CFD IN ROAD VEHICLE AERODYNAMICS

Werner Seibert
Fluent Inc.

1. Introducing CFD to Road Vehicle Aerodynamics

1.1 Motivation

Computational fluid dynamics (CFD) was and is still an emerging technology. It is the merger of the classical branches of theoretical and experimental science, with the infusion of the modern element of numerical computation. The progress in computational aerodynamics since it~~'~~s dawn about 40 years ago [1] has been extraordinary. Much of this progress has been driven by the phenomenal increases in digital computing speed. The power of digital computing has transformed research and engineering in fluid mechanics, just as it has in virtually all fields of human endeavour.

Computer simulations of fluid flow processes are now used to design, investigate, optimize and operate engineered systems and to determine the performance of these systems under various conditions. Computational fluid dynamics simulations are also used to improve understanding of fluid physics and chemistry, such as turbulence and combustion, and to aid in weather prediction and oceanography. CFD simulations today are widely conducted in industry, government, and academia.

Road vehicles, as vehicles in general, offer a wide field of applications for fluid flow analysis. Thermal management simulation for under-bonnet components including radiation heat transfer, in-cylinder computations needing moving/deforming meshes along with spray-, ignition- and combustion models, multiphase flow modelling including phase-change for analyzing climate control, defrosting and defogging and free surface modelling to simulate tank filling and sloshing are just a few application areas to be mentioned. However, the most challenging topic has been and still is the accurate prediction of the external aerodynamics of a road vehicle.

More recently, ~~also~~ the investigation of ~~a~~ closely related phenomena, i.e. aero-acoustic mechanisms and effects, has become feasible. Improved physical models along with powerful computer resources are now allowing automotive users to study noise generated by external airflows past rear view mirrors or open sun-roofs or window sills, or by internal flows through ducts, valves, manifolds as well as sounds originated from rotating fans.

1.2 Previous History

In the early days of computer-aided simulation of flow, such methods were used mainly in aircraft development and during projection of industrial construction~~s~~. The first record of its application to vehicle-like configurations are from the early 1980's [2, 3, 4], but for the mainstream in aerodynamic design and optimization of car bodies, it did not have any relevance until much later.

The reason for this may be attributed to the strong limitations in available geometry modelling. Indeed, it was possible to prepare vehicle-related basic bodies for the available structured flow solvers, however details of the geometry, which ~~in a sum~~together have a considerable influence to flow around a vehicle, could not be included into such investigations. Methods based on potential theory (so called Panel-Methods), sometimes coupled to integral boundary layer methods, only needed a fairly simple discretization restricted to the surfaces, but were usable only for areas of attached flow. The subsequent Euler methods were working with structured volume grids (Finite-Volume-Methods), but they could not yet handle the viscous effects within a flow field.

During all these early years of applying CFD analysis to automotive design, the preparation of a suitable discretization has always been the most problematic part. Typically, vehicles of any kind and the components inside them have very complex shapes. Multipurpose CAD systems, used to supply geometrical definitions, have no functionality to check or establish topological relationships. But those are absolutely necessary to form the closed and well defined domains for flow analysis. Therefore an analyst usually starts with a set of feature curves and surface patches, that cover most of the configuration, but gaps, overlaps and discontinuities are present [5].

Missing the availability of a well defined closed surface description with a proper topology is still a problem today. Although recent developments in CAD systems brought some relief through volume based formulations – where the surfaces are closed inherently – such systems are still not yet capable to handle full vehicle body descriptions. As a consequence most of the aerodynamicists still have to deal with the older surface based versions and all of their limitations.

Traditionally, large amounts of time have to be spent on cleaning up and completing flawed geometries - a tedious and difficult task, requiring the talents of a wood carver. Additionally, at least in the past, there was the need to design a suitable block topology or decomposition of the domain as required by the available solvers - another difficult task which had to be done manually and without much computer aid.

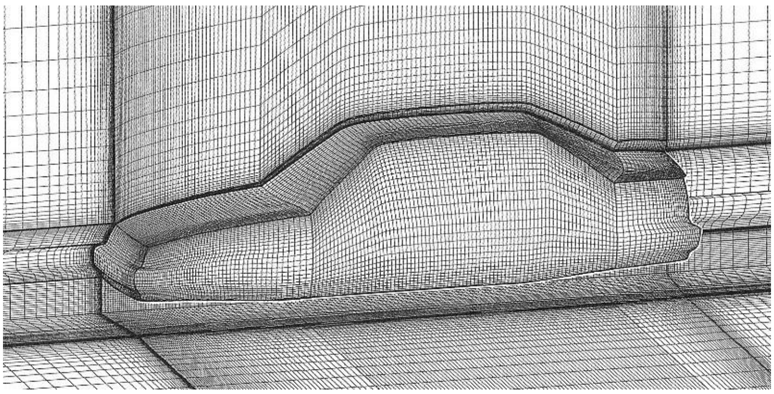

Figure 1
Traditional block structured grid around a 'monolithic' car shape, typical arrangement for CFD during the 80's [6].