.A. Timmer [16]. The airfoil is placed closer to the inlet at a distance of 0.8m in order to visualize the flow separation in the wake region and is equidistant from both the far fields. The numerical simulation flow configuration is shown in Fig. 1.

![Schematic diagram of the domain and boundary](image)

Fig. 1. Schematic diagram of the domain and boundary

<table>
<thead>
<tr>
<th>Table 1. Specifications for the test case</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Dimension</strong></td>
</tr>
<tr>
<td>Channel Height, H</td>
</tr>
<tr>
<td>Channel Length, L</td>
</tr>
<tr>
<td>Airfoil profile</td>
</tr>
<tr>
<td>Chord length, c</td>
</tr>
<tr>
<td>Reynolds number , Re</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Angle of attack, ( \alpha )</td>
</tr>
</tbody>
</table>
3.2 Mesh Generation

The generated Chimera grid consists of two different mesh components: the background mesh that covers the fluid domain and inner solid mesh that covers the airfoil. The airfoil grid spans the background Cartesian mesh and the overset solid domain completely immersed in the fluid domain is modeled. Coupling between the two overlapped grids is done by interpolation between the two overlapped grids by using the cell centers of both the acceptor and donor points. Interpolation is carried out by finding the inter-grid boundary location value of individual grids and interpolating it over the solution of the overset grid [16]. An important point to be considered while generating the overset mesh on the background mesh is the difference between the sizes of the cells in the two meshes. The difference should be kept as least as possible for proper interpolation between the two meshes. In this present study, a Python script is used to generate single block mesh around the airfoil coordinates. The Python script develops a layer width of about 0.025m from each of the airfoil coordinates and a straight line creating separate block on the airfoil surface then connects the layer widths. These blocks have horizontal layers and vertical layers forming a structured grid around each block as shown in Fig. 2. Finally, the domain grid is modeled in Gmsh 3.0.1 with the block mesh airfoil wrapped in it, forming an overset mesh patch due to the overlapping of the airfoil mesh and the fluid domain mesh as in Fig. 3. Around 1.5 million hexahedral elements have been generated out of which 22680 are overset elements around the airfoil surface and 1444203 elements are for the fluid domain. In addition, the near wall distance from the airfoil surface is kept as 0.002mm to maintain a $y^+$ value of 11 for the overset mesh [16]. Standard k-ε turbulence model is used with a turbulence intensity of 0.03%.

3.3 Overset Grid Semantics

Major and Minor grid: The concept of overset mesh consists of a stationary major and minor grid. The minor grid, which is placed over the major grid, is free to move during dynamic analysis.

Hole grid: The region inside the geometry in the background and overset Cartesian grid where the corresponding flow equations are not solved by the solver.
Fringe points (major and minor): The governing equations are solved separately on each cell and the transmission of the solved results between the grids is performed by interpolation between the fringe cells in each grid. The fringe points that transfer data are called donors and those that accept data are called acceptors.

3.4 Boundary Condition

A no-slip ($U_w = 0$) condition is given on the airfoil walls and on both sides of the far field walls, which is set at a distance of 0.6m away from the airfoil to produce negligible wall effect. Uniform inflow condition for velocity field and zero gradient for pressure field is applied on the upstream side of the domain (Dirichlet boundary condition) while uniform outflow condition and pressure field values are given on the downstream side. Standard atmospheric air properties are selected for the study with an air density $\rho = 1.225$ kg/m$^3$ and kinematic viscosity $\nu = 1.48\times10^{-5}$ Pa/s.

3.5 Solver setting

The flow past airfoil is numerically solved using OpenFOAM® solver overSimpleFoam, provided for the overset utility an extension of simpleFoam solver. A pressure-based, coupled, steady state solver is applied to solve incompressible flow using the SIMPLE algorithm. Depending on the boundary condition specified, the algorithm solves the momentum matrix to measure the velocity field with proper under-relaxation factors. Based on conservation of mass, it iteratively calculates the updated pressure field, velocity field from the corrected pressure field and then updates the boundary condition. This loop goes on until the solution converges [18]. The under-relaxation factor maintained in the present work for pressure is 0.3, velocity is 0.7, epsilon is 0.5 and for turbulence kinetic energy is 0.5. The residual levels for the iterative process are set to 1E-06 as a convergence criterion for the pressure field, velocity field and turbulence field. For all the computational simulations, a calculated time-step size of $\Delta t = 1E-05$ seconds is maintained, in addition to keeping the Courant number below 0.9.

<table>
<thead>
<tr>
<th>Terms</th>
<th>Discretization scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time scheme ($\partial / \partial t$)</td>
<td>Steady-state (time derivatives to zero)</td>
</tr>
<tr>
<td>Divergence ($\nabla$)</td>
<td>Second order, Upwind-biased, Linear interpolation</td>
</tr>
<tr>
<td>Gradient ($\nabla$)</td>
<td>Second order, Gaussian integration, Linear interpolation</td>
</tr>
<tr>
<td>Laplacian ($\nabla^2$)</td>
<td>Gaussian integration, Linear interpolation</td>
</tr>
<tr>
<td>Interpolation</td>
<td>Linear interpolation</td>
</tr>
</tbody>
</table>

Table 2. List of Numerical schemes selected for the present study [18].

4. Results and discussion

4.1. Distribution of the aerodynamic coefficient at $Re = 3\times10^5$

Fig. 4 shows the lift coefficient, drag coefficient and moment coefficient comparison on the airfoil surface with experimental study at $Re = 3\times10^5$ for six different angles of attack, $\alpha$. The overset mesh patch and k-ε turbulence model adequately predicts the lift, drag and moment coefficient trend with that of the experimental wind tunnel test conducted by W.A. Timmer [16]. The lift curve shows a steady plateau phase before the stall regime, as the airfoil angle of incidence increases, the lift coefficient also increases until $\alpha = 15^\circ$ beyond which there is a constant decline in lift. This regime is known as the stall regime. The results suggest that the overset mesh was able to predict the stall regime where the flow separation occurs and gradually moves towards the airfoil upstream. Therefore, we can observe that until $\alpha = 15^\circ$, the airflow is smooth over the airfoil surface above which the flow detached. In the post stall regime, massive flow separation takes place with two-bubble separation that can be visibly seen in the contour plot in Fig. 9. Single large bubble forms downstream towards the leading edge and a small bubble forms at the trailing edge. This results in huge wake vortex formation in the domain leading to the increase of drag coefficient. Sudden jump in drag coefficient has been predicted at $\alpha = 10^\circ$. All the three plots shows good correlation with the experimental results. Small discrepancies in the plot are due blockage effect that occurs in wind tunnel testing or the experimental flow field may not be fully turbulent when compared to fully turbulent simulation results.
The coefficient of moment, calculated from about a quarter of the length of chord of the airfoil suggests that the pitching moment coefficient for the given airfoil is zero at $\alpha = 0^\circ$ and $\alpha = 5^\circ$. At higher angle of attack, it falls to negative. The plot also clearly defines that there is very little variation in the airfoil’s moment coefficient with respect to increasing angle of attack as the value of pitching moment of a symmetrical airfoil always stays very close to zero. The slope in the plot indicates the longitudinal stability of the airfoil.

![Graphs](image)

**Fig. 4.** Effect of Reynolds number on (a) drag curve (b) lift curve & (c) moment curve

**Fig. 5.** Effect of Reynolds number on lift and pitching moment coefficients
4.2 Distribution of the aerodynamic coefficient at \( Re = 5 \times 10^5 \)

The results for \( Re = 5 \times 10^5 \) show similar trend with that of \( Re = 3 \times 10^5 \). The coefficient of lift and drag increases with increase in Reynolds number until the stall regime. From Fig. 6, we can observe that the lift plot correlates with the experimental data with a smooth spline curve until \( \alpha = 0^\circ \). After \( \alpha = 15^\circ \), the curve predicts the stall regime similar to that shown in the experimental data, separated bubbles from the leading and trailing edge of the airfoil, captured clearly by overset mesh, tend to decrease the lift coefficient significantly. In addition, as the Reynolds number is increased, the sudden burst of separated bubble is also visualized, as the flow transition to turbulence phase. After the stall regime, the coefficient of lift increases rapidly while the coefficient of drag increases gradually as angle of attack increases. However, as seen in \( Re = 3 \times 10^5 \) case, there is a sudden jump in the drag plot which is due to the large flow separation in the stall regime. The moment, on the other hand, falls steadily from \( \alpha = 0^\circ \) and forms a slope until \( \alpha = 30^\circ \) which stabilizes the airfoil’s aerodynamic center post stall.

![Graphs showing lift and drag coefficients](image)

**Fig. 6.** Effect of Reynolds number on (a) drag curve (b) lift curve and (c) moment curve

4.3 Distribution of the aerodynamic coefficient at \( Re = 7 \times 10^5 \)

As expected at \( Re = 7 \times 10^5 \), the results are in line with those obtained in the previous two cases. The flow is completely attached to the upper and lower surface of the airfoil for low angle of attacks from \( \alpha = 0^\circ \) to \( \alpha = 5^\circ \) which can be observed from both the lift and drag plots. From the smoothness of the lift curve, we can conclude that the flow on the suction side of the airfoil is smooth, and hence the flow is in the laminar flow regime. There is a small turbulent wake region at the trailing edge. Further, as the angle of attack is increased, stall occurs as the velocity of flow over the airfoil surface decreases due to the pressure difference in boundary layer as an effect of adverse pressure gradients. The sudden shift in the plots at \( \alpha = 15^\circ \) can be attributed to the increased flow velocity and higher angle of attack which pushes the flow separation regime towards the leading edge of the airfoil. The drag plot shows good correlation with the experimental results as the pressure drag increases with increase in flow velocity. The pressure drag pushes the transition phase from the suction side to move
further towards the leading edge developing laminar separation at about $\alpha = 15^\circ$ which has been discussed in the velocity contour plot below. Around $\alpha = 15^\circ$ fully developed turbulent flow generates separation bubbles at the trailing edge reducing lift after $\alpha = 15^\circ$. The generation of lift is possible on increasing the angle of attack beyond $\alpha = 20^\circ$ even though there is loss of boundary layer formation due to vortices.

Fig. 7. Effect of Reynolds number on (a) drag curve (b) lift curve and (c) moment curve

From Fig. 8, we can observe that at $\alpha = 0^\circ$, the flow over the foil is smooth creating a laminar slipstream on the surface that is visibly calculated by the overset mesh. Later at $\alpha = 5^\circ$, the flow is influenced by laminar separation bubbles building up turbulent regime in the suction region with strong adverse pressure gradient this region is called as the transition region. When the angle of attack is increased to $\alpha = 15^\circ$, the flow detaches from the surface and gives rise to chaotic. The chaotic flows generate vortices from the mid-section of the suction region that directly leads to the increase in drag on the airfoil. Generation of induced drag due to the vortices formed at the trailing edge of the airfoil at $\alpha = 20^\circ$ reduces the lift on the airfoil. Such vortices are called as trailing vortices. The overset mesh is able to capture the high-pressure curling of air pushing up from the leading and trailing edges of the airfoil leading to flow reversal over the airfoil surface. These counter rotating vortices at the trailing edge define the unstable boundary layer at the leading edge. As the separation bubble moves to the leading edge of the airfoil, a complete separated flow can been noticed at $\alpha = 25^\circ$ and $\alpha = 30^\circ$. The reduction in lift over the airfoil surface occurs when these separated turbulent vortices reattach with the airfoil surface.
5. Conclusion

The main objective of this work was to validate OpenFOAM® v1706’s new overset mesh function’s capability for finite volume calculations. Flow over an airfoil was discretized using overset mesh and modeled with k-ε turbulence modeling. The new overset functionality of OpenFOAM® v1706 was able to capture the attached and separated flow field over the airfoil surface and accurately predict the airfoil stall regime above $\alpha = 15^\circ$. In addition, the trailing edge wake formulation and small-scale vortices were perfectly captured by the overlapping grids. In terms of validation, the aerodynamic coefficients obtained using overset grid were in reasonable agreement with the experimental data. This proves that the new overset utility works well for cell-to-cell mapping to form a single composite domain. As part of future scope of this work, we aim to use the overset grid functionality in dynamic analysis on wind turbine blades.
6. References


[10] Andreas Groß, Development and Application of the overset grid library Bellerophon, 4th OpenFOAM User Conference 2016, Cologne - Germany


