White Paper

Advanced Turbulence Modeling Methods Provide Accurate, Efficient Results in Any Fluid Flow Application

By Gilles Eggenspieler, Ph.D.
Senior Product Manager, ANSYS, Inc.

Turbulence plays an important role in the vast majority of industrial fluid flow applications. It constitutes a classic multiscale problem in which turbulent flow structures of many different scales interact with each other. Accurate prediction of a system's aerodynamics, heat transfer characteristics, mixing performance and other factors is key to determining performance with high precision — so accurate and robust turbulent modeling capabilities are critical in computational fluid dynamics (CFD). Resolving all turbulent flow scales present in industrial applications via simulation is not possible with today's computational resources, but formulations can be used that reduce a problem's complexity while still delivering accurate information on flow turbulence and its effect on product performance.

Because turbulence is a very complex phenomenon, no single do-it-all formulation has been found to date. There are so many turbulence models in existence that the challenge for CFD software developers is to incorporate the right subset of models, resulting in a package that is robust, accurate and validated — and that covers applications that users need. Leaders in the field don't stop there; they offer best practices so a user knows which model to use for a specific turbulence problem. They also advance physics by innovating, testing and validating new hybrid and transition models that deliver the best mix of accuracy and computational intensity for today's challenging applications.

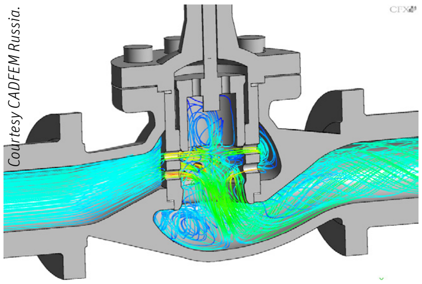
Turbulence Simulation Challenges

Most engineering flows are turbulent, and these flows are inherently multiscale, three-dimensional and unsteady. Fluid flow applications involving complicated geometries and complex physics present the greatest challenges from a turbulence-modeling standpoint, particularly in the industries listed below.

Aerospace

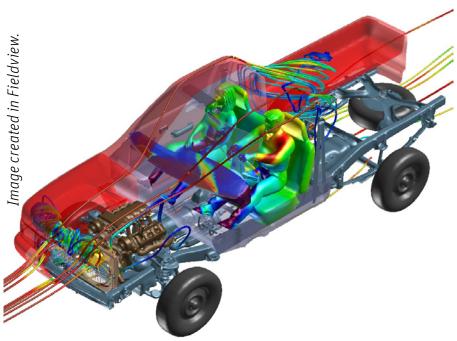
Accurate turbulence modeling is critical to major aerospace challenges such as optimizing lift/drag ratio of a wing to ensure that the plane will fly safely while consuming as little fuel as possible. Turbulence modeling is involved in many other aerospace design problems, such as designing the engine to minimize both external and cabin noise.

Courtesy CADFEM Russia.

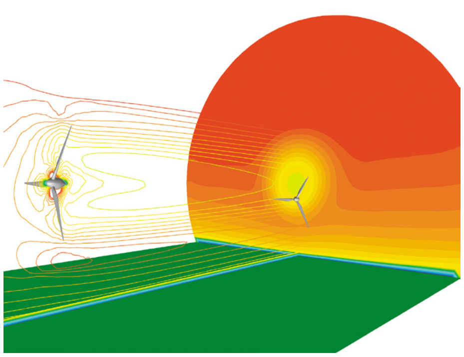


Accurately predicting the performance of a complex valve system requires turbulence models that can capture flow characteristics in both the large main pipe and the small openings. Wall effects, which generate a large amount of turbulence, need to be captured as well.

Image created in Fieldview.



Car manufacturers rely on accurate computational fluid dynamics of turbulent flows to compute the cooling and heating of a passenger car.



Only an accurate turbulence model can predict velocity contours behind a wind turbine. This is critical when studying wake-effect impact on a second down-wind turbine.

Gas and Oil Pipelines

Turbulence dissipates flow energy, so turbulence modeling is critical for companies involved in moving fluids over long distances. For example, it is important to accurately predict energy loss to determine the optimal distance between pumping stations. If this distance is too large, the flow moves too slowly or even stops flowing. If the distance is too small, the pumping system is overdesigned and underoptimized, leading to higher-than-necessary pump installation, operation and maintenance costs. The ability to accurately predict flow behavior and pressure drop induced by turbulence helps companies to design the most efficient and cost-effective oil and gas transport systems.

Automotive Aerodynamics

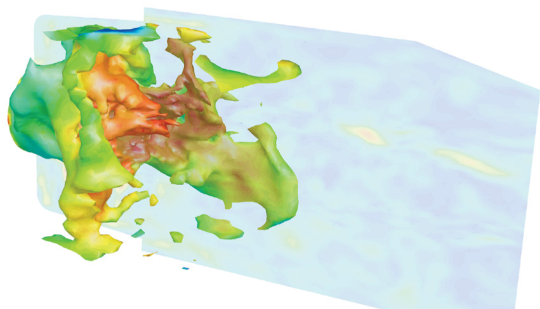
Automotive aerodynamics is all about trade-offs, particularly striking the right balance between body style needs and aerodynamic concerns. Predicting the drag of alternative designs is the key to delivering an appealing design that is as fuel efficient as possible. Turbulence dictates a design's aerodynamic performance, so accurate prediction of a design's drag requires accurate, tested and validated turbulence models.

Mixers in Chemical and Pharmaceutical Industries

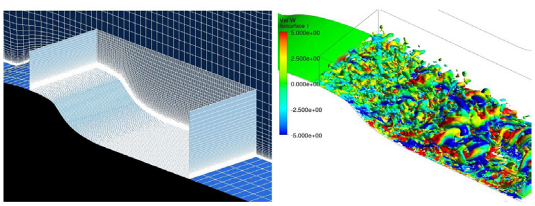
Mixing is a crucial step in most chemical and pharmaceutical industry processes. The throughput of many process operations depends on achieving homogeneous mixing in as short a time as possible. Designers of mixing equipment utilize CFD to evaluate alternative agitator and tank designs and operating conditions to determine optimal configuration. Turbulence plays a critical role in mixing, so accurate turbulence modeling is essential to getting the design right the first time.

Siting Turbines on Wind Farms

For power companies that operate wind turbines, the goal is to generate the maximum amount of power at the lowest cost from a given wind farm site. The challenge is that the wake of a turbine has significant effects — including reduced power output and shorter turbine life — for units located downwind. Wind farm developers are demanding more accurate methods for calculating these wakes as well as any terrain effects that impact a turbine's performance. By accurately predicting these phenomena, power companies can optimally place each wind turbine to produce maximum energy from a given parcel of land.



For some turbulence applications, LES models are needed for accurate results. In this combustion chamber simulation, accurate simulation of the flame location, heat release and pollutant emissions requires resolution of flame-front wrinkling by the larger turbulent scales. This is possible using the LES turbulence model.



ANSYS offers a wider range of hybrid models than other CFD software providers. Software development experts at ANSYS recently progressed the state of the art with two advanced hybrid models: wall-modeled LES (WMLES) and embedded LES (ELES) turbulence models. WMLES allows the simulation of wall-bounded flows without the excessive computational costs associated with classical LES modeling. The ELES model offers even more flexibility by allowing an embedded LES zone within a larger RANS-simulated steady-state domain. This results in accurate simulation without excessive computational expense.

This image shows (left) the grid for NASA hump simulation and (right) turbulent structures in LES domain of embeded-LES simulation.

Turbulence Models for Complex Flows

Turbulence modeling makes it possible to account for turbulence effects using CFD at a reasonable computational cost. No single model or modeling approach can cover all types of turbulent flow, so different types of turbulence models have been developed in the past decades. For combustion, acoustics and other similar applications, some of the turbulent structure needs to be resolved to ensure accurate results.

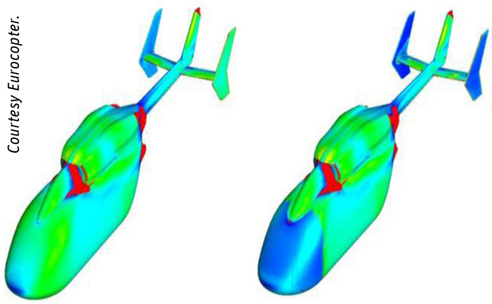
Steady-State Models

Steady-state or Reynolds-averaged Navier–Stokes (RANS) models reduce the complexity of modeling turbulent flows by averaging the velocity field, pressure, density and temperature over time. The steady-state approach calculates mean flow quantities, and no attempt is made to resolve turbulent structures in time and space. Therefore, RANS methods are computationally frugal. If selected and applied properly, the models offer engineers a highly attractive solution to predict the effect of turbulence without having to explicitly capture all scales involved in turbulent flows. RANS techniques are very accurate for the vast majority of applications. However, for some applications, more advanced models that resolve some of the turbulence scales are required.

Large-Eddy Simulation and Hybrid Models

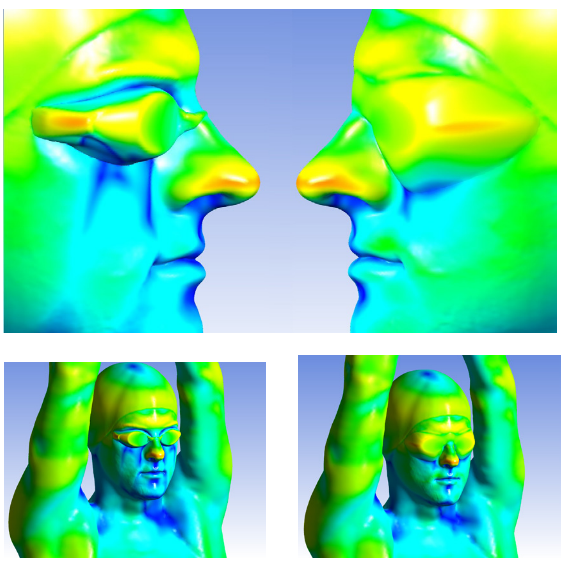
Large-eddy simulation (LES) turbulence models resolve the large turbulent structure in both time and space and simulate only the influence of the smallest, nonresolved turbulence structures. These models are many orders of magnitude more computationally expensive than the RANS approach, especially for complex industrial applications. However, actual simulation time can be greatly reduced using high-performance computing resources.

As turbulent structures become very small in the near-wall region, the associated computational resources needed would make it too expensive and time consuming to use in the product development process. This poses severe challenges for LES simulations of applications in which wall effects impact product/design performance. The solution to this problem is hybrid models, such as detached-eddy simulation (DES) and scale-adaptive simulation (SAS) models that combine steady-state and LES treatments for the model's wall boundary layer and free shear portions, respectively.



Courtesy Eurocopter.

Drag coefficient prediction on a helicopter. Results (left) using a fully turbulent RANS model overpredict actual drag of the helicopter body; (right) using a transition turbulent model better predicts actual drag.



In analyzing the near-surface flow velocity fields of existing goggle designs (left) and a next-generation design concept (right), Speedo engineers were able to see an immediate improvement in hydrodynamic performance.

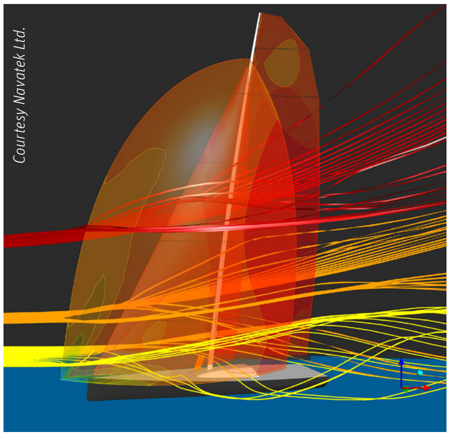
Transition Models

The flow around turbine blades, wings and many other applications often features upstream laminar boundary layers that transition into a turbulent flow further downstream. Capturing this phenomenon is key to predicting wing lift or compressor performance, for example. To aid in analyzing this widely observed physical behavior, a new class of models predict behavior of a flow over a surface that starts in the laminar regime and transitions to the turbulent regime. Turbulence transition models significantly expand the range of applications for CFD software.

Case Study: Speedo

In Olympic swimming competitions, the difference between gold and silver medals can be mere milliseconds, so tiny improvements to multiple details can be critical. Legendary swimsuit manufacturer Speedo is using CFD to remove swimwear imperfections that increase drag while the athlete is swimming. Speedo Aqualab engineers studied three key elements of the gear: swimsuit, goggles and cap. The combination of flow around these elements and the athlete's body features creates an extremely complex and turbulent environment. Hence, very accurate turbulence models are needed to predict which design will create the least resistance in the water and which design will give a competitive edge to the swimmer, even before the race starts.

For many years, Speedo Aqualab has relied on ANSYS Fluent to study water flow in designing such high-performance swimgear. For the most recent world-class races, the engineering team focused on improving the hydrodynamics of goggles, cap and suit. The innovative suit precisely fits the athlete's body and reduces drag. The cap reduces significant flow-field disruptions leading to improved performance. The goggle design blends with the athlete's facial features and results in minimum flow disruptions. Performance gains were real: for example, the goggle's CFD simulation-led design resulted in a performance improvement of 2 percent. When added to improvements that come from the suit and cap as well, this innovative swimwear system can enable an athlete to reduce race time by milliseconds, enough to turn a silver medal into gold. According to Dr. Tom Waller from Speedo Aqualab, "Engineering simulation has been absolutely critical in launching this world-first concept."



Accurate turbulence modeling is critical to designing high-performance racing sailboats.

Conclusion

Turbulence is inherent to most flow problems and is usually the major limiting factor in accurate simulation. No single model or modeling approach can cover all types of turbulent flow, so different types of turbulence models have been developed in the past decades. As CFD applications become more complex, more sophisticated turbulence models are needed. Choosing the right turbulence model to match the application is critical to accuracy and computational resource optimization. ANSYS is a technology leader in this area, offering a wide range of the most advanced model formulations, including WMLES, ELES and transition models. ANSYS turbulence models are tested and validated by the CFD industry's largest research team along with its largest community of users. ANSYS technical support teams fully understand each of its available turbulence models and can help users to select the right model and apply it correctly to achieve optimal results without taxing available computing resources.

For in-depth details about turbulence modeling

Download Florian Menter's technical paper
Turbulence Modeling for Engineering Flow
at ansys.com/tmfe

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.

724.746.3304
ansysinfo@ansys.com