



HAL
open science

Anisotropic mesh refinement in automotive flow and hybrid rans-les models

E Guilmineau

► **To cite this version:**

E Guilmineau. Anisotropic mesh refinement in automotive flow and hybrid rans-les models. Direct and Large-Eddy Simulation 14, Apr 2024, Erlangen, Germany. hal-04785854

HAL Id: hal-04785854

<https://hal.science/hal-04785854v1>

Submitted on 15 Nov 2024

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Public Domain

ANISOTROPIC MESH REFINEMENT IN AUTOMOTIVE FLOW AND HYBRID RANS-LES MODELS

E. Guilmineau¹

¹LHEEA, CNRS UMR 6598
Centrale Nantes, France

emmanuel.guilmineau@ec-nantes.fr

INTRODUCTION

A prediction of aerodynamic coefficients on road vehicles is crucial for an efficient design and optimization process. One critical aspect of CFD modeling is the selection of turbulence models, which play a important role in accurately capturing the complex flow phenomena around vehicles. The objective of the Automotive CFD Prediction Workshop is to assess the predictive capability of CFD codes for road-cars geometries. The third edition of this workshop held at Barcelona in 2022 and two test-cases were studied: the Windsor body at yaw and the notchback version of the DrivAer.

For this workshop, several meshes were generated by the organizing committee. But the solution was not mesh-converged, despite the fine mesh generated, 197 million cells. One solution may be to use automatic grid refinement and vary its parameters to converge into a mesh. Automatic grid refinement algorithms dynamically adapt the mesh based on the local flow conditions, concentrating computational resources where they are most needed. This adaptive approach allows for a more accurate representation of critical flow features without unnecessarily refining regions where the flow is relatively simple.

TEST CASE

In this paper, the so-called Windsor is considered, see Figure 1. This model preserves the dimensions and aspect ratio of the Ahmed body but also presents a slanted front-end that is more representative of passenger cars. The model is 1.04425 m in length (L), 0.288 m in height (H) and 0.290 m in width (W), supported by four cylindrical struts of 50 mm in diameter. The ground clearance (G) is 0.050 m. The frontal area is rounded to be 0.112 m². The reference length used for pitching moment is the wheelbase 0.6375 m. The model has its origin on the ground plane, in the symmetry plane midway between the feet. The coordinate system has X in the streamwise direction, Z upwards and hence positive Y is towards the right of the model. The model is yawed by -2.5 degrees around the Z -axis. The forces and moments are in the coordinate system of the yawed model. The inlet velocity condition is $U_\infty = 40$ m/s which corresponds to a Reynolds number, based on the vehicle length, of 2.9×10^6 . Measurements were taken at the Loughborough University wind tunnel by Varney [1].

FLOW SOLVER

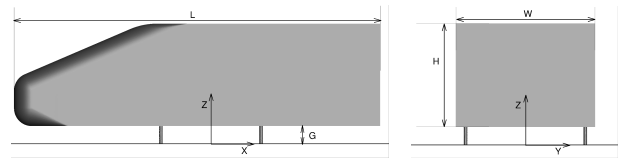


Figure 1: Windsor model squareback.

The in-house solver ISIS-CFD developed by CNRS and Centrale Nantes, also available as a part of the FINETM/Marine computing suite worldwide distributed by Cadence Design Systems, is an incompressible multiphase unsteady Reynolds-averaged Navier-Stokes (URANS) solver mainly devoted to marine hydrodynamics. It is based on a fully-unstructured (face-based) finite volume discretization with specific functionalities needed for multiphase flows [2, 3].

The method features several sophisticated turbulence models: apart from the classical two-equation $k-\epsilon$ and $k-\omega$ models, the anisotropic two-equation Explicit Algebraic Reynolds Stress Model (EARS_M), as well as Reynolds Stress Transport Models (RSTM), are available [4, 5]. Hybrid RANS/LES turbulence models based on Detached Eddy Simulation (DES-SST, DDES-SST, IDDES-SST) are also implemented and have been thoroughly validated on automotive flows characterized by large separations [6] and ships at steady drift [7]. All models are available with wall-function or low-Reynolds near wall formulations.

Moreover, the solver accepts sliding and overset grids and features an anisotropic adaptive grid refinement functionality [8, 9] applied to unstructured hexahedral meshes. The automatic grid refinement performs by local division of meshes. The decision where to refine is based on a metric refinement criterion, a tensor field computed from the flow. The tensor is based on second derivatives of the flow variables. The refinement criterion is the flux-component Hessian. The grid is refined until the dimension of each cell satisfy the threshold Tr .

NUMERICAL SIMULATION SET-UP

The computational domains extends from $X = -5$ m to $X = +6$ m. The model nose is located at $X = -0.56$ m and the base at $X = +0.48$ m. The width and height of the domain matches the wind tunnel for which the cross section is 1.92 m wide \times 1.32 height.

In this paper, two types of mesh are used. The first mesh is

generated by the organizers of the workshop AutoCFD 3 using the trimmer mesh and prism layer approach from Simcenter STAR-CCM+. Three grids are created: baseline g2, coarse g1 and fine g3. All three meshes have the same wall normal grid spacing and prism layer thickness and vary the number of cells by adjusting all other cell dimensions consistently. The second mesh is generated by using Fidelity™. This mesh is similar to the coarse mesh proposed by the organizers of the workshop. Then the automatic grid refinement is used from this mesh. To obtain several meshes, the value of the threshold Tr is modified which is in the range $Tr \in [0.4L, 0.025L]$ for coarse to fine grids.

RESULTS

Figure 2 shows convergence of the aerodynamic coefficients, drag coefficient see Figure 2a and lift coefficient see Figure 2b, for both sets of meshes. The characteristic mesh size is computed as $h = N^{-1/3}$, where N is the number of cells. A first observation is that all simulations underestimate the coefficient obtained in the experiments. These results are consistent with those presented during the workshop using the same turbulence models. A second observation is the converged solution for the aerodynamic coefficient depends on the turbulence model. With the $k-\omega$ SST turbulence model, both mesh types converge to the same solution. However, the solution obtained with automatic mesh refinement converges much faster. The difference in drag between the two final solutions is 0.8%. However, only 15.1 million cells, mesh $Tr = 0.1L$, are used with the adapted mesh instead of the 197.5 million cells, mesh g3, of the committee mesh. With the DDES-SST and IDDES-SST models, the solution is not yet converged with the committee meshes while using the automatic grid refinement, the solution converges towards a mesh with 24.7 million cells for the DDES-SST and 27.8 million cells for the IDDES-SST, mesh $Tr = 0.2L$. The use of automatic mesh refinement allows to converge faster in mesh.

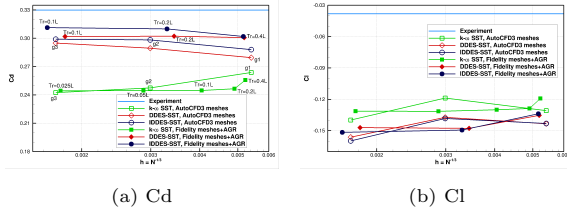


Figure 2: Grid convergence of aerodynamic coefficients.

Figure 3 presents a comparison of the base pressure coefficient, Cp . For a given turbulence model, the solution is mesh-independent. The flow predicted with the $k-\omega$ SST turbulence model is not in agreement with the experiments. The shape of the RANS contours is different from the experiment and it is indicative of a toroidal vortex structure in the wake. With the hybrid RANS/LES models, the base pressure is in agreement with the experiment. However, the Cp level on the leeward side is overestimated.

REFERENCES

[1]Varney, M. : Base drag reduction for squareback road vehicle, *PhD thesis, Loughborough University* (2020).
 [2]Leroyer, A. and Visonneau, M. : Numerical methods for RANSE simulations of a self-propelled fish-like body, *Journal of Fluids and Structures*, **20**, 975–991 (2005).

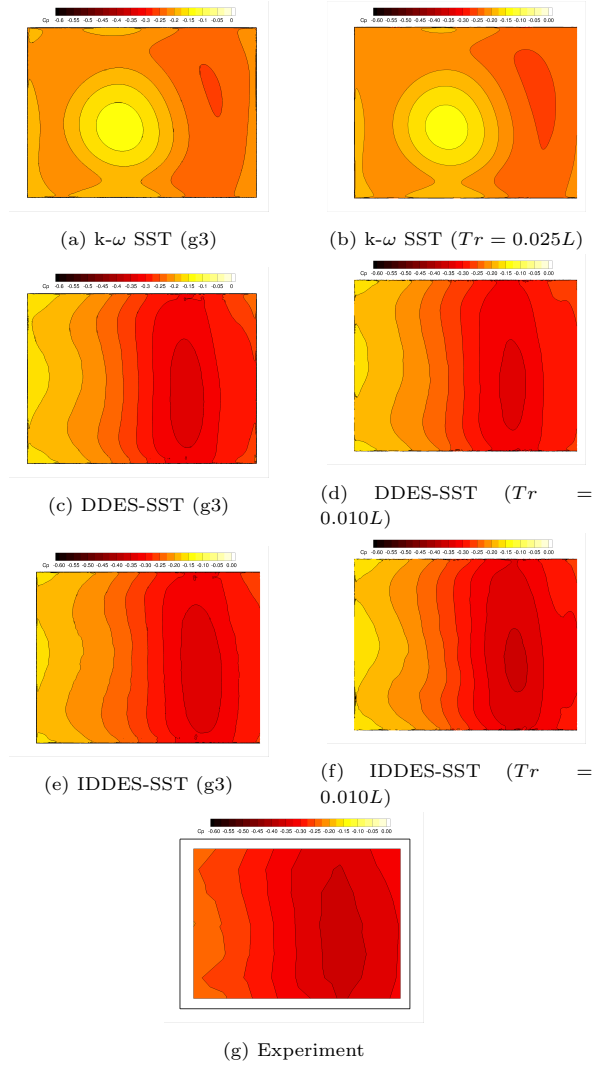


Figure 3: Base pressure coefficient.

[3]Queutey, P. and Visonneau, M. : An Interface Capturing Method for Free-Surface Hydrodynamic Flows, *Computers & Fluids*, **36**, 1481–1510 (2007).
 [4]Deng, G.B. and Visonneau, M. Comparison of explicit algebraic stress models and second-order turbulence closures for steady flow around ships, *Proc. of 7th Symposium on Numerical Ship Hydrodynamics, July, Nantes, France*, (1999).
 [5]Cecora, R.-D., Radespiel, R., Eisfeld, B. and Probst, A. : Differential Reynolds-Stress Modeling for Aeronautics, *AIAA Journal*, **53**(3), 48–65 (2015).
 [6]Guilmineau, E. Deng, G.B., Leroyer, A., Queutey, P., Visonneau, M. and Wackers, J. : Assessment of hybrid RANS-LES formulations for flow simulation around the Ahmed body, *Computers & Fluids*, **178**, 302–319 (2018).
 [7]Visonneau, M., Guilmineau, E. and Rubino, G. : Local flow around a surface combatant at various static drift conditions: The role played by turbulence closures, *Proc. of 33rd Symposium on Naval Hydrodynamics, Osaka, Japan*, (2020).
 [8]Wackers, J., Deng, G.B., Guilmineau, E., Leroyer, A., Queutey, P., Visonneau, M. and Liverani, A. : Can adaptive grid produce grid-independent solutions for incompressible flows?, *Journal of Computational Physics*, **244**, 364–380 (2017).
 [9]Mozaffari, S., Guilmineau, E., Visonneau, M. and Wackers, J. : Average-based mesh adaptation for hybrid RANS/LES simulation of complex flows, *Computers & Fluids*, **232**, 105202 (2022).