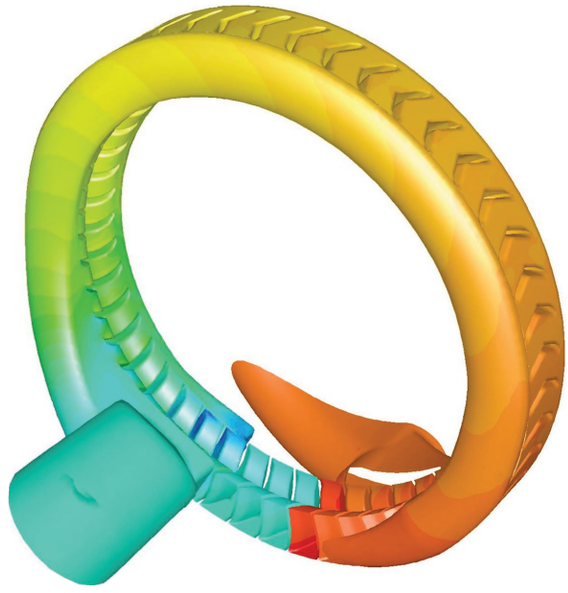
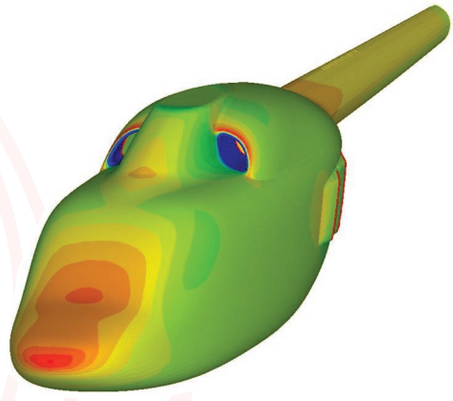
Transportation industry

About us

CFD.HU Ltd. is a Computational Fluid Dynamics consultation company closely associated with the Department of Fluid Mechanics at the Budapest University of Technology and Economics and the distributor and support center of ANSYS simulation software in Hungary. With the support of market-leading ANSYS Fluid Dynamics, Mechanical and Multiphysics software and our own computer cluster the company's internationally acclaimed staff can precisely analyse complex fluid problems with large cell numbers in order to optimise future engineering designs.



The company is co-lead by the former Head of the Fluid Mechanics Department, professor Tamás Lajos and Gergely Kristóf, PhD., the chairman of the sub-committee of Fluid Mechanics and Heat Technology at the Hungarian Academy of Sciences (MTA). All colleagues have a rich academic background with years of experience in the field of Computational Fluid Dynamics. The company's close relationship with the University also provides the opportunity to perform wind tunnel investigations to validate CFD results and to involve experts of other disciplines.

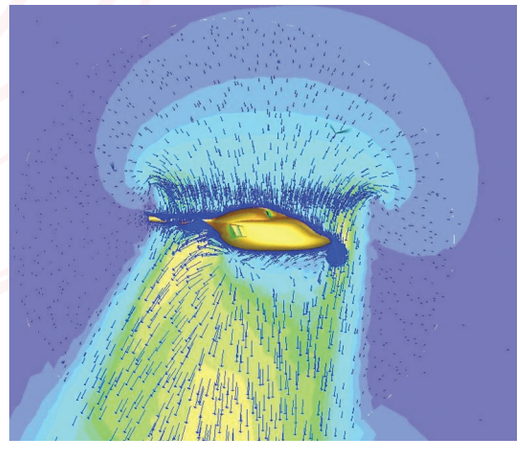


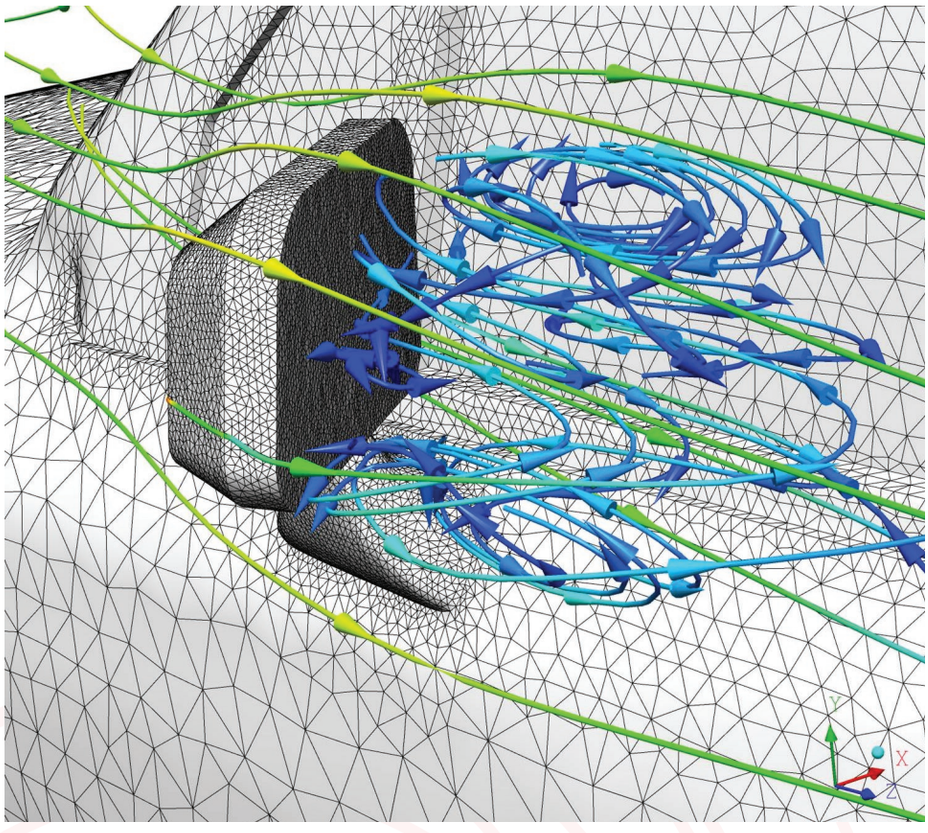
We use CFD, which makes it possible to:

- Analyse complex flow characteristics:
  - Investigate flow profiles
  - Optimise vehicle design and passenger comfort
- Evaluate wind forces acting on vehicles to:
  - Reduce costs associated with construction materials
  - Ensure low noise and efficient fuel consumption
  - Reduce drag force
- Investigate effective ventilation systems:
  - Reduce energy costs by determining optimal configuration
  - Evaluate configurations in a simulated environment reducing investigation time and cost
- Examine pollution distribution:
  - Ensure that the new design will meet current and proposed environmental regulations

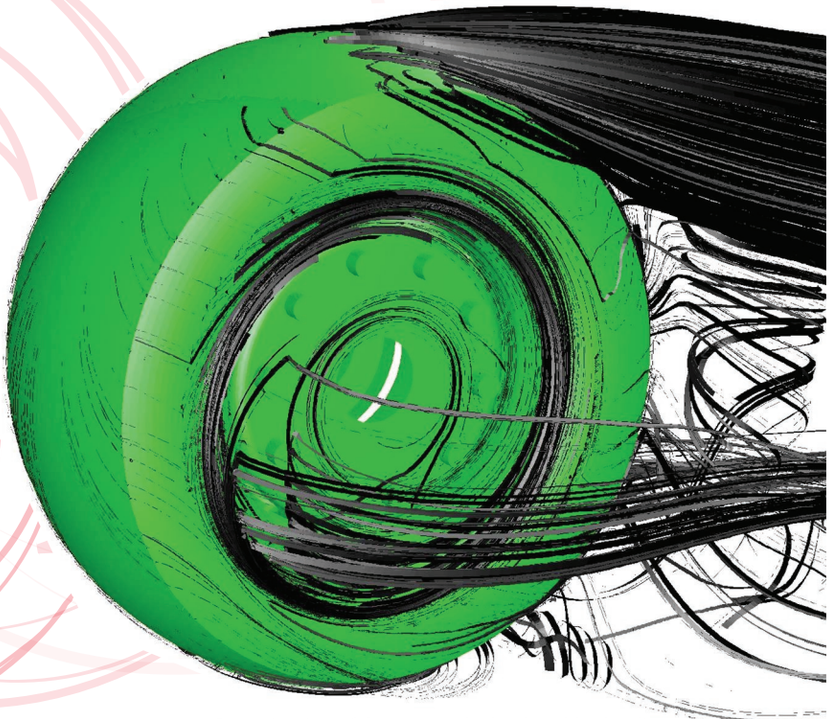
Aerodynamic investigation of a helicopter/ airplane body

In the case of a helicopter or airplane, an investigation of aerodynamic features can provide valuable design information. An accurate distribution of the static pressure can be used to determine the location for the inlet and outlet of cooling and combustion air through the fuselage. Therefore, accurate modelling of the complex processes acting in the boundary layer can help to optimise the body's final design.

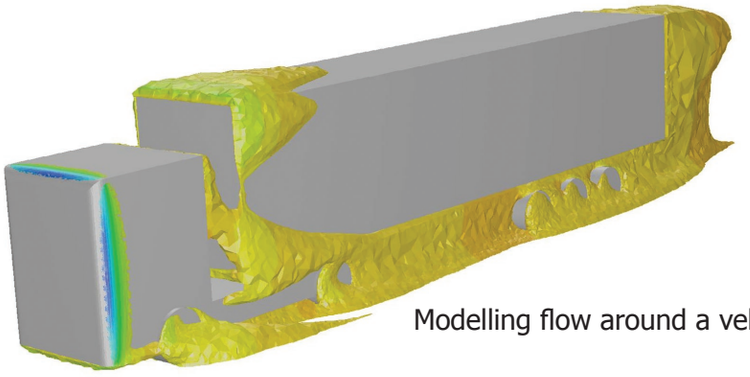




Investigation of the effect of side-view mirrors on noise induction and side panel pollution



Investigation of flow around wheel and brake disc



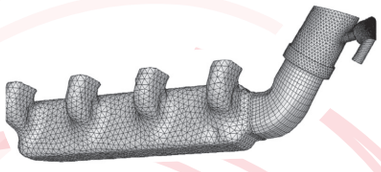
Modelling flow around a vehicle

An aerodynamic investigation of a vehicle can be carried out in two ways: in a simulated CFD environment or with a scaled down model in a wind tunnel. Simulation is becoming increasingly popular due to its time efficiency and low cost, while wind tunnel experiments are generally used as a means to validate CFD results.

Vehicles present many areas where flow optimisation can improve overall design and operation. The aerodynamics surrounding side-view mirrors affect the dynamic loss coefficient, noise induction, and introduction of pollution through the windows. The flow around a wheelhouse is responsible for approximately 30% of a vehicle's dynamic loss coefficient and is also shown to affect the cooling of the vehicle's brake discs.

For a project commissioned by the German company BPW Bergische Achsen, our colleagues used numerical simulation to conduct an investigation of the flow past a hot brake disc of a truck.

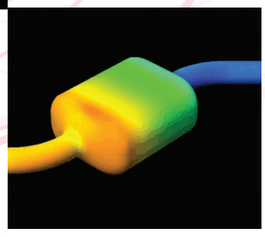
Simulation of a suction system



Turbocharged engines exhibit a delay, referred to as turbo lag, when the vehicle's accelerator is initially depressed. Many solutions have been developed to reduce this delay, each with their own advantages and disadvantages. In order to improve upon the solutions and ultimately determine the one most appropriate, CFD simulation can be used. In this case, numerical simulation requires the use of the new and innovative option of deforming meshes. Our colleagues have extensive experience with this option, using coupled simulations to provide comprehensive results.

Investigation of an exhaust system

In the regulated catalytic converter located in the exhaust pipe of a passenger car, an exothermal catalytic reaction occurs. To lengthen the lifetime of the catalyst, an even heat load is necessary which can be obtained by ensuring even gas flow. The computational calculation is complicated because the installed configuration consists of amorphous pipes and the catalytic body of the converter is made of a material with porosity dependent on flow direction. Using CFD simulations, the velocity and heat load distributions can be determined to optimise the geometry of the exhaust system.



Development of a fuel-feed system

To optimise the fuel-air mixture, there must be a better understanding of the pressure profile of a fuel pump. The amount of electrical power used by these pumps is one of the main load considerations when designing the electrical system of a car. Despite this fact, there is a lack of knowledge about their exact operation. CFD simulations can provide reliable information for the development of equipment operating with multiphase flow.
An investigation of the flow field in a side channel fuel pump was commissioned by the automotive supplier Visteon in order to establish the theoretical background necessary to improve the current pump design.

Visit us at www.cfd.hu to learn more about our references,
or contact us at info@cfd.hu.