Status and Trends in Large Scale CFD Applications for Vehicle Design at Volvo Car Corporation

Andreas Borg

Volvo Car Corporation, Fluid Dynamics Center, Computational Fluid Dynamics
Dept. 96630, PVT 3 40531, Sweden
+46 31 3259527, aborg1@volvocars.com

Abstract. The role of Computational Fluid Dynamics, or CFD as an important tool for solving fluid dynamical issues and designing vehicle sub systems in the automotive industry is yet to grow. In this paper we discuss the current use of Large Scale CFD computations at Volvo Car Corporation. A short historical survey is given on the development of meshing and computational techniques to solve problems with CFD for aerodynamics, thermodynamics, climate control, soiling and aerodynamically induced noise at Volvo Cars. We briefly go through the status of CFD computations for the attributes named above. Finally we discuss some of the main challenges for CFD in the vehicle industry in the near future.

1 Introduction

At Volvo Car Corporation Computational Fluid Mechanics (CFD) has played an active part of the vehicle design and development process for many years. Since the early nineties, CFD has been use to support aerodynamic development of vehicles in close cooperation with physical testing on clay models in model and full scale wind tunnels. In the beginning early studies of mesh quality, mesh resolution and choice of turbulence model for accurate prediction of stagnation and base pressure were done, see [1-2]. Later in the nineties the importance of wheels, wheel housings and underbody, was investigated. Conclusions from investigations regarding these questions and results from CFD simulations of cars with detailed underbodies can be found in [3-4]. In the mid-nineties the group started carrying out computations on the vehicle climate systems. CFD methods supporting thermodynamics, especially cooling performance, were started in 1997. The methodology for cooling performance computations has undergone extensive development, and is today a robust and accurate method for modeling of one-phase heat exchangers in complete vehicle computations [5]. In 2001 simplified methodologies for handling soiling of the exterior and chassis were developed. These techniques were then applied on other areas like snow packing in engine air filters. Today CFD are used at Volvo Cars for these applications regularly in project work. They are used for concept judgment and concept improvement in the early phases of the project and for preliminary verification and failure detection in later stages in the car programs. The vehicle industry is continuously shortening their product development cycles. This translates in to shorter lead times for model build and analysis time. Techniques to handle this are discussed in this paper.

2 Application Areas

2.1 Aerodynamics

Today aerodynamic simulations performed at Volvo Cars are mainly of three types

1. Analysis of exterior design concepts with simplified underbody in the preconcept phase
2. Analysis of exterior design details like mirrors with simplified underbody
3. Complete vehicle aerodynamic simulation with detailed underbody and engine bay.

Models of type 1 consist of models with mesh sizes of around 8 million cells. The boundary layers are resolved by using 5 prismatic layers. A high Reynolds number formulation of the k-ε model with wall functions is employed. The model is run in steady state mode. Typically these runs are performed on 16 CPUs on a Linux cluster from IBM. Models of type 2 are performed on meshes in the range of 10-20 million cells depending of what details that are resolved in the simulation. Models of type 3 are performed on meshes with 20-30 million cells. They include modeling of rotating wheels and the porous resistance from heat exchanger and modeling of the cooling fan. Typical simulation times are 24 h for type 1, 48 h for type 2 and 64-72 h for type 3 on 16 CPUs.

2.2 Thermodynamics

The majority of thermodynamic simulations performed at Volvo Cars are aimed at controlling and improving cooling performance. The computational models typically consist of a full vehicle exterior with a very detailed front geometry and engine bay. The underbody of the car is usually simplified. To achieve high quality predictions boundary conditions like heat release and cooling package performance as well as fan characteristics have to be handled accurately. Typical model sizes range from 3-8 million cells. These simulations which predict top hose temperature are usually run on 8 CPUs on a PC Linux Cluster over night. Other large scale thermodynamic simulations are break cooling simulations and engine bay temperature distribution.

2.3 Climate Control

Simulations for climate control are done to analyze the performance and improve solutions of the complete HVAC with duct systems and ventilation nozzles. Nozzle optimizations are done with complete vehicle cabin geometry and models of typical passengers inside the compartment. It is important that the ventilation nozzles are designed in such a way that the air can be directed away from the body to reduce draft, as well as be able to direct it towards the body for fast cool down. The floor nozzles should be designed such that the feet are heated in a comfortable way. The computational models are run in steady state with the k-ε model. Typical mesh sizes range from 3-8 million cells and typically run over night on an 8 CPU PC Linux cluster.

2.4 Contamination

Simulations of contamination or soiling are fairly new at Volvo Cars. Currently Lagrangian particle tracking is used to analyze body side and chassis soiling. Another application is snow ingestion into the engine air filter when driving in winter climate. The computational models consist of 8-12 million cell meshes of a complete vehicle. The particle tracking is performed as a post processing step after the air flow field has been computed.

2.5 Aerodynamically Induced Noise

Designing exterior shapes to reduce wind noise is an important area which is currently under examination by a Volvo Car Corporation. Simulations with LES have been made for simplified mirror shapes and are currently tested on real car geometries. Typical mesh sizes are 10 million cells and the simulations are run for several weeks on a 16 CPU Linux cluster.

3 Challenges

The most dominant trend in CFD simulations for complete vehicles in vehicle industry is to cut lead-time while maintaining a sufficient level of accuracy. Although the accuracy of the type of simulations outlined in the previous section is far from perfect the current most urgent need is to cut lead-time. A breakdown of the components in the lead-time of performing many of these simulations show that geometry preparation and surface and volume meshing is the most time consuming part of the process. The reason for this is that the complete car geometry is very complicated and often the CAD is "dirty" from a mesh generation point of view. Great effort is currently spent on improving this part of the process. The following list summarizes how we work with this issue at different levels in the CAD->mesh chain.

1. If possible organize CAD in an efficient way and use solid models with history tree to be able to remove unnecessary CAD details for CFD analysis automatically.
2. Use Cartesian shrink-wrap techniques on non watertight geometries to create closed tessellated geometry representations that can be surfaced meshed or volume meshed directly with Octree based meshing tools
3. Use volume meshers that does not require high quality surface meshes or create surface meshes with an Octree based meshing tool.

By using these kind of volume/surface meshers the mesh topology gets simpler and requires less manual work. However due to the isotropic nature of meshes created by Octree meshers the mesh sizes grow very fast. Thus this type of approach by necessity leads to larger computational models compared to more manual designed meshes where grid points can be clustered more clever. However it seems unavoidable that this is the way to go for increased automatization and shorter lead time. Naturally this approach has to be supplemented by high performance clusters to carry out computations on these meshes. For example models for complete vehicle aerodynamical simulations are likely to reach mesh sizes close to 50-100 million cells within the next few years.

On the physical modeling side Volvo Cars put effort in to be able to predict sources of wind noise by using large eddy simulation as described above. Another active field were we improve rapidly is more detailed multiphase models to model water run off. Both these applications are very computer demanding and will require significantly more computer power to be extensively used in project work

References

1. Axelsson, N., Ramnefors, M. and Gustafson, R.: Accuacy in Computational Aerodynamics Part 1: Stagnation Pressure, SAE Technical paper 980037, SAE International Congress & Exposition, Detroit, USA, 1998
2. Perzon, S., Sjögren, T. and Jönson, A.: Accuracy in Computational Aerodynamics Part2: Base Pressure, SAE Technical Paper 980038, SAE International Congress & Exposition, Detroit, USA, 1998
3. Sebben, S.: Numerical Flow Simulations of a Detailed Car Underbody, 2001-01-0703, SAE International Congress & Exposition, Detroit, USA, 2001
4. Sebben, S.: Numerical Simulations of a Car Underbody: Effect of Front-Wheel Deflectors, 2004-01-1307, SAE International Congress & Exposition, Detroit, USA, 2004
5. Jerhamre, A. and Jönson, A.: Development and Validation of Coolant Temperature and Cooling Air Flow CFD Simulations at Volvo Cars, SAE 2004-01-0051, SAE International Congress & Exposition, Detroit, USA, 2004