# Case Study

# Turbulent Models in Ansys Fluent®

Gautham Varma Raja Kochanattu

Developed and curated by the Ansys Academic Development Team

education@ansys.com

## Software Used

Ansys Fluent®, fluid simulation software
Ansys Mechanical™, structural finite element analysis software

## Summary

Ansys Fluent®, a fluid simulation software that is used to solve various problems related to fluid flow, heat and mass transfer, chemical reactions, and more. It uses advanced physical models like turbulent modeling, multiphase modeling, battery modeling, combustion, and fluid- structure interactions to solve the given problem to high level of accuracy.

In this case study, Ansys Fluent software is employed to explore the performance of various turbulence models and their ability to capture distinct physical phenomena during simulations. A single simulation scenario is analyzed using a NACA 0012 airfoil at a high angle of attack, providing insight into how each turbulence model characterizes flow behavior under challenging aerodynamic conditions.

# Table of Contents

## 1. Introduction

Turbulent flows, characterized by chaotic and unsteady fluid motion, present significant challenges for accurate simulation in engineering applications. In computational fluid dynamics (CFD), capturing the complex interactions of turbulence requires robust modeling techniques. Ansys Fluent software offers a range of turbulence models, each tailored to balance computational cost and accuracy for specific flow scenarios. Among the most widely used models are the $\boldsymbol{k-\varepsilon}$ , $\boldsymbol{k-\omega}$, and **Detached Eddy Simulation (DES)** models, each excelling in different aspects of turbulence prediction.

This case study explores the application and performance of these turbulence models by simulating the flow over a NACA 0012 airfoil at a high angle of attack. By comparing their predictive capabilities, the analysis highlights their strengths and trade-offs in capturing critical flow features such as boundary layer behavior, separation, and wake dynamics.

## 2. Physics Behind the calculations

### 2.1. $k-\epsilon$ Turbulence Model

The $\boldsymbol{k-\epsilon}$ **turbulence model** is one of the most widely used models in computational fluid dynamics (CFD) for simulating turbulent flows. As a **two-equation model**, it solves transport equations for two turbulence properties: the **turbulent kinetic energy (k)** and the **turbulent dissipation rate (ϵ)**. The model provides a balance between computational efficiency and accuracy, making it suitable for a broad range of industrial and engineering applications.

#### 2.1.1. Governing Equations

The standard $k-\epsilon$ model consists of two partial differential equations, one for $k$ and one for $\epsilon$. These equations describe the transport, production, dissipation, and diffusion of turbulence.

##### 2.1.1.1. Turbulent Kinetic Energy (k) Equation

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=P_k-\rho\epsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$

**Explanation of Terms:**

1. $\frac{\partial(\rho k)}{\partial t}$ : Time rate of change of k.
2. $\frac{\partial(\rho k u_i)}{\partial x_i}$ **:** Convection of k due to fluid motion.
3. $\boldsymbol{P_k}$ **:** Production of turbulent kinetic energy:

$$P_k=\mu_t\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

   Here, $\mu_t$ is the turbulent viscosity.
4. $-\boldsymbol{\rho\,\epsilon}$**:** Dissipation of k due to viscous effects.
5. $\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$**:** Diffusion of k, where $\sigma_k$ is the turbulent Prandtl number for k.

##### 2.1.1.2. Turbulent Dissipation Rate ($\epsilon$) Equation

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho\epsilon u_i)}{\partial x_i}=C_1\frac{\epsilon}{k}P_k-C_2\rho\frac{\epsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right]$$

**Explanation of Terms:**

1. $\frac{\partial(\rho\epsilon)}{\partial t}$**:** Time rate of change of $\epsilon$.
2. $\frac{\partial(\rho\epsilon u_i)}{\partial x_i}$: Convection of ϵ due to fluid motion.
3. $C_1\frac{\epsilon}{k}P_k$**:** Production of ϵ proportional to the production of k, where C1C_1C1 is a model constant.
4. $C_2\rho\frac{\epsilon^2}{k}$**:** Dissipation of ϵ, where $C_2$ is a model constant.

5. $\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]$: Diffusion of $\epsilon$, where $\sigma_\epsilon$ is the turbulent Prandtl number for $\epsilon$.

#### 2.1.1.3. Turbulent Viscosity ($\mu_t$)

The turbulent viscosity ($\mu_t$) is calculated as:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

where $C_\mu$ is an empirical constant.

#### 2.1.1.4. Model Constants

The standard k−ϵ model relies on empirically determined constants. Their typical values are:

| Constant | Symbol | Value |
|---|---|---|
| **Turbulent Prandtl number for $k$** | $\sigma_k$ | 1.0 |
| **Turbulent Prandtl number for $\epsilon$** | $\sigma_\epsilon$ | 1.3 |
| **Empirical constant for $\mu_t$** | $C_\mu$ | 0.09 |
| **Empirical constant for $P_k$ term** | $C_1$ | 1.44 |
| **Empirical constant for dissipation** | $C_2$ | 1.92 |

#### 2.1.1.5. Strengths of the k−ϵ Model

- **General-Purpose Use**: The standard $k-\epsilon$ model is suitable for a wide range of flows, including free-shear flows (e.g., jets, mixing layers) and boundary layers.
- **Computational Efficiency**: The simplicity of the equations allows for relatively fast computations compared to more advanced turbulence models.
- **Robustness**: Its robustness and ease of implementation make it a common choice for industrial CFD applications.
- **Good for Free-Shear Flows**: It performs particularly well in flows dominated by turbulence away from walls, such as jets and wakes.

#### 2.1.1.6. Limitations of the k−ϵ Model

- **Near-Wall Accuracy**: The model struggles to resolve near-wall turbulence accurately, especially for low-Reynolds-number flows. Wall functions are often required to approximate the effects of turbulence near solid surfaces.
- **Flow Separation**: The standard $k-\epsilon$ model often fails to predict flow separation accurately, especially in cases with adverse pressure gradients.
- **Anisotropy Assumption**: The model assumes isotropic turbulence, which can lead to inaccuracies in flows with significant anisotropy (e.g., swirling flows or secondary flows).
- **Free-Stream Dependency**: The model's performance can be sensitive to the specification of free-stream turbulence parameters.

## 2.2. $k-\omega$ Turbulence Model

The **k-ω turbulence model** is a two-equation model widely used in computational fluid dynamics (CFD) for modeling turbulent flows. It is designed to provide a balance between accuracy in near-wall regions and robustness for complex flow scenarios. The model solves two transport equations: one for the **turbulent kinetic energy (k)** and another for the **specific dissipation rate (ω)**. Here's a breakdown of its physical basis and capabilities:

### 2.2.1. Governing Equations

The equation for turbulent kinetic energy (k) is derived from the Navier-Stokes equations and describes how k is produced, transported, and dissipated in the flow:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_j}\left[(\mu+\sigma_k\mu_t)\frac{\partial k}{\partial x_j}\right]$$

Terms Explanation:

1. $\frac{\partial(\rho k)}{\partial t}$: Time rate of change of $k$.
2. $\frac{\partial(\rho k u_i)}{\partial x_i}$: Convection of $k$ due to fluid motion.
3. $\boldsymbol{P_k}$: Production of turbulent kinetic energy due to velocity gradients, given by: $P_k = \mu_t \frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$ Here, $\mu_t$ is the turbulent viscosity.
4. $\boldsymbol{\beta^* \rho k \omega}$: Dissipation of $k$, where $\beta^*$ is a model constant and $\omega$ represents the specific dissipation rate.
5. $\frac{\partial}{\partial x_j}\left[(\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_j}\right]$: Diffusion of $k$ due to molecular ($\mu$) and turbulent ($\mu_t$) viscosities. $\sigma_k$ is the turbulent Prandtl number for $k$.

##### 2.2.1.1. Transport Equation for Specific Dissipation Rate ($\omega$)

The specific dissipation rate ($\omega$) is governed by its own transport equation, which describes how $\omega$ is generated, transported, and dissipated:

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho\omega u_i)}{\partial x_i} = \alpha\frac{\omega}{k}P_k - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right]$$

**Terms Explanation:**

1. $\frac{\partial(\rho\omega)}{\partial t}$: Time rate of change of $\omega$
2. $\frac{\partial(\rho\omega u_i)}{\partial x_i}$: Convection of $\omega$ due to fluid motion.
3. $\boldsymbol{\alpha\frac{\omega}{k}P_k}$: Production of ω proportional to $P_k$ Here, α is a model constant.
4. $\boldsymbol{-\beta\,\rho\,\omega^2}$: Dissipation of ω, where β is a model constant.
5. $\frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right]$ : Diffusion of ω due to molecular (μ) and turbulent ($\mu_t$) viscosities. $\sigma_\omega$ is the turbulent Prandtl number for ω.

##### 2.2.1.2. Turbulent Viscosity ($\mu_t$)

The turbulent viscosity is calculated as:

$$\mu_t = \frac{\rho k}{\omega}$$

This relation links the turbulent kinetic energy and the specific dissipation rate to the turbulent viscosity, which influences the momentum, heat, and mass transfer in the flow.

### 2.2.2. Closure Coefficients and Constants

The k-ω model relies on empirically determined coefficients for closure. The standard values are:

| Parameter | Value |
|---|---|
| $\boldsymbol{\beta^*}$ | 0.09 |
| $\boldsymbol{\beta}$ | 0.075 |
| $\boldsymbol{\alpha}$ | 0.52 |
| $\boldsymbol{\sigma_k}$ | 2.0 |
| $\boldsymbol{\sigma_\omega}$ | 2.0 |

### 2.2.3. Boundary Layer Modeling

The k-ω model is particularly effective in capturing turbulence in the **near-wall region** because $\omega$ inherently resolves the boundary layer without requiring wall functions. The model can predict the steep velocity and turbulence gradients near walls accurately, making it suitable for low-Reynolds-number flows and applications with adverse pressure gradients.

### 2.2.4. Extensions of the k-ω Model

k-ω SST (Shear Stress Transport) Model:

- Combines the k-ω model near walls with the k-ε model in free-stream regions.
- Includes a blending function to switch between the two models smoothly.
- Introduces a shear stress limiter to improve predictions for separated flows.

The SST variant is widely used in engineering applications, such as turbomachinery, aerodynamics, and heat transfer, due to its enhanced accuracy and robustness.

## 2.3. GEKO Model

The **GEKO (Generalized k-ω)** model in Ansys Fluent software is a robust and flexible turbulence model developed to provide improved control over the turbulence behavior in Computational Fluid Dynamics (CFD) simulations. It is based on the **k-ω** framework but includes additional parameters to allow customization for specific flow scenarios, combining the strengths of various turbulence models.

### 2.3.1. Key Features of the GEKO Model

1. **Customizable Parameters**: GEKO includes tunable coefficients that allow users to tailor the model to match experimental data or specific flow behaviors. These parameters help adapt the model for different applications such as external aerodynamics, internal flows, or free shear flows.
2. **Enhanced Flexibility**: By adjusting parameters, GEKO can mimic the behavior of other popular turbulence models like:
    - Standard **k-ω** model
    - SST (Shear-Stress Transport) model
    - k-ε model (approximately)
3. **Improved Accuracy for Diverse Flows**: The GEKO model is designed to handle a wide range of flow regimes, including:
    - Wall-bounded flows
    - Free shear flows
    - Flows with adverse pressure gradients
    - Transitioning between laminar and turbulent flows
4. **Boundary Layer Modeling**: Like the SST model, GEKO incorporates features to capture the correct behavior of the boundary layer, particularly for flows with separation or reattachment.

### 2.3.2. GEKO Parameters

GEKO introduces six primary adjustable parameters to control various aspects of the turbulence behavior:

- $C_{SEP}$ : The most important coefficient for most applications is $C_{SEP}$. It controls the separation points/lines from smooth body-separation. Increasing $C_{SEP}$ will lead to stronger/earlier separation. When changing $C_{SEP}$ one should as a first step keep all other coefficients at their default values.
- $\boldsymbol{C_{NW}}$ : The coefficient $C_{NW}$ should only be changed if detailed near wall or surface information needs to be matched and if this cannot be achieved by optimizing $C_{SEP}$ alone. The most prominent example would be optimizations with respect to heat transfer coefficients or oil-flow pictures from experiments. Increasing $C_{NW}$ will increase heat transfer and wall shear stress levels in non-equilibrium regions.
- $\boldsymbol{C_{MIX}}$: In some cases, standard settings (or models) underestimate the turbulent mixing in free shear flows. The coefficients $C_{MIX}$ will allow an adjustment under such scenarios. Increasing $C_{MIX}$ will increase eddy-viscosity levels in such zones. It should be noted that this is only possible within physical limits.
- $\boldsymbol{C_{JET}}$: The coefficient $C_{JET}$ are considered when jets are present in the domain. Regions with round jets should best be computed with $C_{SEP} = 1.75 - 2.00$ as otherwise the effect of $C_{JET}$ is not strong enough to achieve the desired effect.

These parameters make GEKO a versatile tool for fine-tuning simulations to achieve optimal accuracy without requiring a complete change in the turbulence model.

### 2.3.3. Advantages

- **Adaptability**: Ideal for users who work with varied flow conditions and need a single model with customizable parameters.
- **Ease of Use**: While flexible, GEKO parameters are intuitive, and Ansys Fluent software provides default values that work well for many applications.
- **Consistency**: Maintains a unified framework while providing the flexibility of different models

## 3. Geometry and Meshing

The geometry was constructed around a **NACA 0012 airfoil** with a focus on capturing aerodynamic characteristics at a high angle of attack (20 degrees). A computational domain was carefully designed around the airfoil, ensuring sufficient space for freestream flow and downstream wake development. This setup ensures that boundary effects and flow recirculation are captured accurately.

The mesh for this simulation was generated using Ansys Mechanical software**,** with an emphasis on achieving a high-resolution mesh around the airfoil surface and critical flow regions. Fine meshing was applied in the vicinity of the airfoil to resolve sharp velocity and pressure gradients effectively, as illustrated in the Figure 1 below. Special attention was given to the boundary layer region, where a well-structured inflation layer was applied. This inflation consisted of multiple layers with a gradual growth rate, ensuring precise resolution of the boundary layer separation and flow detachment phenomena.

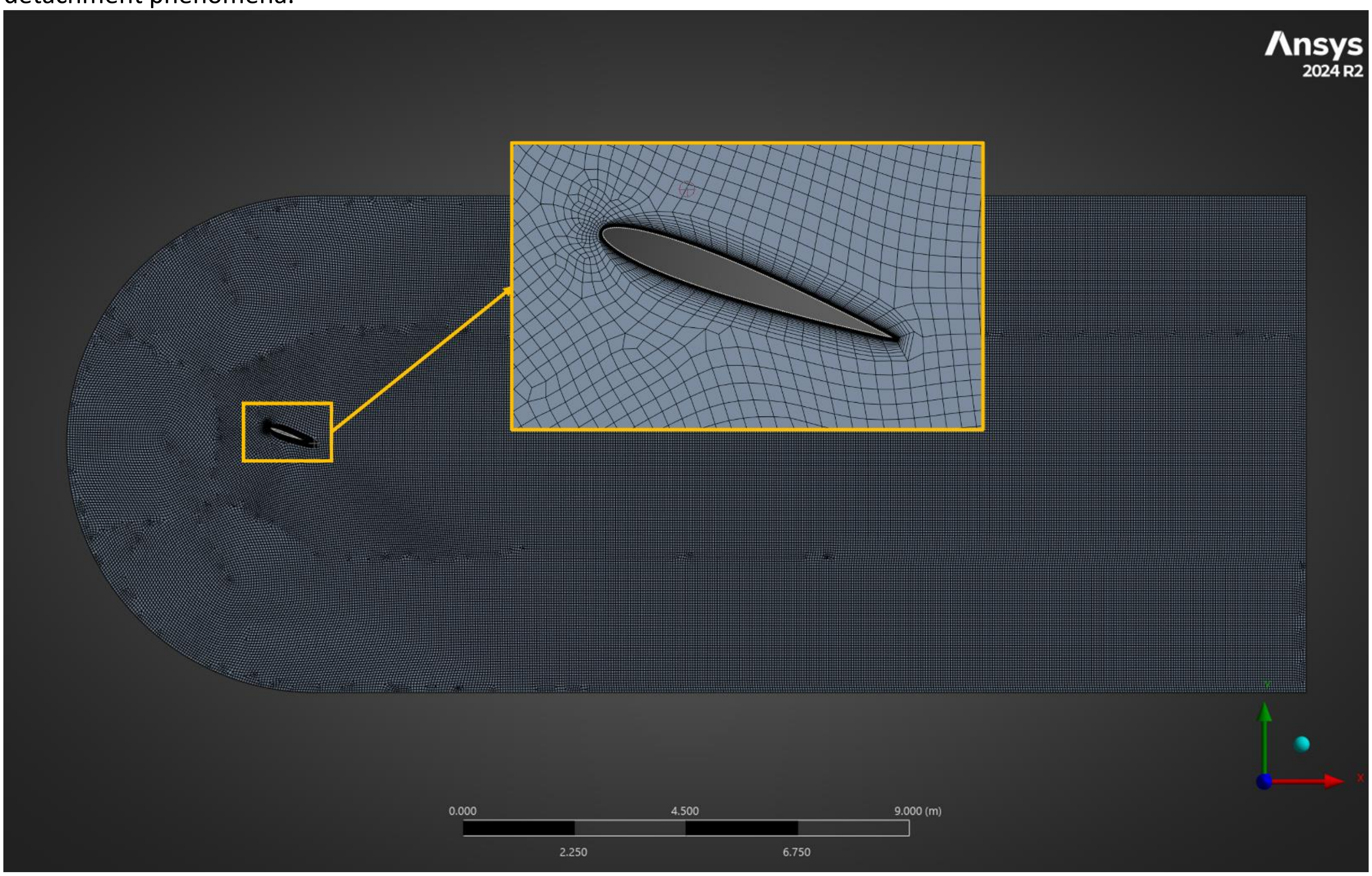


*Figure 1: The Mesh Generated for the simulation also showing the inflation layer near the airfoil for better boundary layer calculations*

## 4. The Results

The simulation was conducted using three different turbulence models with an inlet velocity of 89 m/s, a pressure outlet boundary condition, and air as the working fluid. A coupled pressure-velocity coupling scheme and second-order upwind discretization were used for all spatial terms. The simulations ensured consistent drag and lift coefficients across iterations. While the drag and lift coefficients are generally consistent for all three models (refer

Table 1), the lift coefficient for the GEKO model is slightly lower. This could be attributed to GEKO's tunable parameters, which may influence how it handles flow separation and reattachment near the airfoil surface. Compared to the k-ω SST model, which excels at capturing boundary layer effects and maintaining accurate flow attachment, the GEKO model might predict earlier or stronger separation due to less precise near-wall turbulence resolution, resulting in a slight reduction in lift generation.

| Model | $C_d$ | $C_L$ |
|---|---|---|
| $k-\omega\ SST$ | 0.265878 | 0.717367 |
| $k-\epsilon$ Realizable | 0.271273 | 0.704073 |
| $GEKO$ | 0.253094 | 0.667967 |

*Table 1 : The Lift and Drag coefficients that was obtained from the three turbulent models.*

The velocity contour profiles for the four turbulence models — k-ω SST, k-ε realizable, and GEKO — present distinct differences in how each model captures flow dynamics around the NACA 0012 airfoil at a 20° angle of attack, particularly regarding boundary layer resolution, flow separation, and wake characteristics.

In the k-ω SST model (top left), the velocity gradients near the airfoil surface are sharp, demonstrating the model's ability to accurately resolve the boundary layer. This sharp gradient indicates that the flow close to the surface is well predicted, and the model effectively captures the effects of the adverse pressure gradient, leading to a more realistic representation of flow separation near the trailing edge. As a result, the wake structure downstream of the airfoil is narrow and well-defined, reflecting the model's high fidelity in capturing turbulent behavior in both the boundary layer and wake regions.

In comparison, the k-ε realizable model (top right) shows smoother velocity gradients near the surface, which suggests a less accurate representation of the boundary layer. The flow near the airfoil appears more diffused, and the onset of flow separation is delayed or overpredicted. This leads to a less defined separation zone, which can cause inaccuracies in the prediction of aerodynamic forces, such as lift and drag. The wake region is noticeably broader than in the k-ω SST model, indicating that the k-ε model struggles to resolve the finer turbulent structures and vortex shedding that characterize the wake flow.

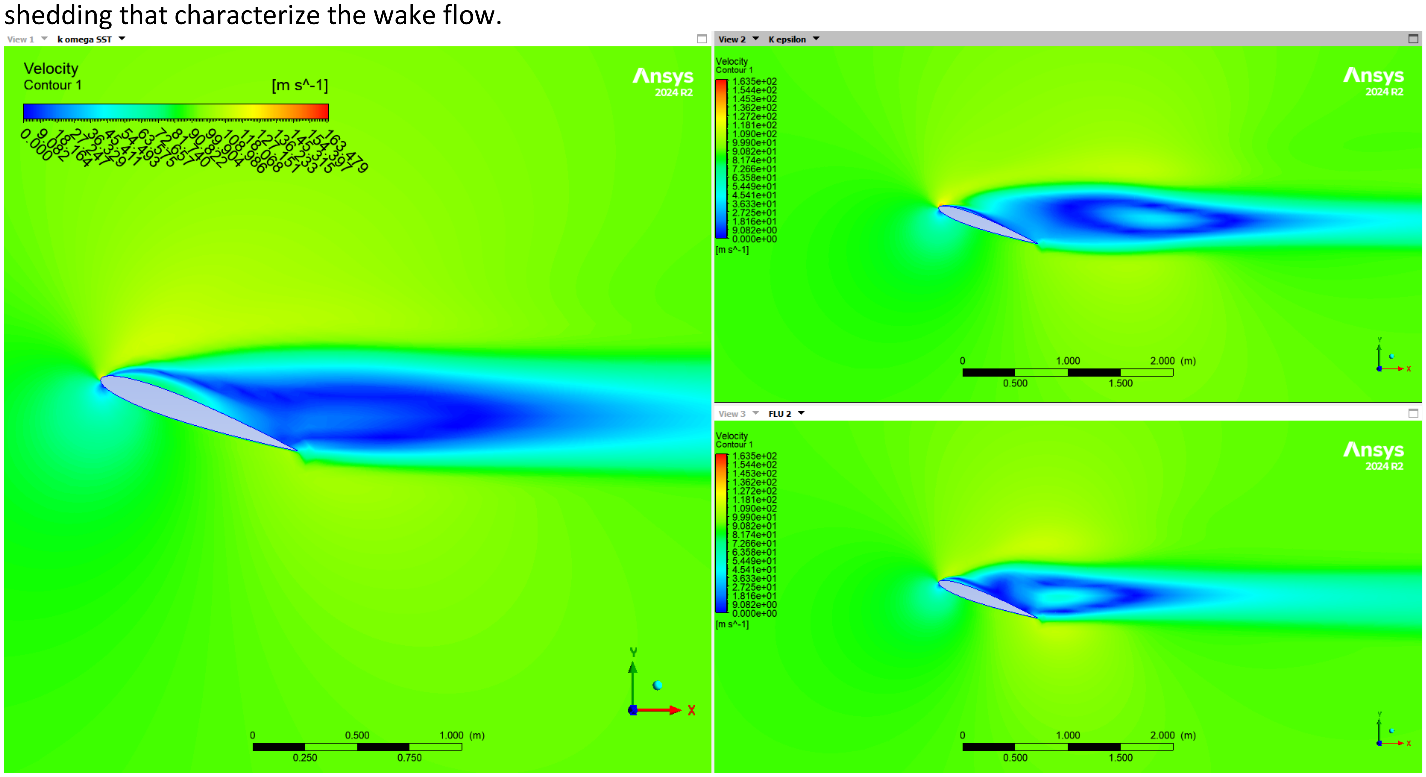


*Figure 2: The figure shows the velocity contours of a) $k-\omega$ SST ( left), b) $k-\epsilon$ (top right), c) GEKO (bottom right)*

Finally, the GEKO (Generalized K Omega) model (bottom right) strikes a balance between the other models. While its boundary layer resolution is not as refined as the k-ω SST or DES models, it provides a reasonable prediction of flow separation. The wake structure is narrower than in the k-ε case but broader than in the k-ω SST and DES simulations. This suggests that GEKO offers moderate accuracy, resolving some of the turbulent features but not to the extent of the more specialized models. The model provides a good compromise between computational cost and accuracy, though it requires proper tuning for the best performance.

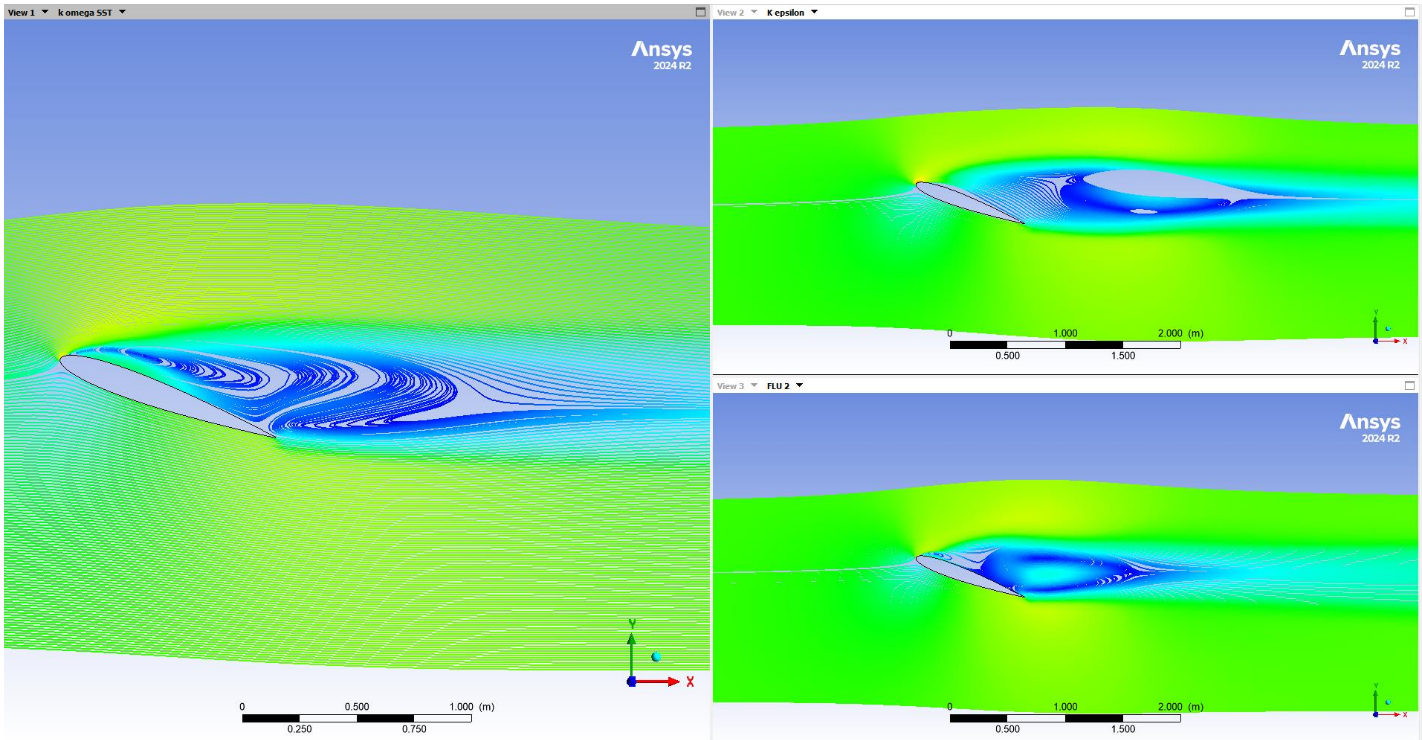


*Figure 3: The figure shows the streamline contours of a) k-ω SST (top left), b) k-ϵ (top right), c) GEKO (bottom right)*

The vortex structures (Figure 3) behind the airfoil differ significantly across the three turbulence models. The **k-ω SST** model provides the most detailed and accurate prediction, capturing well-defined, elongated vortices with clear recirculation zones due to its hybrid formulation, which handles near-wall and free-stream turbulence effectively. In contrast, the **k-ε** model produces broader, more diffuse vortices, as it struggles with flow separation and tends to overpredict turbulence dissipation, leading to less precise vortex resolution. The GEKO model offers intermediate performance, showing moderately detailed vortices and compact recirculation zones, as it balances the robustness of k-ε with the fidelity of k-ω SST. This makes SST ideal for precise aerodynamic studies, k-ε suitable for less demanding applications, and GEKO versatile for scenarios requiring tunable performance.

## 5. Conclusions

The Ansys Fluent simulations provided valuable insights into the aerodynamic behavior of the airfoil under different turbulence models. The velocity and streamline profiles revealed notable differences in how each model predicts flow separation, recirculation, and reattachment. The k-ω SST model demonstrated superior capability in resolving detailed streamline curvature and capturing elongated, well-defined vortices behind the airfoil, reflecting its strength in handling near-wall turbulence and adverse pressure gradients. In contrast, the k-ε model produced more diffuse streamline patterns and broader recirculation zones, indicative of its tendency to overpredict turbulence dissipation and struggle with separation accuracy. The GEKO model offered a balanced prediction, with moderately detailed streamlines and compact vortex structures, though its slight underprediction of lift suggests less effective flow attachment compared to k-ω SST. These differences highlight the critical role of turbulence model selection in accurately resolving flow features, particularly for applications where lift, drag, and flow separation significantly impact performance. Overall, k-ω SST emerges as the most suitable choice for detailed aerodynamic analysis, while GEKO provides flexibility for tunable simulations, and k-ε offers computational efficiency for less demanding scenarios.

**Use and Reproduction**

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

**Document Information**

This case study is part of a set of teaching resources to help introduce students to designing and simulating high-frequency structures.

**Ansys Education Resources**

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

**Feedback**

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

**ANSYS, Inc.**
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com