

Aerodynamic CFD Based Optimization of Police Car Using Bezier Curves

Ali Reza Taherkhani and Carl Gilkeson PhD
University of Leeds

Philip Gaskell PhD
Durham University

Rob Hewson PhD
Imperial College

Vassili Toropov PhD and Amin Rezaienia PhD
Queen Mary University of London

Harvey Thompson
University of Leeds

ABSTRACT

This paper investigates the optimization of the aerodynamic design of a police car, BMW 5-series which is popular police force across the UK. A Bezier curve fitting approach is proposed as a tool to improve the existing design of the warning light cluster in order to reduce drag. A formal optimization technique based on Computational Fluid Dynamics (CFD) and moving least squares (MLS) is used to determine the control points for the approximated curve to cover the light-bar and streamline the shape of the roof. The results clearly show that improving the aerodynamic design of the roofs will offer an important opportunity for reducing the fuel consumption and emissions for police vehicles. The optimized police car has 30% less drag than the non-optimized counter-part.

CITATION: Taherkhani, A., Gilkeson PhD, C., Gaskell PhD, P., Hewson PhD, R. et al., "Aerodynamic CFD Based Optimization of Police Car Using Bezier Curves," *SAE Int. J. Mater. Manf.* 10(2):2017, doi:10.4271/2017-01-9450.

INTRODUCTION

Over the past several decades, a significant amount of effort has been invested to reduce the aerodynamic drag of heavy goods vehicles since this directly influences the fuel consumption and dynamic performance, a reduction in total aerodynamic drag leads into a significant improvement in the fuel economy. In recent years the number of studies which have applied Computational Fluid Dynamics (CFD) and optimization methods to improve the aerodynamic design of road vehicles has increased significantly and is recognized as providing a powerful and efficient means of generating improved aerodynamic performance, particularly when detailed experimentation in wind tunnels is not achievable, see e.g. [1, 2, 3, 4, 5, 6]. The focus of recent studies has been on the drag reduction of Heavy Goods Vehicles, see e.g. [1, 4, 5, 6, 7]. Currently there have been relatively few studies on the aerodynamic improvement of light trucks and cars [8, 9], while a small number still have reported reduced aerodynamic drag using automated design optimization techniques [2, 8, 10].

The present work focuses on the aerodynamic design optimization of a modified BMW 5-series police car, such as those commonly by our police sources and whose aerodynamic optimization has not been considered previously in the literature. It complements and extends a recent study into aerodynamic optimization of ambulances [10, 11].

One of the most important ingredients in numerical optimization is the choice of design variables and the parameterization methodology employed [12, 13]. In order to reduce the number of parameters needed to describe the shape of the roof, several mathematical shape-capturing methods have been proposed in the literature [14]. In the present study, a third order Bezier curve is used to describe the geometry. The advantage of this choice is that, with only two design variables, a wide range of roof dimensions can be assessed [12]. Past methods often involve complex parameterization schemes which use arcs, circles and polynomials for geometry creation. Many of these methods require large data sets to describe the profiles. In addition, they do not provide simple, intuitive parameters for the designers to control [15]. Bezier curves are used to fit an existing data set while maintaining tangency and curvature conditions. Bezier curves have

since become a popular method for creating parametric curves and they have wide applications including turbine blade design [16], wind turbine airfoil design [12, 16], face recognition [17].

Computational Fluid Dynamics (CFD) is used within a multi-objective optimization framework to minimize the drag force on a police car, based on the BMW 5-series design. Design parameterization of a three-dimensional real car model is carried out to enable optimization for a wide range of roof configurations. Optimal Latin Hyper-cubes for surrogate model building and model validation points are constructed using a permutation genetic algorithm and design points are evaluated using CFD. Surrogate models are built using a Moving Least Squares approach. Optimizations of the roof configuration, was undertaken using a genetic algorithm with responses calculated from the surrogates. This approach results in a set of optimized designs, from which appropriate roof designs with lower drag can be obtained. The results presented show significant potential for aerodynamic drag and emissions reductions of police cars.

The choice of design variables in the geometry parameterization is one of the most important part of the optimization process [12]. Previous studies have often used complex parameterization schemes which use arcs, circles and polynomials for geometry creation [15]. Many of these methods require large data sets to describe the profiles. In addition, they do not provide simple, intuitive parameters for designers to control [15]. Therefore there is a need for a method flexible enough to fill the design space. Bezier curves are one such option since they can be controlled by any desired control points.

The equation of the n-order Bezier curve is:

$$p(t) = \sum_{i=0}^n B_i^n(t) \cdot P_i \quad (1)$$

where P_i represents the set of $n+1$ control points, t is a parameter that varies between 0 to 1, and $B_i^n(t)$ is the blending function:

$$B_i^n(t) = \binom{n}{i} \cdot (1-t)^{n-i} \cdot t^i, \quad i = 0, 1, \dots, n \quad (2)$$

In the blending function formula $\binom{n}{i}$ is the binomial coefficient given by $\frac{n!}{i!(n-i)!}$, where n defines the Bezier curve's degree of freedom. From figures 1 and equations 1 and 2, it can be seen that the blending functions act as a link between the control points and the points on the Bezier curve. In each case the first and last control points lie on the actual curve while the other control points form a control envelope for the curve. Often a single Bezier curve is insufficient for describing a complex profile and several of them are joined together. In our case only one Bezier curve with four control points is sufficient as it would give us wide range of roof dimensions in our design space.

Figure 1 shows Bezier curves with different number of control points:

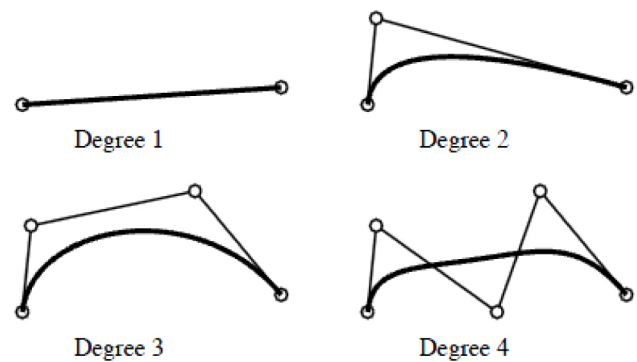


Figure 1. Bezier curves of various control point and degree of freedom

CFD METHODOLOGY

CFD Modeling

The geometry of the vehicle is shown in figure 2 which is based on a simplified model of the BMW 5-series used in the UK police fleet with the dimensions below.

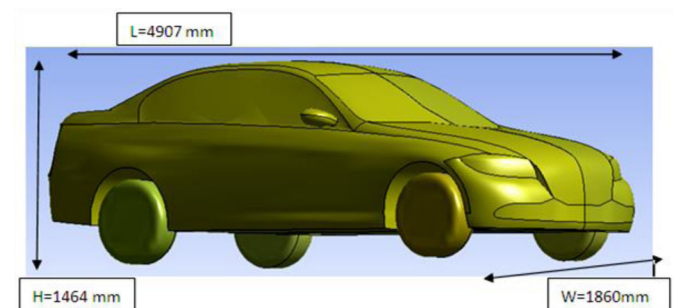


Figure 2. Simplified model of BMW 5series

Only symmetrical, zero yaw angle air flow cases are considered, enabling the flow domain to be reduced by half in order to reduce the computational costs. The semi-cylindrical domain has the following dimensions (See figure 3):

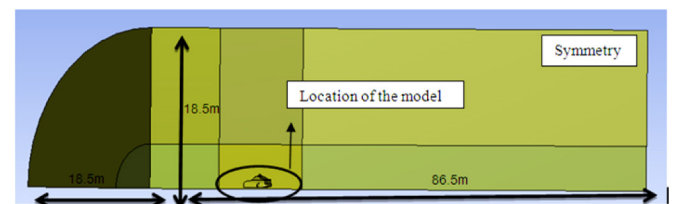


Figure 3. Wind tunnel domain with its 3D dimensions

These dimensions are large enough to minimize any flow blockage effects and to adequately capture turbulent flow behind the vehicle [11], the domain was decomposed into 6 zones to allow suitable mesh control. The volume immediately surrounding the vehicle was discretised by tetrahedral elements, in order to accurately define the complexity of the vehicle geometry, with mesh refinement increasing towards the vehicle surface. Other volumes were meshed using structured elements, see figure 4.

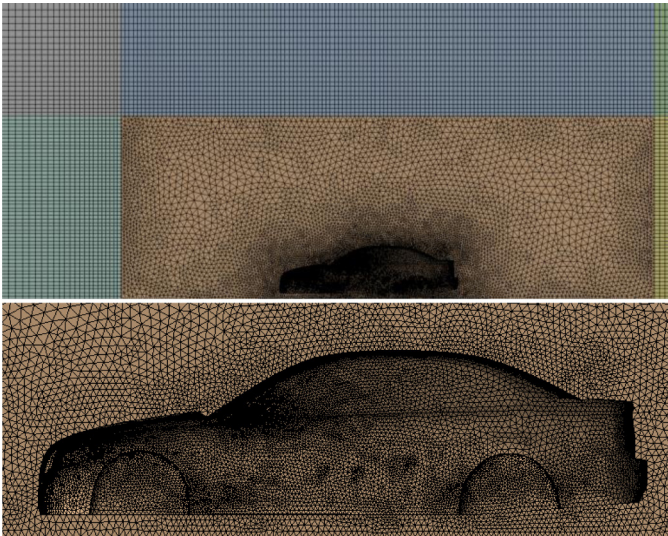


Figure 4. Illustration of the mesh type and size, wind tunnel (top, unstructured around the vehicle and structured further away from the vehicle to reduce the number of elements) and around the vehicle (bottom, smaller elements close to the surface of the vehicle and specially around small parts like mirrors)

In practice the mesh density used throughout the optimization is a compromise between the computation time and simulation accuracy. In this example the generated mesh contains around 9.7 million elements with a cell size of 10 millimetres on the vehicle surface, which correspond to a y^+ value of 75 which is within the range of 30-500 in order to reduce the affects of the turbulence boundary layer [18]. Such mesh densities have been shown to provide good agreement with corresponding wind tunnel data for emergency response vehicles [11].

Boundary Condition

At the inlet flow boundary, located upstream of the vehicle, a uniform stream-wise velocity of 26.8 m/s (60 mph) was prescribed, while at the outlet an atmospheric pressure condition was specified.

Table 1. Boundary conditions applied for the generic ambulance model

Boundary	Boundary type	Parameters
Inlet	Velocity Inlet	26.8 m/s
Outlet	Pressure Outlet	0 Pa
Road	Moving wall	26.8 m/s
Tyres	Rotating wall	70.5 rpm
Vehicle surface	Stationary wall	No slip
Domain top	Stationary wall	Zero shear stress

The same velocity was set for the moving ground. At the wheels, the velocity boundary condition was set to provide a rotation with the circumferential speed being equal to that of the moving ground. On the surface of the vehicle, a no-slip wall boundary condition was specified. A symmetry boundary condition was applied at the vertical edge of the flow domain. The boundary conditions are summarized in Table 1.

Mesh Independence

A mesh dependency study was performed to assess the effect of mesh size on CFD predictions for the police car. Different mesh sizes with two different turbulence models were assessed. As well as the mesh on the surface of the vehicle, the mesh for the whole wind tunnel was iteratively changed.

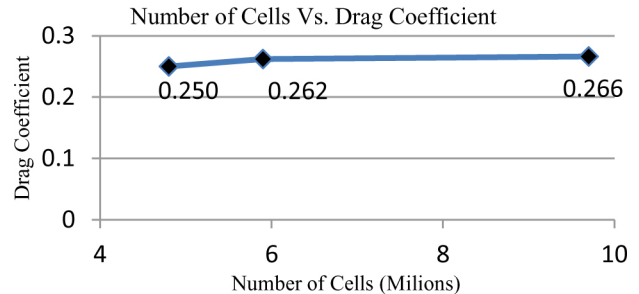


Figure 5. Graph of mesh dependency for Standard k - ε

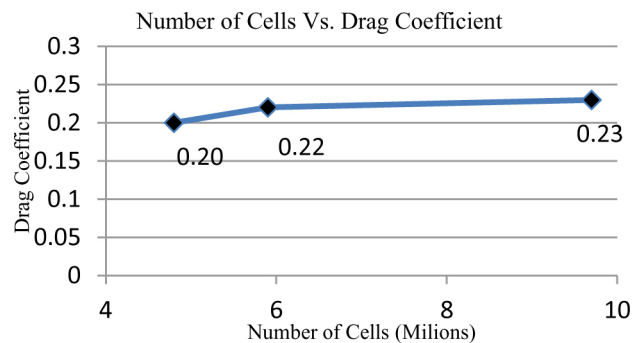


Figure 6. Graph of mesh dependency for Realizable k - ε

This study indicates the independency of the results from the mesh density and as the mesh density increases from 5 to 10 million elements, the drag coefficient ranges from 0.250 to 0.266 for *Standard k - ε* and 0.20 to 0.23 for *Realizable k - ε*. 5 million cells, which relates to cell size of 10mm on the surface of the vehicle (minimum cell size), were used to mesh the model based on these results.

Turbulence Model Validation

Several previous studies have shown that the family of two equation $k-\epsilon$ models can be used to successfully predict aerodynamic drag on vehicles with dimensions similar to ambulance geometries, see e.g. [8, 9, 19, 20]. The predictions from the two such models were compared with experimental data for a real model of BMW 5-series which has a drag coefficient of 0.27 [21]. The following turbulence models (see table 2) were used to simulate the drag coefficient and find the most suitable with the drag coefficient closest to experiment: Table 2 presents the results obtained from the simulations with the element size of 10 millimeters:

Table 2. Drag coefficients for different turbulent model

Turbulence Model	Drag Coefficient
$k - \epsilon$ standard	0.266
$k - \epsilon$ Realizable	0.230
Experimental	0.27