

A Computational Study of the Flow Around an Isolated Non-Pneumatic Tire

Hyeonu Heo
Korea Aerospace University

Jaehyung Ju
University of North Texas

Doo Man Kim and Harkbong Kim
Korea Aerospace University

ABSTRACT

An understanding of the flow around a tire in contact with the ground is important when designing fuel-efficient tires as the aerodynamic drag accounts for about one third of an entire vehicle's rolling loss. Recently, non-pneumatic tires (NPTs) have drawn attention mainly due to their low rolling resistance associated with the use of low viscoelastic materials in their construction. However, an NPT's fuel efficiency should be re-evaluated in terms of aerodynamic drag: discrete flexible spokes in an NPT may cause more aerodynamic drag, resulting in greater rolling resistance. In this study, the aerodynamic flow around a non-pneumatic tire in contact with the ground is investigated for i) stationary and ii) rotating cases using the steady state Reynolds-Averaged Navier-Stokes (RANS) method. A sensitivity analysis was carried out with a varying mesh density. The flow into cavity by the discrete spoke geometry of the NPT does not significantly affect the overall aerodynamic drag.

KEYWORDS: Aerodynamic flow, Non-Pneumatic Tire (NPT), Reynolds-Averaged Navier-Stokes (RANS), Computational Fluid Dynamics (CFD)

CITATION: Heo, H., Ju, J., Kim, D., and Kim, H., "A Computational Study of the Flow Around an Isolated Non-Pneumatic Tire," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(1):2014, doi:10.4271/2014-01-9123.

INTRODUCTION

Recently, some tire engineers have attempted to develop innovative non-pneumatic tires (NPTs) by building flexible polygon type lattice spokes to replace the air of the pneumatic tire [1, 2, 3, 4, 5, 6, 7, 8]. NPTs have drawn attention mainly due to the potential to overcome current disadvantages of the conventional pneumatic tires; catastrophic damage (i.e. flats) while driving, required maintenance of the air pressure, and complex manufacturing processes. Moreover, the potential reduction of rolling resistance with the use of low viscoelastic materials in the construction of NPTs is getting tire engineers more excited.

In spite of the low rolling resistance of NPTs and their contributions to better fuel economy, their fuel efficiency may need to be re-evaluated in terms of aerodynamic drag: Discrete flexible spokes in an NPT may cause more aerodynamic drag,

resulting in higher fuel consumption. Eventually, we attempt to estimate aerodynamic performance of an NPT at its relatively early stage of development.

By the perspective of aerodynamics, tires are known to account for up to one third of the total drag on a vehicle's momentum [9]. For this reason, the aerodynamic characteristics of the flow around a tire are important in designing a tire, which helps a vehicle to achieve greater fuel efficiency [10]. Several numerical and experimental studies on the flow around a tire in contact with the ground have been reported [11, 12, 13, 14]. A rotating tire was reported to produce less lift and drag than a stationary tire due to the cumulative effect of separation areas, distribution of pressure of the tire surface and vortices near the rotating tire [11, 12, 13, 14]. Fackrell and Harvey measured the aerodynamic force and pressure of the air around an isolated and undeformed tire with moving ground [11]. Drag and lift forces were characterized according to a tire model with varying tread widths and groove profiles. They observed a unique feature of the rotating wheel

flow and one that had an important influence on the surrounding flow field. They saw a sharp positive pressure peak which was the result of a “pumping” effect. This effect occurs as the boundary layers on the rotating tire and ground surfaces are forced towards the line of contact before emerging in a jet at the edges of the tire.

Axon and Garry [12, 13] conducted a computational study of the flow and the aerodynamic characteristics around a tire in contact with the ground. The Steady Reynolds-Averaged Navier-Stokes (SRANS) method was used for calculating the flow around the tire with the two-layer Renormalization Group (RNG) $k-\epsilon$ turbulence model to validate the experiments of Fackrell and Harvey [11]. Their model was simplified in its hub and sidewall profiles. The results for the stationary and rotating tires were close to those of the experiment by Fackrell and Harvey [11]: For the rotating tire, the computed lift (C_L) and drag (C_D) coefficients were 0.476 and 0.602, respectively, which were within error ranges of 8.2% and 3.8% respectively, compared with the previous experimental results by Fackrell and Harvey. They showed that computational fluid dynamics (CFD) could provide qualitative results for the flow around a rotating tire on a moving ground and a stationary tire on fixed ground.

McManus and Zhang [14] conducted a computational study on the flow around an isolated tire in contact with the ground using the Unsteady Reynolds-Averaged Navier-Stokes (URANS) method for two cases; a stationary tire on stationary ground and a rotating tire on moving ground. Using the same geometry of the tire that was physically tested by Fackrell and Harvey, they compared their numerical simulations with experimental results. The computational results showed a good agreement with the previous experimental results in predicting the surface pressures, pressure lift and drag forces.

Advanced methods to measure the flow around a tire were suggested considering a deformed tire and a camber angle which may induce an asymmetric lateral flow through the tire width [15, 16, 17, 18, 25, 26]. The aerodynamic models were especially targeted for a formula one pneumatic tire with various turbulent codes. In spite of claiming the deformed shape of a tire is imperative to predict the flow field accurately, they didn't clearly distinguish the deformed effect. They show an overall effect of actual shape with the full brake geometry, brake cooling ducts, and a loaded deformable pneumatic tire with treads. Even if the references [16, 17, 18] claim that deformed tire geometries were used for aerodynamic simulation, it appears to be hard to claim its deformed tire geometry without considering deformed shapes by a vertical vehicle load; the deformed tire shape was justified with a contact area on the ground, not actual deformation in the width.

In this paper, considering a relatively early stage on development of NPTs, we investigate aerodynamic flow around an NPT in contact with the ground for i) stationary and ii) rotating NPTs using the Steady Reynolds-Averaged Navier-Stokes (SRANS) method through computational fluid

dynamics. This paper is organized by the geometry of an NPT, boundary conditions of bulk airflow, and computational setup, followed by simulation results and discussion. We will leave a computational study with the URANS method to our future study. The NPT has more complex airflow and a higher drag force than the pneumatic counterpart has due to its sharp-edged shoulders.

COMPUTATIONAL MODEL AND METHOD

In this section, we introduce our computational setup for simulation of aerodynamic performance of an NPT. The setup was made on the basis of the previous experimental and numerical studies on aerodynamic flow of a pneumatic tire [11, 14].

Tire Geometry

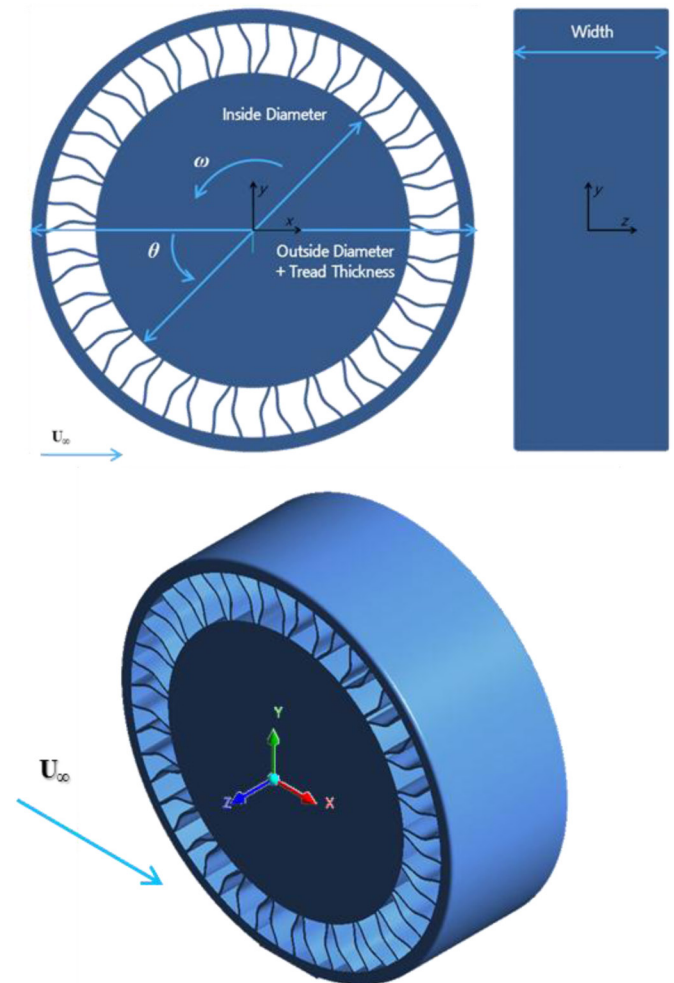


Figure 1. Geometry of the NPT [19, 20]

An NPT's geometry is selected from Narasimhan et al.'s work [19, 20] and is shown in Figure 1. The selected model does not necessarily mean an optimized geometry. We simply selected it based on its availability in the public domain. It consists of 25 spoke pairs. Each spoke is curved, paired and placed at equal intervals along the circumferential direction.

and wheel of the NPT are modeled as a cylinder. An angular velocity, ω , is considered for analysis of the rotating condition of the NPT (Figure 1).

θ refers to the angle in the circumferential direction (Figure 1). The outer diameter of the NPT is 584 mm and its lateral width is 210 mm. The thickness of each spoke is 4.2 mm and their curvature is 8 mm. The detailed geometric dimensions of the NPT are summarized in Table 1.

Table 1. Dimension of parts of the NPT [19, 20]

Parts of the NPT	Dimension
Outside Diameter	584 mm
Inside Diameter	422 mm
Tread Thickness	5 mm
Ring Thickness	15 mm
Number of Spoke Pairs	25
Spoke Thickness	4.2 mm
Width	210 mm
Spoke Curvature	8 mm

Computational Domain and Grid

The computational grid is modeled on the basis of the previous experimental and numerical configurations for studying aerodynamics of pneumatic tires [11, 14]. A Reynolds number of 5.3×10^5 was used for the physical and numerical experiments when the pneumatic tire had a diameter of 0.417 m and a width of 0.191 m, corresponding to a flow velocity of 18.6 m/s (70 km/h) [11, 14]. To match the same Reynolds number of 5.3×10^5 , we used an inlet air velocity of 14.0 m/s (50.4 km/h) for the given NPT diameter (0.584 m) and width (0.210 m).

The geometry of the computational grid is shown in Figure 2. All the geometric quantities for a three dimensional (3D) flow-field are non-dimensionalized with the NPT diameter, d (=584 mm). The distance from the inlet boundary, x_1/d , is set to be 5. The distance to the outlet boundary, x_2/d , is set to be 15. The domains on the width, z_1/d , and height, y_1/d , are 3.66 and 2.93, respectively. Figure 2 shows the geometry of the computational domain. A symmetric geometry for the lateral (z) direction is considered to reduce the computational cost as shown in Figure 2.

In the ground vehicle, the contact patch geometry is of importance because it affects aerodynamic performance around a tire. Following previous studies, it was found that the length of the contact area corresponds to contact angles of approximately between 80° and 100° for the θ defined in Figure 1 [12]. In this study, we used the contact patch for the contact angles at $\theta = 80^\circ$ and 100° .

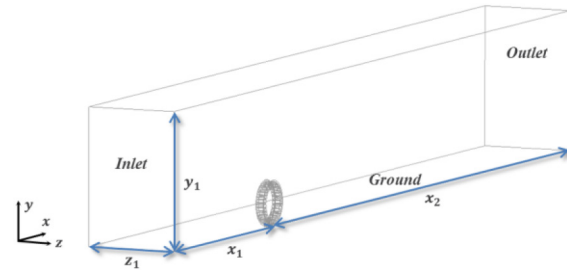


Figure 2. Computational fluid domain for the aerodynamic analysis of the NPT

The computational grid was made with tetrahedral blocks applied to the fluid (air) region. The number of the grid was about 6.01, 5.83 and 4.64 million in fine, medium and coarse grids, respectively, for a grid sensitivity study. The minimum edge size of the fluid around the spokes and the NPT region was set to be 0.1 mm. The grid size on the rest of the fluid region was linearly relaxed by a growth factor of 1.2 when the grid was moved away from the NPT profile region.

Boundary Conditions

A uniform inlet velocity profile at 14.0 m/s (50.4 km/h) is applied to match the reference Reynolds number of 5.3×10^5 considering the NPT geometry. On the outflow boundary, the gauge pressure at the outlet is set to 0 Pa when the reference pressure at the inlet is 1 atm. A slip boundary condition was applied on the tunnel walls and a symmetric boundary condition is applied through the x - y plane perpendicular to the lateral direction of the NPT (Figure 2). Both top and side surfaces were modeled as slip walls on which a free stream condition applies. A no-slip boundary condition was applied on the NPT surface and the ground. Different velocities on the boundary conditions were set to simulate stationary and rotating cases of the NPT. For the stationary case, the velocity on the surface of the NPT and the road is set to zero. For the rotating case, an angular velocity of 47.14 rad/s was applied on the surface of the NPT, which is equivalent to the free-stream velocity of 14.0 m/s (50.4 km/h). A moving ground with a velocity of 14 m/s in the x direction is applied for the rotating case of the NPT. A no-slip condition is applied on the ground surface.

Computational Method

The steady state Reynolds-Averaged Navier-Stokes (RANS) equations are applied for incompressible air flow's continuity and momentum. They are the most widely used approach to calculating vehicles' air flow. The RANS equations are solved using an implicit, segregated, and 3D finite volume method [20]. The continuity and momentum equations are, respectively,

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \quad (1)$$