

CFD Simulation with Automatic Mesh Refinement for the Flow around Simplified Car Models

Emmanuel Guilmineau and Jeroen Wackers
Ecole Centrale De Nantes

ABSTRACT

This paper presents simulations for the prediction of the flow around a passenger vehicle. The flow solver used is ISIS-CFD developed by the Numerical Modeling Group of the Fluid Mechanics Laboratory of Ecole Centrale de Nantes. The CFD simulation is carried out with the Explicit Algebraic Stress Model (EASM) turbulence model.

For improved precision, the recently developed automatic adaptive grid refinement procedure of the flow solver is used. This procedure is built for unstructured hexahedral grids and naturally supports both isotropic and anisotropic grid refinement to keep the grid sizes small for 3D simulations. The refinement criterion used to indicate the locations of grid refinement is based on the second spatial derivatives of the pressure. This criterion is efficient for flows with strong local vorticity.

Three models are used to validate this approach: the Ahmed model, the Willy model and a pickup truck model. The main goal of the simulation is the accurate computation of the trailing vortices of these models. The automatic grid refinement is used to provide the local high grid density needed to create and track these vortices. Comparisons between the numerical results and experimental data are reported.

CITATION: Guilmineau, E. and Wackers, J., "CFD Simulation with Automatic Mesh Refinement for the Flow around Simplified Car Models," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(1):2012, doi:10.4271/2012-01-0589.

INTRODUCTION

The external aerodynamics of a car determines many of its relevant aspects of an automobile such as stability, comfort and fuel consumption at high cruising speeds [1]. The flow field around a vehicle is characterized by separations. Two primary types of separations are to be distinguished: the wake and longitudinal vortices; these separations influence the drag. As passenger cars operating on roads are commonly exposed to side wind conditions, aerodynamic development of such vehicles cannot be limited to the zero yaw condition. In this case, for numerical simulation, it is difficult to have a mesh adapted to the flow. So, an automatic adaptive mesh is a good solution to follow the flow.

Automatic adaptive grid refinement is a technique for optimizing the grid in the simulation of fluid flow, by adapting the grid to the flow as it develops during the simulation. This is done by locally dividing cells into smaller cells, or if necessary, by merging small cells back into larger cells in order to undo earlier refinement. Grid refinement is effective for flows that have localized structures; fine cells can be used to keep the error near these structures low and to

get good resolution, while coarse cells are used in the rest of the domain to reduce the total costs for the simulation.

An automatic mesh adaptation method has been developed for ISIS-CFD, the flow solver developed by the CFD group of the Fluid Mechanics Laboratory. The goal of the development was to produce a method that can be used in daily practice for all the different applications of this code. Also, it should be easily combined with the existing capabilities of ISIS-CFD, such as complex geometries, steady and unsteady flows, and grid deformation for the imposed or resolved motion of bodies. Furthermore, the method needs to be easily maintainable as the code develops over the years. For all these reasons, the mesh adaptation method has been made as general as possible: it allows unstructured grids, directional refinement to keep the size of 3D refined grids low, and derefinement of refined grids to enable unsteady flow simulation. The method is flexible to allow the easy changing of refinement criteria (that indicate where the grid is to be refined) and, like the flow solver, it is completely parallel. The refinement method is integrated in the flow solver.

This paper describes the application of the grid adaptation method to three different test cases. First, this technique is

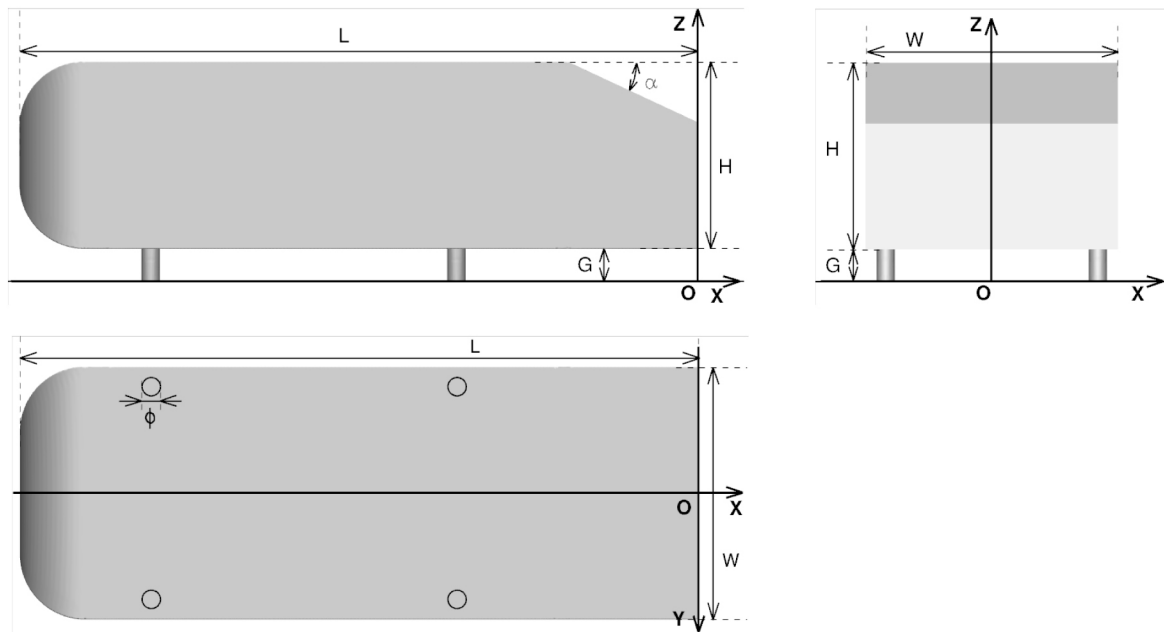


Figure 1. Ahmed model. Dimensions are in mm.

used to simulate the flow around the Ahmed body which has been studied experimentally for the first time by Ahmed et al. [2]. The second test case is the simulation of cross wind around the Willy model which is a simplified car model. This model was first used in an experimental study of Chometon et al. [3]. The last simulation is around a more realistic car: a pickup truck which was investigated experimentally by Al-Garni et al. [4].

CAR MODELS AND COMPUTATIONAL PARAMETERS

Three car models are used in this paper. The first is the Ahmed body, the second is the Willy model and the last is a more realistic car: a pickup. These models are now detailed.

THE AHMED MODEL

The Ahmed body is presented in Figure 1. The slant angle α is adjustable and is the main variable model-parameter in the experimental investigations. Here, only the 25° slant angle is investigated. The length of the model is $L = 1044$ mm, the width is $W = 369$ mm, the height is $H = 288$ mm. The ground clearance is $G = 50$ mm and the diameter of the four feet, which are used to secure the model to the floor of the wind tunnel, is $\phi = 30$ mm. The reference axes (X,Y,Z) are linked to the model. The origin of these axes is the point O located on the floor of the wind tunnel at the base of the model and in the symmetry plane of the model, see Figure 1.

The upstream velocity is $V_0 = 40$ m/s. The Reynolds number, based on the length of the model, is $Re = 2.78 \times 10^6$.

The computational domain starts $2L$ in front the model and extends to $5L$ behind the model. The cross-section of the computational domain is 1.87 m and its height is 1.4 m. This

cross-section, which gives a blockage ratio of 4.3%, is identical to the open test section of the wind tunnel used in the experiments of Lienhard and Becker [5]. The boundary conditions used for this computational domain are no slip walls for the Ahmed body geometry and the floor of the computational domain, while the roof and the side walls of the computational domain are treated as slip walls. At the inlet, the uniform velocity V_0 is imposed while at the outlet boundary, the pressure is imposed.

The mesh is generated using HEXPRESS™, an automatic unstructured mesh generator developed by NUMECA International. This software generates meshes containing only hexahedrons. The initial mesh is composed of 8 million nodes with approximately 7.8 million cells. The Ahmed body is described by 166,000 faces. Unsteady flow is computed; the time step used is 0.001 s.

THE WILLY MODEL

Numerical simulations are performed on the square-back Willy model, see Figure 2, which was designed to satisfy the following criteria:

- The geometry is realistic, comparable with a real vehicle,
- The plane underbody surface of the model is parallel to the ground,
- The separations are limited to the region of the base for a moderate yaw angle, i.e. $\beta = 10^\circ$.

The length of the model is $L = 675$ mm, the width is $W = 240$ mm, the maximum height is $H = 192$ mm and its reference area is the maximum cross section $S_{ref} = 41,791$ mm². The ground clearance is $G = 29$ mm and the diameter of the four feet (f), which are used to secure the model to the

floor of the wind tunnel, is $\phi 1 = 20$ mm. A cylinder (c) with a diameter $\phi 2 = 40$ mm is used in experiments to protect the pressure tubes passing from the pressure taps to the multimanometer. Other dimensions are defined in Guilmineau and Chometon [6]. The (Pt) curve indicated in the Figure 2 is the position where the measurements of pressure will be used for comparison with experimental data.

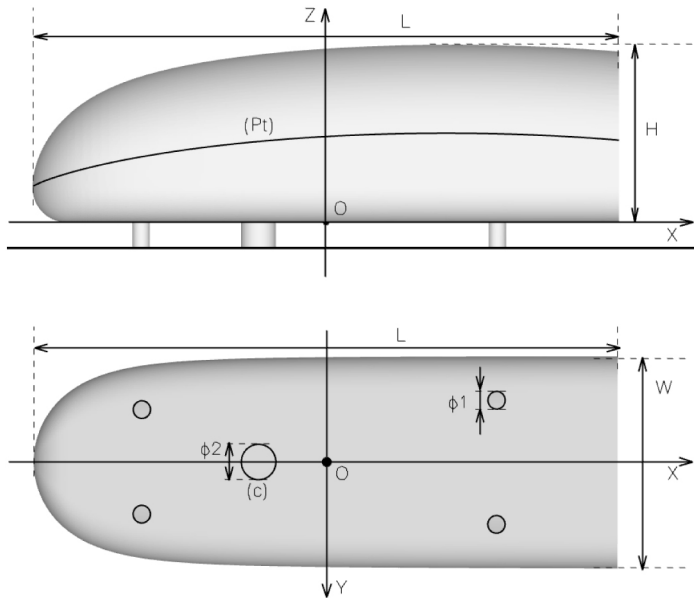


Figure 2. Willy model.

Two coordinate systems are distinguished. One is wind-tunnel fixed (X_o, Y_o, Z_o), X_o being parallel to the upstream velocity V_o . The other one is model fixed (X, Y, Z). The origin of the axes lies at point O located on the floor of the model at 330 mm behind the front of the model, see Figure 2. This point O is the center of rotation of the model.

The upstream velocity is $V_o = 20$ m/s. The Reynolds number, based on the length of the model, is $Re = 900,000$. Cross flow is simulated by turning the Willy model with respect to the upstream velocity by an angle β , called yaw angle. The value of this angle is positive when the right side of the car is to windward. Here, only the yaw angle $\beta = 30^\circ$ is investigated.

The computational domain starts at $3.9L$ in front of the model and extends to $5.4L$ behind the model. The width of the domain is $3L$ and its height is $1.6L$. This cross-section is identical to the open test section of the wind tunnel used in the experiments [6]. The boundary conditions used for this computational domain are no slip walls for the Willy body geometry while a wall function is applied at the floor of the computational domain. Thus, for the walls of the Willy model, we use the low Reynolds number turbulence model. At the outlet, the pressure is prescribed, while for the other boundaries a Dirichlet condition for the velocity and turbulence is employed.

The mesh, generated using HEXPRESSTM, is composed of 5.4 million nodes with approximately 5.2 million cells.

The Willy body is described by 141,000 faces. The time step used is 0.006 s.

THE PICKUP MODEL

The generic extended cab pickup model used in this study represents a 1:12 scale of the full-scale pickup truck. The model is designed with smooth underbody, enclosed wheelwells and without opening for cooling airflow. A schematic view of the pickup truck with dimensions is depicted in Figure 3. The length of the model is $L = 0.432$ m, the width is $W = 0.152$ m, and the height is $H = 0.148$ m. The maximum cross section area is 0.0195 m². The coordinate system is attached to the front tip of the model in the symmetry plane as shown in Figure 3.

The inlet velocity is $V_o = 30$ m/s. The Reynolds number, based on the length of the pickup, is $Re = 864,000$.

The computational domain starts at $2.6L$ from the origin in front of the model and extends to $5L$ behind the model. The section of the computational domain is identical to the test section of the wind tunnel used for the corresponding measurements [4], which gives a blockage ration of 5.2%.

For the surface of the pickup, a no-slip boundary condition is used. For the walls of the wind tunnel, a wall function is used. This treatment of the walls of the wind tunnel allows to take into account the effects of blocking. In addition, there is no separation on these walls. So, this type of boundary layer is sufficient. It is not necessary to use the version of the low Reynolds number turbulence model. At the inlet boundary, velocity and turbulent quantities are imposed while at the outlet of the computational domain, the pressure is prescribed.

The mesh, generated using HEXPRESSTM, is composed of 12.5 million nodes with approximately 11.5 million cells. The Willy body is described by 695,000 faces. The time step used is 0.0002 s.

THE ISIS-CFD FLOW SOLVER

ISIS-CFD, developed by the CFD group of the Fluid Mechanics Laboratory and available as a part of the FINETM/Marine computing suite, is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) method. The solver is based on the finite volume method to build the spatial discretization of the transport equations. The unstructured discretization is face-based, which means that cells with an arbitrary number of arbitrarily shaped faces are accepted. A second-order backward difference scheme is used to discretize time which is treated in a fully implicit manner. Thus, there is no stability limit on the CFL number. This solver can simulate both steady and unsteady flows. A detailed description of the solver is given by Duvigneau and Visonneau [7] and Queutey and Visonneau [8].

The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. In the case of turbulent

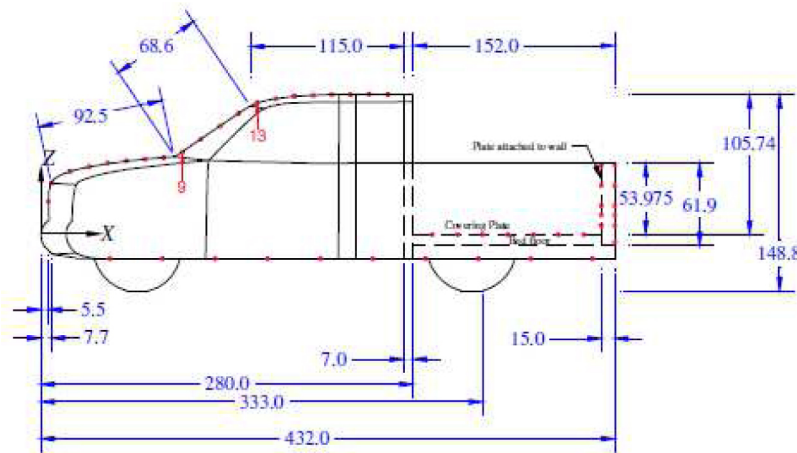


Figure 3. Pickup model. Dimensions are in mm.

flows, transport equations for the variables in the turbulence model are added to the discretization. Free-surface flow is simulated with a multi-phase flow approach: the water surface is captured with a conservation equation for the volume fraction of water, discretized with specific compressive discretisation schemes, see [8].

The method features sophisticated turbulence models: apart from the classical two-equation $k-\epsilon$ and $k-\omega$ models, the anisotropic two-equation Explicit Algebraic Stress Model (EASM), as well as Reynolds Stress Transport Models, are available, see [9] and [7]. The technique included for the 6 degree of freedom simulation of motion is described by Leroyer and Visonneau [10]. Time-integration of Newton's laws for the motion is combined with analytical weighted or elastic analogy grid deformation to adapt the fluid mesh to the moving body. Furthermore, the code has the possibility to model more than two phases. For brevity, these options are not further described here.

REFINEMENT TECHNIQUE

During a flow calculation with adaptive grid refinement, the refinement procedure is called repeatedly to keep the grid permanently adapted to the developing solution. Globally, the method works as follows: the flow solver is run on an initial grid for a limited number of time steps. Then the refinement procedure is called. If a refinement criterion, based on the current flow solution, indicates that parts of the grid are not fine enough, these cells are refined and the solution is copied to the refined grid. On this new grid, the flow solver is restarted. Then the refinement procedure is called again, to further refine or to derefine the mesh. This cycle is repeated several times. When computing steady flow, the procedure eventually converges: once the flow starts to approach a steady state and the grid is correctly adapted to this state according to the refinement criterion, then the refinement procedure keeps being called, but it no longer changes the grids. As the mesh generator creates only hexahedral cells,

the refinement strategy has been developed for this type of cells.

REFINEMENT DECISION

For maximum flexibility and to reduce the chance of errors, the code is divided in three separate parts that exchange only minimal information between them: the calculation of the refinement criterion, the refinement decision, and the actual (de)refinement. The refinement criterion indicates where the grid is to be refined; it can be based on any desired aspect of the flow solution. To permit the user choice of refinement criteria and the easy incorporation of new refinement criteria in the code, the criterion is computed as a field variable (comparable to the velocity or the pressure). It does not depend explicitly on the type or the orientation of the cells. A more detailed description of the refinement criterion computation is given in a following section.

In the second step, this refinement criterion is transformed into the decision of which cells to refine or to derefine. While this decision may depend on the type of the cells, it does not depend on the specific way in which the refinement criterion is calculated. It remains the same for any refinement criterion.

During the refinement decision step, the decision in each cell is adapted to its neighbour cells. To guarantee the quality of the mesh, extra cells may need to be refined, or derefinement of cells may be prevented. The most important quality criteria are given in Figure 4. A face of a cell may not be divided two times, which would cause too great differences in the sizes of its neighbour cells. And the angle between face normals and lines cell centre - face centre may not be too great: this reduces the quality of the state reconstruction at the faces, as indicated by Queutey and Visonneau [8].

Finally, specific measures are taken in order to protect the wall-aligned boundary layer meshes, if these are present. In general, to keep a good grid quality for these high-aspect