Validation of open source CFD methods used for race cars

Pedro Rubio, p.rubio@*iconcfd.com,* Icon Technology & Process Consulting GmbH., 20354 Hamburg, Germany
Baptiste Jacquet, baptiste.jacquet@audi.de, Audi AG - Aerodynamics, 85045 Ingolstadt, Germany
Dr. Ries Bouwman, *r.bouwman@iconcfd.com*, Icon Technology & Process Consulting Ltd., SL4 1QN Berkshire, UK

Abstract

The goal of the research was to apply a process based on open source CFD technology to a race car design environment. Based on the Audi Sport R14 and R15 racing model, simulations have been conducted and validated using race track data. Different setup methodologies were deployed, such as a symmetrical approach and complete car geometry with high refinements on feature edges. Such methodologies required a process that enhanced existing open source meshing. In order to achieve realistic simulation times, strategies were developed to reduce and optimise mesh sizes, without loss of accuracy.

In order to make the validation more accurate, parametric studies have been conducted on rear wing geometry and boundary conditions. The influence of rotating wheels in the simulations has been investigated in a wind tunnel simulation and was subsequently validated with wind tunnel experiments.

Since the level of accuracy is more demanding for racing cars than passenger vehicles, different turbulence models have been tested such as one equation and two equation models (k-omega). The simulations have been conducted using an open source CFD –based process, called AeroFOAM, developed by Icon for the Volkswagen Group.

Key Words: Race car, Audi Sport, open source CFD, DES, mesh, feature edges, rotating wheels, wind tunnel, race track.

Introduction

Open Source technology has been applied in many engineering fields over the past decades, which is clearly shown by the likes of Linux and Open Office. However, it was not until the beginning of the 21st century, that a break-through was made in open source Computational Fluid Dynamics (CFD).

Firstly, CFD itself is only just becoming an established engineering tool, not only in the verification process, but more and more in the design process. Automatic optimisations, parametric studies and complex multiphase studies are only the tip of the iceberg. Some industries already rely solely on CFD before a first prototype will be constructed.

Secondly, it was not until recently that the first open source CFD codes became available. However, their community was modest and some barriers still had to be overcome before widespread adoption could take place.

The largest car manufacturer in the world, the Volkswagen Group (VWG), decided to make a change and together with third party companies, they created a complete process in order to conduct aerodynamic, under hood thermal management (UHTM), climatisation and headlight simulations [1]. Over the past three years, an enormous amount of improvements have been developed for the VW Group in all CFD application fields [2]. The process was first applied only to passenger cars, but is now being slowly integrated in other areas such as sports cars.

This study describes the first validations of the open source CFD process as implemented at the VWG and applied to two recent models of Audi Sport, i.e. the R14 (DTM) and R 15 (Le Mans). Note that although the core mesher and solver technology is open source, other parts of the process (including a GUI) have been developed as proprietary to the VWG.

Methodology

The process described above is mainly based on open source CFD meshing and solver technology with enhancements made by ICON [3]. Since the start of the project, it has been verified for passenger cars throughout each stage, i.e. meshing, pre-processing, solving and post-processing. However, up to now, only very limited validation on sports cars have been conducted.

Audi Sport started validation on two racing cars, the R14 DTM and R15 Le Mans (see Figure 1) . These validations are most of the time not conducted in wind tunnels, but on racing tracks. The CFD methodology applied is therefore very different. The validation of each step of the process is described below.

Figure 1 R14 DTM (left) and R15 Le Mans (right) in action

Mesher

The requirements on the mesh for a sports car are far higher than the ones on a passenger car. However, the mesh size should remain acceptable in order to have fast enough turnaround times.

The initial meshing process [1] was not accurate enough to simulate the fine edges and spoilers, without creating too many cells. Also, the quality of the boundary layer mesh, the so called ‘prism layer’, had to be improved. New techniques were developed in order to refine the mesh, not only on the surfaces, but also along the edges. At the same time, new algorithms for boundary layer termination improved the quality of the mesh significantly without excessive amounts of cells (Figure 2).

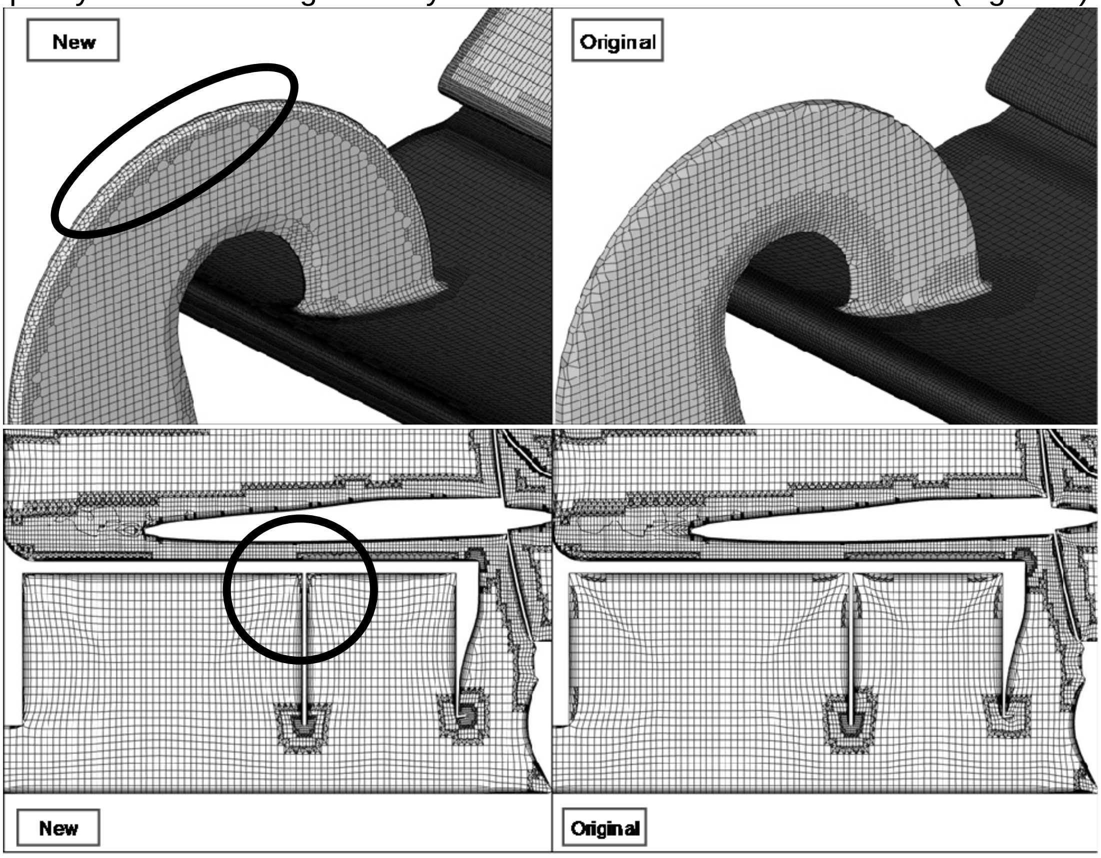

Figure 2 Mesh refinement by feature lines (top) and boundary layer termination improvements (bottom)

The wall functions used in turbulence modeling are very sensitive to the local mesh size and velocity. This dependency is often expressed by a non-dimensional wall distance, y^+.

$$y^+ = \frac{u_* y}{\nu}$$

where u_* is the friction velocity at the nearest wall and y is the distance to the nearest wall. ν is the local kinematic viscosity of the fluid.

In order for the wall functions to work properly, a fairly uniform y^+ value within a certain range is required. For race cars, the velocity gradient can vary enormously around certain geometric parts (e.g. spoiler) hence a local refinement depending on the flow field should be integrated. Using a combination of the open source process and the tool ANSA [4], automatic zones of mesh refinement were created during the simulation at locations of high velocity gradients in order to improve lift/drag calculations.

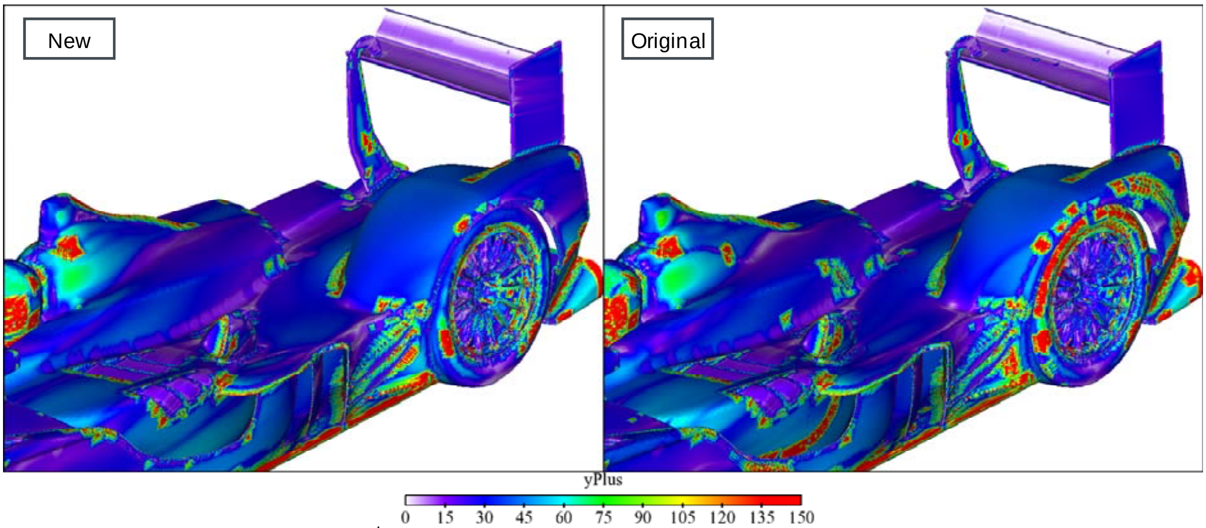

Figure 3 Improvement of y^+ value due to automatic local refinement strategy

Pre-processing

One of the main characteristics of the publically-downloadable open source CFD solver and mesher is that it is solely text-file based. For some CFD engineers (and their companies) who are used to working with traditional proprietary CFD codes, the change from a Graphical User Interface (GUI)-based process to text files represents a barrier to adoption.

In order to overcome this obstacle, a highly intuitive GUI has been created that allows an efficient setup of the simulation (see Figure 4).

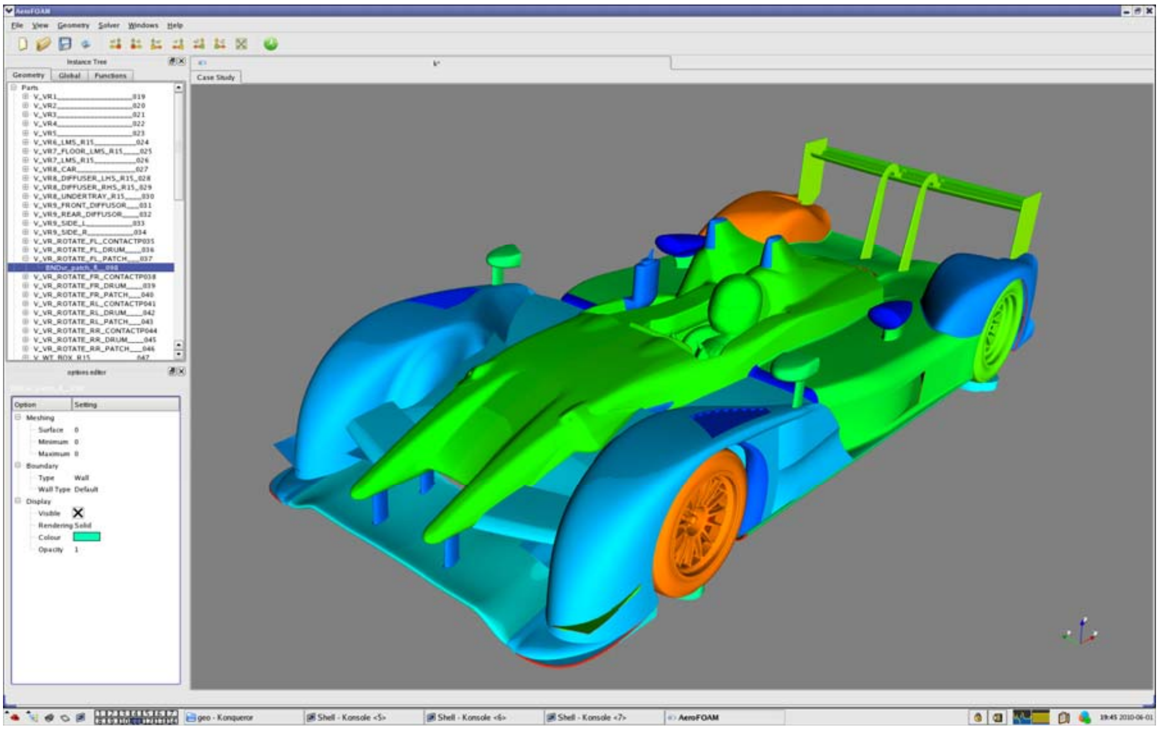

Figure 4 R15 as setup using source purpose-built GUI for the open source meshing/solver process

Solvers

At this stage, the open source CFD process allows steady-state Reynolds-averaged Navier-Stokes (RANS) simulations and Large Eddy Simulations (LES). For the present validation study, only RANS simulations have been used.

The open source CFD solver provides several turbulence models for the RANS simulations, but we narrowed our choice down to three models: one equation, two equation and hybrid two equation models (k-omega).

Post-processing

The open source CFD process allows for post-processing in both commercial packages such as Ensight [5], but also open source packages such as ParaView [6]. The latter does not even require conversion of the result files.

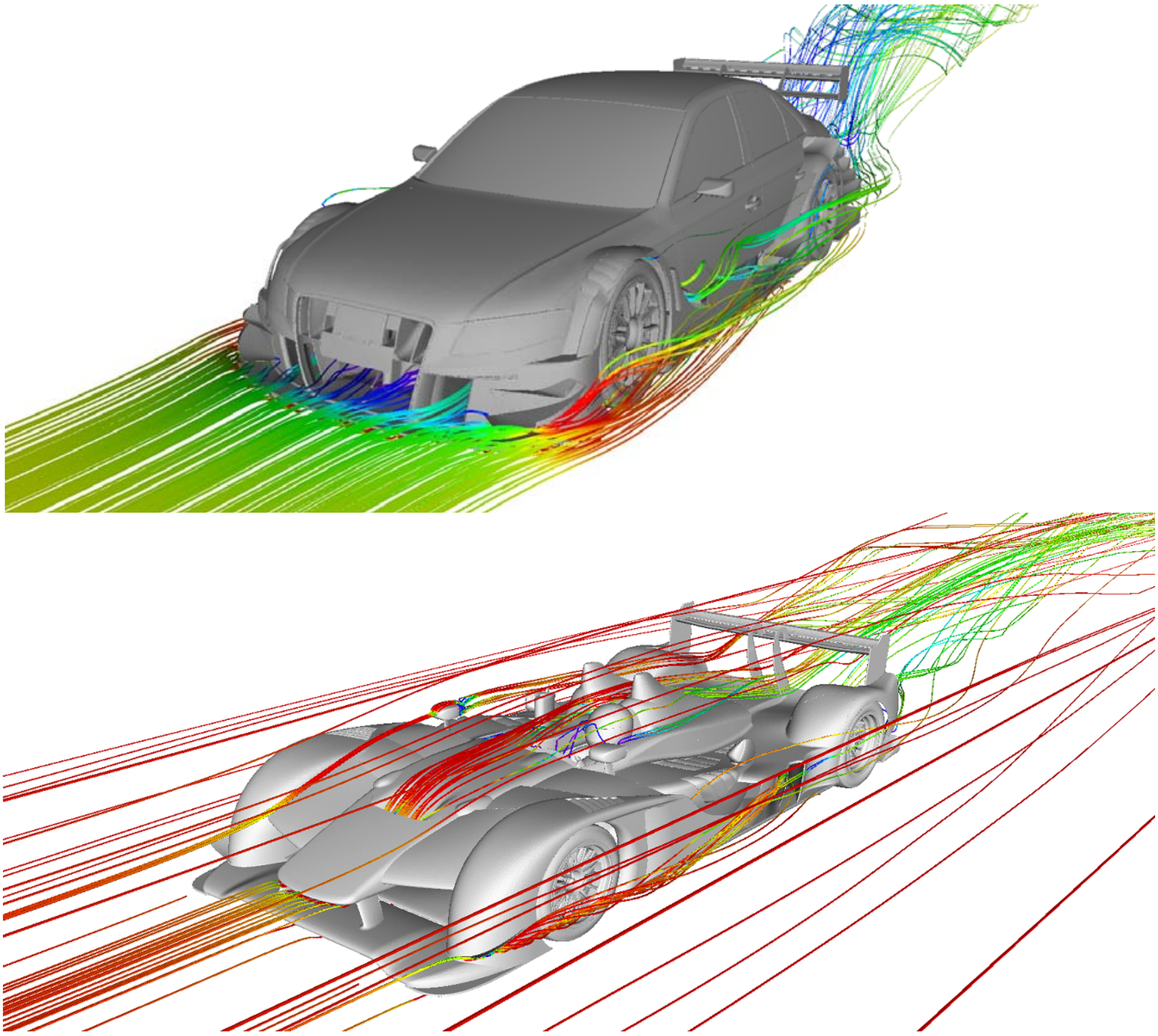
Figure 5 Streamlines over R14 DTM (top) and R15 Le mans (bottom)

Results
Based on the improved methodology described above, several validation simulations have been conducted on both the R14 and the R15. The results are described in the following paragraphs.

R14
Using the R14 geometry, first the open source process is benchmarked against race track data. Second, different flow angles and car positions are simulated.

Solver validation
Based on racetrack measurements and simulations using an existing CFD process three turbulence models have been tested on the same mesh in the open source process. The results are summarised in the table below (Figure 6).

	Cz	Cx	Eff	Bal	Czfr	Czrr
R14 Racetrack	100%	100%	100%	100%	100%	100%
R14 Open Source two equation	94.0%	95.6%	98.3%	105.8%	99.5%	90.3%
R14 Open Source hybrid two equation	90.9%	94.7%	96.0%	107.4%	97.6%	86.4%
R14 Open Source one equation	93.6%	98.2%	95.4%	104.5%	97.9%	90.8%

Figure 6 Comparison of racetrack data with open source simulation results.

where

C_z *Total down force coefficient* $\frac{Vertical\ Force}{\frac{1}{2}\rho \cdot S_{ref} \cdot V^2}$

C_x *Total drag coefficient* $\frac{Drag\ Force}{\frac{1}{2}\rho \cdot S_{ref} \cdot V^2}$

Eff *Efficiency* $\frac{C_z}{C_x}$

Bal *Balance = front axis down force/total down force* $\frac{C_{zfr}}{C_z}$

C_{zfr} *Front downforce coefficient*

C_{zrr} *Rear Downforce coefficient*

The results clearly show that the two equation turbulence model is most appropriate for the R14 simulations.

Finally, two different front end configurations of the R14 have been measured and simulated. The difference between the two configurations has been plotted in Figure 7. The results clearly show the correct trend for front and rear down force of the open source process. Drag and efficiency predictions could be further improved.

Car position

The previous validations showed that the open source process can be applied to race car simulations. In a next step, typical car positions have been simulated in order to further test the applicability of the process.

First, the R14 was simulated using different angles of deviation. The open source process showed results in accordance with the track data, i.e. slight balance shift forward and a loss of down force.

Second, different heights (Figure 9) of the R14 have been simulated and compared with experimental results. Figure 10 shows the so called aerodynamic maps, where lift (Cz), drag (Cx) and balance are plotted against the front and rear down force. Again, the trend looks good, but there seems to be an offset with the track data. The results for both the front and the rear lift are plotted in Figure 10.

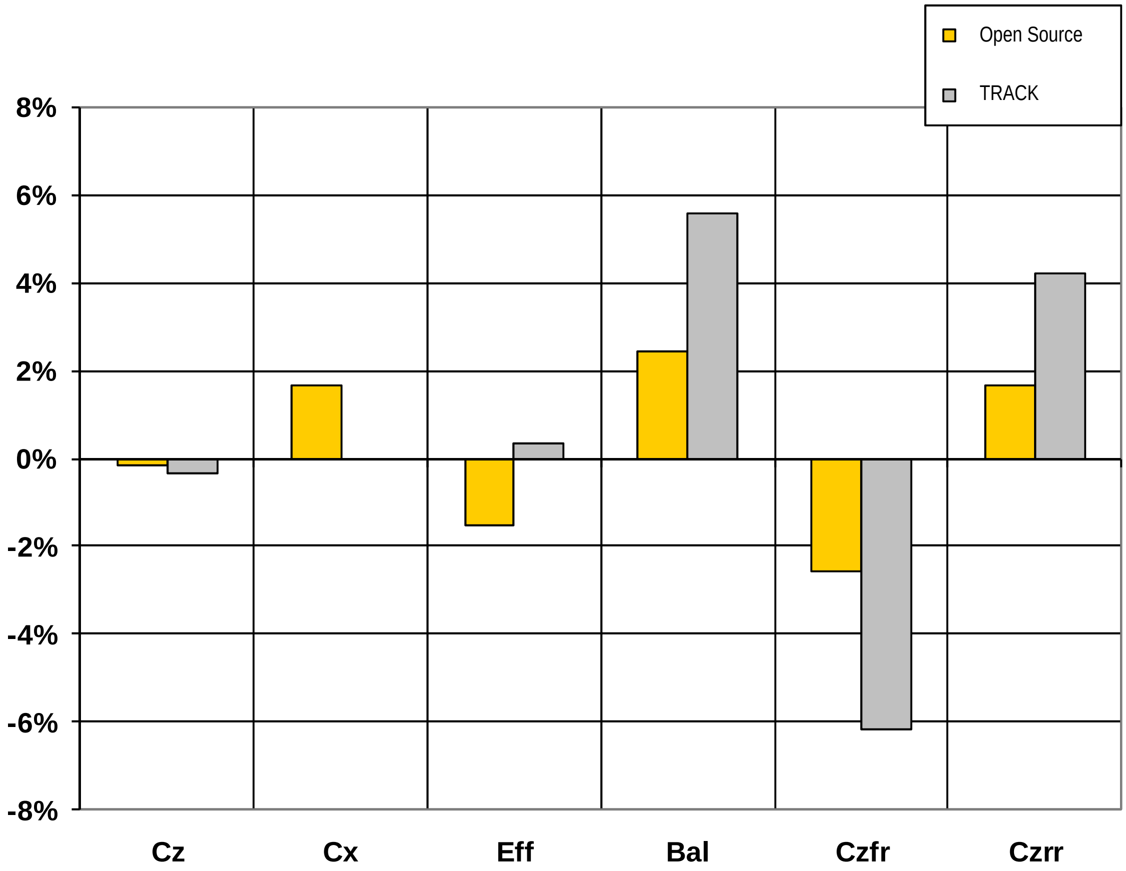

Figure 7 Relative comparison between two different R14 configurations by measurement data (TRACK) and simulations

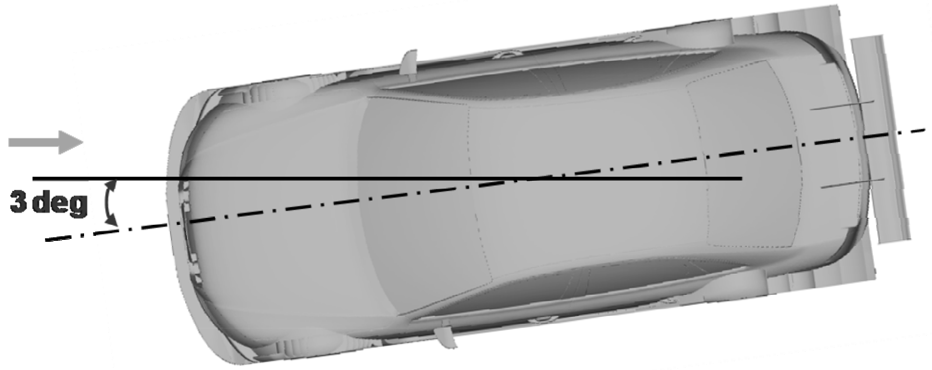

Figure 8 R14 simulated at different angles of deviation.

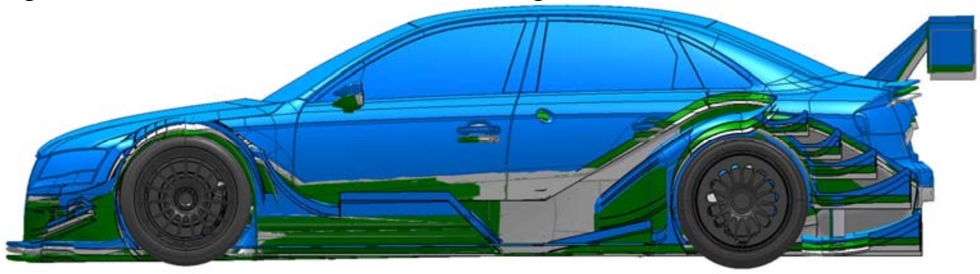

Figure 9 R14 at different heights, compared to the standard setup.

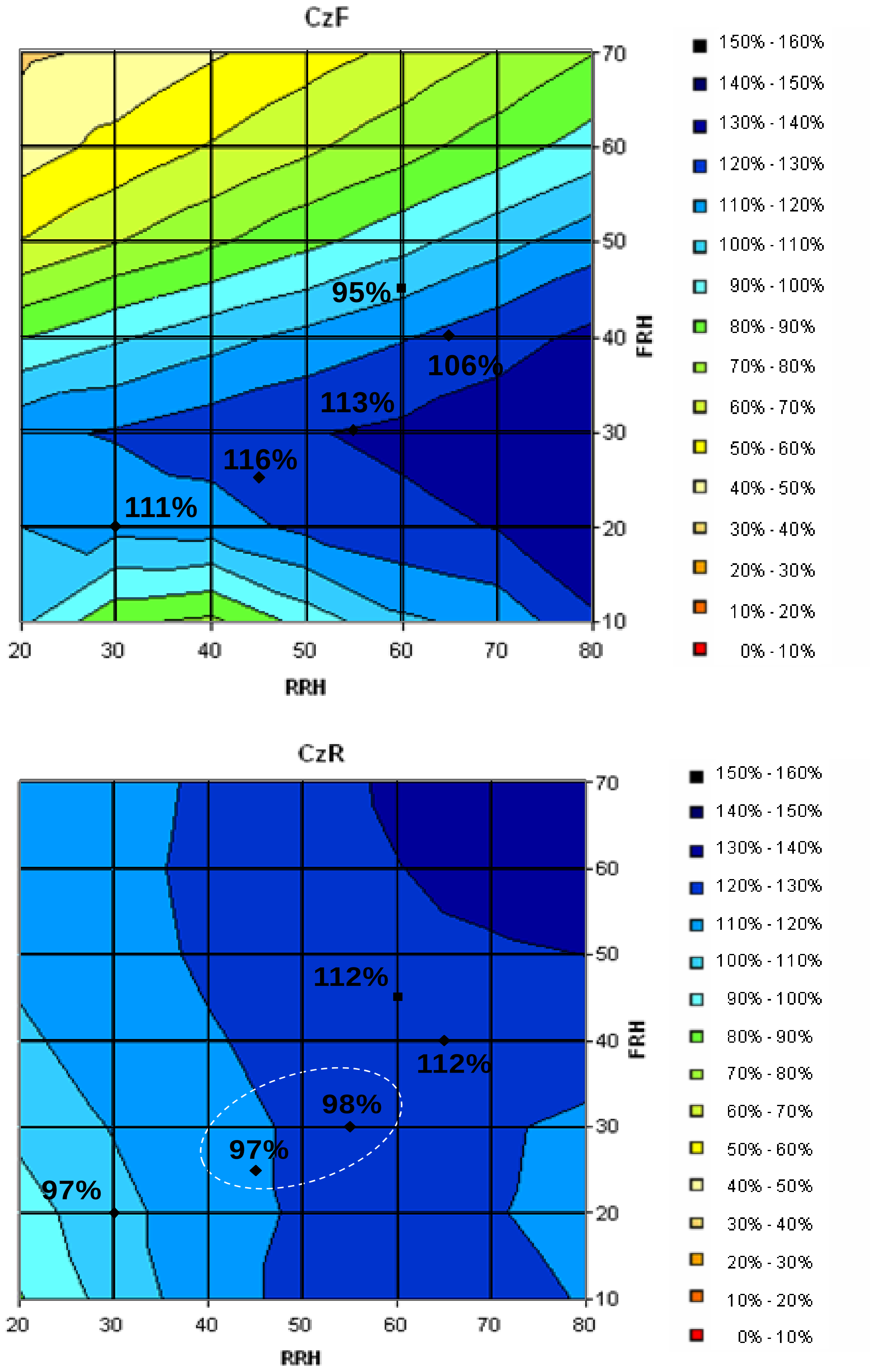

Figure 10 Front lift (top) and rear lift (bottom)

R15

The R15 geometry has been used to conduct a parametric study over six positions on the front spoiler (see Figure 11). The spoiler has been rotated over different angles and the simulation results compared with experimental results from the racetrack. On the track, the vertical forces in the pushrods are measured using linear strain gauges and drag with the moment in the driveshaft.

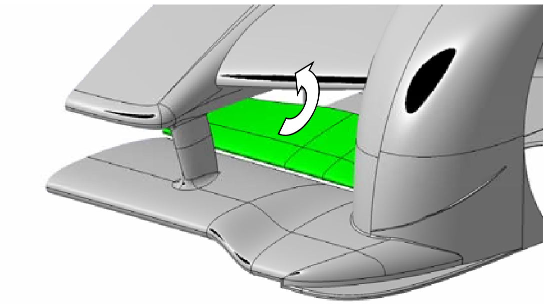

Figure 11 Parametric study over 6 positions of the front spoiler of the R15.

Parametric study

The results of the 6 simulations at different front spoiler configurations are seen in Figure 12. According to the track measurements, the front balance and total down force should increase with the spoiler's angle. The open source results do predict the balance shift correctly. Nevertheless, the total down force predictions show some scatter. A linear interpolation of the results show that the absolute values are close to the track data.

Although there are no large variations in head numbers, the flow structures change a lot when increasing the front spoiler configuration. Local mesh refinements helping to accurately model the parts directly linked to the front wheel wake and different turbulence models could improve this.

These results were promising and showed the applicability of an open source CFD process on racecar simulations.

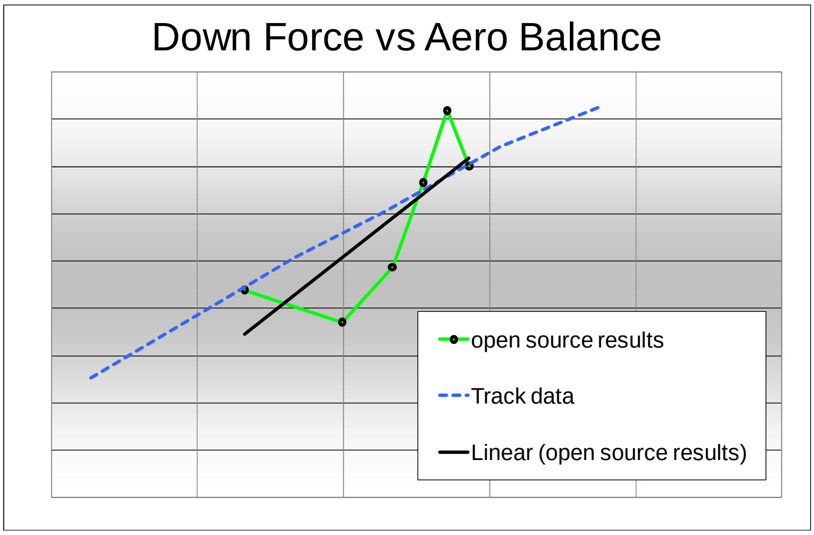

Figure 12 Simulation results of the open source CFD process against racetrack data

Discussion and Conclusions

The investigation clearly showed that the open source process produces valid results and can be applied to racecar simulations.

More development in mesh techniques as well as turbulence modeling was required. However, the flexible nature of open source software allowed engineers at Audi Sport and Icon to adapt the process to their needs.

Another great advantage of open source technology is the scalability of the process. In times where hardware costs are becoming marginal, there is almost no limit to the model size and simulations with over 100 million cells are turning into normality instead of the exception,
Also, local, diversified post-processing during the simulation run helped identifying local improvement potential.

Acknowledgements

The authors would like to thank the Audi AG for permission to use the R14 and R15 data for this work.

References

[1] M. Islam et al., Application of Detached-Eddy Simulation for Automotive Aerodynamics Development, SAE 2009-01-0333, SAE International, 2009
[2] Volkswagen Group Press Statement, Volkswagen Group Adopts New Computational Fluids Dynamics (CFD) Process Developed with ICON, July 10, 2009, Wolfsburg, Germany
[3] http://www.iconCFD.com
[4] http://www.beta-cae.gr/ansa.htm
[5] http://www.ensight.com/
[6] http://www.paraview.org/