Numerical Investigation on Effect of Turbulence Model Selection for Aerodynamic Prediction of Axial Flow Fan

Bharanitharan K J¹, S Senthilkumar¹* and Bhavesh Dadhich¹
Department of Aerospace Engineering, SRM Institute of Science and Technology, Kaatankulathur, Chennai, India – 603203.

Abstract: Axial fans are one of the major power consuming cooling systems in most of the devices being used in daily basis. So, it is very important to predict the aerodynamic performance of the fan design. One way of estimating the fan performance in full 3D scale is by using Computational fluid dynamics. Turbulence models plays an important role in application of CFD for Axial flow fans. It is important to choose proper turbulence model in order to get the most accurate results. In this study, a baseline model experiment was chosen and validated. Both steady and transient studies have been carried out. Steady state simulation was performed using Reynolds Averaged Navier Stokes turbulence models such as Spalart Allmaras, RNG k-ε with swirl dominant flow, k-ω SST model and Transition SST model. Transient simulation is performed using RANS model such as Spalart Allmaras, Standard k-ε with standard wall function, RNG k-ε with swirl dominant flow, k-ω SST model, Detached Eddy Solver and Standard k-ω with embedded Large Eddy Solver. Both studies reveal that with coarse mesh URANS model can predict more accurate than Hybrid URANS models. Out of all the models DES showed a very good prediction in terms of pressure drop.

1. Introduction
Axial fans are one of the important components in most of the mechanical design. It is designed in order to maintain a peak pressure difference between suction and pressure side. More details on aerodynamic characteristics are stated by R.A. Wallis [1]. Unsteady Scale Adaptive Simulation for axial flow fan carried out by Younsi M and Lavedrine J [2] have predicted aerodynamic performance of fan with 1% error with experimental results. Scientific review carried out by Corsini A et al. [3] revealed that in most cases for industrial fan RANS model like k-ε and k-ω can predict more accurately but whereas for axial fans which reaches peak pressure conditions the stability limit of RANS models was reached. Simões et al. [4] computation study on axial flow compressor using RANS turbulence models like k-ε, standard k-ω and SST k-ω revealed that SST k-ω to predict result more accurately. Scientific review by Casey M [5] states that there is no single universal turbulence model which can be applied for any turbomachinery computational studies. Panigrahi D and Mishra D [6] computational study for mine ventilation fans showed k-ε model to predict well with experimental data. Numerical study carried out by Casseer D and Ranasinghe C [7] for ceiling fan with various RANS turbulence model have shown Spalart Allmaras can predict more accurately than other models with 2% error when compared with experimental results. Numerical study of Axial Fan with Detached Eddy Solver by Junger C et al. [8] shows that DES predicts with an error of 9.6% in Ansys Fluent and 2% with OpenFOAM. In this study it is decided to perform both steady and transient simulation for various RANS model like k-ε, Spalart Allmaras and k-ω. Along with Detached eddy simulation and Embedded Large eddy simulation.

*Corresponding Author: Dr. S. Senthilkumar, Associate Professor, Department of Aerospace Engineering, SRM Institute of Science and Technology (E-mail: s.senthilms@gmail.com/senthils7@srmist.edu.in)
2. Geometry and Mesh
In order to perform computational study, it is important to validate with experimental results. For this study the benchmark design in reference [9] was used and the sectional diameters of the following fan is obtained from reference [10]. The 3D CAD model of the fan was designed using Autodesk Fusion 360 and shown in Figure 1. The fan design parameters are shown in Table 1.

| Design Parameter           | Specification |
|----------------------------|---------------|
| Hub diameter               | 248mm         |
| Fan diameter               | 495mm         |
| Duct diameter              | 500mm         |
| Clearance                  | 2.5mm         |
| Chord length of blade at hub| 103mm         |
| Chord length of blade at tip| 58mm          |
| Blade Profile              | NACA 4510     |
| Number of Blades           | 9             |

For external flow analysis it is important to design proper flow domain. The flow domain was the replica of the experimental test section [9] with several modifications on outlet side. Duct and the fan rotor section is shown in figure 2. The flow domain used for analysis is shown in figure 3.
Meshing of the whole flow domain was carried out using Ansys Mesher. A hybrid mesh of 12.1 million elements with both tetrahedral and hexahedral mesh was created using multizone technique. The minimum element size was set to 0.5mm near the blade regions and 1mm tetrahedral and prism elements are created around 10mm from the fan surfaces. Using Fluent Meshing the tetrahedral elements are combined into polyhedral elements without degrading the quality of elements. Table 2 Shows the element quality of the mesh generated in terms of skewness.

| Skewness  | Values |
|-----------|--------|
| Maximum   | 0.85   |
| Average   | 0.1964 |

The following and shows the hybrid mesh developed and polyhedral mesh developed using fluent meshing respectively.
Figure 4. Meshing of Flow Domain

Figure 5. Polyhedral Meshing of Flow Domain

Figure 6 and figure 7 shows the polyhedral mesh on fan and close view of mesh at duct section respectively. Figure 7 also shows well refined mesh around the fan surface. A non-conformal interference was created in between rotor and stator intersections in order to perform rotational flow studies in Fluent.

Figure 6. Polyhedral Mesh at fan surface

Figure 7. Mesh at Duct Section

During any computational study there are lot of errors introduced. One such error is due to spatial discretization. One way to reduce the spatial discretization error is by performing grid convergence study. Table 3 shows the variation of pressure drop with change in number of grids.

Table 3. Grid Convergence Table

| Number of elements | 5x10⁶ | 1.1x10⁷ |
|--------------------|-------|---------|
| Pressure drop in Pascals | 139.87 | 133.7 |
3. Numerical Setup
The numerical simulation was carried out using Ansys Fluent. The study was carried out for both steady state assumption and transient flow conditions. CFD study was carried out for various turbulence model with constant flow rate of 1.4 m$^3$/s and fan rotational speed of 1486 RPM. Since the operating speed is less than Mach number 0.3 the flow is assumed to incompressible. In order to predict the experimental results more accurately the experimental turbulent intensity of 5% is specified.

3.1. Steady state assumption
Though the Axial flow fans are unsteady in real conditions sometime steady state assumption can give some good accurate results because the axial flow fan tends to reach its steady condition after several revolutions. The rotational motion of fan for steady state condition is applied using multiple reference frame with absolute velocity formulation conditions by creating sections in flow domain and non-conformal mesh between rotor and stator. Using steady state assumption, the simulation is carried out with Reynolds averaged Navier Stokes turbulence models namely Spalart Allmaras, RNG k-ε with enhanced wall function with swirl dominant flow condition, SST k-ω and Transition SST. Equation (1) and (2) are the mass conservation and momentum conservation equation respectively for 3-dimensional steady incompressible flow.

$$\nabla \cdot (\rho \mathbf{v}) = 0 \tag{1}$$

$$\nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot (\tau) + \mathbf{F} \tag{2}$$

Equation (3) shows the transport equation for Spalart Allmaras turbulent model having one transport variable known as kinematic eddy viscosity [11].

$$\frac{\partial}{\partial x_i} (\rho \mu_i) = G_\theta + \frac{1}{\sigma_\theta} \left[ \frac{\partial}{\partial x_j} \left( (\mu + \rho \tilde{\theta}) \frac{\partial \tilde{\theta}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \tilde{\theta}}{\partial x_j} \right)^2 \right] - Y_\theta \tag{3}$$

The transport variable equation for kinetic turbulent energy and turbulent dissipation rate are (4) and (5) respectively for RNG k-ε model.

$$\frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M \tag{4}$$

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \frac{\varepsilon^2}{k} - R_\varepsilon \tag{5}$$

Transport variable equation for kinetic turbulent energy and specific turbulent dissipation rate are (6) and (7) respectively for SST k-ω model.

$$\frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k \tag{6}$$

$$\frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega \tag{7}$$

The governing equation for transition SST 4 transport variable turbulent kinetic energy, specific dissipation and intermittency are (6),(7) and (8) respectively. The another transport variable for this model is transition onset criteria in term of momentum thickness Reynolds Number.

$$\frac{\partial (\rho \mu_j \gamma)}{\partial x_j} = P_{\gamma 1} - E_{\gamma 1} + P_{\gamma 2} - E_{\gamma 2} + \frac{\partial}{\partial x_j} \left( \mu_j \frac{\partial \gamma}{\partial x_j} \right) \tag{8}$$
Using moving reference frame allows the cell zone with rotating reference as unsteady zone with relative to inertial reference. Application of Frame motion in Ansys Fluent allows to incorporate acceleration due to rotational motion in the governing equations [12]. Velocity formulation in moving reference frame is shown from (9-11).

\[
    \bar{V}_r = \omega \bar{a}
\]

\[
    \bar{V}_r = \bar{V} - \bar{u}_r
\]

\[
    \bar{u}_r = \bar{V}_t + \bar{\omega} \times \bar{r}
\]

### 3.2. Transient Simulation

Transient simulation of Axial fans are carried out using sliding mesh model. In sliding mesh model insteading velocity formulation the whole mesh of the cell zone selected as moving mesh will be updated each time step based on the user defined rotational motion. The transient simulations were carried out with RANS model such Spalart Allmaras, standard k-ε with standard wall function, RNG k-ε with standard wall function and swirl dominant flow formulation and SST k-ω. Along with RANS model embedded Large eddy simulation was carried out by assigning the duct section as LES zone and remaining cell zones with standard k-ε turbulence model. Detached eddy simulation with k-ω SST model was also carried out which is converse of embedded LES by formulating near wall region with URANS and other region with LES [13].

The mass conservation and momentum conservation for transient study are (12) and (13) respectively.

\[
    \nabla \cdot (\rho \mathbf{v}) = 0 \quad (12)
\]

\[
    \frac{\partial}{\partial t} (\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot (\bar{r}) + \bar{F} \quad (13)
\]

Unsteady transport equation for kinematic eddy viscosity, turbulent kinetic energy for RNG k-ε model, dissipation rate, turbulent kinetic energy for k-ω SST model and specific dissipation rate are (14 – 18) respectively.

\[
    \frac{\partial}{\partial t} (\rho \theta) + \frac{\partial}{\partial x_i} (\rho \theta u_i) = G_\theta + \frac{1}{\sigma_\theta} \left[ \frac{\partial}{\partial x_j} \left( \mu + \rho \theta \frac{\partial \theta}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \theta}{\partial x_j} \right)^2 \right] - Y_\theta \quad (14)
\]

\[
    \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \epsilon - Y_M \quad (15)
\]

\[
    \frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right) + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \frac{\epsilon^2}{k} - R_\epsilon \quad (16)
\]

\[
    \frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \frac{\omega}{\Gamma_\omega} \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega \quad (17)
\]

For Detached eddy simulation with k-ω SST model the dissipation term \(Y_k\) is replaced as shown in (19).

\[
    Y_k = \rho \beta k \omega F_{DES} \quad (19)
\]

\[
    F_{DES} = \max \left( \frac{l_t}{l_{des} \delta_{max}}, 1 \right) \quad (20)
\]

### 3.3. Numerical Discretization

Continuity equation is solved using second order discretization scheme. Remaining all the governing equations are solved using second order upwind scheme except the case of DES and LES. For DES and LES the momentum equation is solved using bounded central difference scheme. The unsteady term for
all the cases except DES and LES were solved using First order implicit scheme whereas for DES and LES, second order bounded central difference scheme was used. Pressure velocity coupling was performed using SIMPLEC algorithm. The convergence factor was set to $10^{-6}$ for all the flow variable. For transient simulation convergence was monitored for every time steps. A time step size of 0.002s was employed in order to satisfy the courant number for the current minimum grid size. In case of steady state study mass balance is monitored for each iteration. In order to perform a comparative study time sampling of the data were taken as the flow is unsteady. The simulations were carried out up to 0.12s of flow time at which the fan reaches stable operating condition.

3.4. Measuring Technique
In this study two important flow properties are measured importantly, that is pressure drop and meridional velocity in the suction side of the fan. Pressure drop of the fan is calculated by estimating the total pressure difference between the suction side of the fan and outlet (standard atmospheric condition). The meridional velocity is obtained along the radius of fan from hub to tip at 20mm from origin of fan at suction side.

4. Results and Discussion

4.1. Validation
Validation is an important part of any CFD study. The simulation results are first validated using Unsteady k-ω SST model in terms of pressure drop and velocity profile in suction side.

![Figure 8. Validation of Meridional Velocity profile of fan suction side using Unsteady k-ω SST model](image)

|               | Experiment | CFD  |
|---------------|------------|------|
| $\Delta P$ (Pascals) | 126.5      | 133.7|

Figure 8 and Table 5 shows the validation of CFD model with experimental results using unsteady k-ω SST model. The error in pressure drop is 5.7% with experimental results. Also, from Figure 8 it is clear that k-ω SST model is predicting experimental results along the profile quite accurately along with some distortion near tip region. The overall average error percentage of meridional velocity along the radius of fan is 4.9%.

4.2. Steady state characteristics
Show the pressure drop obtained for various turbulence models and the percentage of the error with experiment results.
Table 5. Pressure Drop for Steady State Analysis

| Turbulence model | Experiment | Spalart Allmaras | RNG k-ε with Enhanced wall function | k-ω SST | Transition SST |
|------------------|------------|-----------------|-------------------------------------|---------|---------------|
| ∆P (Pascals)     | 126.5      | 246.24          | 214.79                              | 150.6   | 151.2         |
| Error %          | -          | 94.6            | 69.8                                | 19      | 19.5          |

From Table 5 it is clear that assumption of steady state for a turbomachinery problem like axial flow fan can’t predict as accurate results equivalent to the transient studies. For steady state assumption it is also clear that one equation Spalart Allmaras model and RNG k-ε model even with enhanced wall function fails. The steady state study shows that among all the turbulence model being used shear stress transport models were seem to predict some results nearer to experimental results although with an error of 19%.

4.3. Unsteady Characteristics

Unsteady simulations are carried out using two different type of turbulence models such as URANS which includes Spalart Allmaras, standard k-ε with standard wall function, RNG k-ε with standard wall function and k-ω SST and Hybrid URANS which includes Detached eddy simulation and embedded Large eddy simulation.

4.3.1. URANS Models. Figure 9 shows the meridional velocity profile for various URANS models. It clearly shows that all the RANS model attains the velocity profile but with some percentage of errors with the base experimental results. The average error estimation of meridional velocity shows that the Spalart Allmaras model has the minimum error percentage of 2.7% followed by RNG k-ε model with 3.4% of error. It is clear that the standard k-ε model with standard wall function fails with error of 18%. From Figure 9, at the hub and tip region one equation Spalart Allmaras model and k-ω SST model have a good capability in predicting the velocity profile all along the blade radius.
Figure 9. Meridional Velocity profile of fan suction side for URANS models

Shows the pressure drop for various URANS turbulence model. The following table infers that RNG k-\(\varepsilon\) model predicts pressure drop more accurate than most of the RANS model including one of the most used k-\(\omega\) SST model.
Table 6. Pressure Drop for URANS models

| Turbulence Model | Experiment | Spalart-Allmaras | Standard k-ε with standard wall function | RNG k-ε with standard wall function | k-ω SST model |
|------------------|------------|------------------|----------------------------------------|-----------------------------------|---------------|
| ∆P (Pascals)     | 126.5      | 139.4            | 136.07                                 | 132.29                            | 133.7         |
| Error %          | -          | 10.2             | 7.56                                   | 4.3                               | 5.69          |

4.3.2. Hybrid URANS models. Table 7 Show the pressure drop prediction of hybrid URANS model such as DES and embedded LES.

Table 7. Pressure Drop for Hybrid URANS models

| Turbulence model | Experiment | Detached Eddy Simulation with k-ω SST model | Large Eddy Simulation with standard k-ω model |
|------------------|------------|---------------------------------------------|-----------------------------------------------|
| ∆P (Pascals)     | 126.5      | 127.8                                       | 142.008                                       |
| Error %          | -          | 1.02                                        | 12                                           |

Table 7 clearly shows that Large eddy simulation fails to predict the pressure drop properly and detached eddy simulation predicts the pressure drop with vary less percentage of error equal to 1.02%. The failure of Large eddy simulation possibly could be due to the grid resolution in the large eddy simulation zone. Grid fineness is one of the important factors for LES.

![Figure 10. Meridional velocity of fan suction side for Hybrid URANS model](image_url)

Figure 10 clearly shows that the grid quality was in sufficient for both the hybrid models in order to use large eddy solver. Detached eddy simulation predicted the experimental results with an average error of 5.25% but embedded large eddy simulation resulted with an average error of 18%.
4.4. Comparison of URANS and Hybrid URANS

From Figure 11, it is clear that in a coarse grid RANS models with wall functions have a good ability to predict the results even more accurately than hybrid URANS models. Among all the turbulent models used one common thing noticed is all the model has some distortion at the tip region of Fan. Though DES failed in predicting the accurate velocity profile at tip region it predicted the pressure drop more accurately than any other turbulence model. A small increment in grid can improve the results of Detached eddy simulation.

![Figure 11. Meridional velocity profile of fan suction side for unsteady simulations](image)

Table 8 shows the pressure drop and error for all the unsteady turbulence model used and their error percentage.

| Turbulence Model                  | ΔP (Pascals) | Error % |
|-----------------------------------|--------------|---------|
| Experiment                        | 126.5        | -       |
| Spalart Allmaras                  | 139.4        | 10.2    |
| Standard k-ε with standard wall function | 136.07      | 7.56    |
| RNG k-ε with standard wall function | 132.29      | 4.3     |
| k-ω SST model                     | 133.7        | 5.69    |
| DES with k-ω SST                  | 127.8        | 1.02    |
| LES with standard k-ω             | 142.008      | 12      |
5. Conclusion
A numerical investigation was carried out for various turbulence model like RANS, URANS and Hybrid RANS for the application of CFD of Axial flow fan. For an Axial flow fan steady assumption fails and most of turbulence model was not able to predict proper results. Only Shear stress transport models were capable of predicting a nearest possible experimental value with a minimum error of 19%. For transient study among all the URANS model RNG k-ε with enhanced wall function has a good predictability compared to all other URANS models. Hybrid model embedded LES failed in terms of predicting pressure drop and velocity profile because of coarse grid. DES was the most accurate turbulence model in term of pressure drop even with coarse grid. Even with coarse grid DES was able to predict velocity profile well at near hub region. For coarse grid the study reveals that URANS models can perform well in terms of accuracy and computational time. Improving grid little more can enable DES to be perfect model for Axial flow fan simulations. Another important finding includes that all the turbulence model used in Ansys Fluent in this study failed to capture proper velocity profile at tip region.

6. Nomenclature

| Symbol | Nomenclature |
|--------|--------------|
| v      | Velocity vector |
| p      | Pressure |
| \( \tilde{\theta} \) | Kinematic eddy viscosity |
| \( \tilde{\tau} \) | Shear tensor |
| \( \mu \) | Kinematic viscosity |
| \( \rho \) | Density |
| \( C_{92} \) | Spalart Allmaras turbulent constant |
| Y      | Dissipation |
| G      | Generation |
| \( C_{1\epsilon}, C_{3\epsilon} \) and \( C_{2\epsilon} \) | k-ε turbulent constants |
| \( k \) | Turbulent kinetic energy |
| \( \epsilon \) | Rate of dissipation |
| \( D_{\omega} \) | Cross diffusion |
| \( \omega \) | Specific rate of dissipation |
| \( \alpha \) | Effective Prandtl number |
| \( \Gamma \) | Effective diffusivity |
| \( \tilde{u}_r \) | Relative velocity during Moving reference frame |
| \( \dot{a} \) | Acceleration |
| \( \dot{V}_t \) | Tangential velocity |
| \( \Delta P \) | Pressure drop |
| \( C_m \) | Meridional velocity |
| \( r/r_{duct} \) | Ratio of radius of blade to duct |

Reference

[1] Wallis R A 1961 Axial Flow Fans (Academic Press) pp 13-28
[2] Younsi M and Lavedrine J 2015 Unsteady flow and acoustic behaviour of an axial fan: Numerical and experimental investigations 11th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics ETC2015-248, pp 1–13
[3] Corsini A, Delibra G and Sheard A G 2013 A critical review of computational methods and their application in industrial fan design International Scholarly Research Notices vol. 2013 625175 (doi.org/10.1155/2013/625175)
[4] Simões M R, Montojos B G, Moura N R and Su J 2009 Validation of Turbulence Models for Simulation of Axial Flow Compressor Proceedings of COBEM 2009, vol. 0, pp 1–9

[5] Casey M V 2002 Validation of turbulence models for turbomachinery flows – a review (Oxford: Elsevier Science Ltd) pp 43–57

[6] Panigrahi D C and Mishra D P 2014 CFD Simulations for the Selection of an Appropriate Blade Profile for Improving Energy Efficiency in Axial Flow Mine Ventilation Fans Journal of Sustainable Mining vol. 13, pp 15–21

[7] Casseer D and Ranasinghe C 2019 Assessment of Spallart Almaras Turbulence Model for Numerical Evaluation of Ceiling Fan Performance Moratuwa Engineering Research Conference pp 577–582 (doi.org/10.1109/MERCon.2019.8818888)

[8] Junger C, Zenger F, Reppenhagen A, Kaltenbacher M and Becker S 2016 Numerical simulation of a benchmark case for aerodynamics and aeroacoustics of a low pressure axial fan Proceedings of the INTER-NOISE 2016 - 45th International Congress and Exposition on Noise Control Engineering: Towards a Quieter Future pp 741–7

[9] Zenger F, Junger C, Kaltenbacher M and Becker S 2016 A Benchmark Case for Aerodynamics and Aeroacoustics of a Low Pressure Axial Fan SAE Technical Papers 2016-01-1805 (doi.org/10.4271/2016-01-1805)

[10] Krömer F 2017 Sound emission of low-pressure axial fans under distorted inflow conditions (Erlanger: FAU University Press) vol 20 (doi.org/10.25593/978-3-96147-089-1)

[11] Spalart P and Allmaras S 1992 A one-equation turbulence model for aerodynamic flows 30th Aerospace Sciences Meeting and Exhibit (doi:10.2514/6.1992-439)

[12] ANSYS Academic Research 2018 ANSYS Fluent Theory Guide ANSYS Help System

[13] Baker C, Johnson T, Flynn D, Hemida H, Quinn A, Soper D and Sterling M 2019 Chapter 4 - Computational techniques (Butterworth-Heinemann) pp 53–71