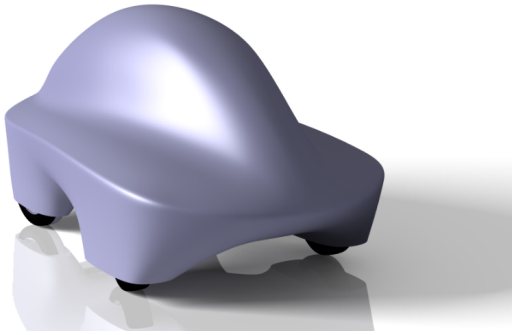
Master Thesis 2016-2017

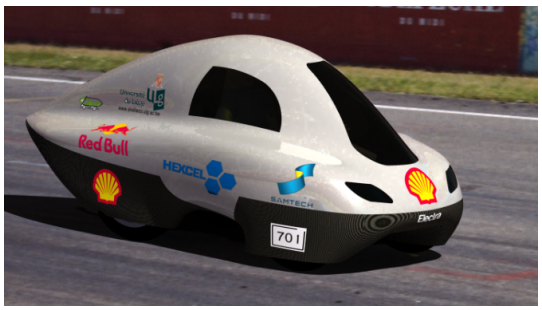
Title:	**"Aerodynamic optimization of the LTAS Eco-Shell vehicle prototype using CFD"**
Laboratory:	ULg-MTFC
Location:	ULg
Date:	February - June 2017 (but start in the fall recommended)
Target sections:	Aerospace Engineering, Mechanical Engineering, Engineering Physics
Contacts:	Vincent Terrapon
Advisors:	Vincent Terrapon, Pierre Duysinx

BACKGROUND

The University of Liège has been regularly taking part these past years in the Shell Eco Marathon. The objective of this contest is to drive 7 laps on a circuit within a given time (corresponding to a speed of ∼30km/h) using as little energy as possible. To achieve this goal requires very innovative vehicle and engine concepts, and the optimization of all aspects of the car. Among others, the aerodynamic drag on the vehicle is a major contributor to energy efficiency.

The flow around such a bluff body is complex, e.g., three-dimensional effects, turbulence, separation. While large eddy or direct numerical simulations would lead to more accurate results, the computational cost is too high. Therefore, a Reynolds-Averaged Navier-Stokes (RANS) approach is typically used for such geometries. Additionally, when flow separation is important Detached-Eddy Simulations (DES) can improve the simulation quality, but at a higher computational cost than RANS simulations.





OBJECTIVE

This study will help the team improve the aerodynamics of their prototype using numerical simulations. The goal is to compute the aerodynamic forces on the

LTAS vehicle using RANS and possibly DES simulations, and to propose potential improvements of the vehicle shape to minimize drag. The simulations will be performed with OpenFOAM, SU2, FINE/OPEN or another CFD package and different turbulence modeling approaches will be investigated.

TASKS

- Conduct a literature survey on the topic
- Perform steady RANS simulations using different turbulence models to compute the aerodynamic forces on the prototype
- Perform unsteady RANS simulations and possibly DES simulations and compare with RANS results
- Compare the simulation results with wind tunnel experimental measurements that are available for a model of the vehicle
- Propose improvements to further reduce the aerodynamic drag

PROFILE

The student must have some familiarity with fluid mechanics, CFD and turbulence modeling and be motivated. Following courses are recommended:

- AERO0004 - Turbulent flows (Fall semester)
- AERO0030 - Computational Fluid Dynamics (Fall semester)
- INFO 0939 - High Performance Scientific Computing (Fall semester)

REFERENCES

- http://www.shelleco.ulg.ac.be/