Modeling Cornering Aerodynamics - A Review

Hugues PERRIN
Department of Aerospace Engineering, École de technologie supérieure, Montréal, Canada

Abstract

This work aims towards providing a review of the literature tied to modeling aerodynamics of an FSAE car, from Computational Fluid Dynamics (CFD) theory to experimental methodologies. It is part of a Master's thesis on FSAE cornering aerodynamics, as are references FETS-2025-02 [1] and FETS-2025-03 [2].

Introduction

Averaging 240 hours of Wind Tunnel testing and dozens of Mega Allocation Unit hours (MAUh) of CFD computations, F1 aerodynamics engineers embody the pinnacle of aerodynamics in motorsport with open-wheel prototypes reaching speeds upwards of 300km/h and lateral accelerations of 5 to 6 Gs. However, these performances could only be reached because of the decades of intensive research and engineering in fluid mechanics, starting in the years 1820 with Claude-Louis Navier and Georges Gabriel Stokes and the creation of the Navier-Stokes equations, diverging from the first time from the Euler inviscid fluid model.

This paper will review, in a logical order, every theory and paper relevant to our end goal of modeling and simulating the aerodynamics of a FSAE prototype in straight line and cornering situation.

Literature Review

Fluid Mechanics Historical Theories

In 1738, Daniel Bernoulli formulates the Bernoulli principle in *Hydrodynamica* [3], a key relation in modern aerospace and ground vehicle aerodynamics stating that, along a streamline of an ideal fluid $p + \frac{1}{2}\rho v^2 + \rho g z = cst$. This principle also lays the fundamental hypotheses behind the Venturi effect, and ground effect commonly referred to in modern high aerodynamic motorsport.

In 1756, Leonhard Euler completes Bernoulli's work by deriving the Euler equations — the differential form of momentum conservation for an inviscid fluid [4] — together with continuity. The modern Euler model is characterized by the following equation :

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} \right) = -\nabla p + \rho \mathbf{g}$$

By the mid 18th century, the existence of both a continuity of mass equation and the momentum of balance for perfect fluids laid the ground for modern fluid mechanics. To this day, the Euler model is useful for first approximations in aerospace and research application, and is well adapted for computational fluid dynamics.

However, this model was questioned by Jean le Rond d'Alembert in 1752, highlighting the differences between model and experimental data. In particular, following Euler's equation would lead to physically absurd results such as zero drag on a body. [5] The D'Alembert paradox proves the existence of a missing component in fluid mechanics, later discovered as viscosity by joint work between Poiseuille and Hagen in the late 1830s. Poiseuille and Hagen confirmed the existence and influence of viscosity in real flows, developing the Hagen-Poiseuille law linking pressure drop to viscosity in laminar pipe flows. [6] Stokes later proved that the Hagen-Poiseuille is an exact solution of the Navier–Stokes equations for axisymmetric, fully developed, incompressible flow in a circular tube, highlighting Hagen-Poiseuille as the first modern viscous flow formulation.

Unfortunately, this law only applies for very specific conditions (pipe flow, low-Re) and would benefit from a generalization for flows interacting with solids at higher Reynolds, typically for aeronautical applications. Prandtl's 1904 research [7] theorizes the necessary elements to lay the foundations of boundary layer works by remarking the effects of friction only apply to a thin layer adjacent to the solid's surface. Within that thin layer are confined great viscous stresses and velocity gradients while the outer region of the flow roughly behaves as the Euler inviscid formulation predicts. This discovery solved the D'Alembert paradox [5] by tying the experimental rise in drag to viscous shear and separation processes within the boundary layer of the studied object.

The first viscous equations of motion were established by Claude-Louis Navier in two of his thesis in 1822 and 1823 [8] where he modifies the Euler formulation to include viscosity, trying to derive the extra terms from a simpler molecular attraction model. In this paper he assumes a newtonian relation between stress and velocity gradients which ultimately leads to the first formulation of the modern Navier-Stokes equations, although historical work (particularly from Darrigol [9, 10]) highlights a severe lack of rigor, jumping from a molecular assumption to continuum Partial Derivative Equations (PDEs) describing macroscopic stress-strain relations. Navier's successors (Cauchy, Poisson, Saint-Venant and ultimately Stokes) judged that his work lacked too much intermediate steps to be properly considered, despite the novel formulation.

Based on Cauchy's mathematical toolbox including a stress tensor formulation [11], being a general way of representing stress in a deformable medium and momentum balance for a continuum, and

Newton's second law of motion, Stokes was able to further improve Navier's work in the 1840s with his paper "On the theories of the internal friction of fluids in motion" [12]. Stokes particularly used continuum arguments instead of molecular models to derive the form of viscous stress tensors for an isotropic Newtonian fluid and inserted it into Cauchy's general momentum balance to obtain the modern form of the Navier-Stokes equations. He also theorizes the need for different boundary conditions, like the no-slip wall condition to conform these equation to Prandtl's research.

Using Einstein's index formatting, the modern Navier-Stokes equations are defined as :

$$\rho \frac{D\mathbf{u}}{Dt} = \rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} \right) = \underbrace{-\widetilde{\nabla} p}_{\text{Pressure}} + \underbrace{-\widetilde{\rho}\,\mathbf{g}}_{\text{Gravity}} + \underbrace{-\widetilde{\nabla} \cdot \boldsymbol{\tau}}_{\text{Viscosity}}$$

Subsequent research work of Blasius [13] and Von Karman [14] introduced similar solutions and integral formulations for laminar boundary layers while suggesting empirical laws based on friction-factor correlations for turbulent boundary layers modeling, all correlated to the Navier-Stokes formulation. This research on specific solutions, whether they're for internal or external, laminar or turbulent flows and empirical laws being scattered across countries and journals, Schlichting offered a synthesis of boundary layer theories into an unified theory [15] that lays the ground for all modern applications of wall-bounded flows research since its publication in 1979.

Fundamentals of Turbulence

After the Navier-Stokes equations were universally adopted as the most general formula to predict fluid flows, research shifted towards turbulent flows and turbulence modeling. Essentially, if the Navier-Stokes equations describe exactly the behavior of fluids, their complexity make it very difficult, if not impossible, to solve them for complex, high-Re, compressible or even transsonic flows. In fact, even modern computing power through the means of heavily parallelized computing and high-performance computing (HPC) struggle to precisely compute exact Navier-Stokes due to its very complex nature. Direct Numerical Simulation (DNS) is generally reserved for research cases, helping the development of new turbulence models for Large Eddy Simulation (LES) models or RANS models [16].

The main problem leading to increasing computational costs of Navier-Stokes simulation lies on how scales of turbulence are modeled and calculated. The reduced computational power at the time of Navier and Stokes work highlighted the need for simpler models based on the Navier-Stokes equations that would be easier to implement in industrial settings without sacrificing simulation accuracy. From that postulate, research focus on intelligent hypotheses that would simplify the Navier-Stokes equations enough to render them a useful tool for engineers.

The first hypothesis on turbulence modeling appeared with Boussinesq in his *Essai sur la théorie des eaux courantes*[17] where he introduced the idea that turbulent momentum transport could be modeled as an "eddy viscosity" with turbulent shear stress proportional to mean shear stress. This idea became widely known as the "Boussinesq hypothesis" and is the conceptual ancestor of most of the all eddy-viscosity RANS turbulence models.

In 1883, Osborne Reynolds discovers a critical parameter controlling the transition between laminar and turbulent flow by visualizing flows in glass pipes with dye filaments [18]. This number, later named the Reynolds number, relies on comparing inertial and viscosity effects for known flow conditions, effectively predicting to an extent if the flow will be in a laminar, turbulent or transitional state. Later, in 1895, Reynolds decomposes the velocity in the Navier-Stokes equations into a mean and a fluctuating component, featuring extra components for the fluctuations called Reynolds stresses [19]. The Reynolds decomposition and the Reynolds-Averaged Navier-Stokes equations are key elements in modern CFD and fluid dynamics, allowing for the computation of complex cases with reduced computational costs.

To obtain the balance equations for the mean quantities, the instantaneous flow variables are decomposed according to Reynolds' rule as

$$u_i = \overline{U}_i + u_i' \qquad p = \overline{P} + p' \qquad \tau_{ij} = \overline{\tau}_{ij} + \tau_{ij}'$$

These decompositions are then introduced into the balance equations, like the mass conservation

$$\nabla \cdot \mathbf{u} = \frac{\partial u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\overline{U}_i + u_i'\right) = 0$$

for which applying the averaging operator and subtracting from the previous equation gives

$$\frac{\partial \overline{U}_i}{\partial x_i} = 0 \qquad \frac{\partial u_i'}{\partial x_i} = 0$$

The decomposition implies that the mean and fluctuating velocity fields are incompressible, which will prove useful to transform the Navier-Stokes equations. Specifically, the conservative form of the Navier-Stokes equations using this incompressibility relation is written as

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$

The introduction of the Reynolds decomposition of the velocity and pressure components leads to

$$\rho \left[\frac{\partial}{\partial t}\left(\overline{U}_i + u_i'\right) + \frac{\partial}{\partial x_j}\left(\overline{U}_i + u_i'\right)\left(\overline{U}_j + u_j'\right) \right] = -\frac{\partial}{\partial x_i}\left(\overline{P} + p'\right) + \frac{\partial}{\partial x_j}\left(\overline{\tau}_{ij} + \tau_{ij}'\right)$$

and applying the averaging operator finally gives

$$\frac{\partial \overline{U}_i}{\partial t} + \frac{\partial (\overline{U}_i \overline{U}_j)}{\partial x_j} + \frac{\partial (u_i' u_j')}{\partial x_j} = -\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial \overline{\tau}_{ij}}{\partial x_j}$$

For (i=1,2,3), this form is known as the Reynolds equations. An alternate form, highlighting the Reynolds stress tensor is

$$\rho \left(\frac{\partial \overline{U}_i}{\partial t} + \frac{\partial (\overline{U}_i \overline{U}_j)}{\partial x_j} \right) = -\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\overline{\tau}_{ij} - \rho\, u_i' u_j'\right)$$

where $\rho u_i' u_j'$ is analogous to an additional stress due to turbulent fluctuations called the Reynolds stress tensor. The Reynolds stress can also be described as

$$\rho\, u_i' u_j' = \rho \begin{pmatrix} u'^2 & u'v' & u'w' \\ u'v' & v'^2 & v'w' \\ u'w' & v'w' & w'^2 \end{pmatrix}$$

where the diagonal components are referred to as normal stresses and off-diagonal components as shear stresses. Shear stress terms are typically zero when studying laminar flow cases, but cannot be neglected when facing turbulent conditions.

This tensor can also be written in a PDE form as

$$\rho \left(\frac{\partial \overline{U}_i}{\partial t} + \overline{U}_j \frac{\partial \overline{U}_i}{\partial x_j} \right) = \rho \frac{D\overline{U}_i}{Dt} = \frac{\partial \overline{\sigma}_{ij}^t}{\partial x_j}$$

or written in a vector form as

$$\rho \frac{D\overline{\mathbf{U}}}{Dt} = \nabla \cdot \boldsymbol{\sigma}_t$$

where σ^t_{ij} is the tensor of mean turbulent stresses

$$\overline{\sigma}^t_{ij} = -P\,\delta_{ij} + \overline{\tau}_{ij} - \rho\, u'_i u'_j$$

However, although responding very well to laminar flow cases, the introduction of turbulence in RANS-driven simulation cases creates a new critical issue, known as the "closure problem". As Pope [20] and Wilcox [**?**] later described it, for turbulent flows, the RANS equations present 4 equations (conservation of the mass and 3D momentum) for 10 variables : the 6 non-diagonal terms of the Reynolds stress tensor are the ones that create the closure problem, and the ones that induce a need for additional equations that will model turbulence.

The Boussinesq hypothesis is the first one trying to tackle the turbulence problem with the introduction of the eddy viscosity to describe the Reynolds tensor [17]. The tensor would be written

$$\rho\, u'v' = -\mu_t\, \frac{\partial \overline{U}}{\partial y}$$

where μ_t is the eddy viscosity (or turbulent viscosity).

In tensor form, the Boussinesq approximation may also be written as

$$\rho\, u'_i u'_j = -\mu_t \left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i} \right) + \frac{2}{3}\,\rho\,\overline{k}\,\delta_{ij}$$

where $\overline{k}$ is the turbulent kinetic energy per unit of mass, defined as

$$\overline{k} = \frac{1}{2}\, u'_k u'_k = \frac{1}{2}\left(u'^2 + v'^2 + w'^2\right)$$

The isotropic term $\frac{2}{3}\,\rho\,\overline{k}\,\delta_{ij}$ ensures that the trace of the Reynolds stress tensor is consistent with the definition of $\overline{k}$, i.e.

$$\rho\,\overline{k} = \frac{1}{2}\,\rho\, u'_i u'_i$$

The turbulent viscosity μ_t is, in principle, a local function of space and time in contrast to the molecular viscosity which is a fluid property. Although this closure proposition is conceptually simple and widely used, it is also restrictive as the turbulence is inherently anisotropic and governed by energy cascade and hydrodynamic instabilities for which a linear relationship between Reynolds stresses and mean velocity gradients has no fundamental justification.

Later, in 1925, Prandtl introduced a developed turbulence, also know as the mixing-length turbulence theory [21]. By looking at pipe flows and boundary layers, Prandtl theorizes that a fluid parcel can move a distance l_m, analogous to a mean free path, while keeping its momentum before mixing. The general equations underlined by Prandtl's work can be summed up as

$$-\rho\,\overline{u'v'} = \mu_t\,\frac{\mathrm{d}U}{\mathrm{d}y} = \rho\,\nu_t\,\frac{\mathrm{d}U}{\mathrm{d}y} \;\approx\; \rho\,\ell_m^2 \left|\frac{\mathrm{d}U}{\mathrm{d}y}\right| \frac{\mathrm{d}U}{\mathrm{d}y}$$

This model is a prototype one-equation, eddy viscosity based model that laid the ground for more complex formulations such as the Prandtl-Kolmogorov, $k-\epsilon$ and wall functions. It is also the first engineeringable closure model for the Reynolds stress in wall flows a is still a conceptually used law in more modern closure models.

Parallel to the research aiming towards closing the RANS equations, researchers like Leray questioned the viability of the original Navier-Stokes, specifically the uniformity and the existence of smooth solutions that would validate the Navier-Stokes equations as a perfect fluid model [22]. However, the pure PDE theory work achieved by Leray only proved the existence of global weak solutions, and local smooth solutions that couldn't be generalized due to hypothetical energy surges and singularities. His work gave a rigorous foundation for the idea that turbulent flows may have a highly irregular structure with chaotic singularity appearances. This research is often cited as the key results leading to the classification of the Navier-Stokes equation's regularity as a millennial problem, deemed impossible to solve.

After his famous 1941 similarity theory [23], Kolmogorov proposed a two-equation turbulence model for high-Re flows by using a first transport equation to model the turbulent kinetic energy k and a second one for the turbulent frequency ω [24]. This Kolmogorov formulation is the ancestor of the more modern $k-\omega$ turbulence model introduced by Wilcox.

The first transport equation focusing on the turbulent kinetic energy can be written as

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho U_j k)}{\partial x_j} = P_k - \beta^*\,\rho k\omega + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right]$$

while the second, turbulent frequency focused, transport equation is defined by

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho U_j \omega)}{\partial x_j} = \alpha\,\frac{\omega}{k} P_k - \beta\,\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right]$$

Those 2 transport equations are tied with Boussinesq hypothesis

$$P_k = 2\mu_t S_{ij} S_{ij}, \qquad \mu_t = \rho\frac{k}{\omega}$$

and the Reynolds stress tensor is formulated as

$$\rho\,\overline{u'_i u'_j} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$

This closure model was historically the first to introduce the influence of frequency on turbulence and use ω terms in its transport equation. This model was later mixed with Prandtl's mixing length theory to create the Prandtl–Kolmogorov one-equation model [25] in 1945.

This model introduces a transport equation for the turbulent kinetic energy k and couples it with Prandtl's length scale to define an eddy viscosity as $\nu_t \sim l\sqrt{k}$. This model is a direct evolution of Prandtl's mixing-length idea that was turned into a dynamical model were the energy level is solved by a PDE rather from a local mean gradient.

Reynolds stresses are defined by

$$-\rho\overline{u'_i u'_j} = 2\mu_t S_{ij} - \frac{2}{3}\rho k\,\delta_{ij}$$

Eddy viscosity formulation is still derived from Boussinesq hypothesis such as

$$\nu_t = C_\mu\,\ell\,\sqrt{k} \qquad \mu_t = \rho\nu_t$$

In 1951 explores another way of modeling turbulence in his paper *Statistische Theorie nichtstationärer und nicht homogener Turbulenz* [26] by writing transport equations for each componenents of the Reynolds stress tensor as $R_{ij} = \overline{u'_i u'_j}$, derived from the fluctuating momentum equations. These equations contain production terms, a pressure-strain correlation, a turbulent viscous diffusion and dissipation terms. In this research, Rotta focused on the pressure-strain term, theorizing a "return to isotropy" model where anisotropic stresses relax towards isotropy at a rate proportionnal to their deviation from isotropic form.

This formulation is the first of its kind that does not collapse the Reynolds stresses to a simple eddy viscosity and follow Boussinesq hypothesis, but rather offer PDEs to determine each Reynolds stress. Rotta's Reynolds Stress Model (RSM) is a second moment model where the tensor itself is primary unknown. The pressure-strain correlation is exposed as

$$\phi_{ij} = -C_R\,\frac{\varepsilon}{k}\left(R_{ij} - \frac{2}{3}k\,\delta_{ij}\right)$$

implying that anisotropy decays back towards isotropy with a time scale of k/ϵ.

Modern RSM models are generally used as benchmarks for "weaker" Boussinesq-based turbulence models due to its higher fidelity of real turbulence. However, this increased fidelity comes with a higher computational cost, requiring to solve 7 PDEs for turbulence instead of 1 or 2 for more classical eddy viscosity based models. It is generally good practice for complex, high gradient, high-Re flows to set a benchmark through a RSM-based solution to ensure the validity of simpler turbulence models.

In the early 1970s, Jones and Launder proposed a low-Re $k-\epsilon$ model [27, 28], quickly adapted to high-Re situation through the addition of wall functions by Launder and Spalding's work [29] that ultimately offered the modern standard $k-\epsilon$ formulation widely used in engineering to this day.

This model is based on two transport equations, one for the turbulent kinetic energy $\overline{k}$

$$\frac{\partial(\rho\bar{k})}{\partial t} + \bar{u}_j \frac{\partial(\rho\bar{k})}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \bar{k}}{\partial x_j}\right] + P_k - \rho\bar{\varepsilon}$$

and the second equation to calculate the dissipation rate $\overline{\epsilon}$

$$\frac{\partial(\rho\bar{\varepsilon})}{\partial t} + \frac{\partial(\rho u_i\bar{\varepsilon})}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_e}\right)\frac{\partial \bar{\varepsilon}}{\partial x_j}\right] + \frac{\bar{\varepsilon}}{k}\left(C_{1e}P_k - C_{2e}\rho\bar{\varepsilon}\right)$$

with both parameter cleanly tied up in the definition of the eddy viscosity

$$\mu_t = \rho C_\mu \frac{\overline{k}^2}{\overline{\epsilon}}$$

and a Reynolds stress implementation still being strictly Boussinesq's.

The standard $k-\epsilon$ closure model is widely used to simulate turbulent flows. However, for areas bounded walls, flow dynamics change as the turbulence is severely cushioned by wall interaction mechanics, modifying flow characteristics beyond the envelope of the model. For some complex boundary layers and high shear flows, the standard $k-\epsilon$ model shows some limits and inaccuracies.

Inspired by the models of Jones, Launder and Spalding, Wilcox revisited Kolmogorov scale-determining theory to formulate a new two-equations closure model based on Boussinesq eddy viscosity and the concept of a dissipation rate dictated by Kolmogorov turbulent frequency [30].

This model theorizes that the larger scales of the turbulence are determined by the dissipation rate and turbulent kinetic energy. The rate $\omega = \epsilon/k$ serves as the characteristic scale for larger vortices and represents the frequency at which the kinetic energy is dissipated in the system. Moreover this model makes the hypothesis of a linear relation between turbulent viscosity and mean shear without using the same eddy viscosity definition as precedent work

$$\mu_t = \alpha\frac{\rho k}{\omega}$$

Transport equations for k and ω are respectively

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\Gamma_k \frac{\partial k}{\partial x_j}\right] + G_k - Y_k$$

and

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left[\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right] + G_\omega - Y_\omega$$

The Wilcox $k-\omega$ closure model proved high correlation and efficiency for near-wall behaviors without the use of wall functions, improving wall shear and heat transfer predictions in a compact, robust model. However, it also showed limits and inaccuracies for free-shear flows, typically outside of the boundary layer and higher shear zones.

By 1988, two models co-exist in fluid dynamics engineering : the $k-\epsilon$ model, efficient for freestream applications and low-shear situations, and the $k-\omega$ model, more accurate in highs-hear but struggling to compute less constrained flows. In 1994, Menter designed a hybrid model relying on those two sets of equations so each of them could compensate the flaws of the other by using a blending function [31]. This model, called $k-\omega$ Shear Stress Transprt (SST), uses $k-\omega$ set of equations near walls for its great wall behavior, but is gradually transformed into a $k-\epsilon$ for the outer flow regions as to avoid $k-\omega$'s free stream sensitivity. As a failsafe, the eddy viscosity is also limited based on local shear stress to improve boundary layer separation prediction.

The $k-\omega$ SST model features the same transport equation as the $k-\omega$ closure model, with the addition of two blending functions F_1 and F_2 that switch coefficients between their $k-\omega$ and $k-\epsilon$ forms.

The transport equation for $\overline{k}$ is defined as

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P_k - \beta^*\,\rho k\omega + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k\mu_t)\,\frac{\partial k}{\partial x_j}\right]$$

while the transport equation for $\overline{\omega}$ can be written as

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j\omega)}{\partial x_j} = \alpha\,\frac{\omega}{k}\,P_k - \beta\,\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega\mu_t)\,\frac{\partial \omega}{\partial x_j}\right] 2(1-F_1)\,\rho\,\sigma_{\omega 2}\,\frac{1}{\omega}\,\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

And the blending functions F_1 and F_2 are defined as

$$F_1 = \tanh\left(\Phi_1^4\right) \qquad F_2 = \tanh\left(\Phi_2^2\right)$$

Typically, when $F_1 = 0$ the pure $k-\omega$ model is used, before blending with the $k-\epsilon$ model until $F_1 = 1$ is reached and a pure $k-\epsilon$ model is used for calculation.

$$\Phi_1 = \min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\sigma_{\omega 2}k}{\beta^* y^2\omega}\right]$$

$$\Phi_2 = \max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right)$$

Those functions are used to determine the level of blending between the $k-\omega$ and $k-\epsilon$ models as

$$\phi = F_1\,\phi_1 + (1-F_1)\,\phi_2 \qquad \phi \in \{\alpha,\ \beta,\ \sigma_k,\ \sigma_\omega\}$$

Finally, the eddy turbulent viscosity and the strain magnitude are defined as

$$\mu_t = \frac{\rho\,a_1\,k}{\max(a_1\omega,\ SF_2)} \qquad S = \sqrt{2\,S_{ij}S_{ij}}$$

Although being slightly more computationally expensive due to the calculation of the blending factors, the $k-\omega$ SST is widely used for complex 3D aerodynamics applications due to its high correlation to more complex RANS-based, LES-based models and experimental data. The blending factors being calculated via the wall distance, this ensures a smooth transition without the need for damping functions and without creating local discontinuities.

The last model used in modern CFD applications was proposed by the joint work of Spalart and Allmaras in 1992 [32]. In their paper, they offered a new 1-equation closure model based on Boussinesq eddy viscosity. Instead of solving for k, ϵ or ω, it solves a single transport equation for a turbulent kinematic viscosity $\tilde{\nu}$.

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = C_{b1} \left(1 - f_{t2}\right) \tilde{S}\, \tilde{\nu}$$
$$+ \frac{1}{\sigma} \left[\frac{\partial}{\partial x_j} \left((\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + C_{b2} \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right]$$
$$- \left[C_{w1}\, f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right] \left(\frac{\tilde{\nu}}{d} \right)^2$$

This transport equation translates the convection of $\tilde{\nu}$ by the mean flow where its production is proportional to a modified vorticity magnitude and diffusion/destruction terms depend on the distance to the wall.

The eddy viscosity relation

$$\nu_t = \tilde{\nu}\, f_{v1} \qquad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3} \qquad \chi = \frac{\tilde{\nu}}{\nu}$$
$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$

and the Reynolds stresses

$$\tau_{ij}^{\text{turb}} = 2\rho \nu_t S_{ij} \qquad S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

complete the overall governing equations of the model.

The Spalart-Allmaras model often serves as a baseline for external flow aerodynamics for its very low computational cost despite the need for a very fine mesh around walls and in wake dominated zones. Its accuracy starts to fail when confronted to strong adverse gradients, large separation or strong swirls and often under-predicts separation leading to wrongful prediction of aerodynamic quantities.

The early 2000s witnessed lots of applications of all these equation models for engineering purposes, with many authors describing the dos and don'ts of CFD and turbulence modeling. Spalart was the first discussing the strategies for CFD modeling and simulation [33], highlighting the key differences and uses for RANS, LES, DES and DNS, especially in engineering. His practical comments on computational power and the predominance of RANS - eddy viscosity codes are still relevant to this day.

Several books positioned themselves as CFD turbulence guides. Introductory-level like Davidson's *Turbulence: An Introduction for Scientists and Engineers* [34] offered explanations on turbulence fundamentals and RANS closure while showing canonical comparisons in channels or boundary layers while Wilcox's *Turbulence Modeling for CFD* [35] is the RANS modeling "bible", expliciting what models are best suited for different types of cases and the exact demonstrations leading to each transport equations. Ferziger's *Computational Methods for Fluid Dynamics* [36] was one of the first book to also cover the interaction between the discretizations of the domain, the boundary conditions and the PDEs models.

Solving the Governing Equations

The practical use of CFD has progressed in parallel with two main ingredients: advances in turbulence modeling on one side, and robust numerical methods on the other. From a mathematical point of view, the Navier-Stokes equations combine hyperbolic, parabolic and elliptic behavior, and the balance between these three aspects dictates how the solver must be built. The hyperbolic character of inviscid compressible flow already appears in Riemann's work on finite-amplitude waves in gases [37], where shocks and rarefactions are analyzed in a one-dimensional setting. Problems of this type, now referred to as Riemann problems, later became the building blocks of conservative finite-volume schemes for hyperbolic conservation laws.

A second key ingredient is numerical stability. Courant, Friedrichs and Lewy introduced a simple but fundamental restriction that links the time step, the grid spacing and the local wave speeds, now known as the CFL condition [38]. In one spatial dimension, it is often written in the form

$$\mathrm{CFL} = \frac{u\,\Delta t}{\Delta x} \;\lesssim\; \mathrm{C_{crit}}$$

and it still provides the basic guideline for explicit time stepping in CFD solvers.

In one dimension, a generic conservation law for a variable $u(x,t)$ can be written as

$$\frac{\partial u}{\partial t} + \frac{\partial f(u)}{\partial x} = 0$$

where $f(u)$ is the associated flux. The corresponding Riemann problem is defined by an initial discontinuity,

$$u(x,0) = \begin{cases} u_L, & x < 0 \\ u_R, & x > 0 \end{cases}$$

whose evolution produces shocks, rarefactions and contact discontinuities. The idea of treating each cell interface as a local Riemann problem is at the core of modern upwind finite-volume methods.

On this basis, Godunov proposed the first fully conservative upwind scheme for non-linear hyperbolic systems [39]. At each interface, left and right states are used to define a local Riemann problem, and the resulting solution provides the numerical flux. This guarantees correct shock propagation and conservation of mass, momentum and energy across discontinuities, and it embeds the wave structure of the governing equations directly in the scheme. Later high-resolution methods can largely be seen as extensions of this idea, combining the robustness of Godunov's approach with higher-order spatial accuracy.

For a finite-volume discretization with cell-averaged values u_i^n at time t^n, the semi-discrete update takes the form

$$u_i^{n+1} = u_i^n - \frac{\Delta t}{\Delta x} \left(F_{i+\frac{1}{2}}^n - F_{i-\frac{1}{2}}^n \right)$$

where $F_{i\pm\frac{1}{2}}^n$ are numerical fluxes at the cell faces. These fluxes are obtained from local Riemann problems between reconstructed left and right states $u_{L,i\pm\frac{1}{2}}$ and $u_{R,i\pm\frac{1}{2}}$.

Van Leer extended this approach by introducing the MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) reconstruction [40]. Instead of the piecewise-constant representation used in the original Godunov scheme, MUSCL uses limited linear reconstructions inside each cell. This leads to schemes that are formally second-order accurate while remaining total-variation-diminishing near shocks and strong gradients. MUSCL-type reconstructions are now standard in many compressible RANS solvers.

In a MUSCL scheme, the reconstructed left and right states at an interface are built from limited slopes. For the interface $i + \frac{1}{2}$, the left state is written as

$$u_{i+\frac{1}{2},L} = u_i + \frac{1}{2}\, \phi(r_i)\; (u_i - u_{i-1})$$

where $\phi(r)$ is a slope limiter and

$$r_i = \frac{u_{i+1} - u_i}{u_i - u_{i-1}}$$

is a local smoothness indicator. A similar expression is used for the right state $u_{i+\frac{1}{2},R}$ in the neighboring cell.

Time integration strategies have evolved alongside these spatial discretizations. MacCormack proposed a two-step predictor–corrector scheme for hyperbolic systems, which became a widely used explicit method in early compressible-flow calculations [41]. The scheme combines a forward-difference predictor and a backward-difference corrector and achieves second-order accuracy in both time and space for smooth solutions, while remaining relatively simple to implement. For three-dimensional flows on complex meshes, explicit multi-stage Runge–Kutta schemes are now more common, especially in combination with finite-volume methods.

For the conservation law $\partial u/\partial t + \partial f(u)/\partial x = 0$ the classical MacCormack scheme can be written as

$$\text{Predictor:} \quad u_i^* = u_i^n - \frac{\Delta t}{\Delta x}\left[f(u_{i+1}^n) - f(u_i^n)\right],$$

$$\text{Corrector:} \quad u_i^{n+1} = \frac{1}{2}\left(u_i^n + u_i^*\right) - \frac{\Delta t}{2\,\Delta x}\left[f(u_i^*) - f(u_{i-1}^*)\right]$$

This simple structure illustrates the flavor of early explicit methods used for compressible flows.

Jameson, Schmidt and Turkel later introduced a multi-stage Runge–Kutta method with added artificial dissipation for steady computations of the compressible Euler equations around aircraft configurations [42]. Their approach uses the CFL restriction [38] to accelerate convergence to a steady-state solution by taking large pseudo-time steps, while the artificial dissipation plays a role similar to upwinding in controlling spurious oscillations.

A general s-stage explicit Runge–Kutta scheme for a semi-discrete system $\mathrm{d}U/\mathrm{d}t = R(U)$ can be written as

$$U^{(0)} = U^n \qquad U^{(k)} = U^{(k-1)} + \alpha_k \Delta t\, R\Big(U^{(k-1)}\Big) \quad k = 1, \ldots, s$$

$$U^{n+1} = U^{(s)}$$

with coefficients α_k chosen to obtain second- or higher-order accuracy.

At high Reynolds numbers, explicit time stepping suffers from a very restrictive time step. To relax this constraint, Beam and Warming developed implicit, approximately factored schemes for the compressible Navier-Stokes equations [43]. The main idea is to factor the multi-dimensional implicit operator into a product of one-dimensional operators. This makes large implicit time steps computationally feasible and forms the basis of many modern implicit RANS solvers.

In an approximately factored implicit method, the linearized update

$$(I - \Delta t\, A_x - \Delta t\, A_y)\,\Delta U = \Delta t\, R(U^n)$$

is replaced by the factored form

$$(I - \Delta t\, A_x)\,(I - \Delta t\, A_y)\,\Delta U = \Delta t\, R(U^n)$$

where A_x and A_y are discrete Jacobians of the fluxes in the x and y directions. The factorization reduces the multidimensional implicit problem to a sequence of one-dimensional solves.

For implicit and pressure-based methods, the efficient solution of the resulting linear systems is just as important as the discretization itself. Stone proposed the Strongly Implicit Procedure (SIP), an iterative method tailored to the large sparse matrices produced by finite-difference and finite-volume schemes [44]. SIP and its variants are widely used to solve the pressure-correction and momentum equations in implicit CFD solvers, particularly for incompressible and low-Mach-number flows where elliptic pressure equations dominate the cost.

In the incompressible case, the main numerical difficulty is to satisfy the divergence-free constraint while solving for velocity and pressure. Harlow and Welch addressed this problem with the Marker-and-Cell (MAC) method, which uses a staggered grid and a pressure-correction step to enforce continuity at every time level [45]. Their method can be viewed as an early segregated algorithm: velocity and pressure are solved in separate steps, but are tightly coupled through the incompressibility constraint.

Patankar and Spalding generalized this concept and introduced the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm in a finite-volume framework for three-dimensional flows [46]. In SIMPLE, the momentum equations are solved with a guessed pressure field, a pressure-correction equation is formed from the continuity requirement, and both pressure and velocity are updated iteratively. Combined with turbulence models, this segregated pressure-based strategy forms the backbone of many industrial RANS solvers for incompressible and low-Mach-number applications.

In a typical pressure–correction scheme, an intermediate velocity field $\mathbf{u}^*$ is first obtained from the momentum equations using a guessed pressure. The corrected fields are then written as

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \frac{\Delta t}{\rho}\,\nabla p' \qquad p^{n+1} = p^n + p'$$

and the pressure correction p' is determined by enforcing continuity,

$$\nabla \cdot \mathbf{u}^{n+1} = 0 \implies \nabla \cdot \left(\frac{\Delta t}{\rho}\,\nabla p'\right) = \nabla \cdot \mathbf{u}^*$$

The discrete form of this Poisson equation for p' is central to SIMPLE-type algorithms.

As pressure-based solvers moved towards co-located variable arrangements for greater mesh flexibility, spurious pressure–velocity decoupling became a major concern. Rhie and Chow proposed a momentum interpolation technique that modifies the face velocities in the continuity equation to include a consistent pressure gradient term [47]. This correction couples pressure and velocity on co-located grids in a way that mimics the inherent coupling of staggered-grid methods such as MAC [45], while preserving the implementation advantages of co-located finite-volume schemes. For unsteady flows, Issa introduced the PISO (Pressure-Implicit with Splitting of Operators) algorithm, which extends SIMPLE by performing several pressure-correction steps within each time step [48]. PISO improves the temporal accuracy and stability of segregated solvers in strongly transient flows and is widely used in unsteady RANS and LES computations.

On co-located grids, Rhie-Chow interpolation modifies the face-normal velocity u_f entering the continuity equation to include a pressure-gradient contribution, for example

$$u_f = \widehat{u}_f - d_f\left(\frac{\partial p}{\partial n}\right)_f$$

where $\widehat{u}_f$ is an interpolated momentum velocity, $(\partial p/\partial n)_f$ is the normal pressure gradient at the face, and d_f is a coefficient built from the neighboring momentum equations. This construction prevents the spurious checkerboard patterns that can appear on co-located meshes.

In the PISO algorithm, the predictor step and the first pressure correction are followed by one or more additional correction steps, written symbolically as

$$\mathbf{u}^{(0)} = \mathbf{u}^* \quad \mathbf{u}^{(m+1)} = \mathbf{u}^{(m)} - \frac{\Delta t}{\rho}\,\nabla p'^{(m)} \quad m = 0, 1, \ldots, M-1$$

with each pressure correction $p'^{(m)}$ obtained from a continuity equation

$$\nabla \cdot \mathbf{u}^{(m+1)} = 0$$

These additional corrections improve the coupling between pressure and velocity inside a single time step and increase the accuracy of transient simulations.

Taken together, these contributions define two main families of CFD solvers for the Navier-Stokes equations. On one side, methods based on hyperbolic conservation laws, rooted in Riemann's wave analysis [37], the CFL stability criterion [38] and Godunov-type finite-volume schemes [39], form the basis of many density-based, upwind RANS solvers for compressible flows and are usually combined with high-resolution reconstructions and explicit or implicit time marching [41, 43, 40, 42]. On the other side, pressure-based segregated algorithms, originating from MAC [45] and extended through SIMPLE and PISO [46, 48], rely on robust linear solvers such as SIP [44] and stabilization techniques like Rhie-Chow interpolation [47] to enforce incompressibility and maintain pressure-velocity coupling. Modern industrial RANS codes typically implement variants of both strategies and select the most appropriate combination of pressure- or density-based formulation, explicit or implicit time integration and upwind or central differencing according to the Mach number range, level of compressibility and degree of unsteadiness of the flow under consideration.

Ground Vehicle Aerodynamics Simulation

In the second half of the twentieth century, the study of ground vehicle aerodynamics gradually departed from classical airfoil theory and evolved towards a dedicated body of work treating road vehicles as three-dimensional bluff bodies operating close to the ground. This evolution is synthesized by Hucho in *Aerodynamics of Road Vehicles*, which provides a comprehensive overview of drag, lift and side-force mechanisms, and proposes a decomposition of the flow field into forebody, underbody, upper-body and wake contributions [49]. In a later review with Sovran, he further formalizes the role of base pressure, A-pillar vortices, rear-window separation bubbles and crosswind stability, and relates these features to characteristic trends observed on production vehicles [50]. These works establish the general vocabulary and physical pictures which are now routinely used when interpreting both experimental and numerical results for road vehicles.

To isolate the main mechanisms responsible for drag and lift in a more controlled manner, a number of simplified reference geometries were introduced. Morel investigates the aerodynamic drag of three-dimensional bluff bodies representative of passenger cars, while systematically varying rear truncation, aspect ratio and edge rounding [51]. His results show that moderate changes in rear shape and corner radii can lead to large variations in base pressure and overall drag, thereby emphasizing the central role of separated wake topology for ground vehicles. Ahmed, Ramm and Faltin follow a similar philosophy and propose a generic car-like model consisting of a rectangular body with a slanted rear surface [52]. By varying the rear slant angle, they identify distinct flow regimes, ranging from attached flow to separated and reattaching flow on the rear slant, and ultimately to fully separated wakes, and correlate these regimes with changes in drag and lift coefficients. The so-called Ahmed body has since become one of the canonical validation cases for CFD studies of road-vehicle aerodynamics and is now widely used to assess the ability of RANS, DES and LES approaches to predict bluff-body wakes.

Other generic shapes such as the Windsor body and idealized notchback, fastback or squareback models extend this parametric approach to rear-end design. Cooper, for example, introduces the Windsor body as an idealized sedan-like geometry and documents its drag, surface pressure distribution and wake structures in detail [53]. Together with the configurations proposed by Morel and Ahmed, these geometries define a hierarchy of test cases that bridge the gap between simple bluff bodies and full-scale production vehicles. In the context of CFD, they provide a progressively more complex set of benchmarks on which numerical methods, turbulence models and meshing strategies can be tested before being applied to full-vehicle simulations.

The influence of wheels and their interaction with the ground was clarified in the early 1970s by Fackrell and Harvey, who measure the flow field and pressure distribution around an isolated rotating wheel in close proximity to a plane boundary [54]. Their experiments reveal a highly three-dimensional flow with strong separation at the wheel shoulders, a complex system of vortices generated at the contact patch and significant sensitivity of wheel drag and lift to both rotation and ride height. Cogotti then extends the study of wheel aerodynamics to configurations closer to production vehicles, by measuring the effect of rotating and non-rotating wheels, different ground simulation techniques and wheelhouse details on the overall aerodynamic balance [55]. When combined with Hucho's drag and lift breakdowns [49], these works demonstrate that wheels and underbody flow can account for a substantial fraction of the total drag and can strongly influence front-axle lift, making their accurate representation in CFD models essential for both passenger and race cars.

Underbody and diffuser aerodynamics form another important component of ground-vehicle aerodynamics. Experimental investigations reported by Hucho, Morel, Cooper and others show how ride height, underbody roughness and diffuser geometry control the mass flow under the vehicle, the pressure distribution on the floor and the onset of separation in the diffuser region [49, 51, 53]. These observations are often interpreted through simplified Venturi and diffuser analogies, in which the underbody behaves as a convergent section and the diffuser as a divergent section with limited pressure recovery in the presence of strong boundary-layer growth. For CFD, these results provide practical guidance on the choice of moving-ground or fixed-ground boundary conditions, on the required geometric fidelity in the underbody region and on the need for turbulence models capable of handling separation and corner vortices in diffusers.

For high-downforce configurations, Katz revisits many of these ideas from the perspective of race car aerodynamics, with emphasis on wings in ground effect, underfloors and diffusers [56]. His work combines classical aerodynamics concepts, such as airfoils in ground proximity and vortex lift, with wind-tunnel measurements on single-seater and sports-car geometries, and highlights the strong coupling between ride height, rake angle, wing settings and the resulting distribution of downforce and drag. This race-car oriented synthesis complements the more general road-vehicle treatments by Hucho and Sovran [49, 50], and is particularly relevant when interpreting CFD results for open-wheel cars, diffusers and wing-in-ground-effect systems.

In a first approximation, the underbody and diffuser flow can be idealized as a one-dimensional incompressible stream through a duct of varying cross-sectional area $A(x) = b\,h(x)$, where b is an effective width and $h(x)$ the local gap height. Mass conservation then gives

$$\dot{m} = \rho A(x)\,u(x) = \rho\,b\,h(x)\,u(x) = \text{cst}$$

so that between two stations 1 and 2 along the underbody

$$u_2 = u_1\,\frac{A_1}{A_2} = u_1\,\frac{h_1}{h_2} \qquad \text{for a fixed effective width } b$$

Neglecting viscous losses, Bernoulli's equation along a streamline between the same two stations reads

$$p_1 + \frac{1}{2}\,\rho u_1^2 = p_2 + \frac{1}{2}\,\rho u_2^2$$

which, combined with Venturi's velocity, gives the pressure drop associated with a reduced underbody area. In terms of pressure coefficients based on a reference dynamic pressure $\frac{1}{2}\rho U_\infty^2$, this can be written as

$$C_{p,2} - C_{p,1} = \frac{p_2 - p_1}{\frac{1}{2}\rho U_\infty^2} = \frac{u_1^2 - u_2^2}{U_\infty^2}$$

so that an increase in local velocity under the car ($u_2 > u_1$) is associated with a reduction in static pressure $p_2 < p_1$ and therefore an increase in downforce.

For the diffuser section, the pressure recovery between the throat (station t) and the diffuser exit (station e) is often characterized by a non-dimensional pressure-recovery coefficient

$$C_{p,\text{rec}} = \frac{p_e - p_t}{\frac{1}{2}\rho U_\infty^2}$$

which is bounded by the ideal (inviscid) value obtained from previous equations and reduced in practice by boundary-layer growth and possible flow separation in the diffuser.

Finally, the effect of crosswind and unsteady wake dynamics has been documented using both generic models and full-scale vehicles. Le Good and Garry analyze the aerodynamic forces and moments acting on a simplified road vehicle under yawed inflow, and show how the stagnation pattern, separation lines and vortex structures evolve with yaw angle [57]. Watkins and Vino perform full-scale measurements in natural wind conditions and relate the unsteady wake behavior to lateral forces and yawing moments experienced by the vehicle [58]. These studies complement the steady and quasi-steady bluff-body investigations described above and provide additional validation targets for unsteady RANS, DES and LES simulations of ground vehicles operating in realistic crosswind environments.

Cornering Vehicle Dynamics and Aerodynamics

The analysis of cornering performance requires a coupled understanding of vehicle dynamics and aerodynamics, since lateral force generation, load transfer and aerodynamic downforce all contribute to the achievable lateral acceleration and to the stability margins of the vehicle. On the vehicle-dynamics side, the development of simplified planar models and tyre-force representations has provided a compact framework to describe steady-state and transient cornering. These ideas are brought together in the classical treatments by Gillespie and by Milliken, who present the linear single-track (widely known as bicycle) model and the associated understeer-gradient formulation in a systematic way [59, 60]. In these models, the vehicle is reduced to a single front and rear axle, each represented by an equivalent tyre pair, and the lateral force balance, yaw-moment balance and kinematic relationships are used to derive expressions for lateral acceleration, yaw rate and sideslip as functions of steering input. The resulting framework allows a clear decomposition of cornering behavior into contributions from front and rear cornering stiffnesses, mass and inertia properties, and geometric parameters such as wheelbase and track width.

In a simple single-track (bicycle) model of steady-state cornering, the lateral force balance and yaw-moment balance about the center of gravity (CG) can be written as

$$ma_y = F_{yf} + F_{yr} = m\frac{U^2}{R}$$

$$a\,F_{yf} - b\,F_{yr} = 0$$

where m is the vehicle mass, a and b are the distances from the CG to the front and rear axles, U is the forward speed, R the turn radius and F_{yf}, F_{yr} the front and rear axle lateral forces. Under a linear tyre assumption,

$$F_{yf} = C_f\,\alpha_f \qquad F_{yr} = C_r\,\alpha_r$$

with C_f and C_r the front and rear cornering stiffnesses and α_f, α_r the corresponding slip angles. Solving the equations yields the well-known relationship between steering angle δ and lateral acceleration

$$\delta = \frac{L}{R} + K\,\frac{a_y}{g}$$

where $L = a + b$ is the wheelbase, g is the gravitational acceleration and K is the understeer gradient, defined by

$$K = \frac{W_f}{mgC_f} - \frac{W_r}{mgC_r}$$

with W_f and W_r the static front and rear axle loads. Positive K corresponds to an understeering vehicle, while negative K indicates oversteer [59, 60].

The predictive capability of such models depends critically on the description of tyre forces in combined-slip conditions, where longitudinal and lateral slips, camber angle and vertical load all vary during cornering. This issue is addressed in detail by Pacejka, who introduces the so-called "Magic Formula" tyre model and develops it into a general framework for representing lateral and longitudinal forces and self-aligning moments over wide ranges of operating conditions [61]. In this approach, the tyre forces are expressed as nonlinear functions of slip variables and normal load, using a small number of coefficients that can be identified from test data. Milliken integrate such tyre models into more elaborate quasi-steady and transient handling analyses for race cars, including the effects of camber, roll stiffness distribution and aerodynamic downforce distribution on the front and rear axles [60]. Genta and Rajamani extend the discussion towards multi-degree-of-freedom and control-oriented models, respectively, where cornering dynamics are embedded in full vehicle models including suspension compliance, roll and pitch motions, and active systems such as ABS, ESC and active steering [62, 63]. Together, these works define the current state-of-practice in cornering vehicle dynamics modeling, from simple linear bicycle models to high-fidelity multi-body and control-oriented representations.

In the Magic Formula framework, the steady-state lateral force generated by a tyre at a given vertical load can be expressed as

$$F_y(\alpha) = D\,\sin\left\{C\,\arctan\left[B\left(\alpha - E\left(B\alpha - \arctan(B\alpha)\right)\right)\right]\right\}$$

where α is the slip angle and B, C, D and E are empirical coefficients controlling, respectively, the stiffness, shape, peak and curvature of the lateral-force characteristic [61]. In combined-slip conditions, additional terms are introduced to account for the interaction between longitudinal and lateral slip, but the functional structure there-above remains the basis of the model.

Cornering aerodynamics add another layer of complexity, as the aerodynamic forces and moments depend on yaw angle, roll angle, steer angle and ride height, and thus vary significantly between straight-line and cornering conditions. Hucho and Sovran discuss the influence of crosswind and yawed inflow on the distribution of pressure and separation over road vehicles, and highlight the resulting changes in side force, yawing moment and lift [49, 50]. In steady-state cornering, the vehicle experiences a relative flow with non-zero yaw angle due to the lateral velocity component, and the associated modifications of the stagnation pattern, separation lines and vortex systems can have a strong impact on both the lateral and vertical aerodynamic loads. Katz revisits these questions from a race-car perspective, with emphasis on the behavior of front and rear wings, diffusers and underfloors in ground effect under yawed and rolled conditions [56]. His analysis shows, for instance, how the effective angle of attack and local ground clearance of a wing in ground effect vary across the span in a corner, leading to asymmetric loading and a shift in the center of pressure, and how the combination of lateral acceleration, roll and heave motions influences the performance of diffusers and Venturi tunnels.

The aerodynamic contributions to lateral force, lift and yawing moment can be written in non-dimensional form as

$$F_Y = \frac{1}{2}\,\rho U^2 A\,C_Y(\beta) \qquad F_Z = \frac{1}{2}\,\rho U^2 A\,C_L(\beta)$$

$$M_Z = \frac{1}{2}\,\rho U^2 AL\,C_N(\beta)$$

where ρ is the air density, A a reference area, L a reference length, and C_Y, C_L and C_N are the side-force, lift and yaw-moment coefficients, which depend on the effective yaw angle β seen by the vehicle [49, 50, 56]. In steady-state cornering, β is influenced both by the vehicle sideslip and by any ambient crosswind, and the resulting variations of $C_Y(\beta)$, $C_L(\beta)$ and $C_N(\beta)$ directly affect lateral force capacity, load transfer and stability margins.

The interaction between aerodynamics and cornering vehicle dynamics is further complicated by unsteady effects and by the presence of crosswinds and gusts. Le Good and Garry examine the forces and moments acting on a simplified passenger-car model in yawed flow, and document the evolution of side force, yawing moment and lift with yaw angle and Reynolds number [57]. Their results provide detailed surface-pressure and force data that are now widely used for validating CFD predictions of yawed ground vehicles. Watkins and Vino extend this line of work to full-scale vehicles subjected to natural wind, and relate the unsteady wake structures and fluctuating forces to vehicle stability and handling in crosswind conditions [58]. When combined with the quasi-steady vehicle-dynamics frameworks discussed above [59, 60, 61], these studies provide the foundation for integrated models in which tyre forces, load transfer and aerodynamic forces and moments are treated in a coupled manner.

In addition to these classical vehicle-dynamics frameworks and quasi-steady aerodynamic studies, a number of authors have proposed more detailed mathematical and numerical models aimed specifically at cornering conditions. Preda and Ciolan formulate a vehicle mathematical model tailored to the study of cornering manoeuvres, starting from the planar equations of motion and incorporating tyre-force characteristics, lateral load transfer and steering system dynamics into a unified representation [64]. Their approach extends the traditional linear bicycle model [59, 60] by retaining a higher level of nonlinearity in the lateral and yaw dynamics, thereby enabling the analysis of stability limits and transient responses in more demanding cornering scenarios. Such models provide a natural bridge between the simplified analytical treatments used in conceptual studies and the high-fidelity multi-body simulations employed in detailed vehicle-dynamics investigations.

On the aerodynamic side, Keogh and co-workers focus explicitly on the particularities of cornering flow conditions around ground vehicles. In an SAE paper, Keogh et al. [65] review and compare several techniques for aerodynamic analysis of cornering vehicles, including wind-tunnel test methods, curved-road approximations and CFD approaches based on rotating reference frames or imposed yawed inflow combined with lateral acceleration [65]. Their subsequent doctoral work provides a more detailed examination of the aerodynamic effects of cornering flow conditions, with emphasis on the changes in pressure distribution, vortex structures and aerodynamic force coefficients that arise when a vehicle follows a curved path rather than a straight line [66]. These studies underline that cornering aerodynamics cannot be adequately captured by simply superposing straight-line aerodynamics with a crosswind component, and they highlight the need for flow conditions and boundary conditions that are consistent with the underlying vehicle dynamics.

A complementary, more fundamental perspective on cornering aerodynamics in ground effect is offered by Patel et al., who investigate the effect of cornering on the aerodynamics of a multi-element wing operating close to the ground [67]. In their work, the wing is subjected to kinematic conditions representative of a cornering vehicle, and the resulting changes in lift, drag and pitching moment, as well as in the spanwise load distribution, are quantified. The study shows that cornering flow conditions introduce significant asymmetries in the pressure field and modify both the magnitude and distribution of downforce compared to straight-line, zero-yaw ground-effect operation. These results are particularly relevant for race-car configurations, where multi-element wings in ground effect are key contributors to lateral grip, and they provide useful reference data and physical insight for the validation of CFD models that aim to capture coupled cornering dynamics and aerodynamics.

Taken together, the mathematical vehicle models of Preda and Ciolan [64], the cornering-specific experimental and numerical methodologies of Keogh [65, 66], and the canonical ground-effect wing study of Patel et al. [67] extend the classical frameworks of Gillespie, Milliken, Pacejka and Katz [59, 60, 61, 56] towards a more integrated treatment of cornering vehicle dynamics and aerodynamics. They also provide additional validation targets and modeling guidelines for the coupled simulations presented in this work, in which tyre forces, load transfer and yawed, ground-effect aerodynamics are treated in a consistent CFD-based framework.

Conclusion

References

1. H. Perrin and E. Houri, *Straight Line Simulation Model Towards FSAE Aerodynamics*. Formule ETS, 2024.
2. H. Perrin, *Aerodynamic Simulation of a FSAE Open-Wheel Car During Cornering*. Formule ETS, 2024.
3. D. Bernoulli, *Hydrodynamica, Sive De Viribus et Motibus Fluidorum Commentarii*. Strasbourg: Johann Reinhold Dulsecker, 1738.
4. L. Euler, "Principes généraux du mouvement des fluides," *Mémoires de l'Académie des Sciences de Berlin*, vol. 11, pp. 274–315, 1757.
5. J. L. R. d'Alembert, *Essai d'une nouvelle théorie de la résistance des fluides*. Paris: David l'aîné, 1752.
6. G. Hagen, "Über die bewegung des wassers in engen cylindrischen röhren," *Annalen der Physik*, vol. 122, no. 3, pp. 423–442, 1839.
7. L. Prandtl, "Über flüssigkeitsbewegung bei sehr kleiner reibung," in *Verhandlungen des III. Internationalen Mathematiker-Kongresses*, (Heidelberg), pp. 484–491, B. G. Teubner, 1904.
8. C. L. M. H. Navier, "Mémoire sur les lois du mouvement des fluides," *Mémoires de l'Académie Royale des Sciences de l'Institut de France*, vol. 6, pp. 389–440, 1823.
9. O. Darrigol, "Between hydrodynamics and elasticity theory: The first five births of the navier–stokes equation," *Archive for History of Exact Sciences*, vol. 56, no. 2, pp. 95–150, 2002.
10. O. Darrigol, *Worlds of Flow: A History of Hydrodynamics from the Bernoullis to Prandtl*. Oxford: Oxford University Press, 2005.
11. A.-L. Cauchy, "Sur l'équilibre et le mouvement intérieur des corps considérés comme des masses continues," *Exercices de Mathématiques*, vol. 4, pp. 293–319, 1829.
12. G. G. Stokes, "On the theories of the internal friction of fluids in motion, and of the equilibrium and motion of elastic solids," *Transactions of the Cambridge Philosophical Society*, vol. 8, pp. 287–319, 1845.
13. H. Blasius, "Grenzschichten in flüssigkeiten mit kleiner reibung," *Zeitschrift für Mathematik und Physik*, vol. 56, pp. 1–37, 1908.
14. T. von Kármán, "Über laminare und turbulente reibung," *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 1, no. 4, pp. 233–252, 1921.
15. H. Schlichting, *Boundary-Layer Theory*. New York: McGraw-Hill, 7 ed., 1979.
16. P. Moin and K. Mahesh, "Direct numerical simulation: A tool in turbulence research," *Annual Review of Fluid Mechanics*, vol. 30, no. 1, pp. 539–578, 1998.
17. J. Boussinesq, *Essai sur la théorie des eaux courantes*. Paris: Imprimerie Nationale, 1877.
18. O. Reynolds, "An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels," *Philosophical Transactions of the Royal Society of London*, vol. 174, pp. 935–982, 1883.

19. O. Reynolds, “On the dynamical theory of incompressible viscous fluids and the determination of the criterion,” *Philosophical Transactions of the Royal Society of London A*, vol. 186, pp. 123–164, 1895.

20. S. B. Pope, *Turbulent Flows*. Cambridge: Cambridge University Press, 2000.

21. L. Prandtl, “Bericht über untersuchungen zur ausgebildeten turbulenz,” *Zeitschrift für Angewandte Mathematik und Mechanik*, vol. 5, no. 2, pp. 136–139, 1925.

22. J. Leray, “Sur le mouvement d’un liquide visqueux emplissant l’espace,” *Acta Mathematica*, vol. 63, pp. 193–248, 1934.

23. A. N. Kolmogorov, “The local structure of turbulence in incompressible viscous fluid for very large reynolds numbers,” *Doklady Akademii Nauk SSSR*, vol. 30, pp. 301–305, 1941.

24. A. N. Kolmogorov, “Equations of turbulent motion of an incompressible fluid,” *Izvestiya Akademii Nauk SSSR, Seriya Fizicheskaya*, vol. 6, no. 1–2, pp. 56–58, 1942. English translation in: Proc. R. Soc. Lond. A 434 (1991) 214–216.

25. L. Prandtl, “Über ein neues formelsystem für die ausgebildete turbulenz,” *Nachrichten der Akademie der Wissenschaften in Göttingen, Mathematisch-Physikalische Klasse*, pp. 6–19, 1945.

26. J. C. Rotta, “Statistische theorie nichthomogener turbulenz,” *Zeitschrift für Physik*, vol. 131, pp. 51–77, 1951.

27. W. P. Jones and B. E. Launder, “The calculation of low-reynolds-number phenomena with a two-equation model of turbulence,” *International Journal of Heat and Mass Transfer*, vol. 16, no. 6, pp. 1119–1130, 1973.

28. W. P. Jones and B. E. Launder, “The prediction of laminarization with a two-equation model of turbulence,” *International Journal of Heat and Mass Transfer*, vol. 15, no. 2, pp. 301–314, 1972.

29. B. E. Launder and D. B. Spalding, “The numerical computation of turbulent flows,” *Computer Methods in Applied Mechanics and Engineering*, vol. 3, no. 2, pp. 269–289, 1974.

30. D. C. Wilcox, “Reassessment of the scale-determining equation for advanced turbulence models,” *AIAA Journal*, vol. 26, no. 11, pp. 1299–1310, 1988.

31. F. R. Menter, “Two-equation eddy-viscosity turbulence models for engineering applications,” *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994.

32. P. R. Spalart and S. R. Allmaras, “A one-equation turbulence model for aerodynamic flows,” in *30th AIAA Aerospace Sciences Meeting and Exhibit*, (Reno, NV), 1992. AIAA Paper 92-0439.

33. P. R. Spalart, “Strategies for turbulence modelling and simulations,” *International Journal of Heat and Fluid Flow*, vol. 21, no. 3, pp. 252–263, 2000.

34. P. A. Davidson, *Turbulence: An Introduction for Scientists and Engineers*. Oxford: Oxford University Press, 2004.

35. D. C. Wilcox, *Turbulence Modeling for CFD*. La Cañada, California: DCW Industries, 3 ed., 2006.

36. J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*. Berlin: Springer, 3 ed., 2002.

37. B. Riemann, “Über die fortpflanzung ebener luftwellen von endlicher schwingungsweite,” in *Abhandlungen der Königlichen Gesellschaft der Wissenschaften zu Göttingen*, vol. 8, pp. 43–65, 1860. English translation: On the propagation of plane air waves of finite amplitude, in *Collected Works of Bernhard Riemann*.

38. R. Courant, K. Friedrichs, and H. Lewy, “Über die partiellen differentialgleichungen der mathematischen physik,” *Mathematische Annalen*, vol. 100, pp. 32–74, 1928.

39. S. K. Godunov, “A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics,” *Matematicheskii Sbornik*, vol. 47, no. 3, pp. 271–306, 1959. In Russian; English translation in USSR Comput. Math. Math. Phys., 1969.

40. B. van Leer, “Towards the ultimate conservative difference scheme. V. a second-order sequel to godunov’s method,” *Journal of Computational Physics*, vol. 32, no. 1, pp. 101–136, 1979.

41. R. W. MacCormack, “The effect of viscosity in hypervelocity impact cratering,” in *AIAA Paper 69-354*, 1969. Describes the two-step predictor–corrector scheme widely known as the MacCormack method.

42. A. Jameson, W. Schmidt, and E. Turkel, “Numerical solution of the Euler equations by finite volume methods using runge–kutta time stepping schemes,” in *14th Fluid and Plasma Dynamics Conference*, 1981. AIAA Paper 81-1259.

43. R. M. Beam and R. F. Warming, “An implicit finite-difference algorithm for the compressible navier–stokes equations,” *AIAA Journal*, vol. 16, no. 4, pp. 393–402, 1978.

44. H. L. Stone, “Iterative solution of implicit approximations of multidimensional partial differential equations,” *SIAM Journal on Numerical Analysis*, vol. 5, no. 3, pp. 530–558, 1968.

45. F. H. Harlow and J. E. Welch, “Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface,” *The Physics of Fluids*, vol. 8, no. 12, pp. 2182–2189, 1965.

46. S. V. Patankar and D. B. Spalding, “A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows,” *International Journal of Heat and Mass Transfer*, vol. 15, no. 10, pp. 1787–1806, 1972.

47. C. M. Rhie and W. L. Chow, “Numerical study of the turbulent flow past an airfoil with trailing edge separation,” in *21st Aerospace Sciences Meeting*, 1983. AIAA Paper 83-0325; introduces the momentum-interpolation technique now known as Rhie–Chow.

48. R. I. Issa, “Solution of the implicitly discretised fluid flow equations by operator-splitting,” *Journal of Computational Physics*, vol. 62, no. 1, pp. 40–65, 1986.

49. W.-H. Hucho, *Aerodynamics of Road Vehicles*. Warrendale, PA: SAE International, 4 ed., 1998. Edited by W.-H. Hucho.

50. W.-H. Hucho and G. Sovran, “Aerodynamics of road vehicles,” *Annual Review of Fluid Mechanics*, vol. 25, pp. 485–537, 1993.

51. T. Morel, “Aerodynamic drag of bluff bodies and road vehicles,” *Progress in Aerospace Sciences*, vol. 18, no. 3, pp. 179–222, 1978.

52. S. A. Ahmed, G. Ramm, and G. Faltin, “Some salient features of the time-averaged ground vehicle wake,” *SAE Technical Paper 840300*, 1984.

53. K. R. Cooper, “The aerodynamic performance of an idealized ground vehicle (the windsor body),” in *SAE Technical Paper 850288*, SAE International, 1985.

54. J. E. Fackrell and J. K. Harvey, “The flow field and pressure distribution of an isolated road wheel,” in *Proceedings of the Institution of Mechanical Engineers*, 1973.

55. A. Cogotti, “Aerodynamic characteristics of car wheels,” in *SAE Technical Paper 830138*, SAE International, 1983.

56. J. Katz, *Race Car Aerodynamics: Designing for Speed*. Cambridge, MA: Bentley Publishers, 1995.

57. G. M. Le Good and K. P. Garry, "On the use of reference models in automotive aerodynamics," in *SAE Technical Paper 2004-01-1308*, SAE International, 2004.

58. S. Watkins and G. Vino, "The effect of vehicle spacing on the aerodynamics of a representative car shape," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, pp. 1232–1239, 2008.

59. T. D. Gillespie, *Fundamentals of Vehicle Dynamics*. Warrendale, PA: Society of Automotive Engineers, 1992.

60. W. F. Milliken and D. L. Milliken, *Race Car Vehicle Dynamics*. Warrendale, PA: Society of Automotive Engineers, 1995.

61. H. B. Pacejka, *Tyre and Vehicle Dynamics*. Oxford: Butterworth–Heinemann, 2002.

62. G. Genta, *Motor Vehicle Dynamics: Modeling and Simulation*. Singapore: World Scientific, 1997.

63. R. Rajamani, *Vehicle Dynamics and Control*. New York: Springer, 2006.

64. I. Preda and G. Ciolan, "Vehicle mathematical model for the study of cornering," *Annals of the Oradea University, Fascicle of Management and Technological Engineering*, vol. XI (XXI), no. 2, 2012.

65. J. Keogh, T. Barber, S. Diasinos, and G. Doig, "Techniques for aerodynamic analysis of cornering vehicles," in *SAE Technical Paper 2015-01-0022*, SAE International, 03 2015.

66. J. Keogh, *The Aerodynamic Effects of the Cornering Flow Conditions*. PhD thesis, University of New South Wales, 2016.

67. D. Patel, A. Garmory, and M. Passmore, "The effect of cornering on the aerodynamics of a multi-element wing in ground effect," *Fluids*, vol. 6, no. 1, 2021.

Definitions, Acronyms, Abbreviations

CAD	Computer Assisted Design
CFD	Computational Fluid Dynamics
CSM	Cornering Simulation Model
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
HPC	High Performance Computing
FEA	Finite Element Analysis
FSAE	Formula Society of Automotive Engineers
LES	Large Eddy Simulation
MAUh	Mega Allocation Unit hours (F1 CFD)
MUSCL	Monotone Upstream-centered Schemes for Conservation Laws
PDEs	Partial Derivative Equations
PISO	Pressure-Implicit with Splitting of Operators
RANS	Reynolds-Averaged Navier–Stokes
RSM	Reynolds Stress Model
SLSM	Straight Line Simulation Model
SST	Shear Stress Transport

Contact Information

Hugues PERRIN, M.Sc
contact@huperrin.com