

Critical Assessment of Some Popular Scale-Resolving Turbulence Models for Vehicle Aerodynamics

Suad Jakirlic and Lukas Kutej
 Technical University of Darmstadt

Peter Unterlechner
 Audi AG

Cameron Tropea
 Technical University of Darmstadt

ABSTRACT

Some widely-used scale-resolving turbulence models are comparatively assessed in simulating the aerodynamic behavior of a full-scale AUDI-A1 car configuration. The presently considered hybrid RANS/LES (RANS – *Reynolds-Averaged Navier-Stokes*; LES – *Large-Eddy Simulation*) models include the well-known DDES (*Delayed Detached-Eddy Simulation*) scheme and two further variable-resolution formulations denoted by PANS (*Partially-Averaged Navier-Stokes*; Basara, 2011) and VLES (*Very LES*; Chang et al., 2014). Whereas the DDES method represents the originally proposed formulation based on the one-equation Spalart-Almaras model (Spalart et al. 2006), whose RANS/LES interface position is directly correlated to the underlying grid resolution, the other two models represent ‘true’ seamless formulations, providing a smooth transition from Unsteady RANS to LES in terms of a dynamic “resolution parameter” variation. The latter parameter is evaluated by contrasting the length scale related to the residual turbulence of both PANS and VLES methods to the grid spacing. The dynamics of residual motion in both methods is modelled by a four equation model (Hanjalic et al., 2004). All computations are performed by the OpenFOAM code. The PANS and VLES formulations, in conjunction with the ‘hybrid wall functions’ used for the wall treatment, were implemented by the authors. The car configuration considered accounts for mirrors, detailed underbody accommodating the exhaust system, as well as the rotating wheels including brake discs and rim details. The solution domain representing a regular hexahedron was meshed by two grids consisting approximately of 31 and 62 million cells in total. The results representation includes detailed time-dependent mean flow and turbulence fields, surface pressure distribution and the resulting drag and lift force coefficients. The results’ analysis is performed by discussing the models’ predictive capabilities along with the available experimental results.

CITATION: Jakirlic, S., Kutej, L., Unterlechner, P., and Tropea, C., "Critical Assessment of Some Popular Scale-Resolving Turbulence Models for Vehicle Aerodynamics," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1532.

INTRODUCTION

The RANS (*Reynolds-Averaged Navier-Stokes*) method, employing the statistical turbulence models to describe the dynamics of the Reynolds-stress tensor $\overline{u_i u_j}$, is still the most widely used computational approach within the engineering Computational Fluid Dynamics (CFD). However, its incapability to capture any spectral dynamics of an instantaneous turbulent flow makes it especially unsuitable for simulating the boundary layer separation at complex, variably-curved surfaces, wall-bounded and free reattachments and in particular the wake regions. The highly unsteady separated shear layers and the flow in the car wake are characterized by organized, large-scale coherent structures developing in the presence of the ground. A high Reynolds number flow past a car operating in the ground proximity represents a prime example of such a flow configuration exhibiting high degree of complexity including multiplicity of mean flow and turbulence phenomena and their

interactions. Accordingly more sophisticated turbulence models are required for a reliable description of the aerodynamic behaviour of a vehicle. Correct capturing of the dynamics of the afore-mentioned coherent vortices, influencing significantly the aerodynamic properties of a car configuration, can only be achieved by employing an eddy-resolving, LES-related (*Large-Eddy Simulation*) turbulence model. Relevant computational methods utilizing an appropriately modified RANS model describing the dynamics of residual turbulence in the so-called hybrid LES/RANS computational schemes have experienced intensified use in the CFD community, also for simulating the external vehicle aerodynamics; see e.g. computational studies by Islam et al. (2009), Han et al. (2013), Frank et al. (2013), Guilmineau (2014), Gaylard et al. (2014), Jakirlic et al. (2014, 2016a, 2016b) and Ashton and Revell (2015a, 2015b). A number of relevant proposals for coupling RANS and LES approaches has been reported. The most advantageous ones represent the seamless hybrid LES/

RANS methods utilizing an appropriately modified RANS model which plays the role of a sub-grid-scale (SGS) model in the entire solution domain, see for instance [Jakirlic and Maduta \(2015\)](#) for a relevant overview.

The work reported presently focusses on application of some widely-used scale-resolving, seamless hybrid LES/RANS methods denoted as DDES (*Delayed Detached-Eddy Simulation*; [Spalart et al., 2006](#)), PANS (*Partially-Averaged Navier-Stokes*; [Basara et al., 2011](#)) and VLES (*Very Large-Eddy Simulation*; [Chang et al., 2014, 2015](#)) on the basis of simulating a full-scale AUDI A1 car configuration.

COMPUTATIONAL MODEL

The continuity and momentum equations governing the three-dimensional unsteady flow past a car configuration read:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho \tilde{U}_j) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \tilde{U}_i) + \frac{\partial}{\partial x_j} (\rho \tilde{U}_i \tilde{U}_j) = -\frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \tilde{U}_i}{\partial x_j} + \tilde{\tau}_{ij} \right) \quad (2)$$

These are in fact the filtered, LES-related (LES – Large-eddy Simulation) Navier-Stokes Equations with $\tilde{\tau}_{ij}$ representing the unresolved sub-grid scale tensor. The model for $\tilde{\tau}_{ij}$ is based on the eddy-viscosity concept which relies on the Boussinesq correlation. Accordingly, the turbulent stress tensor is defined in terms of the resolved strain tensor $\tilde{S}_{ij}$

$$\tilde{S}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{U}_i}{\partial x_j} + \frac{\partial \tilde{U}_j}{\partial x_i} \right) \quad (3)$$

as follows:

$$\tilde{\tau}_{ij} = -2\rho\nu_u \tilde{S}_{ij} + \frac{1}{3} \rho \tilde{\tau}_{kk} \delta_{ij} \quad (4)$$

$\tilde{U}_i(x_i, t)$ represents the resolved (‘filtered’) velocity field and $\tilde{p}(x_i, t)$, the corresponding pressure field. ρ and μ denote the fluid density and the dynamic viscosity coefficient respectively. In the framework of a conventional LES the unresolved turbulent viscosity ν_u is traditionally modelled by applying the Standard Smagorinsky model:

$$\nu_u = (C_s \Delta)^2 \tilde{S} \quad (5)$$

Here, the respective length ($\sim \Delta$) and velocity scales ($\sim \Delta \tilde{S}$) modelled in terms of the representative mesh size Δ (representing the grid scale defined as the grid-cell volume: $\Delta = (\Delta_x \times \Delta_y \times \Delta_z)^{1/3}$) and the magnitude of the strain tensor $\tilde{S} = (2\tilde{S}_{ij}\tilde{S}_{ij})^{1/2}$. Accordingly, the stresses residing in subgrid scales ($\tilde{\tau}_{ij} \sim \nu_u \tilde{S}_{ij}$) depend strongly on the

grid spacing, Δ , corresponding directly to the filter width.

Consequently, the Smagorinsky model performs poorly if an irregular, highly nonuniform grid is to be used. Such anisotropic grids with higher cell aspect ratio, especially in the wall vicinity, are usually encountered in everyday Computational Fluid Dynamics. The latter is especially the case in the Computational Car Aerodynamics.

The latter weaknesses motivated the presently applied modelling strategies which include the DDES (*Delayed Detached-Eddy Simulation*, [Spalart et al., 2006](#)), PANS (*Partially-Averaged Navier-Stokes*, [Basara et al., 2011](#)) and VLES (*Very Large-eddy Simulation*, [Chang et al., 2014](#)) scheme formulations, all representing seamless hybrid LES/RANS (Reynolds-averaged Navier-Stokes) models providing a smooth transition from RANS to LES and opposite. The residual turbulence in all three eddy-resolving model schemes is described by appropriately modified RANS model formulations. These RANS-based sub-scale models, representing the constituents of the presently considered simulation methodologies, rely on the eddy-viscosity concept. Accordingly, the basic difference between them is the way of determining the representative turbulence viscosity ν_u .

Delayed Detached-Eddy Simulation - DDES

The one-equation turbulence model by [Spalart and Allmaras \(SA, 1994\)](#), based on the transport equation for turbulence viscosity, was employed to model the influence of the smallest, unresolved scales on the resolved ones in the framework of the DES computational scheme, principally proposed by [Spalart et al. \(1997\)](#). The SA-RANS model determines the turbulent viscosity ($\nu_t = f_{v1} \tilde{\nu}$) from the corresponding transport equation, whose destruction term is modeled in terms of the distance to the nearest wall d_w :

$$\frac{D\nu}{Dt} = \dots - C_{w1} f_v \frac{\tilde{\nu}}{d_w^2} \quad (6)$$

The DES formulation ($\nu_u \equiv \nu_t$) is obtained by replacing the physical wall distance d_w by L_{DES} as follows

$$L_{DES} = \min(d_w, C_{DES} \Delta_{DES}) \quad (7a)$$

where a somewhat modified grid spacing, introduced as $\Delta_{DES} = \max(\Delta_x, \Delta_y, \Delta_z)$, representing actually the ‘DES filter’ (a relevant length scale), enables the DES to operate in an LES-like mode in off-wall regions in accordance with the condition given in [Eq. \(7a\)](#). This length-scale switch in the nominator of the destruction term in [Eq. \(6\)](#) facilitates the model to function in the RANS operating mode in the near-wall regions ($L_{DES} \rightarrow d_w$), i.e. in ‘attached’ boundary layers, whereas away from the walls, especially in the detached regions of the flow, the closure reduces to a ‘one-equation’ subgrid-scale model ($L_{DES} \rightarrow C_{DES} \Delta_{DES}$) describing the eddy viscosity of the unresolved motion. The original value of the corresponding model constant, used also in this work, is $C_{DES} = 0.65$. The presently used DES scheme deals with the so-called Delayed DES (DDES), representing an important physical upgrade of the original DES formulation (denoted

afterwards by DES97). This DDES scheme remedies some anomalies of the DES97 model characteristic for the situations with insufficient grid resolution within the LES-like operating flow region. It can cause the LES-mode of the DES97 scheme propagating into the wall boundary layer and leading possibly to the so-called “modelled stress depletion” (the consequence of which is a substantial underprediction of the skin friction), or even, in more dramatic cases, to the so-called “grid-induced separation”. To avoid such an anomaly, an appropriate modification of the DES97 length scale is proposed (Spalart et al., 2006):

$$L_{DES} = d_w - f_d \max(0, d_w - C_{DES} \Delta_{DES})$$

$$f_d = 1 - \tanh\left[\left(8r_d\right)^3\right] \quad r_d = \frac{\nu + \nu_t}{\kappa^2 d_w^2 \sqrt{(\partial U_i / \partial x_j)^2}} \quad (7b)$$

With this modification the RANS function of the DES scheme within the entire boundary layer is provided.

Partially-Averaged Navier-Stokes - PANS

The PANS method, principally developed by Girimaji (2006), represents a seamless RANS-LES coupling method, where a RANS-type model mimics a sub-grid-scale model. This so-called bridging method provides smooth and seamless transition from RANS to LES in terms of a “filter-width control parameter” variation, representing the unresolved-to-total ratios of kinetic energy ($f_k = f_u/k$) and its dissipation ($f_\varepsilon = \varepsilon_u/\varepsilon$) which varies within the flow domain, enabling the transition from a fully-averaged computation to a completely resolved simulation. The afore-mentioned filter-width control parameter is pertinent to the spectrum cut-off, which is to be moved continuously from the RANS limit towards the full LES limit by partially integrating the spectrum. In the Basara's et al. (2011) PANS model applied presently the parameter (ζ) is implemented in the computational procedure as a dynamic variable, changing at each computational node at the end of corresponding time step, being afterwards used as a fixed value at the same location during the next time step. The equations governing the unresolved turbulence quantities, such as kinetic energy k , corresponding dissipation rate ε_u , as well as the ratio $\zeta_u = (\overline{v^2})_u/k_u$ and associated elliptic-relaxation function f are systematically derived from the Hanjalic's et al. (2004) RANS- $\zeta-f$ model. This model represents an appropriate upgrade of the Durbin's $\overline{v^2}-f$ model (1991) towards increased numerical robustness. Interested reader is referred to original references by Hanjalic et al. (2004) and Basara et al. (2011) for more details. Presently, only the unresolved dissipation rate equation whose destruction term is appropriately modified will be explained more extensively

$$\frac{D\varepsilon_u}{Dt} = C_{\varepsilon 1} P_u \frac{\varepsilon_u}{k_u} - C_{\varepsilon 2}^* \frac{\varepsilon_u^2}{k_u} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{\varepsilon u}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] \quad (8)$$

Here, the eddy viscosity of the unresolved scales and the corresponding time scale switch take the following forms

$$\nu_u = C_\mu^* \zeta_u k_u T_u$$

$$T_u = \max \left[6\eta_{ku}, \frac{k_u}{\varepsilon_u} \right] \quad (9)$$

with $\tau_{ku} = (\nu/\varepsilon_u)^{1/2}$ representing the Kolmogorov time scale. The form of the functional dependency in the model coefficients is of decisive importance. This is especially the case with the coefficient multiplying the destruction term in the dissipation equation. The appropriate formulation is given by

$$C_{\varepsilon 2}^* = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} (C_{\varepsilon 2} - C_{\varepsilon 1}) \quad (10)$$

with the model constants taking their standard values $C_{\varepsilon 1} = 1.44$ and $C_{\varepsilon 2} = 1.92$. Such a formulated model coefficient enables appropriate reduction of the standard model coefficient value of $C_{\varepsilon 2}^* = 1.92$ (corresponding to $f_k = 1$), prevailing in the near-wall region, towards a correspondingly decreased value pertinent to the core region of the flow, Fig. 1.

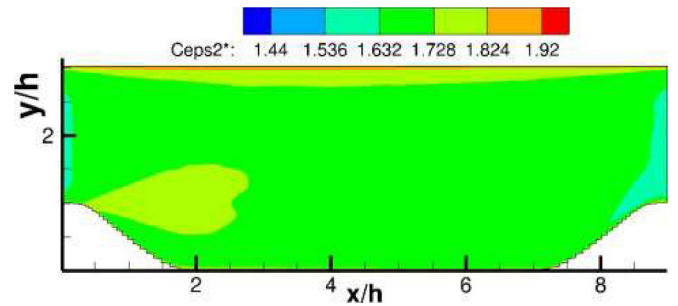


Figure 1. Field of the model coefficient (Eq. 10) in a periodic flow over a 2D hill at $Re_h = 10600$

Herewith, a dissipation rate level will be provided which suppresses the turbulence intensity towards the subgrid (i.e. subscale) level in the region where large coherent structures with a broader spectrum dominate the flow, as for instance in the separated shear layer region, allowing in such a way evolution of structural features. Herewith, a seamless coupling, i.e. a smooth transition from LES to RANS and opposite is enabled. The parameter f_k is formulated in terms of the grid spacing following Basara et al. (2011):

$$f_k = \frac{1}{\sqrt{C_\mu}} \left(\frac{\Delta}{\Lambda} \right)^{2/3} \quad (11)$$

where Δ represents the volume grid cell size (analogue to that used in LES) and $\Lambda = (k^{3/2}/\varepsilon)$ is the turbulent length scale. It was assumed in this derivation that the length scale of the smallest resolved eddies, defined in accordance with the Kolmogorov-scale-like formulation as $(\nu^3/\varepsilon_u)^{1/4}$ corresponds to the grid spacing Δ . This is justified by the equality $\varepsilon_u = \varepsilon$ resulting in $f_\varepsilon = 1$ was assumed. This is justified by the

fact that a PANS-related grid size is still too coarse to be capable of even approximately approaching the resolution being necessary to resolve the dissipation range. The PANS asymptotic behavior goes smoothly from RANS to DNS with $f_k = 1$ decreasing from the value 1 towards zero value. Accordingly, in the computational procedure used here, the lowest value of the parameter $f_k = 1$ is adjusted to the given grid as it was implemented as a dynamic parameter, changing at each grid node. The values obtained at the end of a time step are used in the following time step.

The PANS method has been intensively tested in the past by computing the flows of different complexity, including also the car-relevant configurations; see e.g. Han et al. (2013), Jakirlic et al. (2014, 2016a), Chang et al. (2015), Mirzaei et al. (2015) and Krajnovic et al. (2016).

Very Large-Eddy Simulation - VLES

The turbulence associated with the unresolved residual motion is here represented by the residual stress tensor as follows (nominally in line with the proposal of Speziale, 1998)

$$\tilde{\tau}_{ij} = F_r \tau_{ij}^{RANS} \quad (12)$$

The function F_r blends between the DNS (Direct Numerical Simulation) mode ($F_r \searrow 0$; $\Delta \sim \eta_K$, with $\eta_K = (\nu^3/\varepsilon)$ denoting the dissipative Kolmogorov scales) and pure RANS ($F_r \nearrow 1$; $\tau_{ij}^{RANS} \equiv \overline{u_i u_j}$; $\Delta \sim k^{3/2}/\varepsilon$, with $\overline{u_i u_j}$, $k = \overline{u_i u_i}/2$ and ε representing the Reynolds stress tensor, kinetic energy of the RANS turbulence and its dissipation rate respectively). Herewith, the fully-modelled RANS mode will be seamlessly bridged to fully-resolved DNS in accordance with the vanishing $\tilde{\tau}_{ij}$. Between these two limits an LES or a VLES will be recovered. In the LES, the largest modelled ('u' - unresolved) length scale ($\Lambda_u = k_u^{3/2}/\varepsilon_u$) is proportional to the grid spacing: $\Delta \sim \Lambda_u$. Accordingly

$$\Delta = \frac{k_u^{3/2}}{\varepsilon_u} = \frac{F_k^{3/2} k_{us}^{3/2}}{F_\varepsilon \varepsilon_{us}} = \frac{F_k^{3/2}}{F_\varepsilon} \Lambda_{us} \quad (13)$$

with $\Lambda_{us} = k_{us}^{3/2}/\varepsilon_{us}$ and the ratios (actually resolution parameters) $F_k = k_u/k_{us}$ and $F_\varepsilon = \varepsilon_u/\varepsilon_{us}$ adopted from the PANS (Partially-Averaged Navier-Stokes) methodology (Basara et al., 2011). Analogous to PANS, F_ε can be taken as unity ($\varepsilon_u = \varepsilon_{us}$). The present VLES method employs the Unsteady RANS model applying a built-in function K_L to appropriately suppress the relevant turbulent viscosity (ν_{us}) to the sub-grid scale level ν_u , i.e. $\nu_u = F_r \nu_{us}$. If adopting a $k - \varepsilon$ -type model to define the turbulent viscosity ($\nu \sim k^2/\varepsilon$) the function F_r reduces to

$$F_r = \frac{k_u^2}{k_{us}^2} = F_k^2 = \left(\frac{\Delta}{\Lambda_{us}} \right)^{4/3} \Rightarrow F_r = \min \left[\left(\frac{\Delta}{\Lambda_{us}} \right)^{4/3}, 1 \right] \quad (14)$$

In the flow regions where the modelled length scale is smaller than the filter width ($\Lambda_{us} \leq \Delta$), the 'unsteady' ('us') turbulent properties become equivalent to the fully-modelled properties, implying the URANS operating mode of the VLES method prevails: $F_r = 1$. Fig. 2 shows exemplary the field of the function F_k in a periodic flow over a smoothly-contoured 2D hill. Whereas the high -values are characteristic for the immediate wall vicinity, it reduces significantly towards the core region of the flow. The presently employed VLES method, similar as the PANS method, utilizes the near-wall RANS-based " $\zeta - f$ " model formulation, Hanjalic et al. (2004), as the background URANS model scheme. In addition to the equations governing the kinetic energy of turbulence k_{us} and its dissipation rate ε_{us} it solves transport equations for the quantity ζ_{us} , representing the ratio ν_{us}^2/k_{us} and elliptic function f , with ν_{us}^2 denoting the scalar variable which behaves as the normal-to-the-wall Reynolds stress component by approaching the solid wall. Accordingly, ν_u reads

$$\nu_u = F_r \nu_{us} = F_r C_\mu^\zeta \zeta_{us} k_{us} T_{us} \quad (15)$$

The 'us' turbulent quantities relate to the residual turbulence of the VLES method. T_{us} is a corresponding time scale switch, blending between the Kolmogorov time scale ($\tau_K = (\nu/\varepsilon_{us})^{1/2}$) and turbulent time scale (k_{us}/ε_{us}):

$$T_{us} = \max \left[C_\tau \left(\frac{\nu}{\varepsilon_{us}} \right)^{1/2}, \frac{k_{us}}{\varepsilon_{us}} \right] \quad (16)$$

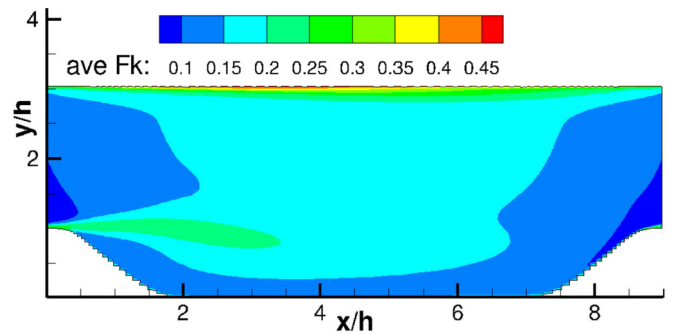


Figure 2. Field of the resolution function F_k (Eq. 14) in a periodic flow over a 2D hill at $Re_h = 10600$

The VLES method is validated interactively in the process of the model derivation by computing a series of generic flows, such as the natural decay of the homogeneous isotropic turbulence, fully-developed flow in a plane channel and a separating flow over a periodic arrangement of smoothly-contoured 2-D hills in a Reynolds number range. Consequently the model's predictive performance is demonstrated by computing more complex flow, such as swirling flow in a vortex tube and flows imposed to mean compression in internal-combustion configurations as well as the flow over the DrivAer car model (Jakirlic et al., 2016b). For a detailed specification of the VLES model interested readers are referred to Chang et al. (2014, 2015).

To summarize, unlike in the LES framework, where the grid spacing Δ represents the characteristic size of the largest unresolved scales (subgrid-scales), the representative length scales of the residual turbulence entering the unresolved turbulent viscosity in the PANS and VLES methodologies is determined by solving the respective differential equations. Accordingly, the grid spacing incorporated into the PANS and VLES models (Eqs. 11 and 14), represents just one of several model parameters serving for the determination of the turbulent viscosity (Eqs. 9 and 15). Eqs. (9) and (15), as well as Eq. (6) describing the evolution of the DDES turbulent viscosity, point to a complex relationship involving different turbulent quantities exhibiting high level of coherence; it is recalled that in the LES method the grid size Δ influences very directly the final outcome (Eq. 5). Hence, in DDES, PANS and VLES the grid spacing Δ is obviously a less influential factor – such a model rationale has inherently much more physics, allowing the use of coarser spatial and temporal resolutions.

In addition to the PANS and VLES simulations, complementary Steady and Unsteady RANS computations employing the $\zeta - f$ model are also performed.

Numerical Method

All computations were performed using the code Open-FOAM®, an open source Computational Fluid Dynamics toolbox (www.openfoam.com), utilizing a cell-center-based finite volume method on an unstructured numerical grid and employing the solution procedure based on the implicit pressure algorithm with splitting of operators (PISO) for coupling between pressure and velocity fields. SIMPLE procedure was applied when computations have been performed using RANS models. Whereas the RANS-SA and DDES-SA models represent the original OpenFOAM-implementation, the RANS- $\zeta - f$ model, as well as the PANS and VLES model formulations, are implemented in the framework of the present modelling and computational activities. The convective transport was discretized by a 2nd-order accurate TVD (Total Variation Diminishing) scheme applying the so-called Sweby flux limiter. For the time integration the 2nd order three point backward scheme was used. The temporal resolution applied is in accordance with the maximum CFL number being smaller than one in the entire solution domain.

CAR CONFIGURATION DESCRIPTION

The presently considered AUDI A1 car model represents a full-scale car geometry. The simulated configuration accounts for the rotating wheels including brake discs and rim details as well as detailed underbody geometry including exhaust system as well as further details, see Figure 3.

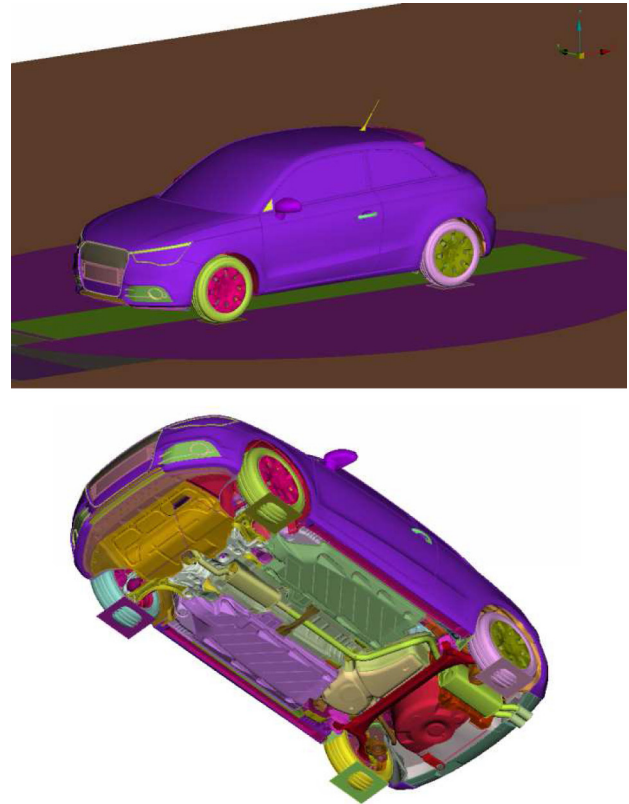


Figure 3. AUDI A1 car model

A detailed three-dimensional surface pressure distribution has been provided experimentally, as well as the database comprising the drag and lift aerodynamic forces, i.e. their relevant non-dimensional coefficients. In addition surface flow patterns at some specific localities – roof/spoiler, back light and trunk car surfaces, reproduced by applying the oil-flow visualization technique are provided.

Computational Details

The solution domain accommodating the full-scale car model has a shape of a regular hexahedron, Figure 4. The origin of the coordinate system coincides with front axle. The computational domain is largely extended in all three coordinate directions to eliminate any significant influence of the boundary conditions adopted at the side, upper and outlet boundary planes.

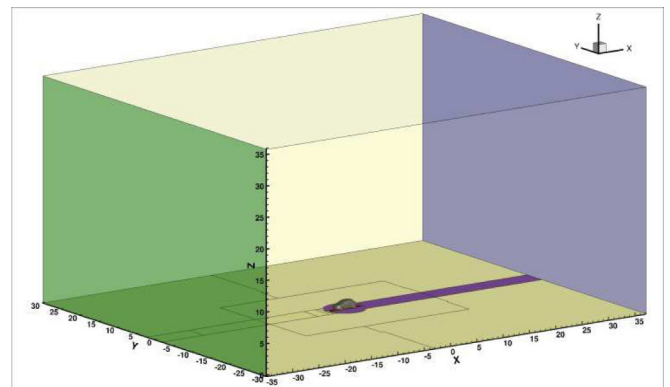


Figure 4. Solution domain: shape and dimensions