

Computational Study of the Aerodynamics of a Realistic Car Model by Means of RANS and Hybrid RANS/LES Approaches

Suad Jakirlic and Lukas Kutej
Technische Universitaet, Darmstadt

Branislav Basara
AVL List GmbH

Cameron Tropea
Technische Universitaet, Darmstadt

ABSTRACT

The aerodynamic properties of a BMW car model, representing a 40%-scaled model of a relevant car configuration, are studied computationally by means of the Unsteady RANS (Reynolds-Averaged Navier-Stokes) and Hybrid RANS/LES (Large-Eddy Simulation) approaches. The reference database (geometry, operating parameters and surface pressure distribution) are adopted from an experimental investigation carried out in the wind tunnel of the BMW Group in Munich (Schrefl, 2008). The present computational study focuses on validation of some recently developed turbulence models for unsteady flow computations in conjunction with the universal wall treatment combining integration up to the wall and high Reynolds number wall functions in such complex flow situations. The turbulence model adopted in both Unsteady RANS and PANS (Partially-Averaged Navier Stokes) frameworks is the four-equation $\zeta - f$ formulation of Hanjalic et al. (2004) based on the Elliptic Relaxation Concept (Durbin, 1991). The latter model mimics the sub-scale model in the PANS method representing a hybrid RANS/LES strategy, proposed recently by Basara et al. (2011). Herewith a seamless transition from Unsteady RANS to the direct numerical simulation (DNS) is provided as the unresolved-to-total ratios of kinetic energy and its dissipation are varied throughout the flow domain. As a result of simulations the three-dimensional wall-pressure distribution, detailed mean flow and turbulence fields are obtained and analyzed along with experimental findings, thus enabling the study of the aerodynamic properties of this ground vehicle including also the unsteady flow phenomena in the car wake.

CITATION: Jakirlic, S., Kutej, L., Basara, B., and Tropea, C., "Computational Study of the Aerodynamics of a Realistic Car Model by Means of RANS and Hybrid RANS/LES Approaches," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0594.

INTRODUCTION

Large majority of the experimental and computational studies performed in the past relate to the steady flow approaching a car configuration situated in a wind tunnel. We mention here only some relevant numerical publications, e.g. Baxendale et al. (1996), Hupertz et al. (1996), Makowsky and Kim (2000) and Basara et al. (2001). Furthermore, in the works of Basara and Tibaut (2002) and Khalighi et al. (2001) the URANS (Unsteady Reynolds-Averaged Navier-Stokes) approach was applied in order to reproduce more accurately the unsteady wake behind the car. The evaluation of the aerodynamic properties of a car is traditionally based on these and similar investigations conducted under the conditions of a steady oncoming flow. The aerodynamic forces obtained from such investigations served as a sole basis for decision to be made

with respect to the aerodynamic properties. Accordingly, the possible time variation of the moments and forces has not been explicitly considered. The influence of their temporal changes on the aerodynamic properties is conventionally regarded to be negligible. The objective of the experimental study conducted by Schrefl (2008), whose results served as reference database for the present computational work, was to check this classical approach utilizing to estimate the aerodynamic performances of a car configuration. The unsteady effects in question originate primarily from the characteristics of the oncoming flow. Relevant configuration corresponds to the flow past a car which is situated in the wake of a truck, a situation encountered frequently during a highway passing maneuver. The aerodynamic effects pertinent to such an overtaking maneuver are still not completely understood. To better understand these effects, a 40% car-truck overtaking

process has been experimentally investigated in the BMW wind tunnel, [Schrefl \(2008\)](#), [Figure 2](#). Prior to considering the truck-car interaction during an overtaking process [Schrefl \(2008\)](#) investigated introductorily the aerodynamics of the isolated car without the truck representing also the flow configurations analyzed in the present computational study. Accordingly, the present analysis represents just the first stage of a longer-term project dealing with the truck interference effects on a car during the overtaking maneuver (see Outlook section).

The highly unsteady separated shear layers and the flow in the car wake are strongly influenced by the large scale dynamics. These coherent vortices, whose characteristic time scale is by far larger than the turbulence time scale, dominate the flow and their correct simulation is of decisive importance for the determination of the aerodynamic forces on the car body. It is well-known that the single point RANS closures are only inadequately capable to capture the transient nature of the turbulence field. Consequently, the effects of the unsteady oncoming flow and the irregular unsteadiness of the oscillating car wake could not be accounted for. Accordingly, the main objective of the present work is the validation and comparative assessment of two computational approaches URANS and a seamless hybrid RANS/LES method denoted as PANS (Partially-Averaged Navier Stokes), formulated principally by [Girimaji \(2006\)](#). The background model of turbulence employed in both computational frameworks is the ERM-based (Elliptic Relaxation Method) eddy-viscosity formulation due to [Hanjalic et al. \(2004\)](#), representing a numerically more robust variant of the Durbin's $\nu_2 - f$ (1991) model. Employment of the latter model within the PANS methodology is proposed by [Basara et al. \(2011\)](#).

COMPUTATIONAL MODEL

The incompressible continuity and momentum equations pertinent to the Unsteady Reynolds-Averaged Navier Stokes (U-RANS) and Partially-Averaged Navier Stokes (PANS) approaches governing the flow field around a car configuration may be written as:

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

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial U_i}{\partial x_j} + \tau_{ij}^t \right) \quad (2)$$

with U_i representing the time-averaged ($U_i = \overline{f(x_i)}$); pertinent to the RANS framework; $\partial/\partial t = 0$) i.e. the ensemble-averaged, time-dependent ($U_i = f(x_i, t)$; related to the Unsteady RANS approach) mean velocity field or the resolved (instantaneous, 'filtered' to an appropriate extent), PANS-related velocity field and p the corresponding pressure field. ρ and μ stay for the fluid density and the dynamic viscosity respectively. τ_{ij}^t denotes

the turbulent stress tensor associated with the time-averaged/ensemble-averaged velocity field (RANS/URANS: $\tau_{ij}^t \equiv -\rho \overline{u_i u_j}$) or with the unresolved (the relevant parameters are denoted with the subscript 'u' throughout the paper) residual motion within the PANS methodology ($\tau_{ij}^t \equiv -\rho \overline{(u_i u_j)_u}$) which is to be modeled.

In both computational frameworks (RANS/URANS and PANS) considered presently the same modeling concept was adopted. The model is based on the eddy-viscosity rationale which relies on the Boussinesq's correlation defining the turbulent stress tensor in terms of the rate of strain tensor S_{ij}

$$S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3)$$

given as (RANS/URANS):

$$\tau_{ij}^t \Rightarrow -\rho \overline{u_i u_j} = 2\rho \nu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij} \quad (4)$$

or (PANS):

$$\tau_{ij}^t \Rightarrow -\rho \overline{(u_i u_j)_u} = 2\rho \nu_u S_{ij} - \frac{2}{3} \rho k_u \delta_{ij} \quad (5)$$

where $k = 0.5 \overline{u_i u_i}$ relates to the total turbulent kinetic energy and $k_u = 0.5 \overline{(u_i u_i)_u}$ represents its unresolved fraction. The latter is to be modelled by the appropriately modified ζ -f model in the PANS framework (with $C_{e,2}$ representing an appropriate function; [Eqs. 6,7,8,9,10,11,12,13,14,15](#)). Contrary to that, in the RANS/URANS framework the total (fully-modelled) k is the outcome of the basic RANS (non-modified) ζ -f model ($C_{e,2} = 1.9$). Herewith, the modeling of the turbulent stress tensor is reduced to the modeling of the total turbulent viscosity ν_t and the unresolved one ν_u .

The Partially-Averaged Navier-Stokes (PANS) approach represents a hybrid RANS/LES (Large-Eddy Simulation) method proposed originally by [Girimaji \(2006\)](#). This so-called bridging method enables a seamless transition from RANS to the direct numerical solution of the Navier-Stokes equations (DNS) in terms of a "filter-width control parameter" variation, that is 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. The PANS-related filter width depends on the unresolved-to-total ratios of kinetic energy ($f_k = k_u / k$) and dissipation ($f_\epsilon = \epsilon_u / \epsilon$) which vary within the flow domain. The turbulence model describing the unknown turbulent stresses is obtained from the equation of motion ([Equation 2](#)) plays here the role of a

sub-scale model. This sub-scale model formulation represents a conventional RANS model formulation being appropriately modified - similar as in some other seamless hybrid LES/RANS strategies (e.g., PITM - Partially-Integrated Transport Model, Chaouat and Schiestel, 2005) the destruction term in the supplying equation, i.e. its coefficient (see Equation 7), is modeled in a way providing the dissipation enhancement in the flow regions dominated by the organized, large-scale coherent structures with a broader spectrum (influencing to a large extent the overall flow behaviour), as e.g. encountered in the separated shear layer regions. These modifications promote the modeled turbulence suppression towards the sub-scale level allowing in such a way the development of the structural properties of the associated turbulence being beyond the reach of conventional RANS approach independent of the modeling level. Accordingly, unlike the ensemble-averaged mean velocity field in the URANS framework, the velocity field in the continuity and momentum equation (Equations 1 and 2) consists of the resolved ($U_i = f(x_p, t)$ which is in the case of the presently computed configuration similar to that of the Unsteady RANS; but, in general these velocity fields are very different) and residual (unresolved, sub-scale) fractions (the latter being modeled by a RANS-type closure formulation) representing the fluctuation about this PANS-related resolved velocity field ($u_i = f(x_p, t)$, which is analogue but not the same as the fluctuations within DNS and LES frameworks). There are basically two variants of the PANS model derived up to now, one based on the $k - \varepsilon$ formulation and another based on the $k - \omega$ formulation (Lakshminpathy and Girimaji, 2006). Recently, Basara et al. (2011) proposed a near-wall PANS formulation whose unresolved kinetic energy (k_u) and dissipation (ε_u) equations are systematically derived from the Hanjalic's et al. $\zeta - f$ model (2004). This is the formulation employed presently. This RANS model relies on the elliptic relaxation (ER) concept providing a continuous modification of the homogeneous pressure-strain process as the wall is approached to satisfy the wall conditions, thus avoiding the need for any wall topography parameter. The variable ζ represents the ratio v^2 / k (v^2 is a scalar property in the Durbin's $v^2 - f$ model, which reduces to the wall-normal stress in the near-wall region) providing more convenient formulation of the equation for ζ and especially of the wall boundary conditions for the elliptic function f . The $\zeta - f$ model is numerically very robust and more accurate than the simpler two-equation eddy viscosity models (Hanjalic et al., 2004). The $\zeta - f$ model represents a near-wall model but is also applicable to a relatively coarse mesh next to the wall providing the distance of the wall-closest node should reach a non-dimensional wall distance y^+ less than 3 as a maximum. Let us recall that Khalighi et al. (2001) have successfully used a compatible ERM model scheme ($v^2 - f$ model of Durbin, 1991) in their work on unsteady wake flow behind a bluff body with a drag reduction device. Readers are referred to the original publications for more specific details about the model developments, here only a detailed specification of the Basara's et al. PANS- $\zeta - f$ model will be displayed. The model equations governing the unresolved kinetic energy k_u and the unresolved dissipation ε_u are given as

$$\frac{\partial k_u}{\partial t} + U_j \frac{\partial k_u}{\partial x_j} = (P_u - \varepsilon_u) + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{k_u}} \right) \frac{\partial k_u}{\partial x_j} \right] \quad (6)$$

$$\frac{\partial \varepsilon_u}{\partial t} + U_j \frac{\partial \varepsilon_u}{\partial x_j} = \frac{C_{\varepsilon 1} P_u - C_{\varepsilon 2}^* \varepsilon_u}{T_u} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{\varepsilon_u}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] \quad (7)$$

with production rate of k_u given as:

$$P_u = - \left(\overline{u_i u_j} \right)_u \frac{\partial U_i}{\partial x_j} \quad (8)$$

and with $\left(\overline{u_i u_j} \right)_u$ defined by Equation (5). The eddy viscosity of the unresolved scales takes its standard form

$$\nu_u = c_\mu \zeta_u k_u T_u \quad (9)$$

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

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

The parameter which determines the unresolved-to-total kinetic energy ratio f_k is defined based on the grid spacing

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

where $\Delta = (\Delta_x \times \Delta_y \times \Delta_z)^{1/3}$ is the representative grid cell dimension and $\Lambda (= k_u^{3/2} / \varepsilon)$ is the turbulent length scale. In this derivation the equality $\varepsilon_u = \varepsilon$ resulting in $f_\varepsilon = 1$ was assumed. The PANS asymptotic behaviour goes smoothly from RANS to DNS with decreasing f_k . In the computational procedure used here, the lowest value of the parameter f_k 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 model is completed by introducing appropriately derived equations for the quantity ζ_u and elliptic relaxation function f_u (the latter equation was originally formulated by using the pressure-strain model of Speziale et al., 1991, Hanjalic et al., 2004):

$$\frac{\partial \zeta_u}{\partial t} + U_j \frac{\partial \zeta_u}{\partial x_j} = f_u - \frac{\zeta_u}{k_u} P_u + \frac{\zeta_u}{k_u} \varepsilon_u (1 - f_k) + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{\zeta_u}} \right) \frac{\partial \zeta_u}{\partial x_j} \right]; \quad \sigma_{\zeta_u} = \sigma_{\zeta} \frac{f_k^2}{f_{\varepsilon}} \quad (12)$$

$$L_u^2 \nabla^2 f_u - f_u = \frac{1}{T_u} \left(C_1 + C_2 \frac{P_u}{\varepsilon_u} \right) \left(\zeta_u - \frac{2}{3} \right) \quad (13)$$

with f_u approaching asymptotically zero value at the wall ($y \rightarrow 0$): $f_{wall} = \lim_{y \rightarrow 0} (-2\nu \zeta_u / y^2)$.

Here, T_u represents a switch between the turbulent time scale $\tau = k_u / \varepsilon$ and the Kolmogorov time scale $\tau_K = (\nu / \varepsilon)^{1/2}$:

$$T_u = \max \left[\frac{k_u}{\varepsilon}, C_{\tau} \left(\frac{\nu}{\varepsilon} \right)^{1/2} \right] \quad (14)$$

The corresponding length scale L_u is obtained as a switch between the turbulent and Kolmogorov length scales:

$$L_u = C_L \max \left[\frac{k_u^{3/2}}{\varepsilon}, C_{\eta} \left(\frac{\nu^3}{\varepsilon} \right)^{1/4} \right] \quad (15)$$

The values of the coefficients appearing in the model equations (Equations 6,7,8,9,10,11,12,13,14,15) are specified in Table 1.

Table 1. Model coefficients

C_1	C_2	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	C_{μ}
0.4	0.65	$1.4 \left(1 + \frac{0.045}{\sqrt{\zeta_u}} \right)$	1.9	0.22

σ_k	σ_{ε}	σ_{ζ}	C_{τ}	C_L	C_{η}
1.0	1.3	1.2	6.0	0.36	85

By setting $f_k = 1$ ($k_u = k$), implying the spectral cut-off shift towards low-frequency (productive) region of the turbulence spectrum pertinent to the RANS method, the PANS equations (Equations 6,7,8,9,10,11,12,13,14,15) governing the unresolved turbulent properties reduce to the conventional RANS/URANS model equations describing the dynamics of the fully-modeled turbulent quantities.

Both computational models are applied in conjunction with the so-called universal wall treatment. This method blends the integration up to the wall (exact boundary conditions) with the high Reynolds number wall functions, enabling well-defined boundary conditions irrespective of the position of the wall-closest computational node. This method is especially attractive for computations of industrial flows in complex domains where higher grid flexibility, i.e. weaker sensitivity against grid non-uniformities in the near wall regions, featured by different mean flow and turbulence phenomena (flow acceleration/deceleration, streamline curvature effects, separation, etc.), is desirable. Popovac and Hanjalic (2007) proposed the so-called compound wall treatment with a blending formula (motivated by the proposal of Kader, 1981) for the quantities specified at the central node P of the wall-closest grid cell as

$$\phi_P = \phi_{\nu} e^{-\Gamma} + \phi_t e^{-1/\Gamma} \quad (16)$$

where ' ν ' denotes the viscous and ' t ' the fully turbulent value. The modeling of the blending function Γ is of crucial importance, keeping in mind that the largest error can be introduced if the wall-next computational cell is situated in the buffer region of the boundary layer: $5 \leq y^+ < 30$. The variable ϕ applies here to the wall shear stress, production and dissipation of the turbulence kinetic energy. A somewhat simplified approach was introduced under the name "Hybrid Wall Treatment" in the numerical code AVL FIRE, [1]. Whereas the original compound wall treatment of Popovac and Hanjalic (2007) includes the tangential pressure gradient and convection, a simpler approach utilizing the standard wall functions as the "upper" bound is used presently. Another departure from the original formulation relates to the calculation of the dissipation rate as proposed by Basara (2006).

Solution Methodology

The model equations are solved by using the Computational Fluid Dynamics (CFD) code AVL FIRE, [1]. The discretization of the governing differential equations is obtained using a cell-centered finite volume approach. In the method used here, the governing equations are integrated term-by-term over the 'polyhedral' control volumes, see Figure 1. Such discretization practice was explored in other publications starting with Demirdzic and Muzafferija (1995), and continuing with Ferziger and Peric (1997), Marthur and Marthy (1997) etc. Hence, the method has been applied and proved on various applications. The method rests on the integral form of the general conservation law.

Thus, for the control volume with the outward surface (cellface) vectors $A = A_k i_k$, all governing equations can be written as:

$$\underbrace{\frac{d}{dt} \int_V \rho \phi dV}_{\text{Rate of change: } R} + \underbrace{\oint_A \rho \phi U_k dA_k}_{\text{Convection: } C} = \underbrace{\oint_A \Gamma_{\phi}^{kk} \frac{\partial \phi}{\partial x_k} dA_k}_{\text{Diffusion: } D} + \underbrace{\int_V s_{\phi}^V dV + \oint_A s_{\phi k}^A dA_k}_{\text{Sources: } S} \quad (17)$$

where a general variable $\phi(x_k, t)$ can represent either scalars or vector and tensor field components. Here, the Cartesian coordinate system (x, y, z) with the unit basis vectors (i, j, k) is used and tensor notation is employed. In the above equation, ρ is the fluid density, t is time, U_k are components of the fluid velocity vector, Γ_{ϕ}^{kk} is the diffusion coefficient pertinent to the variable ϕ (in this case repeated indices do not imply summation), s_{ϕ}^V and $s_{\phi k}^A$ are the volumetric and surface source terms, respectively.

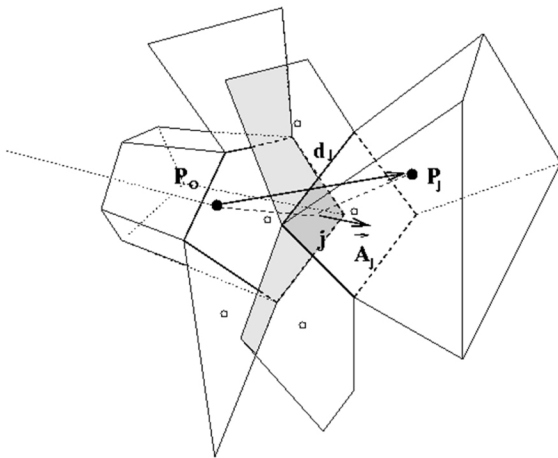


Figure 1. Polyhedral control volume

The cell-face based connectivity and interpolation practices for gradients and cell-face values are introduced to accommodate an arbitrary number of cell faces. A second-order midpoint rule is used for integral approximation. In terms of the notation shown in [Figure 1](#) for a typical grid cell P with the volume V surrounded by its neighbors P_j , the discretized control volume equation can be written as:

$$\frac{d}{dt}(\rho_P V_P \phi_P) + \sum_{j=1}^{n_f} C_j - \sum_{j=1}^{n_f} D_j = (s_{\phi}^V)_P V_P + \sum_{j=1}^{n_f} (s_{\phi k}^A A_k)_j \quad (18)$$

where C_j and D_j represent convective and diffusion transport through the face j , respectively; n_f is the number of cell-faces. With the use of linear interpolation, the cell face values can be calculated as

$$\phi_j = f_j \phi_P + (1 - f_j) \phi_{P_j} \quad (19)$$

where f_j is the cell-face interpolation factor. The cell gradients can be calculated by using either the Gauss' theorem

$$\nabla \phi_P \approx \frac{1}{V_P} \sum_{j=1}^{n_f} \phi_j \bar{A}_j \quad (20)$$

or a linear least-square approach, e.g. [Muzaferija \(1994\)](#). Using Gauss' theorem and utilizing simple mathematical reconstructions to replace vertex values with the contributions from cells P and P_j only, one can arrive at the following formula for the cell-face gradient:

$$\nabla \phi_j = \bar{\nabla} \phi_j + \frac{\bar{A}_j}{\bar{A}_j \bar{d}_j} \left[(\phi_{P_j} - \phi_P) - \bar{\nabla} \phi_j \cdot \bar{d}_j \right] \quad (21)$$

where the overbar gradient is determined by linear interpolation

$$\bar{\nabla} \phi_j = f_j \nabla \phi_j + (1 - f_j) \nabla \phi_P \quad (22)$$

These interpolation practices are used to derive the terms in the discretized [Equation \(18\)](#). A deferred correction approach is also used for the treatment of convection fluxes, thus

$$C_j = \dot{m}_j \phi_j^{UDS} + \gamma_{\phi} |\dot{m}_j| \phi_j (\phi_{P_j} - \phi_P) \quad (23)$$

where mass flux $\dot{m}_j$ is given as $\dot{m}_j = \rho U_j A_j$ (no summation over " j "), and γ_{ϕ} is the blending factor between UDS (1st order Upwind Differencing Scheme) and higher order scheme ($0 \leq \gamma \leq 1$). The underlined term is calculated by using values from the previous iteration step. The flux limiter ϕ_j is provided by the higher order differencing scheme used to ensure a bounded solution. In this work, the SMART (AVL SMART as proposed by [Przulj and Basara, 2001](#)) and MINMOD (2nd order convective scheme; [Sweby, 1984](#)) schemes were used as a compromise between accuracy and convergence properties. The rate of change (see [Equation 17](#)) can be discretized by two implicit schemes, namely first order accurate Euler (two-time-level) scheme and second order accurate three-time-level scheme.

The outcome of all schemes presented above is a set of algebraic equations: one for each control volume and for each transport equation. An algebraic equation can be written concisely as:

$$a_P \phi_P = \sum_{j=1}^{n_i} a_j \phi_{P_j} + S_{\phi}, \quad (24)$$

where n_i is the number of internal cell-faces; a_P and a_j are coefficients and S_{ϕ} is the source term. Thus, for a computational domain with M control volumes, a system of $M \times N$ algebraic equations needs to be solved for N dependent variables ϕ . Considering the non-linearity and coupling of equations, they are solved by the segregated SIMPLE-like algorithm of [Patankar and Spalding \(1972\)](#) (see also [Ferziger](#)