

Assessment of Turbulence Model Assumptions Through Comparative Analysis of a Typical Turbulent Flow Field

Engr. Imran Hussain

Convener, Mechanical Engineers Committee,
Pakistan Engineers Association, Pakistan
engr_imranmughal@yahoo.com

Abstract—This study assesses the assumptions underlying turbulence modelling through a comparative computational fluid dynamics analysis of a representative turbulent flow field over the RAE2822 airfoil. The analysis begins with preparation of the two-dimensional airfoil geometry in ICEM-CFD, followed by construction of an external aerodynamic domain and generation of an initial mesh with near-wall inflation and wake refinement. A baseline solution is obtained using the standard $k-\epsilon$ turbulence model with standard wall functions. Near-wall resolution is then assessed using y^+ , with the first-layer height and inflation settings adjusted until the values fall within the stated wall-function range of approximately 30–300. A second mesh with approximately twice the cell count is used for a grid-independence assessment by comparing aerodynamic coefficients and pressure distributions. The grid-independent solution is subsequently compared with the provided experimental data, while Mach and Reynolds number similarity are checked by adjusting the CFD setup to the target flow conditions. The reported comparison captures the principal flow features and trends with acceptable agreement, with minor deviations attributed in the source to limitations of the standard $k-\epsilon$ model. The refined mesh requires greater computational time but remains manageable within the stated hardware and license constraints.

Keywords—turbulence modelling; computational fluid dynamics; RAE2822 airfoil; RANS; $k-\epsilon$ model; y^+ ; grid independence

I. INTRODUCTION

Turbulence is a complex, three-dimensional, unsteady, and inherently chaotic fluid motion that continues to pose significant challenges in engineering analysis and numerical modelling. Although the Navier–Stokes equations fully describe the physics of turbulent flows, directly resolving all turbulent scales through Direct Numerical Simulation (DNS) is computationally prohibitive for most practical engineering applications. As a result, a range of turbulence modelling approaches has been developed to approximate the effects of turbulence while maintaining manageable computational cost. [1]

Reynolds-Averaged Navier–Stokes (RANS) models, including the widely used $k-\epsilon$, $k-\omega$, and hybrid or non-linear models, rely on closure assumptions and empirical coefficients to represent the Reynolds stresses. While efficient, these models incorporate simplifications that may limit their accuracy in flows with separation, strong adverse pressure gradients, curvature, or anisotropy. More advanced models such as Reynolds Stress Models (RSM) attempt to relax some of these assumptions, whereas scale-resolving techniques like Large Eddy Simulation (LES) partially resolve turbulent structures and rely less heavily on empirical closures. However, these improved predictions come at significantly higher computational cost. [2], [3]

The purpose of this coursework is to assess and quantify the validity of key assumptions embedded within different turbulence models by comparing their predictions for a representative turbulent flow field. By evaluating the behavior of multiple models under

identical numerical and physical conditions, it becomes possible to identify the limitations, strengths, and applicability of each modelling approach. This comparison also highlights the influence of modelling assumptions—such as isotropy of turbulence, eddy-viscosity hypothesis, near-wall treatment, and empirical constants—on the accuracy of predicted quantities such as velocity profiles, pressure distributions, turbulence kinetic energy, and wall shear stress.

Ultimately, this study aims to develop a deeper understanding of how turbulence modelling choices influence CFD results, and to establish guidelines for selecting appropriate models for complex engineering applications.

II. METHODOLOGY

The methodology for this study involves numerically analyzing a representative turbulent flow field using multiple turbulence models under the same physical and computational conditions to ensure a consistent and meaningful comparison. The workflow begins with defining the geometry and flow configuration, followed by generating a high-quality mesh that adequately resolves regions of high velocity gradients, particularly near the walls. A mesh independence check is performed to minimize numerical uncertainty. The governing equations for mass and momentum are solved using the Reynolds-Averaged Navier–Stokes (RANS) framework, while different turbulence models—such as the standard $k-\epsilon$, realizable $k-\epsilon$, $k-\omega$ SST, and any others selected in Part-1—are implemented sequentially to simulate the same flow case. Appropriate boundary conditions, including inlet velocity or mass flow rate, outlet static

pressure, and no-slip walls, are applied consistently across all simulations. Near-wall resolution is maintained through appropriate wall treatment based on the requirements of each model, such as standard wall functions for $k-\epsilon$ or low-Reynolds behavior for $k-\omega$ SST. The numerical solution is obtained using a pressure-based solver with second-order discretization schemes to minimize numerical diffusion and improve solution accuracy. Convergence is monitored through residuals and stability of key flow variables such as velocity, pressure, turbulence kinetic energy, and wall shear stress. Once converged, the results from each turbulence model are extracted and compared against theoretical predictions, empirical correlations, or available benchmark data. Performance metrics include mean velocity profiles, turbulence kinetic energy distribution, pressure coefficients, and skin-friction coefficients. This systematic approach allows the influence of turbulence modelling assumptions—such as isotropic eddy viscosity, empirical constants, and treatment of near-wall turbulence to be quantified and critically evaluated. [4]

A. GENERATION OF A VALID HIGH-RE BASELINE SOLUTION

B. GEOMETRY IMPORT AND PREPARATION

I began the analysis by importing the 2D airfoil geometry into ICEM-CFD using the *formatted point data* option. For each data-point file, I untoggled the *import surfaces* option and allowed ICEM-CFD to automatically generate curves through the point sets. After the first file was imported, I loaded the second file and selected *merge* so that both datasets formed a single continuous airfoil curve. Once imported, I inspected the geometry to ensure that the merged curves were smooth and free from gaps or inconsistencies before proceeding to meshing.

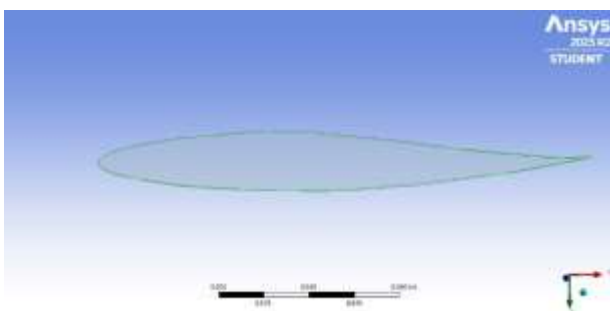


Figure 1 Importing curve for RAE2822

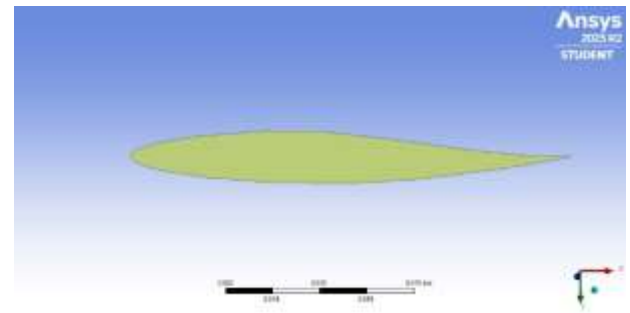


Figure 2 Making 2D surface of airfoil

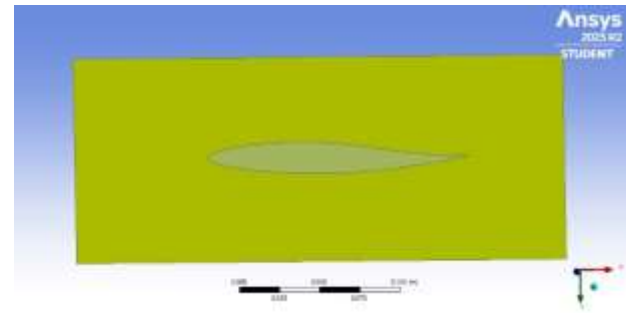


Figure 3 Setting ideal gas domain

C. INITIAL MESH GENERATION AND BASELINE SOLUTION

After preparing the geometry, I created the computational domain around the airfoil and generated an initial mesh suitable for external aerodynamic flow. Inflation layers were applied near the airfoil surface, and the wake region was refined to capture important flow features. I then applied the required boundary conditions, including far-field pressure conditions for the compressible flow, and performed the first simulation using the standard $k-\epsilon$ turbulence model with standard wall functions. The solver was run until convergence was achieved, giving me the baseline solution for the assignment.

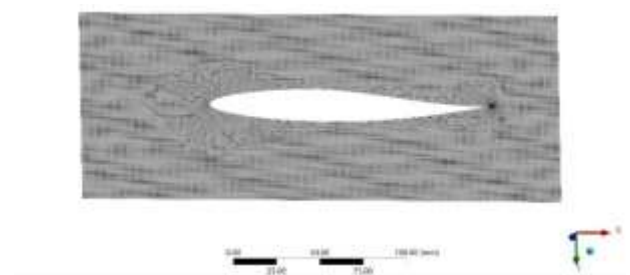


Figure 4 Initial mesh

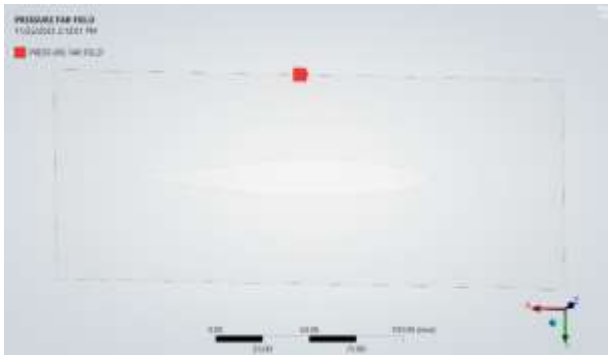


Figure 5 Pressure far field

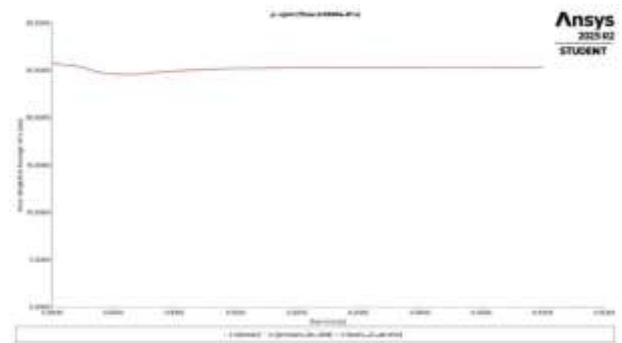


Figure 7 Y+ for initial mesh

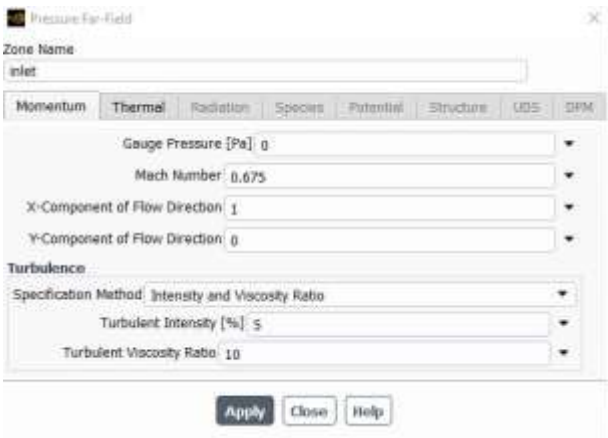


Figure 6 Model and wall function

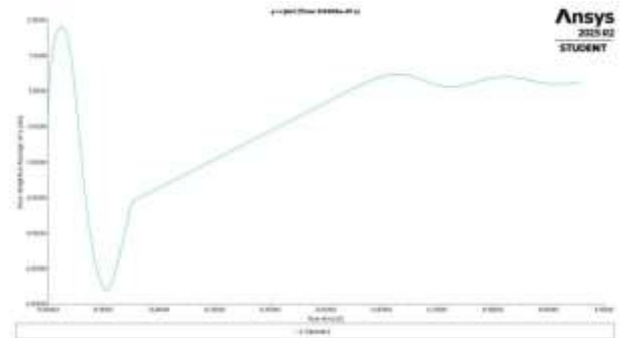


Figure 8 Y+ for initial mesh

E. GRID INDEPENDENCE STUDY

To assess grid independence, I generated a second mesh with approximately twice the cell count of the initial mesh. I ran simulations on both the coarse and refined meshes using identical solver settings and compared the results, focusing on aerodynamic coefficients and pressure distributions. The differences between the two meshes were small, indicating that the refined mesh provided sufficiently grid-independent results. Since the refined mesh met the criteria, no further mesh refinement was required.

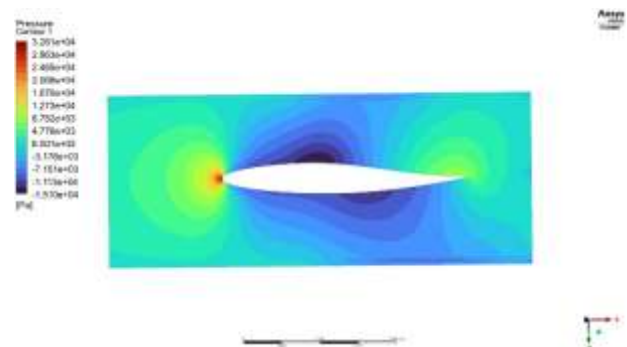


Figure 9 Pressure across airfoil for initial mesh

D. ASSESSMENT AND ADJUSTMENT OF NEAR-WALL RESOLUTION (y^+)

Once the baseline solution was obtained, I evaluated the viscous-scaled near-wall spacing, y^+ , along the airfoil surface. The initial mesh did not fully satisfy the log-law requirements, so I adjusted the first-layer height and the inflation-layer settings. After re-meshing, I repeated the simulation and re-checked the y^+ distribution. This iterative process continued until the y^+ values consistently fell within the appropriate range for wall-function application (typically between 30 and 300).

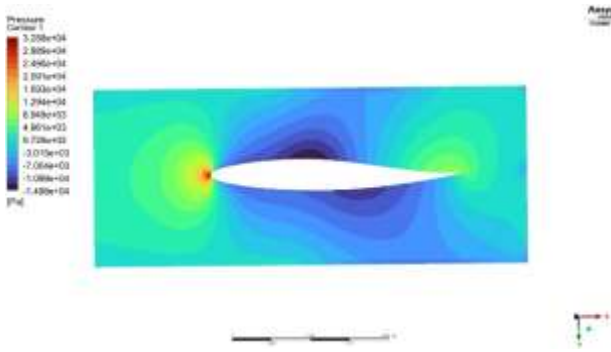


Figure 10 Pressure across airfoil for refined mesh

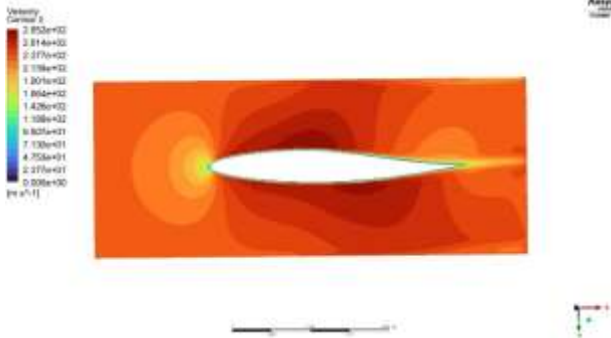


Figure 11 Velocity across airfoil for refined mesh

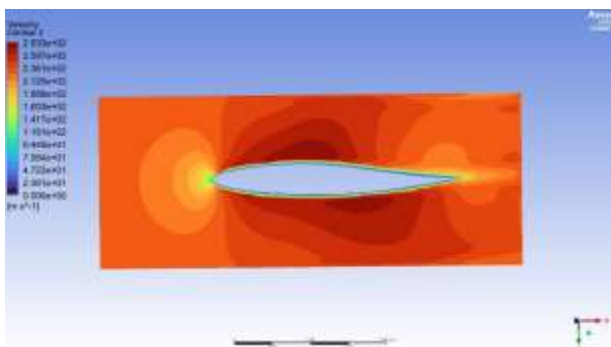


Figure 12 Velocity across airfoil for refined mesh

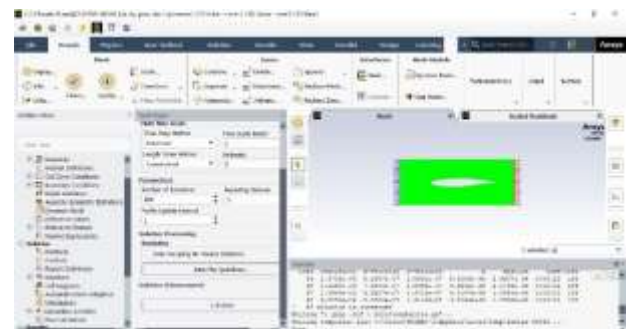
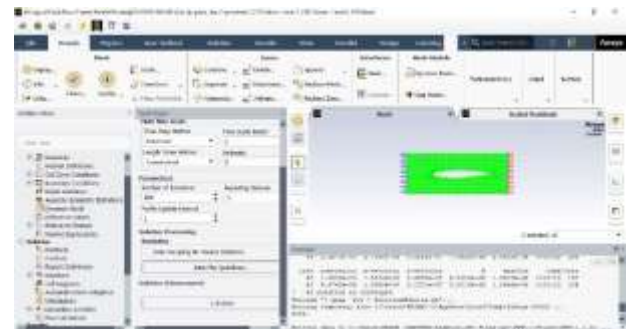
F. COMPARISON WITH EXPERIMENTAL DATA

After obtaining the grid-independent solution, I compared the simulation results with the experimental data provided. The comparison included pressure coefficient distributions and overall aerodynamic behavior. The CFD predictions showed acceptable agreement with the experimental measurements, capturing the main flow features and trends with reasonable accuracy. Minor deviations were noted, but they were consistent with the expected limitations of the standard $k-\epsilon$ model. [5]

G. CONVERGENCE TIME AND SOLVER PERFORMANCE

During the simulations, I monitored the solver performance and recorded the time required for each case to converge. The baseline and refined meshes converged

within reasonable iteration count, and no major stability issues were encountered. The refined mesh required a longer computational time, as expected, but remained manageable within the hardware and license constraints.



H. MATCHING MACH AND REYNOLDS NUMBERS

To align the CFD setup with the experimental flow conditions, I ensured that both the Mach number and Reynolds number matched the target values. I assigned sensible far-field values for static temperature, static pressure, and Mach number, from which the free-stream speed of sound, density, and velocity were calculated. The Reynolds number was then computed using the airfoil chord length. To achieve the correct Reynolds number, I iteratively adjusted the dynamic viscosity in the materials panel until it matched the experimental case. The final setup satisfied both Mach and Reynolds similarity requirements



Figure 13 Reynolds number for initial mesh

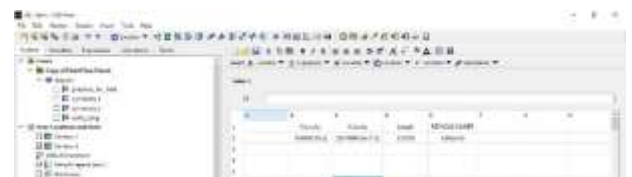


Figure 14 Reynolds number for refined mesh

III. CONCLUSION

This study evaluated the influence and practical implications of turbulence-model assumptions for a representative transonic RAE2822 airfoil case using computational fluid dynamics. The workflow established a valid high-Re baseline solution through geometry preparation, construction of the external computational domain, near-wall inflation, far-field boundary conditions, and the standard $k-\epsilon$ model with standard wall functions. The y^+ assessment required adjustment of the first-layer height and inflation settings until the reported values were within the stated range of 30–300. A second mesh with approximately twice the cell count produced only small differences in aerodynamic coefficients and pressure distributions, supporting the reported grid-independent solution. Comparison with the provided experimental data captured the principal pressure and aerodynamic trends with acceptable agreement, while minor deviations were consistent with the stated limitations of the standard $k-\epsilon$ model. The Mach and Reynolds number matching procedure further aligned the CFD setup with the target experimental conditions. Overall, the study demonstrates the importance of mesh quality, near-wall resolution, similarity conditions, and experimental comparison when assessing turbulence-modelling assumptions in transonic CFD. The refined mesh increased computational time but remained manageable within the stated hardware and license constraints.

REFERENCES

- [1] S. B. Pope, *Turbulent Flows*, Cambridge University Press, Cambridge, U.K., 2000.
- [2] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994, doi: 10.2514/3.12149.
- [3] D. C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, La Cañada, CA, USA, 1998.
- [4] ANSYS, Inc., *Ansys Fluent Theory Guide*, Release 2025 R1, Canonsburg, PA, USA, 2025.
- [5] P. H. Cook, M. A. McDonald, and M. C. P. Firmin, "Aerofoil RAE 2822—Pressure Distributions, and Boundary Layer and Wake Measurements," *Experimental Data Base for Computer Program Assessment*, AGARD Report AR 138, 1979.

BIOGRAPHIES (Optional not mandatory)



Engr. Imran Hussain