Modeling Turbulence in FloWizard

By André Bakker, Fluent Inc.

FloWizard has a unique, wizard-based interface. Its innovative GUI is designed to make it easy to set up and run CFD problems, and to shield users from some of the more difficult modeling decisions by making choices based on other user inputs. One area where this happens is in modeling turbulent flow. FLUENT offers many different turbulence models, each with its own specific sub-models, settings, and constants. FloWizard greatly simplifies the choices that the user has to make for setting up a turbulent flow. A Flow Type Guide panel is available that clearly explains the available options, which are laminar, turbulent, turbulent with strong swirl, and unknown. The objectives are to keep the user from having to make a decision based on esoteric turbulence modeling issues, and to ensure the best possible solution.

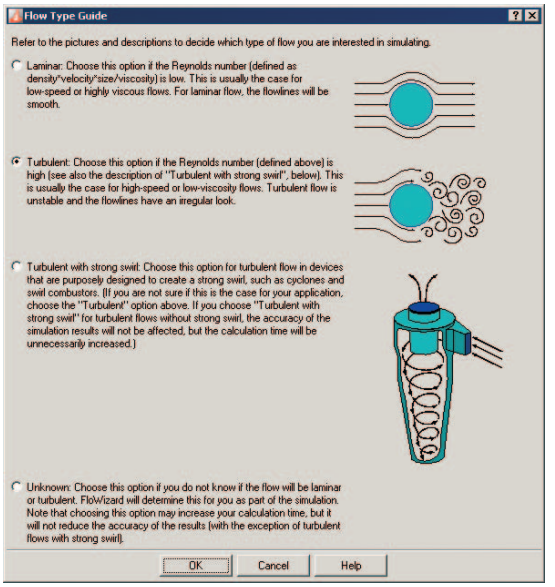
When the user chooses "turbulent", FloWizard enables the realizable k-ε turbulence model, which provides an excellent balance between computational speed and solution quality. For applications with strong swirl, FloWizard activates the Reynolds stress model, the only available steady-state turbulence model that gives correct predictions for such flows. To make this choice, it asks if the equipment being modeled is purposely designed to create strong swirl, as would be the case for a cyclone or a swirl combustor.

When the user chooses "unknown" in the Flow Guide panel, FloWizard will choose a turbulent or laminar flow model based on the inlet Reynolds numbers. For pressure driven flows, where this cannot be calculated in advance, it will periodically recalculate the inlet Reynolds number during convergence, and switch between laminar and turbulent flow as needed.

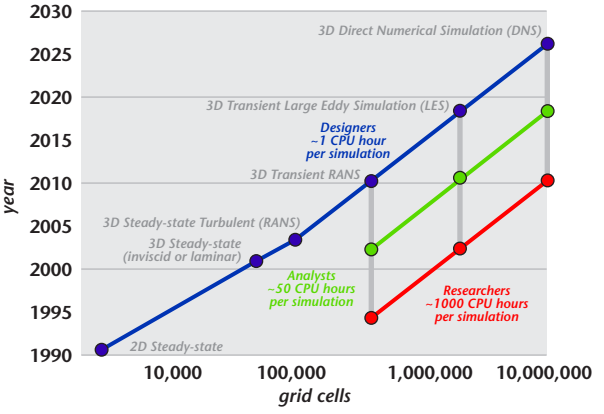
All turbulence models in FloWizard are based on the steady-state, Reynolds averaged Navier-Stokes (RANS) equations because of the much longer computation times associated with transient models, such as large eddy simulation (LES).

In design, timeframes are much shorter than in engineering analysis and scientific research. Researchers may accept CPU times on the order of thousands of hours, employing either large computing systems or patient students. Many detailed CFD analyses at commercial companies will be limited to approximately 50 hours, which corresponds to an overnight run on a typical four CPU cluster. By contrast, design projects often require a review of multiple design permutations in a single day, and are therefore restricted to computing times of an hour or less for each run.

Based on these different requirements, and on the historical progression in computer speed, a forecast can be created for the turbulence modeling methods that will most likely be favored in different types of CFD work in the years to come. While three-dimensional, steady-state RANS models are usable for the typical design project today, it will take several years before transient RANS and LES methodologies will be widely used. In the meantime, the more computationally intensive turbulence models will be used primarily in analysis and research to perform tasks such as validation and process optimization. One day, however, these techniques will become commonplace for engineering design projects as well. ■



The Flow Type Guide panel in FloWizard illustrates different flow regimes



The predicted evolution of typical grid cell counts and turbulence modeling approaches, based on the approximate average time spent per simulation, shown for design engineers, analysts, and researchers