A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking

Jakub Broniszewski and Janusz Piechna

Faculty of Power and Aeronautical Engineering, Warsaw University of Technology, Warsaw, Poland

ABSTRACT
This study investigates the impact of unsteady aerodynamic loads on the behavior of car dynamics during braking maneuvers. In the analysis, two independent solvers were coupled to solve an existing fluid structure interaction (FSI) problem. Transient CFD analysis applying Fluent software was used to obtain realistic aerodynamic loads. A full car, nonlinear dynamic model with elastic suspension system was built in a dedicated multibody dynamic system MSC.Adams/Car. Both physics were integrated into one ecosystem via a block diagram environment for multidomain simulation (Matlab/Simulink). Methodology elements were validated against experimental results for a rectangular beam in cross-flow with vortex-induced vibrations. The final results were validated against a full-scale experiment on a real car. The solution of an untypical flow problem (decreasing car speed during a braking maneuver) required an untypical reference frame. The classic case, with an observer at rest (stationary model and moving air), was replaced with a moving observer. The whole domain was translocated during the analysis with a velocity which varied with time. The problem was further complicated by the presence of movable aerodynamic elements. The overset mesh technique was used to allow movement of the car body and active aerodynamic surfaces. The results obtained show the importance of aerodynamic effects on the braking process. It was shown that in the analyzed scenario, movable aerodynamic features can reduce the distance to stop by 6%. The complex interaction between the active aerodynamic surface and the car body with respect to load split is discussed as well. The methodology proposed to determine the behavior car dynamics with unsteady aerodynamic effects taken into account appears to be a significant improvement when compared with existing methodologies used for such an assessment.

1. Introduction

Unsteady aerodynamic effects are very common in normal life. Leaves on tree or a flag waving in the wind are examples from normal life. From a technical perspective, such phenomena have many implications because it can impact the dynamic behavior of a structure. One of the most spectacular manifestations of unsteady aerodynamic effect was the Tacoma Narrows bridge collapse on 7 November 1940. In the world of competition cars, the most famous examples of the consequences of rapid change in aerodynamic forces occurred during the Petit Le Mans race in 1998, 1999, and 2000. All of these races had cars taking-off and flipping in the air. The front wheels axis lost downforce in the wake of the preceding car. Cases like these show how important the prediction of dynamics behavior could be when unsteady aerodynamic effects are taken into account. These are examples of fluid structure interaction (FSI) problems.

Dynamically car model can be treated as a system of five masses (four wheels and a chassis) connected by system of springs and dampers. The chassis can move up and down and rotate along three axes, according to gravitational, inertia and aerodynamic forces as well forces generated by tires (acceleration, braking, cornering). In the case of a fast car, with a complicated chassis shape that forces a complicated flow structure around it, which results in the generation of aerodynamic forces dependent on the dynamic motion of the chassis, predicting unsteady aerodynamic forces is a crucial task. Instantaneous inclination of the car in relation to the road and the clearance under the chassis have an influence on the unsteady aerodynamic forces. Typically, there exist separate numerical tools used for the simulation of the chassis motion such as MSC.Adams/Car and the calculation of unsteady aerodynamic forces such as ANSYS-Fluent. The logical structure of these software programs is totally different. MSC.Adams/Car integrates in time a set of...
equations of motion of a relatively low number of masses loaded by external forces. Fluent calculates flow parameters inside a great number of small volumes being the result of discretization of the large space around the moving car. This paper presents a new method of simultaneous calculation of the motion of car elements (wheels, chassis, aerodynamic elements) and the resulting aerodynamic forces, exchanging geometrical data and force data between different solvers. A detailed algorithm and tools used for such exchange of data online is presented.

The computational fluid dynamics (CFD) analyses are extensively used to study vehicle steady and unsteady aerodynamics. Fu, Uddin, and Robinson (2018) compared different RANS models for modeling the flow over a NASCAR car. One of the conclusions from the work states that the $k - \omega$ SST model shows some advantages in multiple horse shoe vortex predictions. A direct comparison of detached eddy simulation (DES) and RANS models to the experimental data (Morden, Hemida, & Baker, 2015) shows that the $k - \omega$ SST model can reduce computational expense with a slight reduction in the accuracy in predicting forces and surface pressures (Liu, Chen, Zhou, & Zhang, 2018). Some studies are focused on one, specific aspect of CFD modeling. Hobetka and Sebben (2018) made a study regarding the modeling of a rotating wheel. In their paper, several techniques were compared such as rotating wall (RW), moving reference frame (MRF), sliding mesh (SM) and their combinations. The strengths and weaknesses of each of them are described in detail. It was noticed that the wheels contribute to approximately 25% of the overall aerodynamic drag (Wickern, Zwicker, & Pfadenhauer, 1997). On the other hand, models of vehicle dynamics have been widely used in the automotive industry for many years to check whether dynamic requirements are met. These two types of analyses are sometimes combined to solve the FSI problem. Cheng et al. (2013) in their research studied car body aerodynamic damping during pitching. To do that, they selected the transient numerical simulation based on the LES method and the vehicle motion was realized with the arbitrary Lagrangian–Eulerian (ALE) method. This approach made it possible to perform a fully coupled analysis. The computational mesh contained 12 million cells. This value seems to be rather low in comparison with other studies. The boundary layer was modeled with nine layers and the height of the first layer was 1 mm. This ensured that $y^+ < 150$ everywhere. The quasi-static values of lift and drag coefficients were compared to the wind tunnel data and an almost perfect match was reported. Recent published research activities are focused on analyzing the impact of crosswind on car behavior. If a numerical approach is selected, then the methods developed attempt to reproduce the test procedure according to the ISO standard (ISO 12021:2010, 2010). One of the representative examples of such an analysis is the study undertaken by Forbes, Page, Passmore, and Gaylard (2016). The authors compared the results of a fully coupled closed loop system analysis with the simplified quasi-steady and unsteady one-way coupled method. As a test model, a fastback variant of the DrivAer model was used. The simplest method used in this case consisted of a set of steady-state solutions. The results were interpolated and input for the dynamic model was prepared. This method, however, was unable to predict overshoot and undershoot of the aerodynamic yawing moment nor the drag force present in the transient unsteady static coupling method. The difference in the results between the quasi-static and unsteady-static approaches is magnified for yaw angles greater than 30°. According to the above-mentioned ISO standard, the crosswind and car velocity should be 20.0 and 27.8 m/s (100 km/h), respectively. The Corresponding yaw angle is about 36°. The resultant velocity is 34.2 m/s and the Reynolds number is approximately 10 million (based on vehicle length). The car dynamic model was created in Matlab/Simulink based on an approach proposed by Best (2012). The CFD computational mesh contained 20.3 million hexahedral elements. Eight prism layers with a total thickness of 5 mm were used to model the boundary layer. This ensured $y^+ < 20$. To allow car movement an overset mesh was used. Delayed-detached eddy simulation (DDES) was selected for turbulence modeling. The grid was less dense than typically used for this turbulence model, however, the authors were focused on a large flow structure which could affect car behavior and precise drag prediction was not a main goal. The results obtained show that including the vehicle position correction in fully coupled analysis compared to the unsteady-static approach has limited impact on car behavior. Most important is capturing the transient effects associated with entering and exiting the crosswind zone. This conclusion might change if the considered car inertia were smaller (such as in the case of the car study in this study). It is worth mentioning that even this sophisticated analysis does not utilize the bi-directional coupling approach in the situation when the configuration (i.e. active aerodynamic surfaces) and velocity change at the same time. The co-simulation approach presented in this study goes one step further and removes almost all limitations present in the current state-of-the-art analyses. This makes it possible to take into account the impact of flow unsteadiness on vehicle dynamics in almost any kind of situation. The main advantage over current approaches is the possibility...
of studying a vehicle with changing velocity and configuration in different weather and road conditions. Such an experiment would be very difficult to reproduce in a wind tunnel or in real traffic. The presented methodology was validated against wind tunnel data and a full-scale car experiment. The aerodynamic coefficients obtained for the simplified car body and the braking characteristics obtained for the full car analysis show good agreement with the experimental data.

2. The co-simulation approach

The sophisticated bi-directional co-simulation approach requires the development of a unique system. Complex physics-based analyses, data management, solvers synchronization, etc., need to be taken into account. To ensure high fidelity results, the created tools ecosystem contains high-end software available on the market as presented in Table 1. In all of the analyzed scenarios, the Mach number is below 0.15 and the assumption regarding flow incompressibility is valid. To determine the flow field, ANSYS FLUENT solves the governing integral equations for the conservation of mass, momentum and a scalar, i.e. turbulence, using the control-volume-based technique.

MSC.Adams is a general-purpose multibody system (MBS) analysis program and its Car module is used in vehicle dynamics. A typical model contains multiple rigid body parts joined with some kind of compliance matrix (Blundell & Harty, 2015). For a dynamic analysis, a full definition of all components is required. Data such as mass, center of mass position and the mass moments of inertia need to be provided. Matlab/Simulink is widely used in academic research and in the industry environment for modeling, simulating and analyzing multidomain dynamical systems. A single Simulink block is a model element that defines a mathematical relationship between its input and output. The connection with Adams is made via ADAMS Plant block, which is an encapsulated dynamic model of the mechanism.

Proposed co-simulation analysis requires a client-server connection between Fluent and Matlab. This is made via the component object model (COM) which is a platform independent, distributed, object-oriented system for creating binary software components that can interact. In such cases Fluent acts as a COM server and Matlab is a client.

Matlab is also responsible for data exchange between the solvers: Fluent and MSC.Adams. This is performed through a Simulink model with S-Functions for data management and control over Fluent’s iterations. Figure 1 shows the data flow process. Analyses for both physics (fluid flow and structure motion) are conducted simultaneously with the same, constant time step. Data are exchanged at every iteration. However, this is not a limitation of the presented methodology. Different time steps for flow and dynamic analyses can be used if necessary.

3. Methodology validation

For such a sophisticated analysis it is crucial to validate the methodology because even a small error in one element of the process can lead to a huge error in the final results.

3.1. CFD-system dynamics coupling

This paragraph describes the analysis whose aim was to verify and validate the coupling between fluid flow and the calculation of the system dynamics. An experiment with a square beam in crosswind being a simplified version of the Tacoma Narrows bridge, was designed and conducted in the low-speed Wawelberg wind tunnel at the Division of Aerodynamics, Warsaw University of Technology. The test stand set-up is presented in Figure 2. The beam was mounted with flexible links in the working space. To ensure proper tension, the links were fixed at one end and system with springs, pulleys and masses was constructed at the other end. It was expected that, in certain conditions, vortex-induced
vibrations (transverse) would occur. During the experiment, the inlet velocity was slowly increased up to the point where beam oscillation reached maximum amplitude. Resonance occurred at a velocity of 4 m/s. Taking into account the beam section edge length of 50 mm, the corresponding Reynolds number was \( Re = 13,235 \). Vibrations were recorded with a camera speed of 120 fps. Movement was extracted with motion capture software Kinovea (Charmant, 2016). The frequency obtained and average peak-to-peak amplitude were 2.53 Hz and 0.158 m, respectively. The results of the experiment are in line with the theory. Based on the Strouhal number:

\[
St = \frac{fl}{U}
\]  

where:

\( St \) = Strouhal number  
\( f \) = frequency  
\( L \) = characteristic dimension  
\( U \) = flow velocity

If the experimental data is substituted into Equation (1), the corresponding Strouhal number will be 0.2 which is in agreement with the values found in the literature for this Reynolds number. To simulate beam behavior dedicated models were built for flow (half beam) and system dynamics analyses (full beam). Similar to the CFD test case, a polyhedral mesh was utilized to reduce computational time. Turbulence was modeled with \( k - \omega \) \( SST \). Second order schemes were used for spatial discretization for all quantities and the Least Squares Cell-Based algorithm was used for gradient calculation. As an inlet boundary condition, the velocity inlet was used and the bulk velocity was 4 m/s. The intensity and turbulence length scale were 4% and 50 mm, respectively. These values correspond to the measured value for the wind tunnel used. The pressure outlet boundary condition was used at the flow exit plane. On the remaining outer surfaces, a symmetry boundary condition was applied. The boundary layer parameters correspond to

| Parameter                  | Value | Units |
|----------------------------|-------|-------|
| Section edge length        | 50    | mm    |
| Beam length                | 540   | mm    |
| Beam mass                  | 0.044 | kg    |
| Dead weight                | 0.300 | kg    |
| Springs stiffness          | 0.085 | N/mm  |
| Springs damping            | 0.006 | N s/mm|
Due to the sharp edges of a beam, the separation location is well defined and a lower $y^+$ value is not required in this particular case to correctly predict the transverse force. The dynamic model was fed with measured test stand parameters: dimensions, masses, spring stiffness and spring damping as presented in Table 2.

The numerical results from the co-simulation presented in Figure 3 show a good agreement with the test results. Both frequency and amplitude were captured.
with good accuracy. It can be observed that the forcing function has two dominant frequencies. The first one corresponds to the system response. A complex flow structure can be observed in Figure 4. One can distinguish two zones where vortices are generated. The first zone is just behind the beam, and the second zone is located at the beam ends. The existence of those two vortices can explain multiple harmonics in the forcing function. The results obtained from the performed comparisons show a good agreement between the tests and the numerical prediction. Based on that, it can be concluded that the methodology works as intended. The assumptions regarding mesh construction and coupling mechanism were confirmed.

4. Car braking analysis

4.1. Analyzed car motion scenarios

One of the most popular examples of an active aerodynamic device mounted on a car is a retractable rear wing. It was assumed that a small sports car equipped with such a movable wing is for emergency braking on a road. Two scenarios were analyzed. In the first scenario, the rear airfoil was stationary. In the second scenario, after the ’start braking’ decision the driver fully pressed the brake pedal activating the rotation of the rear wing to its aerodynamic braking configuration. The combination of friction, inertia and aerodynamic forces acting on the car body generates its complicated motion. The flow around the variable position in relation to the road and the variable car body geometry generates aerodynamic forces influencing the car dynamics. Two different physical processes, flow around the car body and body motion had to be simulated simultaneously. Analysis parameters such as the brake demand characteristic, the friction coefficient, etc., were selected based on the data obtained from the real car experiment for the first scenario. The same data were used for the moving airfoil scenario. Figure 5 presents two airfoil configurations: stationary and in the final braking position.

4.2. CFD modeling

During braking, the car velocity decreases. This introduces new challenges to the analysis. The classic approach with a stationary car and inflowing air (Kurec, Remer, Mayer, Tudruj, & Piechna, 2019) could not be used in this case. Due to finite computational domain size and the assumption regarding air incompressibility, pressure in the domain will drop due to air inertia. In this situation, the reference frame had to be changed. This assumption leads to the conclusion that during analysis the car needs to be on the move. To execute this, a few options are available. The first option, with moving reference frame (MRF), cannot be used with the selected set of tools because Fluent (version 18.2) does not yet support the combination of this technique with overset mesh technology, which is also utilized. The second option is to create a computational domain which will cover the whole braking distance and move only the car body. This solution has one main drawback – the computational domain size. Its length should be larger than the braking distance, which is about 110 m for the assumed initial velocity and road parameters. The Element count, needed to ensure proper CFD modeling in such a scenario, is much larger than possible. The third option overcame those issues. The domain is the same as for the steady analysis and all its cells are moved with time. Such an approach keeps the element count at a reasonable level. The same instantaneous, longitudinal velocity is assigned to all computational subdomains. Additionally, the subdomains for the car elements which can have additional degrees of freedom (vertical and pitch) have also corresponding velocity components. The main computational domain dimensions and applied boundary conditions are shown in Figure 6. The air in the computational domain was assumed to be stationary and to execute such a behavior, the velocity inlet boundary condition was set to zero during the whole analysis. As a consequence, the ground velocity was also set to zero. The centers of the wheel rotation were defined as contact points between the wheels and the ground (locations of

Figure 5. Considered airfoil configurations: stationary and in the final braking position.
Figure 6. Half car CFD computational domain with main dimensions and boundary conditions. Dimensions given in millimeters.

Figure 7. Velocity components on the wheel: (a) axial, (b) normal to the road, (c) velocity magnitude.

developments. These points were changing with time). Figure 7 presents how the velocity distribution appears on the wheel just before braking. It is clearly visible that the velocity magnitude is equal to zero at wheel–road contact point. To allow movement of the car body and active aerodynamic surfaces, the overset mesh technique (also known as ‘Chimera’ or ‘overlapping’ mesh) was utilized instead of a time-consuming remeshing process. The other drawback of remeshing is a constraint regarding the time step. It should be small enough to avoid negative cell volume creation. The overset grid technique simplifies the process of mesh creation for complex geometry. This helps to avoid remeshing failures and set-up issues as in the dynamic mesh and maintains grid quality during mesh motion. It ensures easy configuration changes and part swapping (Ramakrishnan & Scheidegger, 2016). Turbulence was modeled with the $k – \omega$ SST model. The $y^+$ values were about 30 everywhere (Figure 8). Second order schemes were used for spatial discretization for all quantities and the Least Squares Cell-Based algorithm was used for gradient calculation.

Based on research by Blocken (2014) even if LES could have better results fidelity compared with Unsteady RANS (URANS) it also has some drawbacks which are not easy to eliminate. Just to name two: computational requirements and appropriate inlet boundary conditions.

The computational domain was split into background a zone and component zones with overlapping boundaries. The placement of the overset interfaces is presented in Figure 9. According to the Ansys documentation and the experience of the authors to avoid the creation of orphan cells, a grid should be created in such a way that at least four elements should exist between the walls which belong to different subdomains. This requirement was met and orphan cells were not created during the analysis. In the analysis, flow symmetry was assumed and a half car model was used. Symmetry boundary conditions were also applied on all remaining exterior walls: on the top and on the side. The background domain was built as a structural, hexahedral mesh. To speed up calculations, tetrahedral grids for the car components were converted to polyhedral. The total reduction factor in the element count was 1.65 (2.14 for tetrahedral domains which were converted). The mesh size statistics for each subdomain are presented in Table 3. To allow communication with external tools, the ‘ Fluent as a server’ launch option was used (ANSYS, Inc., 2017b). This selection enables remote connection to and control of Fluent sessions from an external application, in this case Matlab,
which is crucial for a coupled analysis. Data exchange and control over positions and velocities of car elements were realized via Fluent User-Defined-Functions (UDF) (ANSYS, Inc., 2017a) written for parallel calculation.

### 4.3. CFD grid convergence

Before performing final analysis, a study regarding grid convergence was performed. To determine sufficient grid resolution four steady-state analyses with different mesh sizes were conducted. The range of grid size was from 6.6 million to 14.7 million cells. The background grid was the same for all cases and meshes around the car body and airfoil were modified. In all cases, tetrahedral grids were converted to polyhedral ones and the overset mesh technique was utilized. As a criterion of mesh convergence drag force was used. Figure 10 shows how the calculated force changes with cell count. The flattening of characteristics which starts at a mesh size equal to 7.7 million can be observed. A few factors were taken into account during the mesh size selection process:

- Results fidelity
- Analysis time
- Available computational resources
- Model simplifications which affect the calculated forces (e.g. no internal flows, simple wheels, etc.)

Table 3. Cell count statistics before and after conversion to polyhedral elements.

| No. | Domain            | Before conversion (M) | After conversion (M) | Ratio |
|-----|-------------------|------------------------|----------------------|-------|
| 1   | Background        | 3.34                   | 3.34                 | 1.00  |
| 2   | Car               | 6.04                   | 2.98                 | 2.03  |
| 3   | Rear airfoil      | 2.02                   | 1.07                 | 1.89  |
| 4   | Front wheel       | 0.94                   | 0.21                 | 4.48  |
| 5   | Rear wheel        | 0.40                   | 0.11                 | 3.64  |
| Total|                   | 12.74                  | 7.71                 | 1.65  |
Based on the above criteria, it was decided to select a 7.7 million cell mesh. This appears to be a good compromise between results fidelity and time required to complete an analysis. Additionally, recent data have been compared with data obtained by Kurec et al. (2019) during tests on a 1:2.5 model of a Honda CRX del Sol performed in a wind tunnel.

### 4.4. Dynamic model

The full car dynamic model presented in Figure 11 was created in the MSC.Adams/Car software. The tool utilized multibody dynamic analysis. A detailed description of this approach to vehicle dynamics can be found in the book by Blundell and Harty (2015). The software makes it possible to include nonlinearities in the calculation such as the contact between tire and road (separation allowed) or nonlinear characteristics of springs and dampers. The equations of motion are generated automatically and all defined constraints and joints are taken into account. One classic publication by Genta (1997) describes the derivation of the equation of motion for car dynamics in detail. The required model parameters such as mass, moment of inertia, location of center of gravity (CG) were estimated based on the study by Heydinger et al. (1999). The assumed mass and moment of inertia are presented in Table 4. The spring rates for suspension were 61 and 44 N/mm for the front and rear, respectively. These values are typical for this kind of car and correspond to products available on the market. Instead of standard dampers, customized and adjustable magneto-rheological dampers were used. It was decided to use different damping settings for the front and rear.

| Parameter                        | Value | Units |
|----------------------------------|-------|-------|
| Sprung mass                      | 1300  | kg    |
| Pitching moment of inertia       | 1200  | kg m² |

**Table 4.** Mass properties of the car with two passengers used during the analysis.
Figure 12. Schematic car model and main dimensions with damper and spring characteristics used during analysis.

for the experiment. Figure 12 shows both characteristics which were measured in the laboratory before installation on the car. The same figure presents spring characteristics and CG location relative to the wheel axes. A sensitivity study for the coefficient of friction between the tire and road is out of the scope for this paper and only one value was used, equal to 0.71. This value was selected based on the deceleration data obtained from the experiment. It is also in line with the value reported by Jones and Childers (2001), namely that the coefficient of friction for dry road is about 0.7. The tire model and the tire–road contact mechanism has been developed by the MSC.Software according to the book, Tire and Vehicle Dynamics by Pacejka (2002). The main tire parameters are presented in Table 5. The full vehicle dynamic analysis for the braking maneuver was set up and the input–output parameters were determined. As input data, the aerodynamic forces and moment were used. The output information can be divided into two groups. The first group contains data which are then passed to the CFD solver: the linear and angular velocity for different car elements. The second group, covers additional parameters which help to understand the impact of unsteady aerodynamics on the car behavior: forces on the wheels, center of gravity displacements, etc. The model was exported as an Adams plant using the Adams/Controls module to allow information exchange with Matlab.

Table 5. Tire main parameters used during analysis.

| Parameter                        | Value | Units |
|----------------------------------|-------|-------|
| Free radius                      | 261   | mm    |
| Nominal section width of the tire| 165   | mm    |
| Nominal aspect ratio              | 0.65  | –     |
| Vertical stiffness                | 310   | N/mm  |
| Vertical damping                 | 3.1   | N s/mm|
| Lateral stiffness                | 190   | N/mm  |

4.5. Analyses coupling

Analyses for both physics (fluid flow and structure motion) were conducted simultaneously with the same, constant time step equal to 0.005 s. Data were exchanged at every iteration. However, this is not a limitation of the presented methodology. Different time steps for flow and dynamic analyses can be used if necessary. Because the straight-line maneuver is analyzed, two force components and one moment are required to solve the equations of motion. Fluent provides all of these components: drag, lift and pitch moment with respect to the actual CG location. MSC.Adams calculates the linear and angular velocities as well as the position of the centers of gravity of the car elements. This data was passed to Fluent to adjust the car configuration and the boundary conditions. Figure 13 shows a Simulink block diagram used for the analysis.
5. Full car experiment and analysis results

5.1. Analytical model set-up and initial conditions

In steady or quasi-steady conditions aerodynamic forces are proportional to the velocity squared. This leads to the conclusion that their effects are important at a relatively high velocity (McBeath & Toet, 2015). In that study, the initial velocity for braking was 40.31 m/s (145 km/h). A quasi-steady solution with constant velocity was used as a starting point for braking. Based on research by Mitschke (1977) the time required to reach full braking torque after making an emergency braking decision for 50% of the population is about 0.5 s. This value was confirmed in the experiment and used in the analysis. A linear change in braking torque was assumed. During fully developed braking, the wheels were at the limit of slipping. The aim of this approach was to simulate a simplified version of the ABS system. To determine the impact of aerodynamic effects, two comparative analyses were performed. In the first analysis, the rear airfoil was stationary. The second analysis included rear airfoil movement up to the final braking position which was 50° from the starting position. The airfoil angular velocity was predefined and constant with the value set to 5.82 rad/s. This corresponds to a rotation time equal to 0.15 s. Figure 14 presents the streamlines colored by the velocity magnitude at quasi-steady-state conditions just before braking.

5.2. Experiment set-up

The experiment was conducted at the Sochaczew-Bielice airport where a long straight sector with good and uniform road conditions was available (Figure 15). Figure 16 presents the test car. This is an instrumented car (GPS, accelerometers, airfoil position, etc.) equipped with additional aerodynamic features such as a front splitter and rear movable airfoils. The main purpose of
Figure 15. View of the Sochaczew-Bielice airport where a full-scale test was performed.

Figure 16. Test car with additional aerodynamic features: front splitter and movable rear airfoil, equipped with measurement and data acquisition system.

Figure 17. A comparison of braking characteristics between experiment and analysis results: acceleration, velocity and traveled distance.

The test was data collection for computational model validation. The maximum speed for this particular car with additional aerodynamic features and two passengers inside was 40.31 m/s and this value was used as a starting point for all analyses. The testing procedure was as follows: the car accelerated to the maximum velocity and when this velocity was stable, the driver pressed the braking pedal to achieve maximum braking performance. The data acquisition system collected information continuously throughout. The reference data for the numerical model are based on a braking maneuver with a stationary airfoil, which was in the ‘zero’ position.

5.3. Full car model validation results

To validate the analysis results, the braking characteristics were compared. Only the extreme position (‘zero’ angle) of the rear airfoil was used in this case. The available data contain acceleration, velocity and traveled distance. Figure 17 presents a comparison between the test and
Table 6. Results summary for Figure 17.

| Value          | Unit | Experiment | Analysis | Difference | %   |
|----------------|------|------------|----------|------------|-----|
| Traveled distance | m    | 111        | 112      | 1          | 0.90 |
| Time to stop    | s    | 5.34       | 5.29     | 0.05       | -0.94|
| Max deceleration | g    | 1.00       | 0.95     | -0.05      | -5.00|

6. Results and discussion

Validated and verified computational system comparative analyses for two scenarios as described in the previous section were performed. The aim of this study was to determine the impact of changing the aerodynamic configuration during braking on car dynamics and distance to stop. The results obtained from the transient coupled analyses are aerodynamic forces, CG velocity, CG location, pitch angle and the velocity of car elements (linear and angular). Additionally, the force split between the front axis and the rear axis is also available. Figure 18 presents a time series comparison for linear velocity and traveled distance.

Inclusion of the active aerodynamic surface reduces the distance to stop by 6% (6.7 m) and the braking time was shorter by 7.1% (0.37 s). Even though this was not a subject of this study, it is important to mention that these values depend highly on the coefficient of friction between tires and the road. Its high value reduces the aerodynamic impact whereas on a slippery road aerodynamic forces (especially drag) becomes more important. The data presented in Figure 19 show the airfoil movement impact on the generated forces. It can be observed that when the airfoil rotates the downforce on the car body increases by more than two times (114%). On the other hand, the downforce generated on the airfoil itself is higher only during movement and after reaching its final position it is at the same level as for the case with the stationary airfoil. To show how the car body is influenced by the rear wing in the different setups, the pressure

![Figure 18. Comparison of velocity and traveled distance vs. time characteristics for models with and without a rear airfoil.](image-url)
distribution along its surface is presented in Figure 20. To make the graphs more clear, only the rear part of the car body is presented. When the rear wing is at a high angle of attack, values of pressure coefficient on the car body change significantly. Apart from the rise of pressure on the top side of the car, it should be also noted that on the underside of the car pressure is decreased which also contributes to the generation of the downforce (see also Figure 21). This effect is in line with a 1:4 scale wind tunnel test (Katz, 2006) and resembles the interaction between the components of a multi-element wing (Katz & Largman, 1989). Those data clearly indicate that the downforce generated by a configuration with strong airfoil-car body interaction is much larger than the combined contribution of the isolated car body and the wing alone. The situation is different with drag force. The interaction between the elements results in low drag generated on the airfoil when it does not move. When it is in moving, the value obtained becomes comparable with the value generated on the car body. Additionally, drag on the car body also increases. Such behavior illustrates strong aerodynamic interaction between the car body and the airfoil.

A static pressure comparison for time points at the maximum downforce is presented in Figure 21 and this shows that airfoil movement changes the pressure distribution on the top and bottom side of the car body, but mainly on the rear part (the front part is almost unaffected). The pressure on the top is higher whereas on the bottom it is lower when the airfoil is rotated and this
**Figure 21.** Static pressure distribution comparison at maximum downforce conditions. Wheels omitted for clarity.

**Figure 22.** Selected dynamic characteristics impacted by different car configurations.

**Figure 23.** Force split factor between front and rear axes on both configurations.
explains the downforce increase. The maximum deceleration for the case with moving the airfoil is 6.9% higher when compared to the model with the stationary one (1.02 g vs. 0.95 g). The difference is caused by a two times higher aerodynamic drag and a 60% higher downforce generated when the airfoil changes position. In the analyzed cases, the maximum braking force caused by the aerodynamics (combined effect of drag and downforce) is smaller than the braking force coming from the car weight. Values of 21.3% and 9.6% represent the fraction for the cases with the movable airfoil and the stationary airfoil, respectively. The results summary for the discussed characteristics is presented in Table 7. The car body pitch angle, the generated forces and the deceleration are coupled. All of these values impact each other. A higher deceleration will result in a larger angle, which will affect the forces, and the forces will affect the deceleration, etc. This phenomenon illustrates how important is the coupled analysis of flow and car dynamics, especially when dealing with highly nonlinear effects. Additionally, the pitch velocity is correlated with the airfoil movement (or rather with the airfoil lift force) which can be observed in Figure 22. The pitch velocity peak at the beginning of braking leads to a higher pitch angle during the whole braking process. In both cases, the maximum pitch angle is below 2°. The difference in values is 0.13° (7.1%). The pitch angle also controls the

Table 7. Results summary for comparative analysis: stationary vs. moving airfoil.

| Quantity             | Unit | Stationary | Moving | Delta | %     |
|----------------------|------|------------|--------|-------|-------|
| Traveled distance    | m    | 112        | 105    | −7    | −6.3  |
| Time to stop         | s    | 5.29       | 4.92   | −0.37 | −7.0  |
| Max deceleration     | g    | 0.95       | 1.02   | 0.07  | 7.4   |
| Max drag             | N    | 545        | 1100   | 555   | 102   |
| Max downforce        | N    | 728        | 1165   | 437   | 69    |
| Max pitch velocity   | rad/s| 0.0846     | 0.0827 | −0.0019 | −2.3 |
| Max pitch angle      | deg  | 1.82       | 1.95   | 0.13  | 7.1   |

Figure 24. Static pressure comparison on an adjusted scale to show airfoil movement impact on the rear part of the car body.

Figure 25. Streamlines colored by velocity magnitude at max downforce condition.
clearance between the car and the road, which is an additional parameter that impacts the forces generated on the car body. On the same plot, deceleration characteristics are presented. Peaks in the values around 0.6 s can be observed. This is a result of simplification in ABS modeling. Such behavior is known from the theory for vehicles without an ABS system. The peaks are present because the maximum deceleration is reached before locking the wheels of the vehicle (Kudarauskas, 2007). With locked wheels, braking is less efficient and, for this reason, deceleration decreases after reaching this point. The car body pitch angle is also correlated with the force split between the front and rear wheel axes. Just before braking, the force split between the axes is equal. After pressing the brake pedal, the front axis becomes more loaded mainly due to the inertia effect (Figure 23). However, the difference between the characteristics indicates that aerodynamic forces also contribute to the force split. Loads generated on the airfoil during rotation cause faster front axis loading. At peak, the ratio between the front and rear axis loads is 1.93 with a moving airfoil and 1.83 when the airfoil does not change position. After a stabilization period, these values are almost constant and equal to 1.61 and 1.55, respectively. At this point, after the reviewing dynamic characteristics, it can be stated that, in this particular analysis, the transient effects are important during the first 0.9 s. After this period, when the rear airfoil is in the final position and oscillations have vanished, the system starts to behave nearly linearly. The additional downforce is generated not on the airfoil itself, but due to the strong aerodynamic interaction, on the rear part of the car body. Because a large flat area is affected, the difference in generated force is significant. The detailed pressure comparison for time points corresponding to the maximum downforce on an adjusted scale, as presented in Figure 24, confirms this conclusion. The flow structure around the car geometry is highly three dimensional. In the analyzed scenarios this is even further complicated due to the presence of additional aerodynamic features. Strong separation regions on the rear part of the car body and on the airfoil visible in Figure 25 confirm the complex flow nature. Based on that, it can be concluded that the generated forces cannot be accurately predicted by scaling quasi-steady results. The change in the reference frame (moving observer) opens new options regarding flow field visualization. How flow structures appear in a real situation on the road can be checked. The streamlines in this reference frame are presented in Figure 26. Information obtained in this nonstandard way might help in understanding the interaction between the car elements. A detailed explanation of the different reference frame after-effects can be found in Hucho, Hannemann, Schramm, and Williamson (2007).

7. Conclusions

The main advantages of the proposed methodology over existing solutions are flexibility and robustness. The methodology can be applied to almost any kind of structure without modification. The performed validations against the experimental data confirm that the methodology is suitable to capture a strong interaction between fluids and structures. The proposed methodology works as intended and the dynamic behavior of the vehicle with unsteady aerodynamic forces taken into account was accurately captured. As a general observation from the analysis, it can be said that, for the analyzed scenarios, transient aerodynamic effects are important up to 0.9 s of the analysis which is about 18% of the whole braking time. Beyond this point, the system behavior is nearly linear. The interaction between the airfoil and the car body cannot be captured by a superposition of isolated elements due to the strong interaction between them. In the considered cases, additional aerodynamic forces generated due to airfoil movement can reduce the distance to stop by 6%. Due to the existence of high aerodynamic forces during the initial phases of the braking process,
most important is the development of fast-acting aerodynamic elements and its rapid activation after the decision of braking. Active aerodynamic elements should be combined with safety systems typically used to control the operation of mechanical brakes. Aerodynamic braking is effective in the initial phase but has a strong effect on the final stopping distance achieved much later. The use of aerodynamic elements requires changes in the habits of drivers and their mentality. Braking action requires prevision and early decisions. In the area of autonomous car development, such requirements can be effectively included in the control software. Based on the results presented in this study and the reference documents, it can be concluded that the modern CFD software implementation of near wall treatment is quite robust and ensuring that $y^+ \approx 1$ in some situations is not required to correctly capture the physics of fluid flow. The next step in this research is the analysis of cornering. This will require a full-size car model and all six degrees of freedom will be taken into account.

**Disclosure statement**

No potential conflict of interest was reported by the authors.

**Funding**

This project was funded by the National Center for Research and Development (Narodowe Centrum Badań i Rozwoju), grant number PBS3/B6/34/2015, ‘Aktywny system tłumienia drgań pojazdu’.

**ORCID**

Jakub Broniszewski DOI: http://orcid.org/0000-0002-0914-759X

Janusz Piechna DOI: http://orcid.org/0000-0003-4267-3994

**References**

ANSYS, Inc. (2017a). ANSYS fluent customization manual (18.2) [Computer software manual].

ANSYS, Inc. (2017b). ANSYS fluent as a server user’s guide (18.2) [Computer software manual].

Best, M. C. (2012). *A simple realistic driver model*. Presented at: AVEC ‘12: The 11th international symposium on advanced vehicle control, 9th–12th September 2012, Seoul, Korea.

Blocken, B. (2014). 50 years of computational wind engineering: Past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, 129, 69–102. doi:10.1016/j.jweia.2014.03.008

Blundell, M., & Harty, D. (2015). *The multibody systems approach to vehicle dynamics* (2nd ed.). Oxford: Butterworth-Heinemann.

Charmant, J. (2016). Kinovea (0.8.25) [Computer program]. Retrieved from http://www.kinovea.org

Cheng, S., Tsubokura M., Okada Y., Nourzawa T., Nakashima T., & Doh D. (2013). Aerodynamic stability of road vehicles in dynamic pitching motion. *Journal of Wind Engineering and Industrial Aerodynamics*, 122, 146–156. (The Seventh International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7)) doi:10.1016/j.jweia.2013.06.010

Forbes, D. C., Page G. J., Passmore M. A., & Gaylard A. P. (2016, April). A fully coupled, 6 degree-of-freedom, aerodynamic and vehicle handling crosswind simulation using the DrivAer model. *SAE International Journal of Passenger Cars – Mechanical Systems*, 9, 710–722. doi:10.4271/2016-01-1601

Fu, C., Uddin M., & Robinson A. C. (2018). Turbulence modeling effects on the CFD predictions of flow over a NASCAR Gen 6 racecar. *Journal of Wind Engineering and Industrial Aerodynamics*, 176, 98–111. doi:10.1016/j.jweia.2018.03.016

Genta, G. (1997). *Motor vehicle dynamics: Modeling and simulation* (1st ed.). Singapore: World Scientific.

Heydinger, G. J., Bixel R. A., Garrott W. R., Pyne M., Howe J. G., & Guenther D. A. (1999). Measured vehicle inertial parameters – NHTSA’s data through November 1998. *SAE technical paper*. SAE International. doi:10.4271/1999-01-1336

Hobeika, T., & Sebben S. (2018). CFD investigation on wheel rotation modelling. *Journal of Wind Engineering and Industrial Aerodynamics*, 174, 241–251. doi:10.1016/j.jweia.2018.01.005

Hucho, W. H., Hannemann K., Schramm J. M., & Williamson C. (2007). Aerodynamics. In C. Tropea, A. L. Yarin, & J. F. Foss (Eds.), *Springer handbook of experimental fluid mechanics* (pp. 1043–1155). Berlin, Heidelberg: Springer. doi:10.1007/978-3-540-30299-5_16

Jones, E. R., & Childers, R. L. (2001). *The friction of automobile tires*. Contemporary college physics (3rd ed.). Boston: McGraw Hill.

Katz, J. (2006). Aerodynamics of race cars. *Annual Review of Fluid Mechanics*, 38(1), 27–63. doi:10.1146/annurev.flud.38.050304.092016

Katz, J., & Largman R. (1989, June). Experimental study of the aerodynamic interaction between an enclosed-wheel racing-car and its rear wing. *Journal of Fluids Engineering*, 111(2). doi:10.1115/1.3243616

Kudarauskas, N. (2007). Analysis of emergency braking of a vehicle. *Transport*, 22(3), 154–159. doi:10.1080/16484142.2007.9638118

Kurec, K., Remer M., Mayer T., Tudruj S., & Piechna J. (2019). Flow control for a car-mounted rear wing. *International Journal of Mechanical Sciences*, 152, 384–399. doi:10.1016/j.ijmecsci.2018.12.034

Liu, T., Chen Z., Zhou X., & Zhang J. (2018). A CFD analysis of the aerodynamics of a high-speed train passing through a windbreak transition under crosswind. *Engineering Applications of Computational Fluid Mechanics*, 12(1), 137–151. doi:10.1080/19942060.2017.1360211

McBeath, S., & Toet, W. (2015). *Competition car aerodynamics. A practical handbook* (3rd ed.). Dorchester: Veloce Publishing.

Mitschke, M. (1977). *Dynamika samochodu [Car dynamics]* (1st ed.). Warsaw: Wydawnictwa Komunikacji i Łączności.

Morden, J. A., Hemida H., & Baker C. J. (2015). Comparison of RANS and detached eddy simulation results to
wind-tunnel data for the surface pressures upon a class 43 high-speed train. *Journal of Fluids Engineering, 137*(4), 041108. doi:10.1115/1.4029261

Pacejka, H. (2002). *Tyre and vehicle dynamics*. Oxford: Butterworth Heinemann.

Ramakrishnan, S., & Scheidegger T. (2016). *Overset meshing in ANSYS Fluent*. Presented at: 13th Symposium on overset composite grids and solution technology, 17th–20th October 2016, Mukilteo, WA, USA.

ISO 12021:2010 (2010). *Road vehicles – sensitivity to lateral wind – open-loop test method using wind generator input (standard)*. International Organization for Standardization, December 2010.

Wickern, G., Zwicker K., & Pfadenhauer M. (1997, February). Rotating wheels – their impact on wind tunnel test techniques and on vehicle drag results. *SAE technical paper 970133*. SAE International. doi:10.4271/970133