Supercomputing in F1 – Unlocking the Power of CFD

Larsson T., Sato T., Ullbrand B.
SAUBER PETRONAS Engineering AG,
Wildbachstrasse 9, 8340 Hinwil, Switzerland

ABSTRACT

The highly sophisticated body shape of a modern Formula One (F1) car is dictated by aerodynamic efficiency and performance. With numerous deflectors and external devices added, the coupling and interaction between the front-end and rear-end of the car have become strong. Minute changes in geometrical details or car set-up can have a significant impact on car performance and, therefore, can result in success or failure. Such detail optimisation is accomplished in wind tunnels which increasing numbers of competitors in F1 runs 24 hours a day to discover that last fraction of performance gain. This level of fine-tuning is beyond the current CFD capabilities.

Still, though, this design evolution process through physical testing is somewhat of a heuristic method. The fundamental understanding and knowledge of the underlying physical mechanisms are not necessarily gained. The complexity and nature of F1 aerodynamics can only be fully understood through advanced and highly accurate CFD simulations.

With ever increasing simulation capacities, new horizons are opening up. New investments in computer hardware and CFD technology allow SAUBER PETRONAS to further explore and uncover even some of the most subtle flow phenomena responsible for fundamental changes in car behaviour, handling, and performance.

Keywords: *Formula 1, Sauber Petronas, Supercomputing, Vehicle Development Process, Side-Slip, Linux Cluster, Curved Plates, Engine Cover, Overtaking, AMD*

1. INTRODUCTION

In recent years the automotive industry has experienced significant changes in the development process for new vehicles in order to have:

a) *Improved product performance and quality.*
b) *Reduced development costs.*
c) *Reduced lead-times and “time-to-market”.*

The traditional Vehicle Development Process (VDP) has been replaced with a modern approach that relies more on computational tools and simulations. In the early 90’s the automotive industry was running on an average 60 month VDP’s. Today, most major automotive manufacturers are working to a VDP of 18 months or less (fast VDP). This resulted from considerable effort that has been devoted to the development of computational methods, providing guidance in the design of various components of modern cars. The introduction of *unstructured grid technology*, *accurate and robust numerical methods*, and the availability of *powerful parallel computers* has acted as catalysts in the rapid acceptance of these approaches.

Computational Fluid Dynamics (CFD) is now becoming a cornerstone in vehicle development and is extensively used throughout the complete process, from early concept phase to detailed analysis of a final product. Within the F1 industry, development lead-times have to be kept even shorter. In F1, not only a brand new race car is born before the start of every season, but intense development of the race car also continues throughout the season for every race, which takes place only two to three weeks apart. Hence, the importance of an efficient and fast development process becomes evident to keep up with such a demanding pace.

2. F1 AERODYNAMICS

A modern F1 race car is capable of exceeding speeds of 360 kph, generating steady state cornering forces of 2.5g to 3.0g and straight-line braking forces of 3.5g to 5.0g. Such braking and cornering capabilities require large amount of aerodynamic downforce. In order to generate sufficient levels of downforce, the cars are equipped with front and rear wings, and specially contoured bodywork and underbodies, all operating in close proximity to the ground.

The nature of the flow field around a F1 car is highly complex. The presence of large body and tyre wakes, strong vortices and separated flow regions, all contribute to create a complicated bluff body aerodynamic flow scenario. Figure 1 shows features of the simulated flow field around a F1 car. Further illustrations of CFD applications within F1 are displayed in Ref. (1, 2).

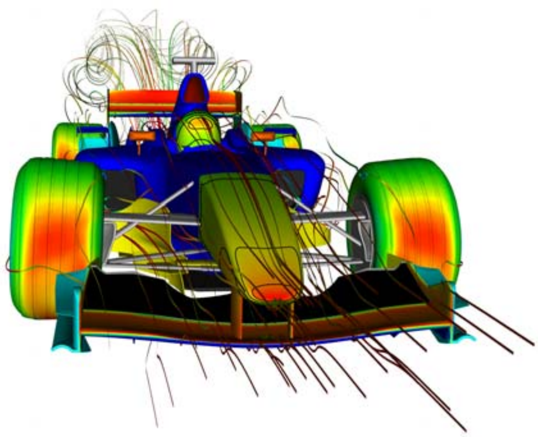

Figure 1: Aerodynamics simulation around a SAUBER PETRONAS C23.

Aerodynamics has nowadays become the dictating factor in the development of a successful car, thus participating constructors have made enormous investments in this area of development, highlighted by the construction of their own dedicated wind tunnels and, in recent years, increased computational resources for CFD analysis.

Today, CFD technology is being applied successfully in many areas of race car development. A few key areas include:

- *Early Concept Phase* - Parametric Studies of Bodywork Concepts
- *System Design* - Engine Cooling, Brake Cooling, Fuel Systems
- *Component Design* - Front Wings, Rear Wings, Diffuser, Deflectors, Brake Ducts
- *Complete-System-Design and Interactions*: - Full Vehicle CFD analysis

CFD has become a vital complement to wind tunnel and race car track testing. With added simulation capabilities, more concepts, ideas, and new components can be evaluated within shorter times. This will save costs, reduce lead-times, and eventually lead to a better utilization of the wind tunnel resources, since only the most promising concepts pre-selected in CFD need to be manufactured and tested.

Moreover, there are certain scenarios that can only be simulated in CFD due to e.g. practical limitations of physical testing. A few examples are; simulations of hot exhaust gases, brake heating/cooling events, tire deformations, and fuel filling and sloshing. CFD will also bring new detailed knowledge and insights into the aerodynamic flow around the car and on how components interact at various conditions. These are pieces of information virtually impossible to extract from physical testing but crucial for the conceptual understanding in the quest for finding performance gains in this the most competitive category of Motorsports.

3. SUPERCOMPUTING IN F1

Introduction of Sauber Petronas' powerful new partner "Albert" earlier this year marked the beginning of a new era for the team's CFD involvement in aerodynamic development. "Albert", a supercomputer, which is equipped with over 500 processors, 1 TB of memory, and over 10 TB of storage space, is a significant and crucial step forward in increased computational capacity for CFD. It could not have come at a better time, as the new wind tunnel is in full operation, the pace of aerodynamic development had increased dramatically. The supercomputer responds to this challenge with the capability to handle models with more complexity, with even higher mesh resolution, and far less time is required to obtain solutions than ever.

Figure 2: Albert- A Linux Cluster dedicated to CFD simulations.

Supercomputing capability is nowadays vital for F1 aerodynamic development. Due to the complexity of the aerodynamics, producing accurate CFD predictions to a tight time schedule becomes quite a challenge. With numerous deflectors and external devices integrated to the bodywork, understanding of the coupling and interaction between the front-end and rear-end of the car has become very important. Minute changes in geometrical details of these components can often have a global impact on the overall flow topology, therefore influencing car performance. Hence, a simplified sub-modelling approach can provide only very limited information, sometimes misleading, and would not be sufficient. Only detailed full-car simulations have a chance of bringing the CFD results to the required confidence level. The CFD simulation must resolve every relevant surface feature of the complete race car.

Grid resolution is of prime importance here; details in numerical and physical modelling are secondary. As a consequence, massive computational resources are needed for accurate F1 external aerodynamics predictions. After the introduction of the supercomputer, further refinements in the modelling techniques are now possible. Even parametric studies using full-car simulations have become a feasible approach.

4. CFD MODEL DEVELOPMENT

CFD models are used to conduct aerodynamic evaluations of various parts of the F1 car and they have evolved at SAUBER PETRONAS along with increasingly available computational resources and improving software capabilities. It is reasonable to say that for development of models, a platform for simulation, is as imperative as the aerodynamic development itself in CFD.

Just in the past couple of years, computational capacity at SAUBER PETRONAS has increased by a thousand as illustrated in Figure 3. This leap has taken the level of complexity of simulation models from a very localized, nearly stand-alone approach with a compromised mesh resolution, to a massive full-car model with much resolved mesh surrounding the whole car and downstream.

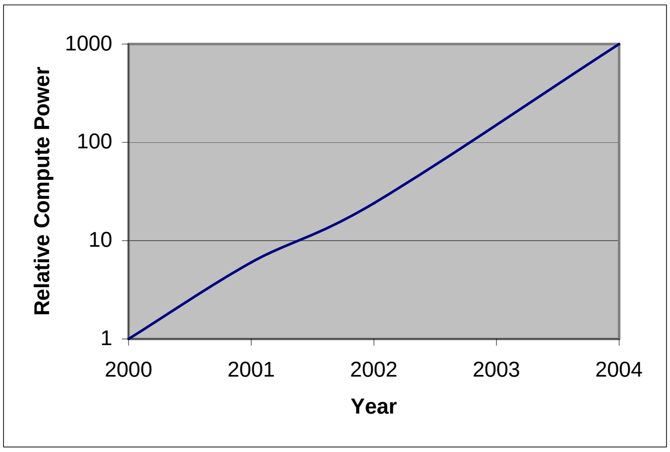

Figure 3: Progress of Sauber CFD computational capacity.

For example, front wing analysis was conducted on a relatively simple front-end model just a few years ago. It was merely capable of evaluating front wings. Cell count only reached about 20% of what is generally used today. It was soon found to be insufficient, as interest broadened beyond the evaluation of the front wing itself. The new model had to allow not only front wing analysis, but also downstream flow field study to further understand the impact of the front wing and its devices to the trailing car bodywork and components. This required a careful meshing strategy, because capturing wakes and vortices was very critical for the task. A refined volume mesh was necessary, but it was also costly in cell count. At this point, even with much reduced rear-end physical details and mesh resolution, it reached 50-60% of today's cell count. The subsequent introduction of a new meshing scheme, which helps to resolve the flow field with better control and a reduced cell count requirement for the same model. This improvement in meshing technique prompted an overhaul of the model.

The new model had all the details of the car with improved surface and surrounding flow field mesh resolution, while keeping the overall cell count similar. Then, there was the arrival of Albert, which was a major step forward also from a modelling point of view. Now the model intended for front wing analysis is essentially a fully detailed model with fine volume mesh in proximity. A jump in cell count was imminent, however it was an affordable penalty as computation time is less than half that for nearly double the cell count.

Current CFD models generally used today at SAUBER PETRONAS are hence fairly large, even in the sense of today's level of computational capability. It is increasingly becoming important to maintain a good balance among available computational resources, pre-processing, solving, and post-processing times. As models become larger, each step becomes a time-consuming task, and it is entirely governed by the cell count. Therefore, an efficient meshing strategy is the key to a successful CFD model.

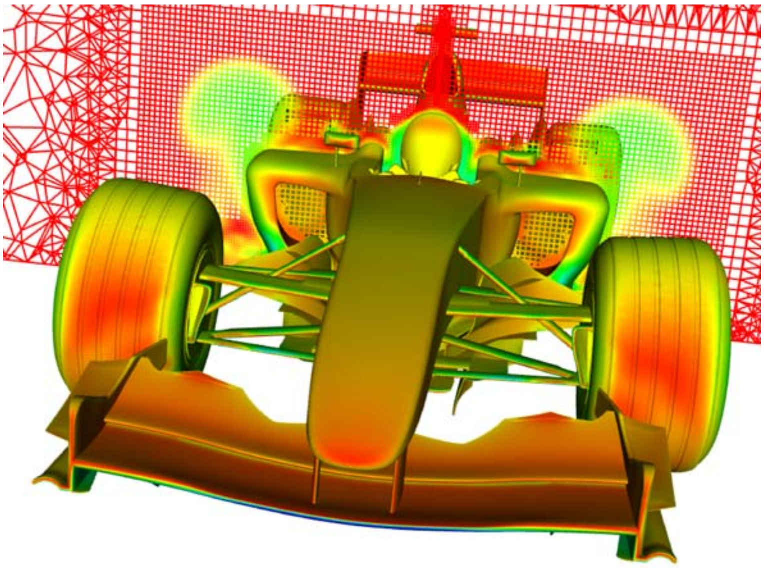

Figure 4: C23 full car simulation solution sample showing mesh resolution around the car.

5. CFD MODELLING - MESHING STRATEGIES

A careful choice of meshing techniques is crucial for correctly capturing all essential flow field phenomena without creating a mesh of prohibitive size. The flow around F1 cars exhibits strong gradients travelling over large distances, and the control of the volume mesh in terms of both resolution and quality becomes of utmost importance. An accurate prediction of viscous vortex flows is known to be a true challenge, and excessive numerical dissipation resulting from insufficient grid resolution is a major concern. Using a hybrid meshing strategy based on tetrahedral, hexahedral and prism elements has here proven to be efficient. To further illustrate the significance of grid influences on capturing vortex flows, a few findings from a simple test case are presented herein.

The simple model used here constitutes a strake type vortex generator placed in a box as depicted in Figure 5. Different levels of tetrahedral and hexahedral meshes were produced to investigate effects from mesh resolution and structure. Pressure data from the simulations were extracted at different downstream locations of the vortex generator in order to investigate the decay of the vortex pressure peak, sees Figure 6 - 7.

It was found that the hexahedral meshes were less dissipative leading to a much better capturing of the streamwise vortex propagation. Mesh convergence using hexahedral cells was achieved with only 25% of the cell count needed for tetrahedral mesh convergence. Altering discretization schemes did not have any major influence on the solutions.

F1 cars are equipped with bodywork and numerous devices that produce vortices and wakes, and they are prone to affect the performance of downstream components, whether intended or not. As discussed previously, local analysis is thus not sufficient as an optimum local solution could be disastrous for the rest of the car. Therefore, consideration of vortex tracking in particular is an important aspect in a mesh generation process. The strategy presented here achieves a high level of vortex tracking capability at reasonable cost (cell count).

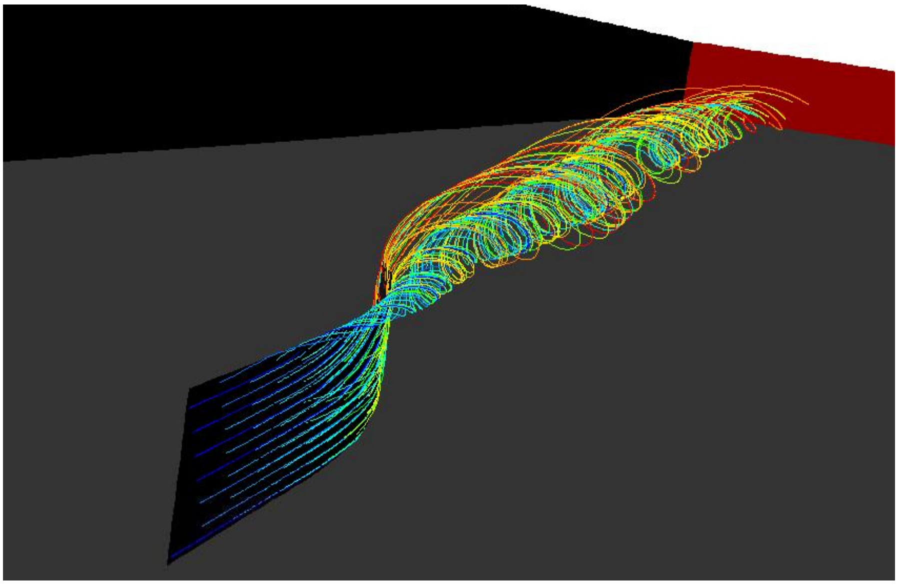

Figure 5: Pathlines released from a curved plate showing vortex flow.

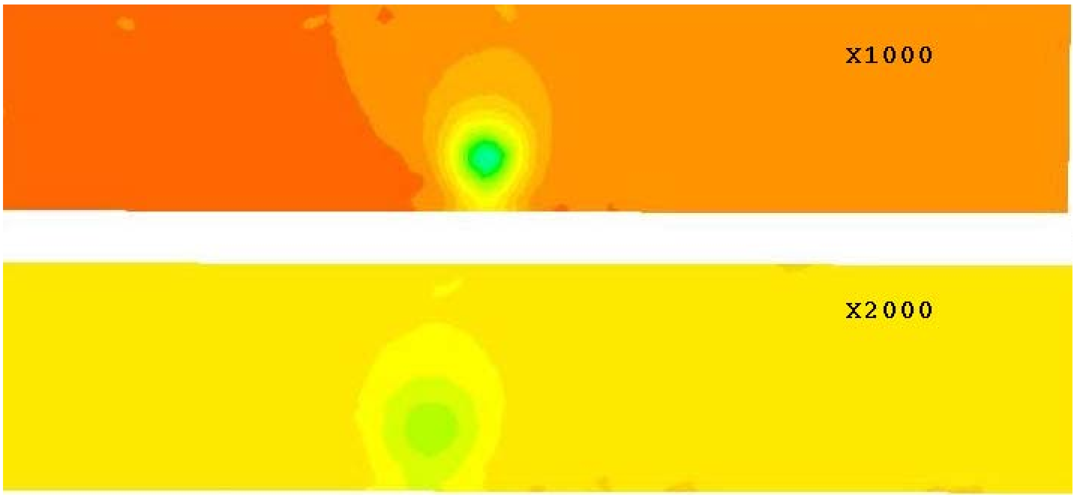

Figure 6: Pressure contours in planes downstream of the plate, tetrahedral mesh.

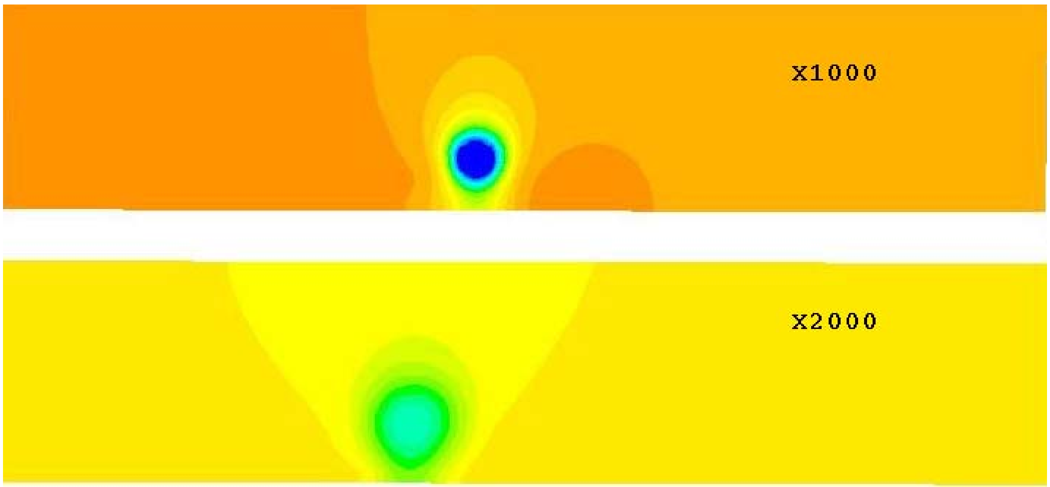

Figure 7: Pressure contours in planes downstream of the plate, hexahedral mesh.

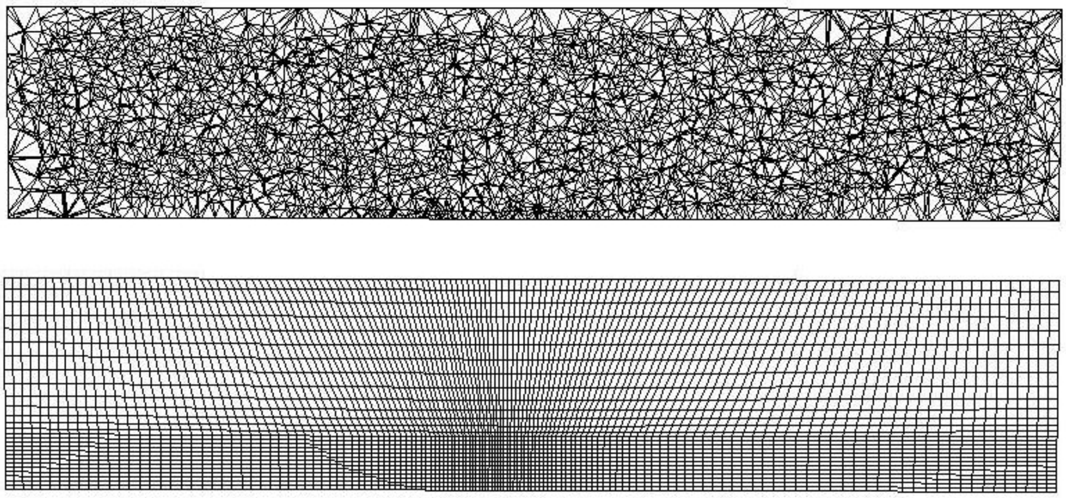

Figure 8: Grid planes at X2000

6. AERODYNAMIC DESIGN - C22 ENGINE COVER

Even well before the introduction of Albert, the Sauber Petronas team was relying heavily on CFD simulations. With the applications often pushed to the limits, a close cooperation with both software and hardware suppliers became necessary to succeed in delivering high quality data at the requested pace. Briefly described below is one such CFD project where the team produced several highly detailed CFD simulations during a very short time period. The complexity of these simulations even demanded some modifications to be made to the commercial CFD solver source code.

Early on in the 2003 season, it was recognized by the team that the C22 race car suffered from some aerodynamic shortcomings while cornering. The race car was found sensitive to changes in side-slip angle (yaw), and it was immediately understood that the rear-end of the car was losing too much downforce while going into low-speed corners. However, despite extensive wind tunnel testing the root of the problem remained somewhat unclear. The wind tunnel used at the time offered limited capabilities for testing at yawed conditions and only angles up to 4.5 degrees were achievable.

To gain further insights into this complex aerodynamic subject, it was decided to conduct a highly detailed CFD analysis. It had been demonstrated on several occasions, through both wind tunnel measurements and track testing, that only minor geometrical changes, e.g. to the front wing endplate or details in the front wheel assembly, could have a global impact on the overall vehicle aerodynamics, clearly revealing a strong aerodynamic interaction/coupling between the front-end and the rear-end of the car. To fulfil such demands, the final hybrid mesh used in this study reached almost 100,000,000 cells.

From this extensive database, valuable results were extracted that helped to better understand the cause of the vehicle yaw sensitivity. Clearly, and with no surprise, the front tire wakes were detrimental for the rear wing performance in yawed conditions. As indicated in Figure 9 and Figure 10, the oncoming flow conditions are significantly affected by the front tire wake being shed onto the rear wing. Furthermore, these CFD simulations revealed some very interesting flow characteristics that would have been virtually impossible to detect in a wind tunnel. Moreover, a few details of the underbody were redesigned based on the simulation results, which also translated into quantifiable performance gain on the race track.

These CFD findings were vital for decisions on the development directions, and a completely revised rear-end bodywork including more efficient deflectors were successfully developed in the wind tunnel during a very short time frame. The car with the new bodywork was first raced at the U.S. Grand Prix where the SAUBER PETRONAS pilots Heinz-Harald Frentzen and Nick Heidfeld finished in 3rd and 5th place, respectively.

This project was a good illustration on how state-of-the-art CFD technology can efficiently complement traditional experimental testing in a fast-paced design environment, and in effect, this became a “trigger” to initiate the Albert project.

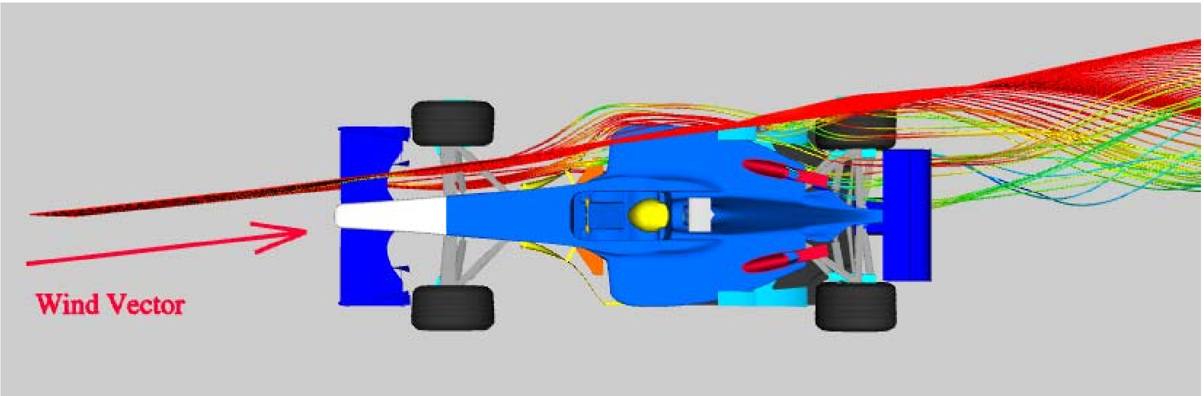

Figure 9: C22 Full Car CFD simulation at 6 deg side-slip angle.

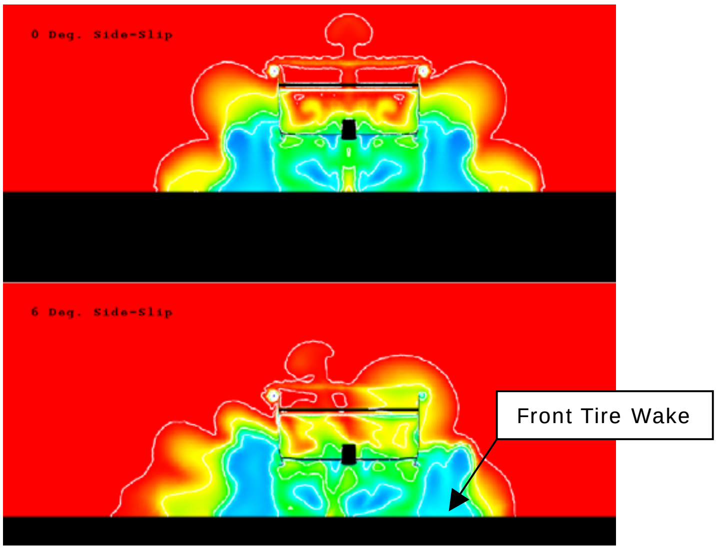

Figure 10: Total pressure iso-contours showing rear wing flow conditions. Full car CFD simulations of the SAUBER PETRONAS C22.

7. HIGH-END APPLICATIONS

Recent investments in hardware and CFD technology have opened up new possibilities in terms of advanced CFD applications at SAUBER PETRONAS. Even time accurate simulations of race car dynamics are now within reach. The team is already capable of delivering transient simulations, e.g. on overtaking manoeuvres (Figure 11). These kinds of time dependent flow simulations can ultimately bring new insights into the aerodynamic interactions of competing race cars running at various conditions.

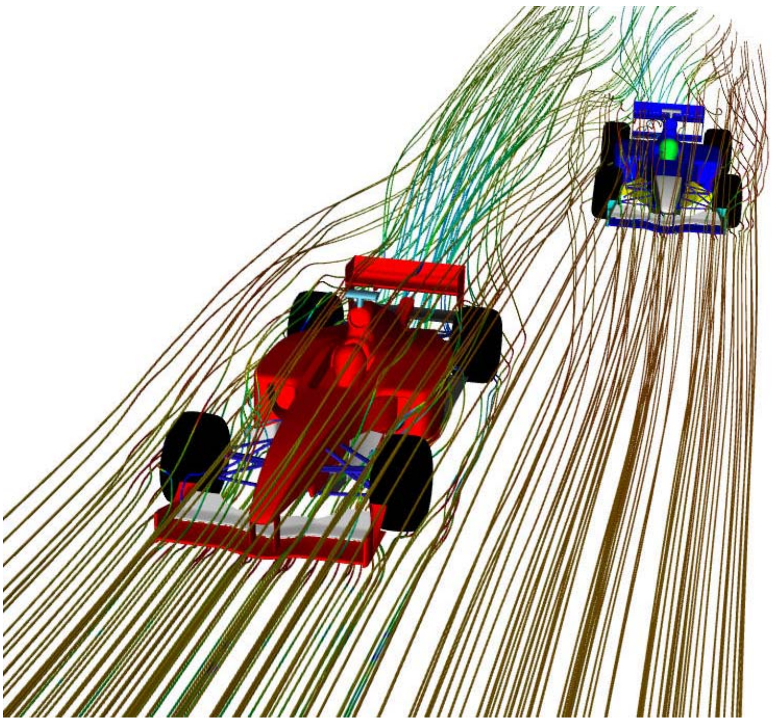

Figure 11: CFD simulation of an overtaking manoeuvre.

8. CONCLUSIONS

A thousand fold increase in computational capacity over the last five years has transformed the way SAUBER PETRONAS approaches and utilizes CFD and simulation technology in the vehicle development process. This leap has taken the level of complexity of simulation models from a very localized, nearly stand-alone approach with a compromised mesh resolution, to a massive full-car model with a much resolved mesh surrounding the whole car and the downstream vicinity. With sights set firmly on much further advanced applications in the near future, the supercomputer is unlocking the power of CFD.

REFERENCES

[1] Akanni S., Larsson T., Bienz C. , “Numerical Modelling of the Aerodynamic Flow Field about a Formula One Car”, Fluent User Group Meeting, Germany 2001.

[2] Bienz C., Larsson T., Sato T., Ullbrand B. “In Front of the Grid – CFD at SAUBER PETRONAS F1 Leading the Aerodynamic Development”, 1st European Automotive CFD Conference, Bingen, Germany, 2003