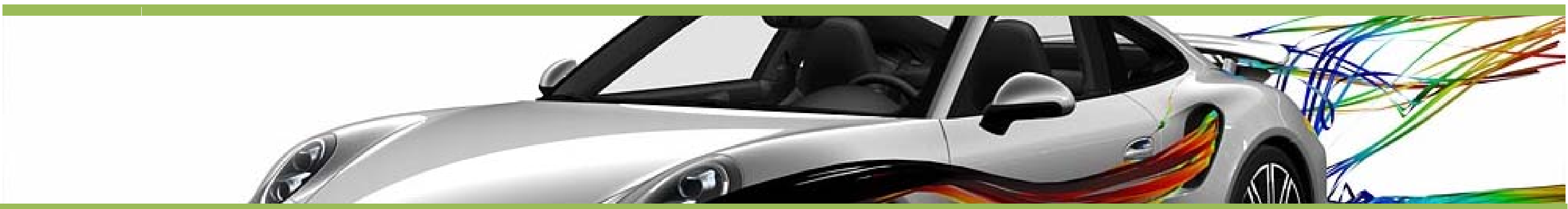
Automotive Simulation Center Stuttgart



2nd Cluster Workshop Vehicle Physics

Simulation Driven Design for CFD

Black box or detailed knowledge?

23 February 2016

AGENDA

9:00 WARM-UP
Arrival of the guests and welcoming them over a cup of coffee

9:30 INTRODUCTION ROUND
Introduction to the workshop

9:40 TAKE THE POLE POSITION - PART 1

KEYNOTE: Spectral/hp element, scale resolving modelling for high Reynolds number F1 Aerodynamics
Professor Spencer Sherwin - Imperial College London

Bringing the Complexity of CFD into the CAD World of an Engineer
Boris Marovic - Mentor Graphics (Deutschland) GmbH

Efficient methods for parametric shape optimization of flow channels
Ioannis Nitsopoulos - ISKO engineers AG

10:55-11:30 PIT STOP
Coffee break

11:30 TAKE THE POLE POSITION - PART 2

The influence of mesh characteristics on OpenFOAM simulations of the DrivAer model
Grigoris Fotiadis, Vangelis Skaperdas, Aristotelis Iordanidis - BETA CAE Systems S.A.

12:20-13:45 PIT STOP
Lunch break

13:45 TAKE THE POLE POSITION - PART 3

On the Need for Expertise when Using CFD - a Teacher's View
Ulrich H. Rist - Institute of Aerodynamics and Gas Dynamics University of Stuttgart

Below the Glossy Surface: Methods, Errors and Complexity in CFD
Prof. Gabriel Wittum - G-CSC, University of Frankfurt

14:35-15:00 PIT STOP
Coffee break

15:00 WIN THE RACE

Information about available subsidies

Presenting and discussing the submitted project appetizers

16:00 CHEQUERED FLAG
End of the workshop – there will be ample opportunities for further technical discussions afterwards.

Sponsored by Mentor Graphics

ABSTRACTS

Prof. Spencer Sherwin | Imperial College London

Spectral/hp element, scale resolving modelling for high Reynolds number F1 Aerodynamics

The use of computational tools in industrial flow simulations is well established. As engineering design continues to evolve and become ever more complex there is an increasing demand for more accurate transient flow simulations. It can, using existing methods, be extremely costly in computational terms to achieve sufficient accuracy in these simulations. Accordingly, advanced engineering industries, such as the F1 industry, is looking to academia to develop the next generation of techniques which may provide a mechanism for more accurate simulations without excessive increases in cost.

Currently, the most established methods for industrial flow simulations, including F1, are based upon the Reynolds Averaged Navier-Stokes (RANS) equations which are at the heart of most commercial codes. There is naturally an implicit assumption in this approach of a steady state solution. In practice, however, many industrial problems involve unsteady or transient flows which the RANS techniques are not well equipped to deal with. In order to therefore address increasing demand for more physical models in engineering design, commercial codes do include unsteady extensions such as URANS (Unsteady RANS), and Detached Eddy Simulation (DES). Unfortunately even on high performance computing facilities these types of computational models require significantly more execution time which, to date, has not been matched with a corresponding increase in accuracy of a level sufficient to justify this costs. Particularly when considering the computing restrictions the F1 rules impose on the race car design.

Alternative transient simulation techniques have been developed within research and academic communities over the past few decades. These methods have generally been applied to more academic transient flow simulations with a significantly reduced level of turbulence modelling. As the industrial demand for transient simulations becomes greater and the computer "power per $" improves, alternative computational techniques, not yet widely adopted by industry, are likely to provide a more cost effective tool from the perspective of computational time for a high level of accuracy.

In this presentation we will outline the demands imposed on computational aerodynamics within the highly competitive F1 race car design and discuss the next generation of transient flow modelling that the industry is looking to impact on this design cycle.

Boris Marovic, Mentor Graphics (Deutschland) GmbH | Frankfurt

Bringing the Complexity of CFD into the CAD World of an Engineer

Phrases such as Simulation Driven Design and Frontloading finding increasing use in the numerical simulation environment. However the actual design environment is far from being driven by the simulation. In most cases the simulation in such a scenario is slowing down the design. With the 3D CAD environment design changes can be created every few minutes and no traditional CFD software can provide that fast answers as to drive the next design any time close to the design cycle times.

With Meshing of most CAD generated models (not idealized models) taking up at least several man-hours (manual meshing) but in most cases rather man-days if not even weeks, the design cycle is abruptly interrupted if it were to wait for the next design suggestions based on the simulation results. Design and product development engineers are creating the new product designs, shapes and structures and often want to know if their design works or what and where they should improve. They also don't have the numerical knowledge of CFD experts to understand what residuals, Y+ and a broad range of turbulence models have to be considered for good result accuracy.

Often the solution is praised as scripted or "App-lified" simplifications of the traditional CFD tools for those engineers. In such a case the experts define scripts or create a simplified GUI for the engineer to enter their boundary conditions and get the results automatically. Those scripts or Apps however are often very limited in capability or application and don't offer any flexibility.

This presentation will show a different approach of a CAD embedded CFD solution which utilizes a special meshing technology that enables automatic meshing of most complex geometry and solver technologies that reduce the user input to their typical boundary conditions and do not require any turbulence model selection or tweaking in order to reach solver stability. Designs can therefore be analyzed in hours compared to days or weeks. Bringing engineers much closer to their desired design cycle compared to traditional tools but does not make them obsolete.

Ioannis Nitsopoulos | ISKO engineers AG | München

Efficient methods for parametric shape optimization of flow channels

An important requirement in the design optimization process is the variation of of geometry in each loop. Until now this was only possible in a limited way in an automated process, since the optimization needs a robust and flexible process which uses the potential of the existing design space.

This presentation provides an overview of shape changing methods based on CAD and network-based models and introduces a new efficient and practical method.

In the lecture the entire procedure, from the geometry description, on the integration of CFD simulation in an optimization process, through to the evaluation of the results is shown by the example of a flow channel.

Grigoris Fotiadis, Vangelis Skaperdas, Aristotelis Iordanidis | BETA CAE Systems S.A. | Greece

The influence of mesh characteristics on OpenFOAM simulations of the DrivAer model

Computational Fluid Dynamics is a sophisticated, continuously expanding field, providing its benefits to an ever increasing number of engineers across the industry. It offers great insight to complex flow problems that would otherwise be difficult, if not impossible, to gain. It has therefore become an integral part of the complete design process in all high-tech industries.

In this study external aerodynamics CFD simulations are performed using OpenFOAM on the three variants of the DrivAer model, a realistic geometry with details representative of current automotive designs. A thorough examination of the effect of different meshes on the solution convergence and accuracy is performed. These meshes differ in terms of generation process and time involved, their density and their quality. Different meshing approaches are followed using the pre-processor ANSA, ranging from standard hybrid penta and tetra meshes to hexa dominant and polyhedral ones. Other factors considered are the steady or transient approaches, as well as the importance of including the wind tunnel in the simulations to exactly match CFD and experimental results. Conclusions are derived with respect to the importance of the mesh, and the optimum pre-processing strategy that ensures robust automation as well as high fidelity CFD simulations with OpenFOAM.

Bettina Landvogt | scapos AG | Schloss Birlinghoven | Sankt Augustin

Using MpCCI to Solve Coupled CFD Problems arising in Automotive Design

Numerical simulation has become a widely accepted and established state-of-the-art method in the field of fluid dynamics, especially in the automotive design and development process. However, there are some applications where the fluid flow cannot be simulated realistically without taking other aspects into account: the deformation of spoilers or wings, for example, influences the fluid flow around the car, which in turn influences the deformation of the spoiler.

For these so-called fluid-structure interactions, and many other multi-physical applications, different simulation codes can be linked together in order to deliver a solution to the whole coupled problem.

Fraunhofer SCAI's code coupling environment MpCCI provides an independent interface between the most widely used specialized simulation codes. MpCCI takes care of the data interpolation, synchronizes the two simulation codes and exchanges data via fast socket communications.

Different examples from automotive applications will be presented:

the simulation of fluid-structure interactions is especially important for the design of wings and spoilers for racing cars, e. g. in Formula 1. The very big CFD model sizes with more than 100 million cells can be handled by MpCCI and the whole coupled simulation can be integrated into a batch system work flow.

Wheel spoilers of every day vehicles are usually made of very deformable plastic components and distort tremendously while driving. A coupled simulation can be used to find the optimal geometrical and material layout for these wheel spoilers.

The thermal management of cars is another important topic:

MpCCI can be used to couple a shell model of the thin underhood parts in RadTherm - a specialised and widely-used code for radiation, conduction and convection - with a standard CFD model of the flow around the car. Using full car models for the CFD and radiation simulation very accurate results can be produced.

To simulate the interaction between the air flow around a vehicle and the chassis and suspension, CFD codes can be coupled to Multi-Body-System Codes. An interesting application is a truck exposed to a strong side wind, e.g. while overtaking another vehicle or crossing a bridge. During the development of off-road vehicles a similar coupling with water instead of air can be employed to model a car driving through deep water.

Ulrich H. Rist | Institute of Aerodynamics and Gas Dynamics University of Stuttgart| Stuttgart

On the Need for Expertise when Using CFD - a Teacher's View

University teaching starts with the mathematical and physical basics of engineering, i.e., the necessary principles and the fundamental equations. At a higher level, numerical methods and more application specific problems (like boundary-layer flows, for instance) are presented. The primary idea behind this approach is conveying background knowledge not only to future users of software packages but also to future deciders. Knowing how a method works is essential to understand it and to foresee where it is applicable and where it might fail. The latter hopefully prevents costly pitfalls. In addition, expertise is always essential to judge the qualitative meaningfulness of new computational results.

A user's expertise will not deliver the perfect solution by itself but it will guide him to look more carefully at certain aspects and force him to carefully study the influence of critical parameters which another user (without that knowledge) would have overlooked.

In the present author's view the best way for building up this kind of expertise is excellent education followed by a year-long learning process using specific software packages. The university can only contribute to the first part of this process. Some examples related to turbulence modeling, prediction of laminar-turbulent transition and boundary layer separation and re-attachment will be used to illustrate the author's view that expertise is indispensable.

Prof. Gabriel Wittum, G-CSC, University of Frankfurt

Below the Glossy Surface: Methods, Errors and Complexity in CFD

Modern CFD software offers great Graphical User Interfaces (GUI) and visualization capabilites. Independent software vendors (ISV) invested a lot into the user interfaces. Easy to use GUIs and convincing visualization are decisive arguments for the ISVs to persuade the customer into bying the software.

Below the glossy surface, however, the core numerics are doing the computational work, which is the critical part of a simulation. This part is about methods, errors and complexity finally deciding on reliability and computational efficiency of the results. Often, this does not get the attention it actually merits to have, since a lot of expertise is necessary to judge this part of the story.

In the presentation, we explain the interaction of methods, errors and complexity, i.e. the critical part of the simulation. We further introduce an approach, how to reach reliability and efficiency on large-scale computations.