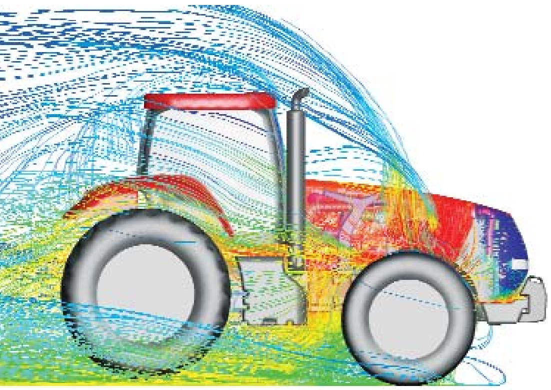
Tractor Cooling Simulation Cuts Prototyping Expenses

By Panos Tamamidis, *Case New Holland*, Burr Ridge, Illinois, USA

Large tractors move slowly over the ground and require extra effort to operate implements such as plows, bucket loaders, mowers, and planters. Because the power demands are high and the tractors cannot rely on high velocity travel to contribute to their cooling needs, specialized cooling systems are needed. At Case New Holland (CNH), where a wide range of agricultural tractors are manufactured, CFD is being used to help design the entire tractor cooling package, saving the company money and reducing fuel costs for the end user.

When CNH introduces a new tractor or substantially upgrades an existing one, it is often necessary to redesign the cooling system. Large tractors require a complex cooling system with separate modules to cool the five different fluid systems in the engine. The modules each contain a heat exchanger. Additional cooling is provided by the lone engine fan.

The trend nowadays is towards larger engines that present greater cooling challenges and take up more space, which means that the space available for the cooling package is



Airflow pathlines emitted from the hood grille

reduced. Increasing the efficiency of the cooling package is also important because the fan can consume a lot of horsepower. This reduces the amount of power available to drive the tractor and implements, and impacts the fuel efficiency as well. Thus the challenge is not so much to design any one of these systems but rather to get them all working together in a given space while consuming a minimum amount of power.

Normally, a single engine fan delivers air to the heat exchangers, driving each of the cooling modules. The fan needs to deliver enough air to each module to meet its cooling requirements while drawing the least possible amount of air away from the engine.

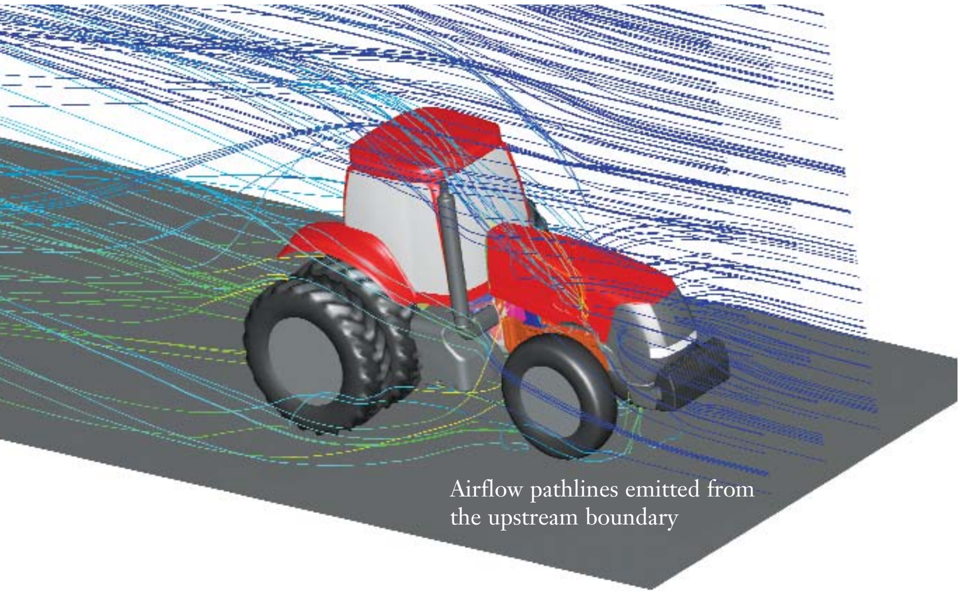
The design process required to meet these requirements may necessitate moving the different modules around inside the engine compartment, first to provide enough air for each module and second to fit them all within the limited space. Within each module it is necessary to trade off increasing the amount of available air against increasing the efficiency of the heat exchanger, which may require more space and increase its cost.

In the past, CNH engineers performed engineering calculations that made it possible to estimate the total airflow provided by the fan and the airflow required by each module. But these calculations did not take the geometry of the underhood compartment into account so they had no way of accurately determining how the fan's airflow was distributed among the different modules. This made it necessary to build and test several prototypes on a typical project to find a design that met the cooling and space requirements. But even a large number of design iterations was not nearly enough to optimize the design, particularly in terms of the amount of power consumed by the fan.

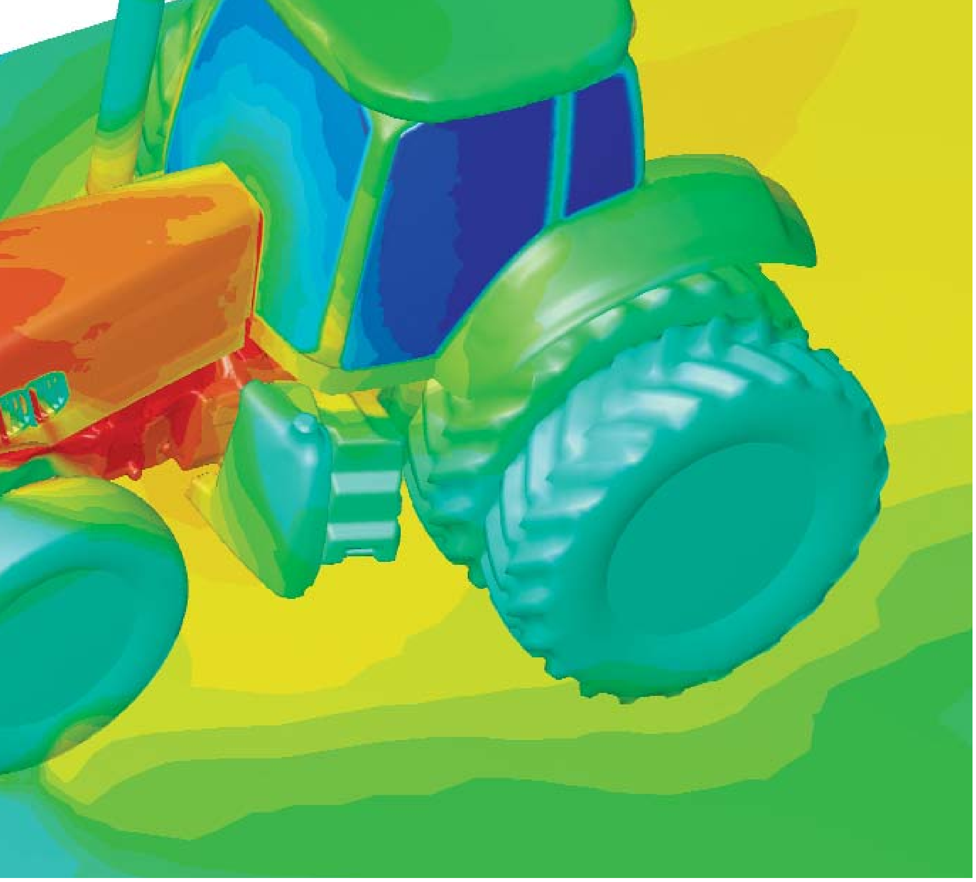
Temperature contours on external surfaces

The engineers now use CFD to simulate the flow of air from the fan, through the engine compartment, and through the heat exchangers used in each of the modules. CFD has the ability to accurately simulate extremely complicated cooling packages while keeping the computational requirements at reasonable levels.

For a typical simulation, they import the computer aided design (CAD) geometry of the engine, heat exchangers, fans, and other components. Performance data for each of the heat exchangers is taken from the respective manufacturers, and the dimensions and properties and the engine compartment sheet metal are



Airflow pathlines emitted from the upstream boundary



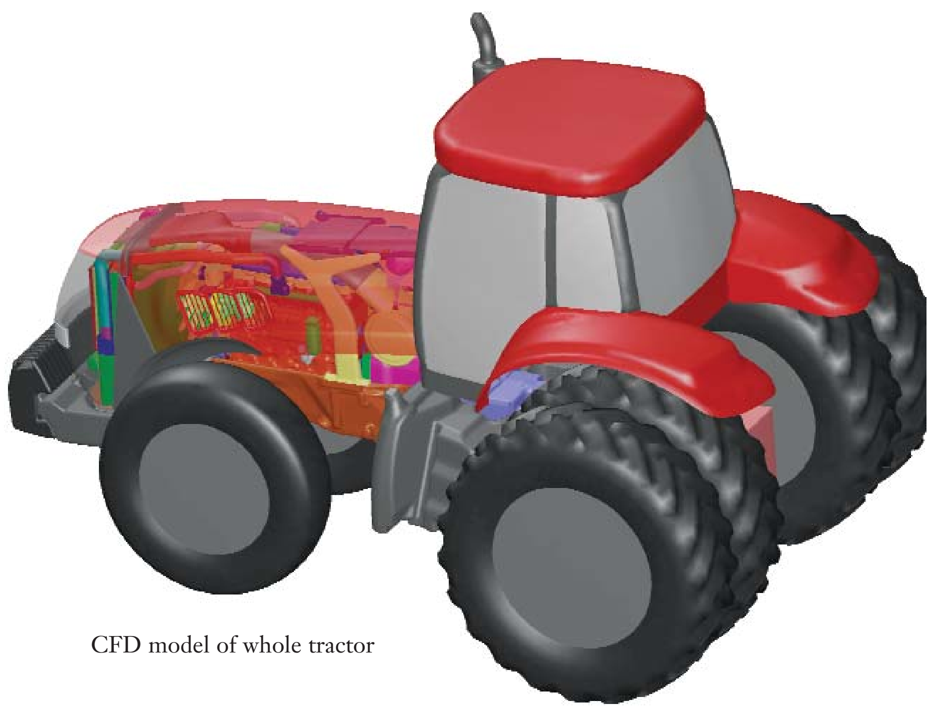
taken from CNH specification sheets.

For a recent project, a simulation of an initial design showed that oil temperatures in one of the modules were too high. Engineers addressed this problem by changing the cooling package configuration and evaluating different oil coolers. The best configuration they found put the charge-air cooler for the intake manifold and the radiator in parallel. This approach reduced the oil temperature to acceptable levels and made it possible to reduce the size of the radiator and charge-air cooler, which helped meet the tight space requirements inside the engine.

Engineers then used CFD to optimize the design of the fan. They simulated several different blade designs looking for a design that would reduce the amount of power drawn. They made further adjustments in the position of the different modules. They added louvers in the engine compartment in order to allow air to move freely and to reduce the amount of work performed by the fan.

Through this iterative process, engineers were able to complete a simulation of each new design quickly, making it possible to evaluate far more design alternatives than would have been possible if they had used the traditional build and test approach.

The result was that CNH engineers were able to significantly reduce the amount of power drawn by the fan compared to the initial design while meeting all cooling module temperature and packaging requirements. Coolant temperatures were also lower than in the previous design. A prototype of the best design was built, and measurements were in very good agreement with the CFD simulations. Overall, the use of CFD in the design process made it possible to bring the product to market substantially faster than the previous generation of products, cutting the costs of building and testing prototypes. ■



CFD model of whole tractor