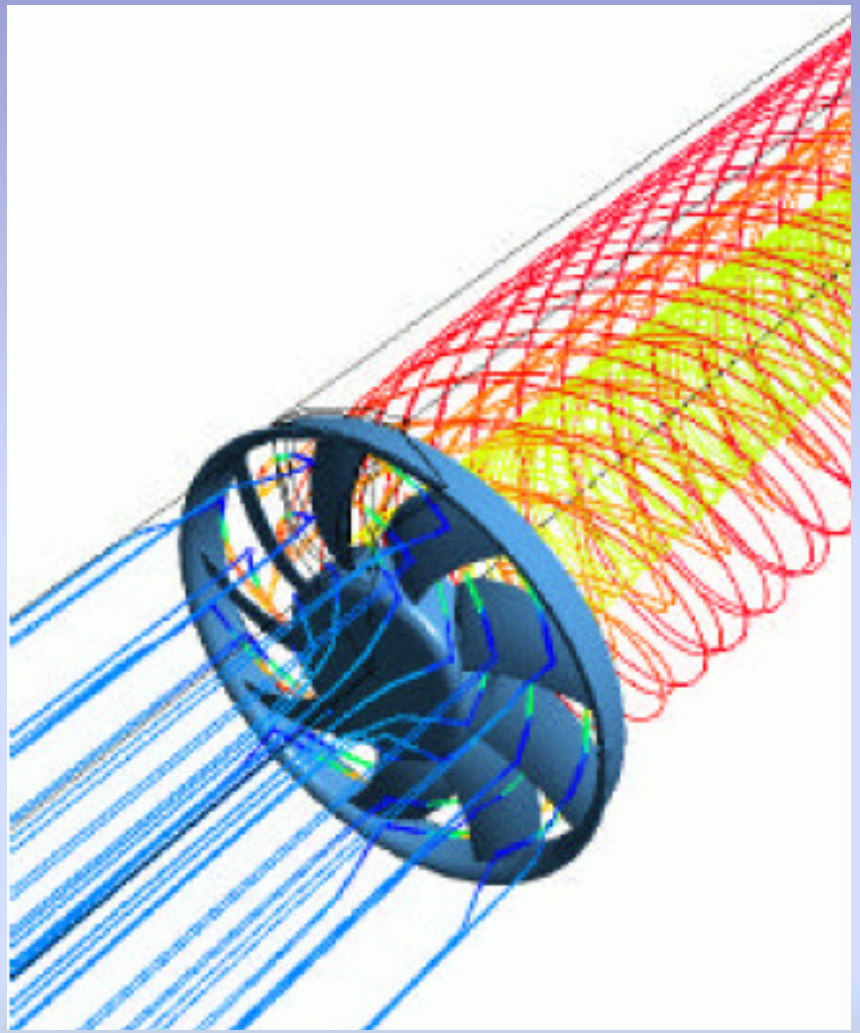
COMPUTATIONAL FLUID DYNAMIC (CFD) MODELING OF ENGINE COOLING FANS

Background

There is an increasing demand for more compact and efficient fan systems in order to improve automotive thermal management. CFD is a part of this design strategy.

Objective

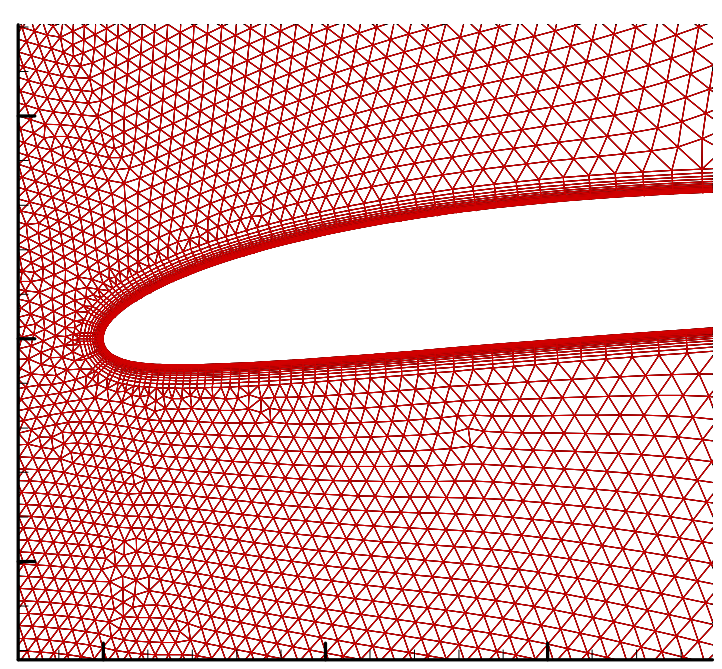
To build two- and three-dimensional CFD models of a fan blade to be validated by experiment and then used in the future for designing automotive fans.



Streak lines from three-dimensional CFD simulation of an automotive fan flow

Research Carried Out

2-D CFD models for different cross sectional profiles of the blade have been built. The flow field, lift and drag coefficient have been obtained. The 3-D CFD model will be validated by data from experiments.



Generated grid for the 2D CFD modeling of the fan blades

Expected Outcomes

The 3-D CFD model will be run for a new fan design under a range of operating conditions. This will allow formulation of a new design procedure.

2-D Contours of velocity magnitude (m/s)