Aerodynamic Simulation of a Standalone Rotating Tread Tire

Ales Alajbegovic 1) Phoi-Tack Lew 1) Richard Shock 1) Bradley Duncan 1) Jamie Hoch 1)

1) Exa Corporation
55 Network Drive Burlington, MA 01803, USA (E-mail: alesro@exe.com)

Received on June 29, 2017

ABSTRACT: The aerodynamics of a rotating tire can contribute up to a third of the overall aerodynamic force on the vehicle. The flow around a rotating tire is very complex and is often affected by smallest tire features. Accurate prediction of vehicle aerodynamics therefore requires modeling of tire rotation including all geometry details. Increased simulation accuracy is motivated by the needs emanating from stricter new regulations. For example, the upcoming Worldwide harmonized Light vehicles Test Procedures (WLTP) will place more emphasis on vehicle performance at higher speeds. The reason for this is to bring the certified vehicle characteristics closer to the real-world performance. In addition, WLTP will require reporting of CO₂ emissions for all vehicle derivatives, including all possible wheel and tire variants. Since the number of possible derivatives can run into the thousands of models, their evaluation in wind tunnels is not be practically possible. Therefore, simulations are the only alternative especially, since their use is allowed by WLTP. As a first step in order to meet these escalating demands, the current study uses a Lattice Boltzmann method (LBM) based computational fluid dynamics (CFD) solver using an immersed boundary method (IBM) based approach to simulate and validate a standalone rotating treaded tire. Simulated wake plane prediction results are in good agreement with experimental wake plane measurements. Effect of tire loading on wake results is also discussed.

KEY WORDS: heat, fluid, Rotating Tire, Cooling Drag, WLTP, Aerodynamic Efficiency, Immersed Boundary Method (D1)

1. Introduction

For vehicles throughout the world, manufactures are required to provide estimated fuel economy values that are used for consumer education, environmental certification and regulation, and in many regions, as a basis for CO₂ and other environmental taxation. The biggest drawback of the estimated fuel consumption reported today is that it is only an estimate for a particular trim. If a consumer wishes to change, add or remove certain features be it aerodynamic, e.g. spoilers or others such as winter tires, performance tires or rims, this will greatly impact the fuel efficiency but is not reflected in reported metrics. In-situ mileage information from OEMs is under current regulations. However, with the upcoming Worldwide Harmonized Light Vehicles Test Procedure (WLTP) proposed for 2017, new emphasis will be placed on high-speed realistic driving conditions of configured vehicles. This means that vehicle OEMs will be required to certify every vehicle configuration available on the market. This places a huge burden on the OEMs to ensure that new vehicle designs will meet fuel efficiency targets, not only for the lowest drag configuration, but for all trim levels and options offered throughout the product line.

This attention to on-road performance is also reflected in the evolving views and buying habits of today’s consumers. Vehicle range is one of the main differentiators for EV and PHEV vehicles. For many drivers of electric vehicles, range is part of the consideration of every trip they make, and thus there are plenty of opportunities for the consumer to compare real world performance to that which is provided by the manufacturer, influencing perceived value and quality. Also, for both electrified and traditional internal combustion (IC) vehicles, consumers can now easily monitor their actual efficiency via on board diagnostics, or increasingly via crowd sourced, real world efficiency metrics. For a given consumer, the on-road performance of their vehicle necessarily implies ‘as configured’. Thus, both during the purchasing process and throughout the product life cycle, the consumer is paying more and more attention to the performance of the actual vehicle, and not just the ‘low energy’ configuration that is typically reported today. Providing accurate information on a per vehicle basis will aid the customer and guide them in their buying decision. However, it also creates significant additional effort for vehicle manufacturers to evaluate emissions values for the many vehicles that they manufacture with all their configurations. Performing wind tunnel measurements for multiple trims and configurations is not only time consuming and cost-prohibitive but can be also plagued with measurement uncertainties. Hence, the use of efficient Computational Fluid Dynamics (CFD) is key in aiding and complementing the design of future vehicles to meet the demands set forth by WLTP and evolving consumer demands.

Due to these pressures, accurate, efficient, and insightful computational simulations for aerodynamics are now a fundamental requirement for the digital vehicle development process. Using detailed aerodynamics simulations of the full...
vehicle geometry, including accurate engine bay, underbody components, suspension, brakes, and wheels, almost every aspect of aerodynamic design can be explored.

It is estimated that 25-30% of vehicle drag is generated from the wheel/rim and tire assembly region\(^2\). This also includes the wheel well area. Because of the high drag from the tires/wheel assembly region, the overall fuel consumption of a vehicle is heavily influenced by the choice of wheels and tires. Furthermore, all tires on light vehicles are treaded tires. Treaded tires have unique aerodynamic characteristics due to the small-scale separations and pressure gradients created by the passage of tread features past the approaching airstream. The tread can significantly affect the flow into the wheel and brake regions, affect the downstream size and direction of the wheel wake, and in turn affect the aerodynamic drag of the entire vehicle. These factors are important for the design of the body, wheel deflectors, and underbody treatments, as well as the development of aerodynamic tire and wheel concepts. They impact brake cooling performance and other design attributes like soiling. If the overall flow features from the wheel/tires are not well understood, it is difficult to make design decisions, not only related to tires and rims, but on the entire vehicle, since the interactions with different wheels and tires may not be known. Correctly predicting the effects of rotating tread early in the design requires multiple technologies to provide predictive capabilities on design iterations with accuracy and efficiency.

However, aerodynamic simulation of a truly rotating treaded tire on a vehicle remains a challenging task due to complex fluid-structure interaction between the tire and ground, i.e. contact patch and the aerodynamic flow forces around that region. Moreover, simply simulating the moving tread in proximity of the ground plane is a challenge, as true moving geometry is required. The region around the tread cannot be handled via sliding mesh or other reference transformations since the interaction of the treads with the ground plane is one of the main areas of interest.

The final objective of this work is a complete aerodynamic vehicle design with rotating treaded tires. As a first step in that direction a preliminary validation study of standalone rotating treaded tire using Lattice-Boltzmann Method (LBM) is performed. As hinted previously, the flow structures in the wheel/tire housing are highly complex and thus it would be naïve to assume that accurate analysis and results of a standalone tire will immediately translate to an efficient vehicle design. Nonetheless, due to the relative simplicity of the setup knowledge can be gained to validate the new methodology towards the final goal.

2. Brief Description of Rotating Treaded Tire Method

The rotating treaded tire method is based on the immersed boundary approach originally developed by Peskin\(^3\) intended to efficiently solve cardiovascular flow problems. In the immersed boundary, the elastic or inelastic material is treated as part of the fluid. Specifically, a Cartesian computational grid does not conform to the shape of a solid body such as a rotating tire. Instead a body force is applied at the interface of the material which therefore acts on the fluid governing equations. As a transient simulation progresses, the material which is tracked in the Lagrangian frame receives and exerts a body force on the surrounding fluid. The attractiveness of this method is that it is efficient and robust when applied to tire/road problems. This method is the basis for the rotating tread simulation in PowerFLOW.

3. Computational Methodology

The commercial CFD package, PowerFLOW 5.4 from Exa Corporation was used to solve the flow field numerically. The software is based on the Lattice–Boltzmann numerical scheme using Very Large Eddy Simulation (VLES) approach for turbulence modeling. The accuracy and robustness of this technology in solving transient aerodynamics over complex domains has been well established\(^4\)-(\(^1\)\(^5\)). The rotating tread is solved in conjunction with the Lattice-Boltzmann method at each time step during the simulation.

4. Simulation Detail & Setup

Figure 1 shows a realistic, high quality computer aided design (CAD) representation of a standalone treaded tire attached to a sting. The computational setup follows as close as possible the experimental wind tunnel setup that were performed by Schnepf et al.\(^16\). The tire with covered rim is type 225/55 R17 or more specifically has a base width of 225mm, 55% side wall height aspect ratio (123mm) and a radius of 17 inches (431.8mm). The top of Figure 2 shows a close-up view from the front of the tire base. The tread channel gaps measure from 2mm at its smallest to 4mm at its largest, respectively.
On the bottom of Figure 2, the contact patch shape is shown. The incoming flow velocity was set to 140 kph (38.89 m/s) and the floor or moving belt in the simulation and experiments is set to the same freestream velocity.

For a standard aerodynamic tire simulation, two key setup features are typically used. First only the rim is enclosed in a Local Reference Frame (LRF), i.e. the rim is truly rotating and not the tire. If the tire was included within the LRF, the periphery of the LRF will intersect the ground and part of the floor will rotate within the LRF. Clearly this is unphysical. Thus, an LRF is applied only to the wheel rim. On the tire, the previous preferred approach was to use a rotating wall boundary condition on a grooved tire. A grooved representation of the treaded tire is preferred due to the moving wall BC’s inability to impart momentum in the normal direction on the treads. By using grooved representation, the maximum amount of tangential momentum is able to be transferred to the fluid by the moving wall BC. For this study, a grooved tire case on the sting was also setup and run with the same resolution but without the use of the immersed boundary as a reference. Figure 3 shows a closer surface comparison between the treaded and grooved tire.

It is important to note that the simulation setup is for an undeformed tire whereas in experiments, the tire is vertically loaded and therefore has a deformed bulge on the contact patch. The main reason is that the current implementation of the rotating tread capability in PowerFLOW requires an axis-symmetric tread pattern. Upcoming versions of PowerFLOW will support a deformed tire, but since those versions were not available, an undeformed treaded tire was used. Differences between experiments and simulation will be addressed in the discussion section.

As a pre-step before comparing tire-ground results, two ‘flying’ wheel simulations were performed. First was using an LRF on the entire tire and second the immersed boundary. The results were nearly identical in terms of flow structures and the drag and lift force difference differed by 5 percent.

5. Results

5.1. Transient Flow-Field
Figure 4 shows the instantaneous snapshot of velocity magnitude filled contours on a centerline slice of the tire.
Note the thin boundary layer on the tire (top left of tire) is preserved until the flow separates at roughly 90deg measured from the stagnation point of the tire. Figure 5 shows the instantaneous Lambda-2 iso-surfaces colored by velocity magnitude.

The Lambda-2 vortex criterion is a method to properly identify vortex structures from a three-dimensional velocity field (17). The Lambda-2 is Galilean invariant which produces the same results even when the velocity field is translated. Note the fine structures that are resolved not only around the contact patch, but within the channels of the treads as well in the channels of the treads. This is further underscored in Figure 6 which shows the instantaneous total pressure contours for a fluid plane slice in the contact patch region.

Figure 7 shows a schematic of two fluid plane measurement regions of the flow field that correspond to what was captured in the wind tunnel. The main wake plane has dimensions of 0.5m in span and 0.7m in height. The main wake plane is placed 20mm behind the tire at an X-coordinate location of x=0.363m with x=0m being the center of the tire. The second plane is near the contact patch which has a span of roughly 0.18m and height of 0.15m. The contact patch measurement plane is located at x=0m with the nearest edge of the plane about 15mm away from the tire sidewall. Further details of the experimental measurement technique can be found in Schnepf et al (14).

5.2. Mean Flow-Field

Figure 8 through 10 show the mean total pressure at the wake plane and contact patch regions for PowerFLOW with rotating tread, PowerFLOW with rotating wall boundary condition (BC) and experimental wind tunnel measurements, respectively. Observing the three figures, the rotating tread result shows a
considerable improvement over the rotating wall boundary condition when compared to wind tunnel wake survey. The overall tire wake profile is slimmer in Figure 8(a) compared to Figure 10(a). Furthermore, the top of the tire wake shape at the height of the separation is at least captured better compared to Figure 9(a) which is non-existent. The lobed vortex structure at the bottom of the tire close to the belt (left or outboard) although an improvement over rotating wall, still shows some discrepancies compared to experiments. Further explanation of this behavior is discussed in the next section.

For Figure 9(a) the wake pressure losses near the floor show total pressure losses spanning the nearly the entire measurement window. Similar observations are also seen for the contact patch, i.e. Figure 9(b).

To gain a better view and understanding on the three-dimensional structures, Figure 11 shows the mean total pressure for iso-surface value of zero viewed from the rear and front. Focusing on floor flow structures, although the deflection location begins at the same point (where tire meets the belt) the deflection angle differs starkly resulting in a large pressure loss as shown in Figure 9(a). One can also observe the separation on the side wall for the treaded tire is smaller and tighter compared the grooved tire. It can be clearly seen that the increased momentum transfer by directly simulating the moving tread blocks allows the flow to stay more attached around the contact patch. From the front view, the iso-surface of total pressure clearly shows separated flow near the top of the tire whereas none is shown for the grooved tire. In this case, the increased momentum transfer from the tread block to the fluid is causing the flow to separate earlier, as on the top of the tire the momentum transfer is in the negative streamwise direction. Furthermore, the wake from the treaded tire has more of an ‘upwash’ again indicating that momentum from the fluid in the contact patch region is properly transferred downstream compared to grooved tire. Centerline mean total pressure comparing rotating tread, rotating wall and experiment are shown in Figure 12.

Due to the limited near wall data in the experimental plot, it is difficult to make a proper analysis as to where the separation location occurs in the wind tunnel measurement (Figure 12(a) vs Figure 12(e)). However, qualitatively, the height of the wake from rotating tread is in good agreement with the measurement. The elevated total pressure exhibited in the measurement could be due...
to the elevated measurement uncertainty in reversed flow regions. Conversely, the rotating wall BC, Figure 12(b), clearly does not show any separation behavior at least on the centerline, as the rotating tread is not able to impart as much momentum in the negative streamwise direction as the rotating tread.

Figure 13 shows the mean y-velocity contours on a fluid plane located at \( Y = -0.15 \text{m} \) which is located on the outboard of the tire. Here there is a very good qualitative comparison between Rotating Tread and the wind tunnel measurements, where the magnitude of the sidewash is well captured, i.e. Figures 13(a) and 13(c). The rotating wall BC however (Figure 13(b)) does not exhibit the same behavior as the grooves do not 'pick-up' and discharge the flow inward. One of the reasons here is that the rotating tread is more efficient in transferring momentum and energy from the contact patch downstream. This is highlighted in Figure 14 which shows the mean total pressure plane cutting through the contact patch between the floor and inner tire casing. The image shows a clearer view of the behavior in the contact patch. On the outboard and inboard, the planar wake deflection angle is smaller for the treaded tire. In fact, the inboard planar wake of the treaded tire is pushed outboard and with less pressure losses compared to the grooved tire.

The streamwise channels on the treaded tire suffer less pressure losses overall compared to the groove tire. Although the two center channels on the groove tire suffer less of a pressure drop, the outer and inner channels do not. The flow appears to be experiencing more blockage and therefore energy of the fluid has to go around the patch peripheral resulting in a larger patch wake. Due to the treaded tire truly rotating and its tread pattern, this flow constriction is alleviated resulting in efficient jetting through the main channels. No forces were reported by Schnepf et al. but based on the simulation, the rotating tread contributes about 20% of the total drag on the tire.

Figure 14 Mean total pressure slice cut between floor and inner tire casing. Rotating Tread (left) and grooved tire (right)
The rest is from the tire casing, rim and rim cover. Overall, the results from the rotating tread method is very encouraging and offers a leap for the study of rotating treads and rim/tire design.

6. Further Discussion on Tire Wakes

Although the main tire wakes predicted by the rotating tread has shown remarkable improvement compared to the grooved tire, there are still some differences in the flow structure near the floor. Observing the tire wake pattern in Figures 8 and 10, the floor wake is weaker and smaller and appears to be have more up-wash in the wind tunnel measurements. As mentioned, the simulated tire is unloaded (bulge-absent) whereas in the wind tunnel experiments, the tire is vertically loaded with approximately 5 kN and therefore a bulge is present at the contact patch. Schenpf (18) studied the effect of different vertical loads on the standalone tire and found that the loading or bulge plays a crucial factor in determining the ultimate wake shape near the floor.

Figure 15 shows the standalone tire under three different types of loads from light to heavy and its effect on the floor tire wake. A lighter vertical load indicates that the tire side wall bulge near the contact patch is smaller compared to a heavily loaded tire, i.e. smaller curvature compared to a large curvature for a heavier load. One could also postulate that as the load gets heavier so does the size of the contact patch which could affect the floor wake structure. As mentioned, when the tire was raised (less of a penetration depth) in the simulation, the wake structure did not improve compared to the wake structure in Figure 9. Hence, the penetration depth of the unloaded simulated tire is analogous with the vertical load applied on the wind tunnel tire. To make a qualitative comparison, the simulation results compare well with the medium loaded tire (3.5 kN) from experiments and this is shown in Figure 16.

7. Summary and Next Steps

Presented is a Lattice-Boltzmann Method using an immersed boundary method extension to study a rotating treaded tire. In this work, the rotating tread has shown considerable improvement and encouraging results when compared to a standard rotating wall boundary condition. The mean total pressure is now in good agreement with the experimental wake survey right behind the tire and at the contact patch. Differences in floor wake structure is most likely due to the absence of the bulge in the simulated tire. From the simulation, the current results closely resemble a moderately loaded tire in the experiments. Other factors that need to be studied and compared to as the effect of further resolution, tire roughness and the implementation of a bulge at the contact patch.

Accurately predicting flow around a standalone tire is a necessary first step towards robustly predicting a typical wheel/tire implementation on a road vehicle. The following step of demonstrating rotating tread and the impact on wheel and tire interaction for a road vehicle will be presented in the near future. That work and further standalone tire work related to investigations into the importance of rotating treads with tire buldge are are currently on-going.

References

(1) “Worldwide Harmonized Light Vehicle Test Procedure”, https://www.dieselnet.com/standards/cycles/wltpph.php, (Accessed November. 2. 2016).
(2) Wickern, G., Zwicker, K., Pfadenhauer, M., “Rotating Wheels – Their Impact on Wind Tunnel Test Techniques and on Drag Results”, SAE Technical Paper 970133, (1997).
(3) Peskin, C. S., “Flow patterns around heart valves: A digital computer method for solving the equations of motion”, PhD
(4) Chen, H., Teixeira, C. and Molvig, K., “Digital Physics Approach to Computational Fluid Dynamics, Some Basic Theoretical Features,” Intl. J. Mod. Phys. C, Vol. 8, No. 4, pp. 675-684, (1997).

(5) Chen, H., Chen, S., and Matthaeus, W. H., “Recovery of the Navier-Stokes Equations using a lattice-gas Boltzmann Method,” Phys. Rev. A, Vol. 45, No. 8, pp. R5339–R5342, (1992).

(6) Qian, Y., d’Humieres, D., and Lallemand, P., “Lattice BGK Models for Navier-Stokes Equation,” Europhys. Lett., Vol. 17, pp. 479–484, (1992).

(7) Chen, H., “Volumetric Formulation of the Lattice Boltzmann Method for Fluid Dynamics: Basic Concept,” Phys. Rev. E, Vol. 58, pp. 3955-3963, (1998).

(8) Chen, S. and Doolen, G., “Lattice Boltzmann Method for Fluid Flows,” Ann. Rev. Fluid Mech., Vol. 30, pp. 329-364, (1998).

(9) Chen, H., Kandasamy, S., Orszag, S., Shock, R. et al., “Extended Boltzmann Kinetic Equation for Turbulent Flows”, Science, Vol. 301, pp. 633-636, (2003).

(10) Chen, H., Orszag, S., Staroselsky, I. and Succi, S., “Expanded Analogy between Boltzmann Kinetic Theory of Fluid and Turbulence”, J. Fluid Mech., Vol. 519, pp. 301-314, (2004).

(11) Li, Y., Shock, R., Zhang, R. and Chen, H., “Numerical Study of Flow Past an Impulsively Started Cylinder by Lattice Boltzmann Method,” J. Fluid Mech., Vol. 519, pp. 273-300, (2004).

(12) Fares, E., “Unsteady Flow Simulation of the Ahmed Reference Body using a Lattice Boltzmann Approach”, Comput. Fluids, Vol. 35 (8-9), pp. 940-950, (2006).

(13) Duncan, B., Sengupta, R., Mallick, S. et.al., “Numerical Simulation and Spectral Analysis of Pressure Fluctuations in Vehicle Aerodynamic Noise Generation”, SAE Technical Paper 2002-01-0597, (2002).

(14) Vaillant, O. and Maillard, V., “Numerical Simulation of Wall Pressure Fluctuation on a Simplified Vehicle Shape”, AIAA Paper 2003-3271, (2003).

(15) Senthooran, S., Crouse, B., Balasubramanian, G. et al., “Prediction of Wall Pressure Fluctuations on an Automobile Side-glass using a Lattice-Boltzmann Method,” AIAA Paper 2006-2559, (2006).

(16) Schnepf, B., Schütz, T., and Indinger, T., "Investigations on the Flow Around Wheels Using Different Road Simulation Tools," Progress in Vehicle Aerodynamics and Thermal Management, Proceedings of the 9th FKFS Conference, Stuttgart Germany (2013).

(17) Jeong, J. and Hussain, F., “On the Identification of a Vortex”, J. Fluid Mechanics, 285 pp. 69-94, (1995).

(18) Schnepf, B. “Untersuchung von Einflussfaktoren auf die Umströmung eines Pkw-Rades in Simulation und Experiment”, PhD Thesis, Technical University of Munich, December (2015).