18th Australasian Fluid Mechanics Conference
Launceston, Australia
3-7 December 2012

The Influence of Compressibility Effects in Correlation Issues for Aerodynamic Development of Racing Cars

James Keogh[1], Graham Doig[1], Sammy Diasinos[2]

[1] The University of New South Wales
School of Mechanical and Manufacturing Engineering, Kensington, Sydney, NSW 2052, Australia

[2] Caterham F1 Team, Hingham Industrial Estate, Ironside Way, Hingham NR9 4L, UK

Abstract

In motorsport, Computational Fluid Dynamic (CFD) simulations of new designs are routinely compared to wind tunnel and track data in cross-validation exercises. Often, relatively poor correlation is found between these three methods. A Formula 1 car can reach speeds in excess of Mach 0.25 and, in addition, parts of the car operate in extreme ground effect thus accelerating the local flow to double or more the freestream Mach number. A 3-dimensonal CFD study was conducted into the significance of compressibility effects occurring over an entire open-wheeled racing car for a typical design-phase scenario whereby a 60% wind tunnel model is matched to Reynolds-scaled full-scale CFD (i.e. lower velocity). A 36% model was also investigated for the same Reynolds number. The overall lift and drag coefficients for the whole car were very similar, which could give a misleading impression of negligible compressibility effects. However, the results demonstrated significant differences (up to several percent) in both lift and drag over the major individual components of the wheels, front wing, diffuser and rear wing due to compressibility effects, when compared with incompressible benchmarks. The behaviour of vortices and separation points would be affected by the density changes, which would in turn have significant consequences for bodywork fine-tuned in incompressible simulations.

Introduction

A designer has three options for examining the aerodynamic characteristics of a racing car or its bodywork components: wind tunnel testing with a model, CFD simulation, and track testing with an actual vehicle. The latter is rare in the context of ongoing design development, and the former two are usually performed in close concert, with CFD both filling in gaps in tunnel testing and simulating scenarios not possible in experimental conditions. Correlation between data sets often throws up significant errors and uncertainties due to inconsistencies and unknowns.

The effects of compressibility for aerospace applications are generally accepted to become prominent in the flowfield at Mach numbers upwards of 0.3. In the case of open-wheel racing cars, the high-downforce components and the speed of the vehicle itself is likely to mean local Mach numbers around the car in excess of twice this value [5]. This would produce significant changes in flow density around the car, but while compressible effects are likely to be prevalent in such scenarios, it has been acknowledged that the issue remains largely unaddressed in the public domain [8]. In Katz's otherwise comprehensive review of the aerodynamics of racing cars [7], for instance, the single mention of compressibility comes in a brief passage about the Mach 0.85 Blue Flame rocket car. Most design teams would likely consider it more important to compute large parametric studies rapidly than to run fewer, more RAM-intensive simulations to account for compressibility if the freestream Mach numbers are perceived to be distinctly incompressible..

Previous studies by Doig et al. described a comparison between compressible and incompressible CFD which was loosely equivalent to comparing incompressible CFD to results obtained by a full-scale vehicle on track as the Reynolds number was left to vary freely with increasing velocity [3,4]. The studies indicated that above a freestream Mach number of 0.15, compressible CFD was required in order to accurately describe the flows, since force coefficients could be wrongly-predicted by incompressible flow by several percent. However, the influence of Reynolds number was not separated from that of Mach number. The present paper outlines a scenario wherein one wishes to correlate full-scale CFD to scale-model wind tunnel tests, where precise Reynolds-scaling may be perceived as being sufficient for comparison due to the relatively low speeds involved.

The automotive racing industry has historically lagged behind the aerospace industry in studying issues relating to scale testing of wind tunnel models and comparisons of such data sets to those from full-scale prototype testing and CFD. This has been due to the much greater reduction in scale necessary for researching the aerodynamics of an aircraft in a wind tunnel than that required for a road vehicle, which can often be tested at 50% or indeed even full-scale. Nevertheless, potential issues of Reynolds-scaling and its influence on transition, tunnel wall effects and ground representation are appreciated.

A simplified and generic open-wheeled racing-car geometry complying with current Formula One regulations was used for all simulations described in this paper, with full-scale dimensions as described here. The front wing geometry used was the double element airfoil of Zerihan and Zhang [9] (T026 geometry with flap), for which detailed wind tunnel data was available for validation. The wing span was set to be 1800mm, featuring a single element mid-section in the region where the nose would ordinarily connect. The rear wing, spanning 750mm, consisted of two S1223 airfoils - a common high-lift, low-Reynolds number design. The lower rear wing element also utilized the S1223 geometry. The diffuser under the car at the rear was constructed within current Formula 1 regulations, at a 10.7 degree angle from the ground reference plane. A nose-down 0.7 degrees of rake was applied to the body of the car. Minimum ground clearance of the floor was 52mm at the front of the body.

Wheels were of a simple construction within regulation size, and with the outside hub removed. Validation on the wheel aerodynamics was carried out on the isolated wheel of Fackrell [6]. All wheels had a diameter of 650mm, with the front wheel maximum width being 350mm, and 380mm for the rear wheels.

A 10mm high (at full scale) contact patch was placed where the wheel meets the ground to allow for simpler meshing and to partially-represent the deformation typically experienced. The lift coefficient of an isolated wheel has previously been shown to be highly sensitive to this region as the high pressure in the contact patch dictates, to a large extent, the overall lift on the wheel [2].

Simple sidepods and an airbox above the cockpit were implemented as pressure outlets to account for the cooling air ingested by the car. Exhaust and cooling flow outlets were placed in similar locations to those seen on most current cars.

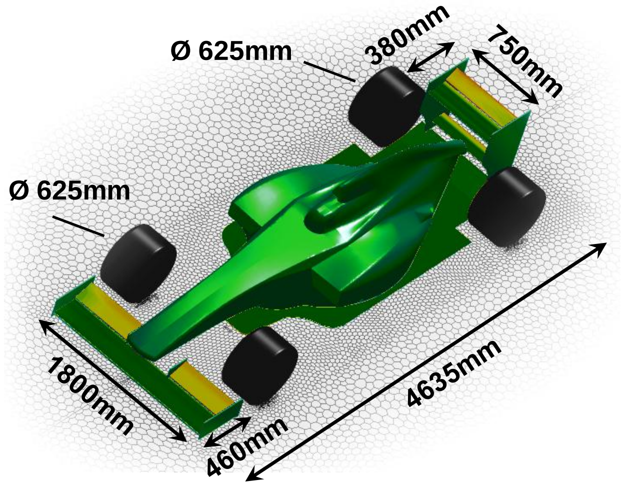

Figure 1- The generic F1-style car, with dimensions and ground mesh.

Current Formula 1 wind tunnel testing restrictions limit teams to testing a 60% scale model at 50m/s [1]. With the assumption of incompressible flow, Reynolds scaling can be used to then determine how the flow will behave for a full scale model at 30ms^{-1}. Benchmark compressible and incompressible simulations were conducted at 100% scale at these conditions, and this approach was repeated at 60% for 50ms^{-1}, and 36% scale at 83.3ms^{-1}, the latter being a more common size for a smaller wind tunnel more common at a university. The flow conditions are tabulated in table 1 for clarity, along with the Reynolds number based on the length of the car (4635mm). All drag force coefficients mentioned in this paper are based on frontal area, and lift coefficients based on top-down plan view area. Negative lift is also referred to as downforce throughout the manuscript.

Scale (%)	Velocity (m/s)	Mach	Reynolds No.
100	30	0.088	9.4×10^6
60	50	0.147	9.4×10^6
36	83.33	0.245	9.4×10^6

Table 1- Flow conditions for Re-scaled simulations

Numerical Method

A commercial finite-volume Reynolds-Averaged Navier Stokes solver, ANSYS FLUENT, was used to generate the results. The software is commonly used in the automotive industry, and with several current Formula 1 teams. A pressure-based, coupled solver was applied to obtain steady-state solutions, and convergence criteria were deemed to be met not only when the mass and momentum scaled-residual errors ceased to change by more than approximately 0.01% over 1000 continued iterations, but also when the aerodynamic forces on the body ceased to change by more than 0.01% over 1000 further iterations. All cases were run in 64-bit single precision using a second order cell-based upwinding discretization scheme. A standard three-coefficient Sutherland viscosity model was applied to simulations involving compressible (ideal gas) flow.

For the incompressible simulations the inlet, exhaust and cooling outlet were set as velocity inlets with the outlet, cooling inlet and engine intake all set as zero static pressure outlets. Turbulence intensity was set at 0.2%, representative of that which would be expected in a wind tunnel.

The mesh was a polyhedral hybrid mesh, whereby a traditional tetrahedral mesh with prism layers growing from the car surface was converted to a polyhedral domain inside FLUENT. Several meshing approaches were examined, and as with the choice of turbulence model, work is ongoing into verification to indicate the most preferable approach. However, with 16.5 million nodes around a half car (with symmetry plane), with a strong bias around areas of geometric complexity and high pressure gradients, it is anticipated that the results will provide reliable trends.

Selection of turbulence model was validated through simulating an isolated wheel and the front wing in isolation and comparing to wind tunnel data – the Realizable k-ε model was chosen from this work, in agreement with that of Doig et al [3,4], due to a preferable match to experimental pressure distributions and force coefficients compared to runs with Menter's kω-SST model and the 1-equation Spalart Allmaras model.

The moving ground of all simulations was set to be equivalent of the freestream appropriate to the scale being simulated, and the wheel rotation was similarly case-matched. Far-field boundaries at full-scale were located 23m upstream of the car and 70m downstream, 10m above the vehicle and 14m to the side, based on sensitivity studies indicating negligible influence on the car's force coefficients with increasing boundary distance.

Results

Initially, negative lift and drag coefficients for the entire car were evaluated, for both compressible and incompressible cases. As logic dictates, the incompressible simulations produced essentially identical results no matter the scale or speed due to Reynolds scaling. Minor (<0.01%) differences were observed, presumably due to the slight alterations to wall y^+ due to the scaling of the mesh.

Figure 2 plots the coefficients vs. speed for the various scales. It is clear that the compressible simulations indicate insignificant changes in downforce compared to the incompressible counterparts, even when the freestream Mach number is at 0.245 and producing a peak local Mach number of approximately 0.5 in the flowfield. The incompressible drag predictions show a trend of increasing under-prediction with Mach number if the compressible result is taken to be the "true" value (i.e. the most realistic), resulting in a 0.45% difference at the 60% scale case, and 1.3% at 36% scale. While these increments may initially seem insignificant, they may dwarf minor improvements to drag typically pursued with the tweaking of bodywork parts, and therefore a significant margin of error is introduced if incompressible CFD is used to compare to the wind tunnel data and track data.

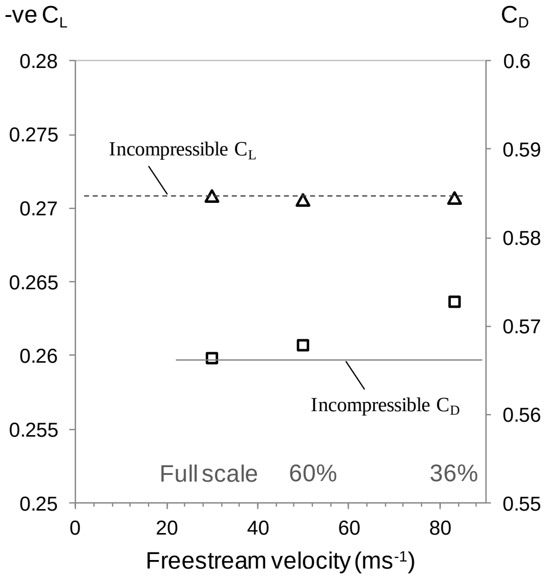

Figure 2- lift and drag coefficients for the entire car for the 3 Re-scaled cases, compressible and incompressible simulations.

The overall the differences could be written off as being small enough to negate considerations of the additional computational expense required to run compressible simulations. However, when the major individual components of the car are examined, particularly those which produce a lot of lift or downforce, a different picture emerges.

Figure 3 indicates the extent of the incompressible under-prediction for lift for the front wing, rear wing, underbody/diffuser arrangement, and the front and rear wheels. Both front and rear wings indicate a distinct discrepancy for downforce of over 1% at 36% scale, consistent with previous findings [3], with the simulations also indicating around 0.5% under-prediction for the lift on the front wheel and over-prediction on the rear. All of this is largely balanced by a significant 3.3% under-prediction of the downforce produced by the underbody/diffuser (around 1.6% at 60% scale), hence the negligible difference seen for the full car in figure 2. Thus, analysis at the macro scale produces a misleading impression – compressibility has an effect on each component in subtly different ways.

As could be inferred from the results in figure 2, drag is more sensitive to compressibility effects, largely due to the greater acceleration of flow around the downforce-producing parts resulting in thicker wakes and slightly earlier separation than would be expected in incompressible simulations. Figure 4 indicates this to be the case – even at 60% scale the incompressible simulations are off by over 1% for the wings and front wheel in terms of under-prediction. At 36% scale, a 2-3% difference is observed for the same components. The rear wheel is only slightly less-sensitive in the same conditions. The underbody/diffuser, for drag same as for lift, shows the opposite trend, and incompressible simulations over-predict by close to 1% at 60% scale, and over 2.5% at 36% scale. Regions of separation exist at the leading edge of the undertray which may make the flow particularly sensitive to small changes such as those brought on by compressibility, and may therefore cause the exaggerated trends that are seen in the graphs, though the performance of the diffuser is strongly linked to the flow around the rest of the vehicle and therefore is linked to compressible effects over the upper surfaces and, in particular, the lowest element of the rear wing arrangement.

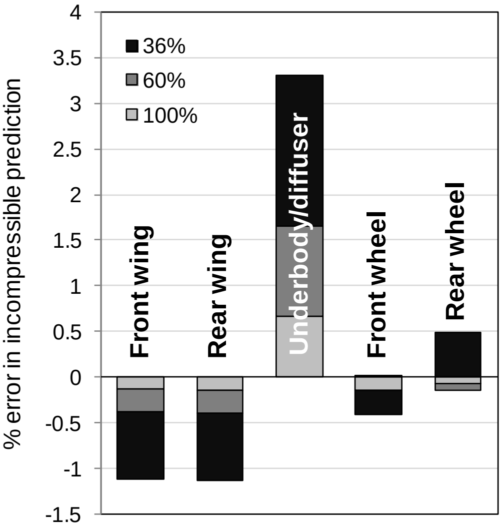

Figure 3- extent of incompressible predicted lift coefficient error for major individual components for the 3 Re-scaled cases.

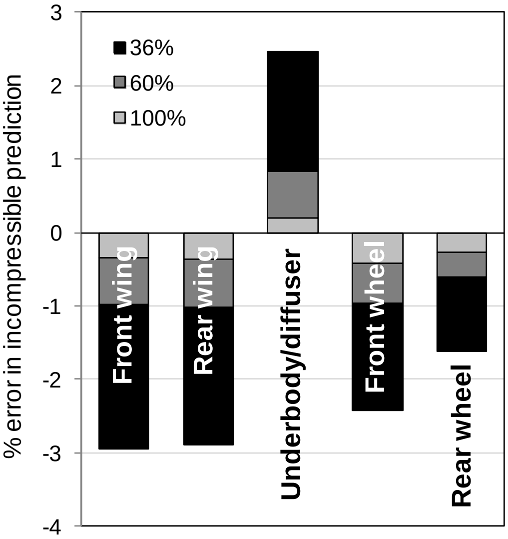

Figure 4- extent of incompressible predicted drag coefficient error for major individual components for the 3 Re-scaled cases.

Figures 5 and 6 highlight density changes around the vehicle on the symmetry plane as non-dimensionalised against the freestream value, shedding further light on the extent of compressibility effects around the wings and the underbody. Clearly, in a situation where parts are designed to be millimetre-perfect for optimal performance, the subtle but cumulative effects of compressibility may start to become a notable source of discrepancy between real-world testing and incompressible CFD. Clearly, the front and rear wings produce the strongest density reductions, which results in an increase in suction due to the greater ability of the flow to accelerate around the surfaces. The general wake region is also highlighted as being increasingly influenced by Mach number.

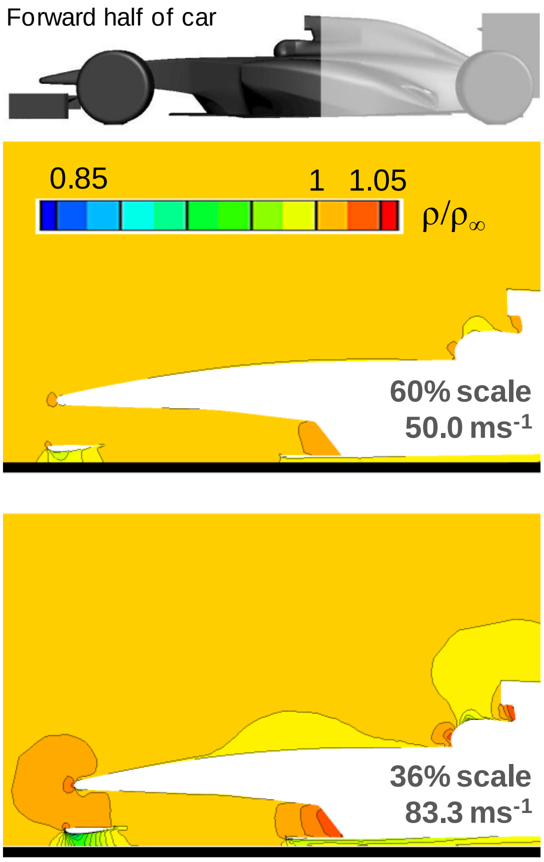

Figure 5- non-dimensionalised density changes around the forward half of the car on the symmetry plane.

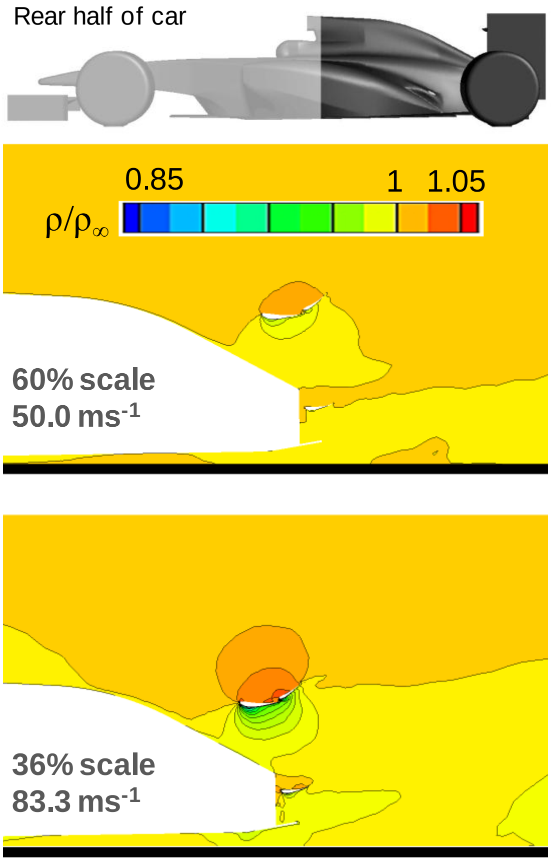

Figure 6 - non-dimensionalised density changes around the forward half of the car on the symmetry plane.

Conclusions

From simulations conducted on a racing car representative of a Formula 1 design, it has been shown that compressibility effects are present on such vehicles even at relatively low freestream Mach numbers of 0.147 and 0.245. There is a tendency for incompressible simulations to under-predict force coefficients for major components apart from the underbody/diffuser, where an over-preditction increases with increasing Mach number. Since all the simulations were run at the same Reynolds number, this presents a scenario similar to that which is common in the industry, whereby small-scale wind tunnel testing is matched to full-scale incompressible CFD at the same Re, and indicates that running compressible simulations for validation may well produce better correlation to wind tunnel and track data. The cumulative discrepancy in force coefficients for compressible compared to incompressible results is not considerable, but this is a consequence of the different parts of the car being affected in different ways by density changes, resulting in a coincidental "cancelling out" of errors, and therefore a more detailed analysis is required to uncover the extent of compressibility effects.

Further validation and verification will be conducted to ensure the trends established here are reliable. Future work will consider compressibility effects up to the maximum speed of the vehicle (close to Mach 0.3), and examine the influence of density changes on vortex behaviour and separation points.

References

[1] "2012 Formula 1 Sporting Regulations", Section 22, Fédération Internationale de l'Automobile. See also URL: http://www.formula1.com/inside_f1/rules_and_regulations/

[2] Diasinos, S. "The Aerodynamic Interaction of a Rotating Wheel and a Downforce Producing Wing in Ground Effect", PhD thesis submitted to the University of New South Wales, 2009.

[3] Doig, G., Barber, T., Neely, A. "The Influence of Compressibility on the Aerodynamics of an Inverted Wing in Ground Effect". ASME J. Fluids Eng. 133, 2011

[4] Doig, G., Barber, T.J., Leonardi E. and Neely, A.J., 2007. "The Onset of Compressibility Effects for an Inverted Aerofoil in Ground Effect". The Aeronautical Journal, 111(1126), pp. 797-806.

[5] Dominy, R. G., 1992. "Aerodynamics of Grand Prix Car". Proceedings of the Institution of Mechanical Engineers, 206, pp. 267- 274.

[6] Fackrell, J.E., "The Aerodynamics of an Isolated Wheel Rotating in Contact with the Ground," Ph.D. thesis, London, Imperial College of Science and Technology, 1975

[7] Katz J. Aerodynamics of Race Cars, Annual Review of Fluid Mechanics, Vol 38, pp 27-63, 2006

[8] Zhang, X., Toet, W. and Zerihan, J., 2006. "Ground Effect Aerodynamics of Race Cars". Applied Mechanics Reviews, 59, pp.33-49.

[9] Zhang, X. and Zerihan, J., 2002. "Aerodynamics of a Double Element Wing in Ground Effect". AIAA Paper 2002-0834.