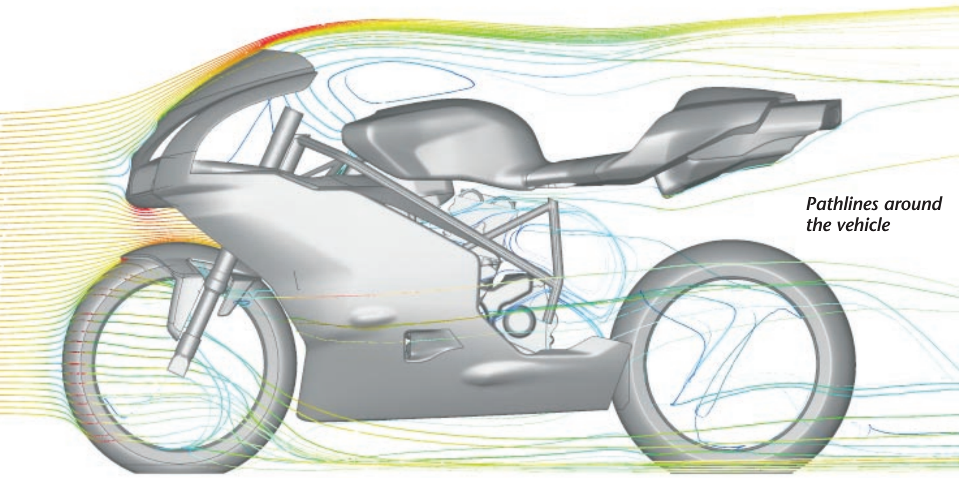
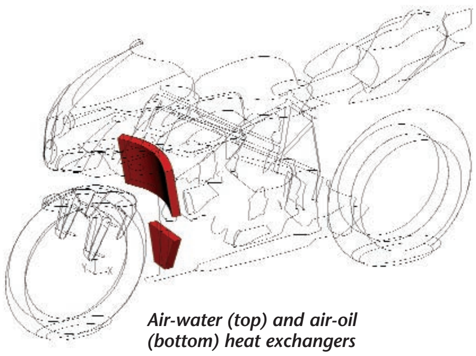


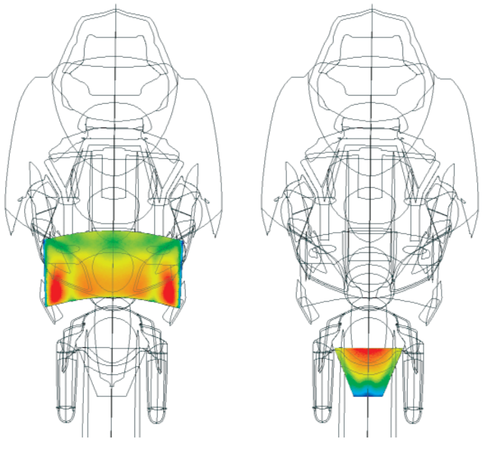
Pathlines around the vehicle

Cool Ducati
Heat Exchangers

By A. Cantagalli and R. Rossi, Laboratorio di Termofluidodinamica Computazionale Seconda Facoltà di Ingegneria di Forlì, Forlì, Italy; S. Di Piazza and A. Ferraresi, Ducati Motor Holding S.p.A., Bologna, Italy



Air-water (top) and air-oil (bottom) heat exchangers



Contours of velocity magnitude on the heat exchanger surfaces

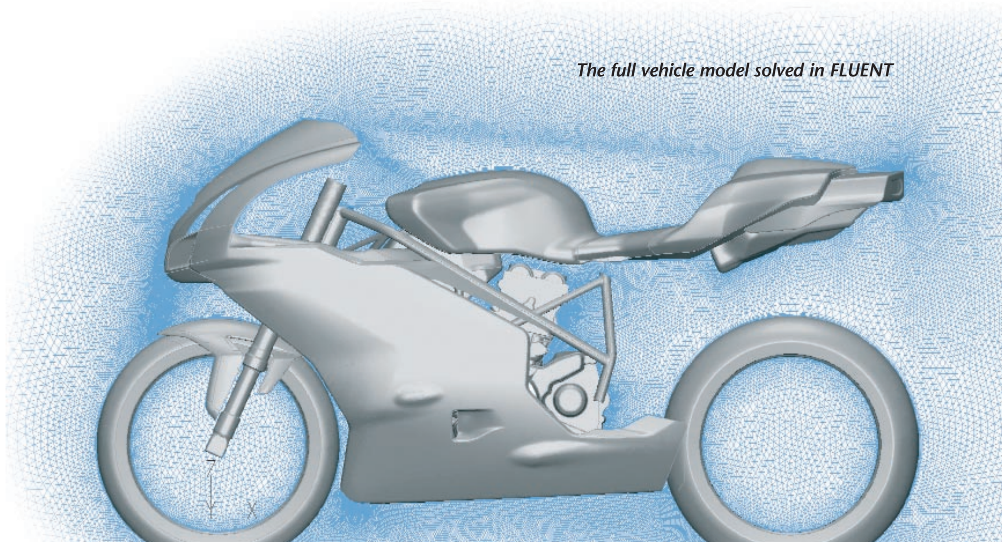
The Computational Thermo-Fluid-Dynamics Laboratory of the Forlì School of Engineering and Ducati Motor Holding have developed an innovative approach to the analysis of motorcycle heat exchanger performance. Called the "numerical multiscale method", the approach makes use of different orders of spatial discretization to analyze two heat exchangers embedded in the complex geometry of a motorcycle. The approach, which has been carried out on the cooling system of the Ducati 999 motorcycle, couples FLUENT with an in-house "distributed parameters" code.

Heat exchangers represent the main component of a motorcycle's cooling system. Two types are commonly used: air to oil, and air to water. Through these heat exchangers, the major part of energy produced by the engine's internal combustion is released to the environment. The auxiliary weight and size of these components are considered drawbacks in terms of performance and efficiency loss. Thus, there is a need to design heat exchangers that guarantee maximal heat rejection with reduced overall dimensions. Usually, the designs are based on the experience acquired from previous projects and from experimental data obtained from costly prototyping. Approximate formulae and correlations for heat exchanger design can also be used, but they do not help designers achieve a proper balance between cost and performance.

For the coupled analysis, a FLUENT simulation was first performed for the entire motorcycle, starting from the CAD surfaces of the vehicle provided by Ducati. The external flow analysis provided velocity distributions on the surfaces of the two heat exchangers. In the FLUENT simulations, the heat exchangers were modeled by a porous media in order to simulate the pressure drop across the components and the deviation experienced by the air when crossing them. The surface velocities predicted by the CFD analysis were then used as boundary conditions for the distributed parameters calculation for the heat exchangers. This approach uses a formulation that is based on the discretization of a system with *n* elements or entities, each of which represents a part of the system analyzed. In each of these entities, an integral form of balance equations is solved. For the heat exchangers, the distributed parameters calculation provides a detailed temperature distribution throughout the heat exchanger.

The CFD results illustrated some interesting features of the flow in the vicinity of the heat exchangers. The velocity distributions on the heat exchanger surfaces clearly show the influence of the front fork and front wheel. The upstream components also give rise to a non-uniform turbulence intensity on the surfaces of the heat exchangers, which may influence their operating efficiency.

The distributed parameter predictions for outlet oil temperature are in good agreement with experimental data. The success of the coupled calculations suggests that the numerical multiscale method is a valuable tool for quantifying the effects of surface velocity distribution on heat exchanger performance. It is cost-effective, too, since the analysis time is considerably less than that of a full CFD analysis of the motorcycle and heat exchanger detail for each proposed design change. ■



The full vehicle model solved in FLUENT