PLAYING IT Cool

CFD SIMULATION OF DRIVE UNIT COOLING HELPS TO IMPROVE RELIABILITY.

By Tadashi Yamada, Drivetrain Unit Engineering Design Division
Toyota Motor Corporation, Toyota, Japan

The cooling performance of oil is critical to the functionality and durability of vehicle drive units, such as transmissions and differentials. Traditionally, automotive R&D teams evaluate cooling performance by building prototypes, installing them in a vehicle, and conducting tests in wind tunnels. The use of free-surface multiphase flow modeling with high-performance computing (HPC) has made it possible to accurately predict oil cooling performance via readily available computing resources. Automotive leader Toyota uses this approach to evaluate more design alternatives in the early stages of the product development process.

A typical drive unit consists of a case containing rotating internal parts, such as gears and shafts supported by bearings that transmit power. These are surrounded by oil and air. The oil serves various functions, including lubrication, power transmission and cooling. The major heat sources within the drive unit include meshing of the gears, sliding friction between bearings and shafts, and stirring oil as a result of gear movement. The heat generated is conveyed to and through the oil to the internal surface of the case; from there, it goes to the case's external surface and surrounding air. Oil flow patterns within the transmission are critical to efficient lubrication, power transmission and cooling performance, and to avoid negative effects, such as churning loss, in which friction between the oil and gears revolving at high rpm can consume several horsepower.

Simulating the cooling capacity of a drive unit requires predicting internal oil flow patterns involving free surfaces, external air flows, and the complex three-dimensional flows of heat from the oil to the air. A key difficulty is that external air flows can be resolved with sufficient accuracy only by modeling the entire vehicle,

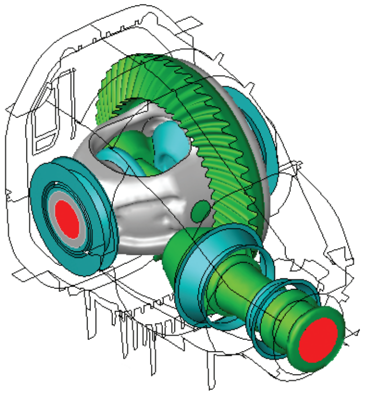
This approach allows Toyota to evaluate more design alternatives in the early stages of the product development process.

whereas the internal temperature distributions must be analyzed for about an hour until a saturation temperature is reached. Until recently, this combination of complex physics, large spatial scale and long duration required such extensive computing resources that simulation could not be completed within a time frame that would positively affect the design process.

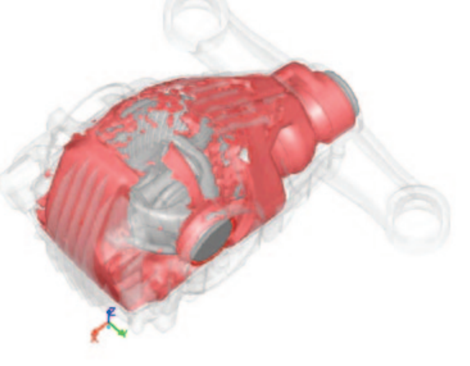
Advancements in physical models and HPC performance in the latest generation of CFD software spurred Toyota engineers to initiate new efforts to simulate oil cooling performance in drive units. A strong coupling approach in which the entire computational domain is formulated is computationally very expensive. For this reason, the internal oil flows and vehicle airflows were simulated separately but coupled to exchange data to determine the entire system's behavior.

The internal oil flows were solved to obtain the heat transfer coefficients between the internal components and fluids, and between the fluids and the case. The whole-vehicle air flow simulations produced heat transfer coefficients between the case and the external air. Toyota engineers used the results of the internal and whole-vehicle simulations along with the heat generation rates of various components as boundary conditions for a heat calculation model consisting of only solid parts of the unit, with oil temperature as an unknown variable. The model was iterated until it converged on a solution that achieved energy balance to calculate the temperature distribution of the oil and internal components. Toyota used ANSYS Fluent to solve the internal fluid flow.

Within the drive unit, as the rotating parts stir the oil, free surfaces are formed at the interface between the oil and air. Engineers used the volume of fluid (VOF) technique within Fluent to track the surface as an interface through a grid and apply boundary conditions at the interface. They employed an explicit geo-reconstruction scheme to solve the interface behavior, as it provides the most realistic interface between phases. Because of the importance of the gear tooth geometry on the free-surface formation, they modeled the tooth geometry



Simplified differential unit used for correlation with CFD simulation



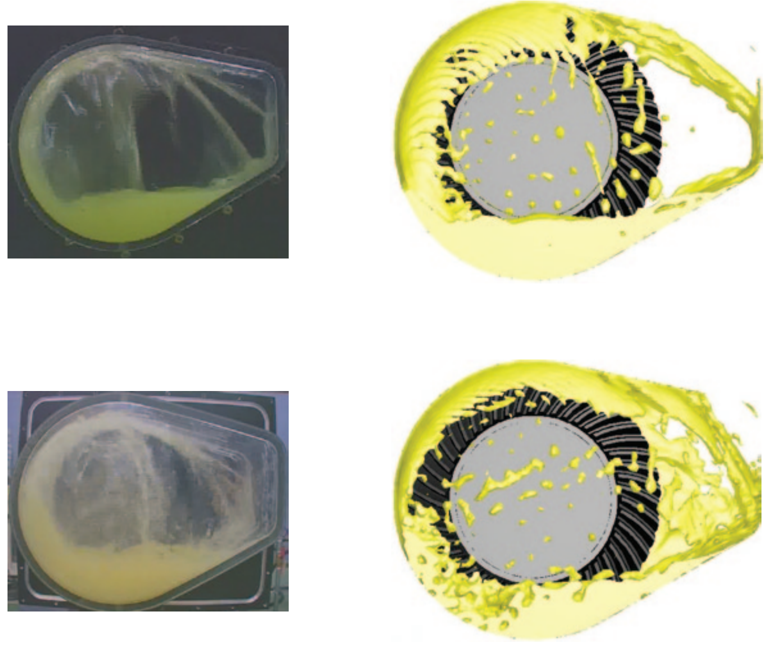
Oil distribution calculated with CFD using VOF model to identify free surfaces as accurately as possible, and the computational grids surrounding the gears were rotated using a moving mesh capability.

Using the von Karman analogy between induced drag on wings and wave drag on bodies, engineers calculated local heat-transfer coefficients from the transient internal oil flow simulation. Then they applied a user-defined function (UDF) to time-average the results. This method was validated by physical testing on a differential drive unit equipped with a single hypoid ring gear. The behavior and flow velocity of the oil stirred by the ring gear were measured and compared to CFD predictions. The behavior of the main flow of oil stirred up by the gear, the amount of oil accumulated on the pan, and the geometry of the free surface correlated well between testing and CFD through multiple revolutions. The team noted relatively minor differences in how droplets and thin films were scattered. Laser Doppler velocimetry (LDV) measurements agreed with the CFD flow velocity predictions for both the magnitude and the direction of flow in the gear stir-up periphery where rapid flow occurred.

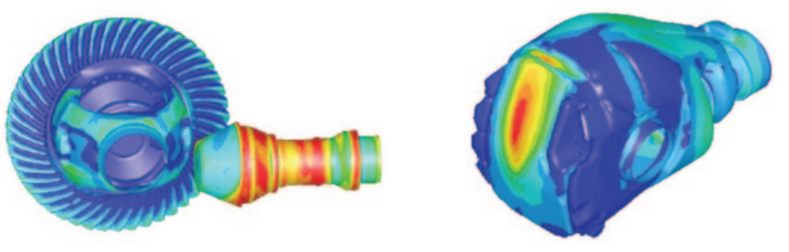
Based on successful correlation of the simplified drive train, engineers developed a model of a real rear independent-suspension differential unit with about 1.3 million cells. The model was solved on an HPC cluster consisting of a network of about 80 personal computers; it was used to determine the oil distribution and local heat-transfer coefficients between the internal parts and the oil, and between the oil and the case. The results again demonstrated good agreement in heat flux distribution and temperature distribution between the actual measurements and simulation, illustrating the value of this simulation technology in the product development process.

Visualization of the results showed that there was sufficient cooling in the bottom of the unit, where the external airflow velocity is high, but insufficient cooling in the upper front section. Based on these results, Toyota modified the simulation model to evaluate the effect of adding cooling fins to areas where oil temperature was high and removing fins from areas where the simulation showed that they were not needed. Rerunning the simulation showed that this change significantly improved the surface temperatures in the differential drive unit while also reducing case weight.

This example demonstrates the ability of simulation to enable engineers to

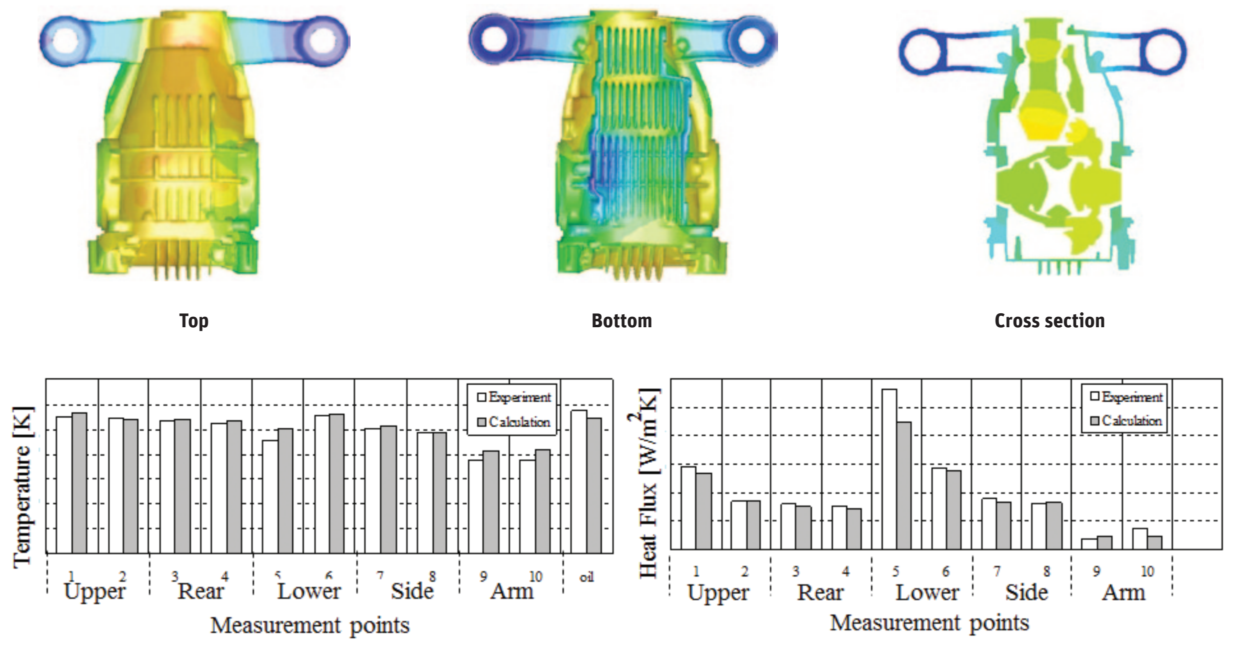


Comparison between physical measurements (left) and CFD visualizations (right) at 550 rpm (top) and 1,000 rpm (bottom) shows good agreement.



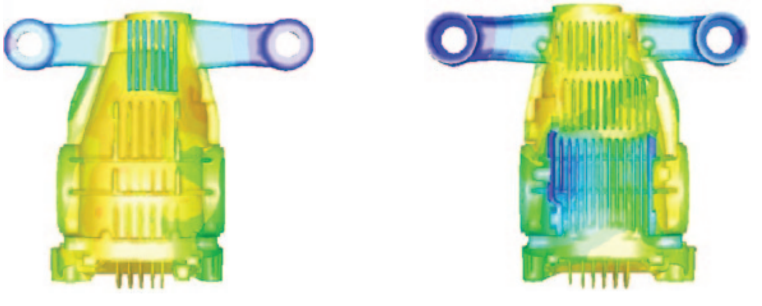
Time-averaged local heat-transfer coefficients between internal parts and oil, and between oil and case

The results demonstrated good agreement in heat flux distribution and temperature distribution between the actual measurements and simulation, illustrating the value of this simulation technology in the product development process.



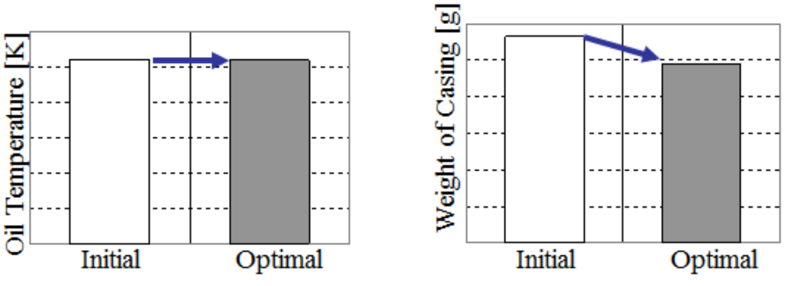
Heat calculation model generated predictions of heat flux distribution and temperatures, which agreed well with measurements.

The team reduced the weight of the drive unit while improving its cooling capacity.



Surface temperatures were reduced in the model on the right after redistributing cooling fins based on earlier simulation results.

evaluate design alternatives in a fraction of the time and cost required for physical prototypes. Simulation also provides more diagnostic information than can be obtained from physical testing, such as the ability to determine flow velocity, pressure and temperature at any point in the computational domain. As a result, the team reduced the weight of the drive unit while improving its cooling capacity. In the near future, Toyota plans to extend its simulation capabilities to model other types of drive units and to perform transient simulation, which should offer improved accuracy. ∧



Design changes made with input from CFD simulation maintained the existing optimal oil temperature while significantly reducing case weight.