ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

MSc THESIS

Veli Kemal AYDIN

EFFECTS OF USAGE DIMPLED SURFACES ON AHMED BODY JENERIC MODEL FLOW CHARACTERISTICS

DEPARTMENT OF AUTOMOTIVE ENGINEERING

ADANA, 2022

ÇUKUROVA UNIVERSITY
FEN BİLİMLERİ ENSTİTÜSÜ

EFFECTS OF USAGE DIMPLED SURFACES ON AHMED BODY JENERIC MODEL FLOW CHARACTERISTICS

Veli Kemal AYDIN

MSc THESIS

DEPARTMENT OF AUTOMOTIVE ENGINEERING

We certify that the thesis titled above was reviewed and approved for the award of degree of the Master of Science by the board of jury on 13/10/2022

……………………………….	……………………………	……………………………
Assoc. Prof. Dr. M. Atakan AKAR	Assoc. Prof. Dr. Hasan SERİN	Assoc. Prof. Dr.İlker SUGÖZÜ
SUPERVISOR	MEMBER	MEMBER

This MSc Thesis is written at the Department of Institute of Natural and Applied Sciences of Çukurova University.
Registration Number:

Prof. Dr. Sadık DİNÇER
Director
Institute of Natural and Applied Sciences

This study supported by Çukurova University Scientific Research Projects Unit.
Project No: FYL-2021-13513

ABSTRACT

MSc THESIS

EFFECTS OF USAGE DIMPLED SURFACES ON AHMED BODY JENERIC MODEL FLOW CHARACTERISTICS

Veli Kemal AYDIN

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES
DEPARTMENT OF AUTOMOTIVE ENGINEERING

Supervisor : Assoc. Prof. Dr. M. Atakan AKAR
Year: 2022, Pages: 33
Jury : Assoc. Prof. Dr. M. Atakan AKAR
: Assoc. Prof. Dr. Hasan SERİN
: Assoc. Prof. Dr. İlker SUGÖZÜ

Vehicle aerodynamics which directly affect fuel consumption values in vehicles, has a great importance in terms of vehicle stability, acceleration ability, vehicle noise and environmental pollution. The Ahmed car model is one of the most important generic models so this popular car model is widely used in the literature on fluid mechanics and vehicle aerodynamics studies. The simple and easy applicable shape of the Ahmed body make the model convenient in these sort of investigations. This study investigates effect of a passive drag reduction technique which it is known as dimple surface, on Ahmed car model with 35° slant angle. In order to take distinctive result, rear wake zone was selected for design optimization area. The flow control methods were applied numerically also these methods were compared with similar works. The selected passive drag reduction method was used with different model configurations to determine optimum parameters and evaluate these parameters how will effect on the vehicle drag and lift coefficients. Analyses made in this study, were calculated with 2.4 million cell number and the maximum drag value reduction was obtained as 3.5 % by using dimple surfaces. Moreover, the applicability and manufacturability factor was also evaluated in terms of implementing the dimpling modification to today's current road vehicles and in order to make this study meaningful for vehicle aerodynamics.

Keywords: Drag; Lift; Aerodynamics; CFD; Ahmed body

ÖZ

YÜKSEK LİSANS TEZİ

AHMED CİSMİ JENERİK MODELİ ÜZERİNDE ÇUKUR YÜZEYLERİN KULLANIMININ AKIŞ KARASTERİĞİNE ETKİSİ

Veli Kemal AYDIN

ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ
OTOMOTİV MÜHENDİSLİĞİ ANABİLİM DALI

Danışman : Doç. Dr. M. Atakan AKAR
Yıl: 2022, Sayfa: 33
Jüri : Doç. Dr. M. Atakan AKAR
: Doç Dr. Hasan SERİN
: Doç. Dr. İlker SUGÖZÜ

Araç yakıt tüketim değerlerini doğrudan etkileyen araç aerodinamiği aynı zamanda araç stabilitesi, hızlanma yeteneği, araç sesi ve çevre kirliliği açısından büyük öneme sahiptir. Ahmed cismi jenerik araç modeli akışkanlar mekaniği ve araç aerodinamiği konusunda yapılan çalışmalarda çok fazla kullanılmaktadır. Çünkü Ahmed cismi günümüz araçlarına göre daha basit ve az detaya sahip olması yönüyle bu tarz çalışmalarda kolaylık sağlamaktadır. Bu çalışma, 35° arka pencere eğim açısına sahip olan model üzerine çukur yüzeylerin uygulanmasını ve bu yüzeylerin araç aerodinamiğine olan etkisini araştırmaktadır. Yapılan optimizasyonlarının daha efektif sonuç vermesi için çukurlar aracın arka kısmına uygulanmıştır. Akış control yöntemlerinin numerik olarak araştırıldığı bu çalışmada aynı zamanda literatür çalışmaları da referans olarak alınmış ve sonuçlar karşılaştırılmıştır. Yapılan çalışmada, çukurcuklamayla sürtünme azaltma tekniği farklı parametrelerle kullanılmış ve akış üzerine etkileri değerlendirilmiştir. Analizler 2.4 milyon mesh sayısı ile çözümlenmiş olup, çukurcuklu yüzeylerin kullanılması ile en iyi model de sürükleme kuvveti katsayısında 3.5% azalma elde edilmiştir. Aynı zamanda bu çalışmanın aerodinami açısından anlamlı olması için uygulanan bu çukur yüzeylerin günümüz araçlarına uygulanabilirliği ve üretilebilirlik faktörü de değerlendirilmiştir.

Keywords: Sürükleme kuvveti katsayısı; Kaldırma kuvveti katsayısı; Araç aerodinamiği; CFD; Ahmed cismi

EXTENDED ABSTRACT

Continuously increasing in fuel prices causes some problems about vehicle fuel economy and main focus points of the automotive industry to overcome this issue are improving engine efficiency, reducing vehicle weight, and minimizing drag force. For a sedan car at 100 km/h, approximately 20 % of total fuel consumption and for a heavy car 50 % of total fuel consumption is used to overcome drag forces.

Fossil fuel resources are reducing day by day with the growing population and increasing usage of

Reducing fossil fuel resources and increasing greenhouse effect day by day make more important lower fuel consumption values on automotive industry. Additionally, lower drag force coefficient is an essential requirement for controllability of vehicles at high velocity values.

That is why, this study aims to enhance vehicle aerodynamic and dependent factors like fuel consumption, greenhouse effect and vehicle controllability.

This study investigates a passive drag reduction technique using a car model which is named Ahmed body. Ahmed body is a generic car geometry generally used in the literature to observe fluid structure of ground vehicles. Its shape is very simple comparing with an actual car shape, so this model can be easily modeled in CFD investigations.

There are so many experimental and numerical investigations in the literature for enhancing aerodynamic performance with using Ahmed car geometry. Three major components of this body are front rounded surface, middle flat section, and rear part with slanted surface. Front surface has limited effect on wake zone and vortex creation by the rounded surface. However, variable slanted surface effect a lot of parameter about vehicle aerodynamic.

Non-smooth surfaces are very effective technique for reducing drag coefficient and make proper flow separation rear surface of Ahmed body. Dimpled surfaces is a good example for non-smooth surfaces and this surfaces on a golf ball which it provides longer distance travel if compared a non-dimpled surface ball are inspired from engineers. Because of longer distance travel is providing lower turbulent air on flow separation area with the aid of dimpled surfaces. Literature studies show aerodynamic efficiency of an airfoil can be improved with addition a single bump at CD 6% can be lower than a smooth air foil. In another literature study was investigated effect of dimpled surfaces on Ahmed body aerodynamic. These dimpled surfaces with different surface roughness were mounted on top surfaces of the geometry. According to study results 1.95 % drag reduction was obtained by using the dimple surfaces.

Simulation models were created with using CATIA 3d modelling software and dimpled surfaces mounted on slanted surface of Ahmed body with 35° slanted surface. In simulations dimpled surface dimension were selected to obtain different surface roughness value for each model. Dimple surfaces were categorized 3 diameter group which they are 10, 15, and 20 mm. Each model in groups has different depth value and this values increase from 1 to 5 mm.

To provide mesh refinement, the enclosure created by body of influence volumes with different poly-hex core mesh size. Watertight Geometry module ins Ansys Fluent was used for mesh structure creation. Mesh independence study was made by using 3 different setups also these setups occur from 1.1 (mesh 1), 2.4 (mesh 2), and 5.5 (mesh 3) million total cell number values. These study results indicated that mesh 2 setup is independent for this conditions.

K-ω SST turbulence model has important adventageous on adverse pressure gradient that is why all simulations in this study was solved these turbulence model. The air velocity has been set to 40 m/s with a Reynolds number of Re = $2{,}86\ x\ 10^6$ was based length of the Ahmed body.

According to results, using dimpled surfaces were reduced drag force coefficient.

Effects of dimpled surfaces shows similar results with reference studies when compared. Used dimpled reduce drag force coefficient by creating friction on pressure gradients. Maximum drag reduction was obtained as 3.5 % with 20d3h model with comparing base model. However, increasing dimple depth after a value causes an increment on values. In comparison wake flow structures, representations include the most optimized model and base model. When evaluated Q-criterion, TKE, vorticity and velocity magnitude contours it can be seen clearly with using dimple surfaces pressure gradients reduced by creating friction on flow boundary layer. Additionally, applied passive methods provide pressure and velocity recovery at the rear of the body.

GENİŞLETİLMİŞ ÖZET

Günümüzde akaryakıt fiyatlarının sürekli olarak artması otomotiv endüstrisinde temel problemlerin başında görülmektedir. Bu olumsuz duruma önlem almak adına yapılan çalışmalar motor verimliliği, araç ağırlığının azaltılması ve araç sürükleme katsayısının düşürülmesi konularını esas almaktadır. Öyle ki, 100 km/s hızda hareket eden bir binek aracın yakıt tüketiminin yaklaşık olarak 20 %'sini sürükleme kuvveti oluşturmaktadır. Bu değer ağır tonajlı bir araç için ise yaklaşık 50 % değerlerini bulabilmektedir. Başka bir konu olan fosil kökenli yakıtların sınırlı olması ise yüksek yakıt tüketim değerlerinden ve kullanılan araç sayısının artmasından olumsuz olarak etkilenmektedir. Bu değerler ile direk olarak bağlantılı olan araç düşük sürükleme katsayısı otomotiv tarihi incelendiğinde araç yol tutuş kabiliyetini artırmak adına da her zaman önemli olmuştur. Bu bilgiler ışığında bu çalışma araç aerodinamiğinde iyileşme sağlamak adına katkı değeri olan sonuçlar elde etmeyi amaçlamaktadır.

Sürükleme kuvveti katsayısının pasif yöntemler ile azaltılması konusunda bir araştırma yapan bu çalışma araç modeli olarak Ahmed cismini kullanmaktadır. Ahmed cismi genellikle literatür çalışmalarında kara taşıtlarının akış karakteristiklerini incelemek amacıyla kullanılmaktadır ve günümüz araçlarına kıyasla daha basit bir şekle sahip olduğu için kolay bir şekilde modellenip, optimize edilebilmektedir. Bu yüzden literatürde deneysel ve nümerik çalışmalarda sıklıkla yer almaktadır. Ahmed cismi yuvarlak bir ön yüzey, düz yan yüzeyler ve değişken arka pencere yüzeyinden oluşan 3 ana bölüme sahiptir. Ön ve yan yüzeyler geometrinin vorkteks yapısına olan etkisi neredeyse çok az iken, değişken açılı arka yüzey akış karakteristiğini önemli ölçüde etkilemektedir.

Pürüzlülük etkisini artıran yüzeyler araç sürükleme kuvveti katsayısının pasif olarak azaltıldığı etkili yöntemler arasındadır. Çukurcuklu yüzeyler pürüzlülük etkisi yüksek olan yöntemlere iyi bir örnek teşkil etmektedir. Çukurcuklu yüzeylerin golf topunun üzerinde kullanılması topun havada daha uzun mesafe yol almasını sağlamaktadır. Bunun temel nedeni pürüzlülük etkisi oluşturan yüzeylerin akış ayrım bölgesinde oluşan türbülans yoğunluğunu düşürmesidir.

Yapılan bir çalışmada kanat profili üzerine uygulanan tek bir kabarcıklı yüzeyin sürükleme kuvveti katsayısını 6 % düşürdüğü sonucuna ulaşılmıştır. Bir başka çalışma ise yüzey pürüzlülük etkisi farklı olan çukurcuklu yüzeyleri Ahmed cisminin üst yüzeyine oluşturmuştur. Elde edilen sonuçlar araç sürükleme kuvveti katsayısında 1.95 % azalma sağlandığı yönünde olmuştur.

Analizlerde kullanılan modeller CATIA 3 boyutlu modelleme proğramı ile oluşturulmuş olup, çukurcuklu yüzeyler Ahmed cisminin açılı arka yüzeyi üzerine yerleştirilmiştir. Her model de farklı yüzey pürüzlülük etkisi oluşturmak adına çukurcuklu yüzeylerin çapı 10, 15 ve 20 mm değerlerinde seçilmiştir. Ayrıca derinlik değerleri her çap grubunda bulunan modeller de farklı değerlere sahip olacak şekilde 1 mm'den 5 mm'ye kadar artırılmıştır. Akış alanında poly-hexcore ağ türünün araç duvarına yaklaştıkça artması sağlanmıştır. Ayrıca ağ yapısını oluşturmak için Ansys Fluent proğramının Watertight Geometri modülü kullanılmıştır. Yapılan analizler de kullanılan toplam hücre sayısı çözüm süresini doğrudan etkilemektedir. Bu yüzden bu çalışma için optimum hücre sayısının tespit edilmesi için ağ bağımsızlık çalışması yapılmıştır. Bu çalışma 3 farklı ağ grubundan oluşmaktadır ve bu gruplar 1.1 milyon, 2.4 milyon ve 5.5 milyon toplam hücre sayılarından oluşmaktadır. Çalışma sonucuna göre 2 numaralı ağ yapısı seçilmiş ve çalışma devamında diğer analizlerde kullanılmıştır.Ters basınç gradyanlarında sağladığı önemli avantajdan dolayı bu çalışmada K-ω SST türbülans modeli kullanılmıştır. Analizler 40 m/s olarak hava akış hızında yapılmış olup bu değer uzunluk olarak araç boyu esas alındığında $2{,}86\ x\ 10^6$ Reynolds sayısına eşdeğerdir.

Referans çalışmaları ile benzer olan analiz sonuçlarına göre çukurcuklu yüzeylerin kullanımı sürükleme kuvveti katsayısında azalma sağlamıştır. Bu yüzeylerin kullanımı ters basın gradyanlarında sürtünme etkisi yapmıştır. Maksimum sürükleme katsayısı azalması 3.5 % ile 20d3h modeliyle elde edilmiştir. Ancak çukurcuk derinliğinin artışı bir noktadan sonra bu değerlerin yükselmesine sebep olmuştur. Baz model ile 20d3h modelinin O-kriter, Türbülans Kinetik Enerji ve hız konturları karşılaştırıldığında açık bir şekilde görülmektedir ki çukurcuklu yüzeylerin kullanımım aracın arka kısmında hız ve basınç geri kazanımı sağlamıştır.

ACKNOWLEDGEMENTS

Firstly, and foremostly, I would like to express my sincere gratitude to my supervisor Assoc. Prof. Dr. Mustafa Atakan AKAR for his continuous support, patience, motivation, and wisdom. Thanks to his mentoring skills and guidance, I managed to complete the research and writing of this thesis. I am grateful to him not only for being a great advisor also being a lifetime mentor.

I also thank to Assoc. Prof. Dr. Hasan SERİN and Assoc. Prof. Dr. İlker SUGÖZÜ for evaluating my thesis.

I specially thank to Prof. Dr. Mustafa ÖZCANLI and Asst. Prof. Dr. Tayfun ÖZGÜR for their help and valuable advices throughout my study.

Besides my advisor, I am thankful to Research Assistant Oğuz BAŞ and Research Assistant Umut KUMLU for their helps throughout this thesis work. Their supports and advices have considerably contributed to my thesis.

I wish to thank all staff of Automotive Engineering and Head of Automotive Engineering Department, Prof. Dr. Ali KESKİN.

In addition, I would like to thank my parents and my brother for their confidence and support.

CONTENTS **PAGE**

LIST OF FIGURES **PAGE**

LIST OF TABLES **PAGE**

ABBREVIATIONS AND SYMBOLS

CFD	:	Computational Fluid Dynamic
k	:	Turbulence kinetic energy per unit mass (m2⁄s2)
ω	:	Turbulence eddy frequency (s^{-1})
ρ	:	Density (kg⁄m3)
Re	:	Reynolds Number
y+	:	Dimensionless wall coordinate
U	:	Local flow velocity (m/s)
U∞	:	Free stream velocity (m/s)
TKE	:	Turbulent kinetic energy (m2/s2)
EASM	:	Explicit Algebraic Stress Model
RANS	:	Reynolds-Averaged Navier-Stokes
LES	:	Large eddy simulation
URANS	:	Unsteady Reynolds-averaged Navier–Stokes

1. INTRODUCTION

Nowadays continuously increasing fuel prices and climate change effect have caused fuel efficiency to become an important issue on vehicle production industry. Also, usage of energy resources increasing significantly with the population growth and developing fuel consumption is being more important. Main objectices are developing fuel economy or fuel consumption values depend on three essential parameters which they are engine performance [1], car weight and coefficient of drag to enhance fuel economy (Abd-Alla GH et al., 2002, Jacques EVP and Richard PD et al., 2010, Frederique M, Laurent D and Vincent H et al., 2004). The drag coefficient of a vehicle plays an important role on total fuel consumption value. For a sedan car at 100 km/h approximately 20 % of total fuel consumption and for a heavy car 50 % of total fuel consumption is used to overcome drag forces (Mc Callen R, Couch R, Hsu J Et al., 1999) Therefore, improving vehicle design and overcome design limitations to reduce coefficient of drag is an important step all over the vehicle design process in the automotive industry (Jeff, H. et al., 2002).

Two types of drag forces exist in aerodynamic which are pressure and friction drag.

The pressure drag occurs by pressure difference between front and rear of the body and it is related with form of aerodynamic design and flow separation. However, friction drag occurs friction between the air and body and it affects by viscosity of air which is directly related Reynolds number and surface area of body. Moreover, consumption of fossil fuels will cause an increment in greenhouse effect by 50-250 % by 2050 (Smith, T. et al 2014) and lower fuel consumption values for commercial vehicles requires good aerodynamic design. Additionally, a good aerodynamic design provides safe driving and more sustainable environment.

In aerodynamics studies, the wind tunnel experiments are very successful to simulate real conditions and obtain reliable results of the analysis. But testing each optimization may lead high cost and high waste of time. However, with the developing technology in CFD softwares has been an alternative for wind tunnel experiments.

With developments in high performance computers and computational fluid dynamics (CFD) softwares become useful tools to improve vehicle design.

In the Automotive industry there are many reference car models, which are called generic ground models. Generic ground models are generally used in literature studies as a simulation model and each generic model has a specific characteristic aerodynamic depends on analysis conditions. These generic models are also named as bluff body and these bodies have flow separation zones and characteristics wake flow region. (Cooper et al., 1993). This flow separation causes a lower pressure in rear wake zone and all the optimization studies aim to reduce big pressure difference between relatively higher pressure zone and lower pressure zone. That is why flow separation regions and recirculation in rear wake area are one of the important parameters in automotive aerodynamic.

Ahmed Body generic model is one of the important simulation tools in literature works. Like the other models easy to create and analyze because of its simple geometric shape compared to commercial vehicles (Le Good and Garry et al., 2004). Even it has a simple shape, creates complex flow characteristics like commercial vehicles. Demonstration of full-scale Ahmed Body dimensions and slant angle variations can be seen from Figure 1.1.

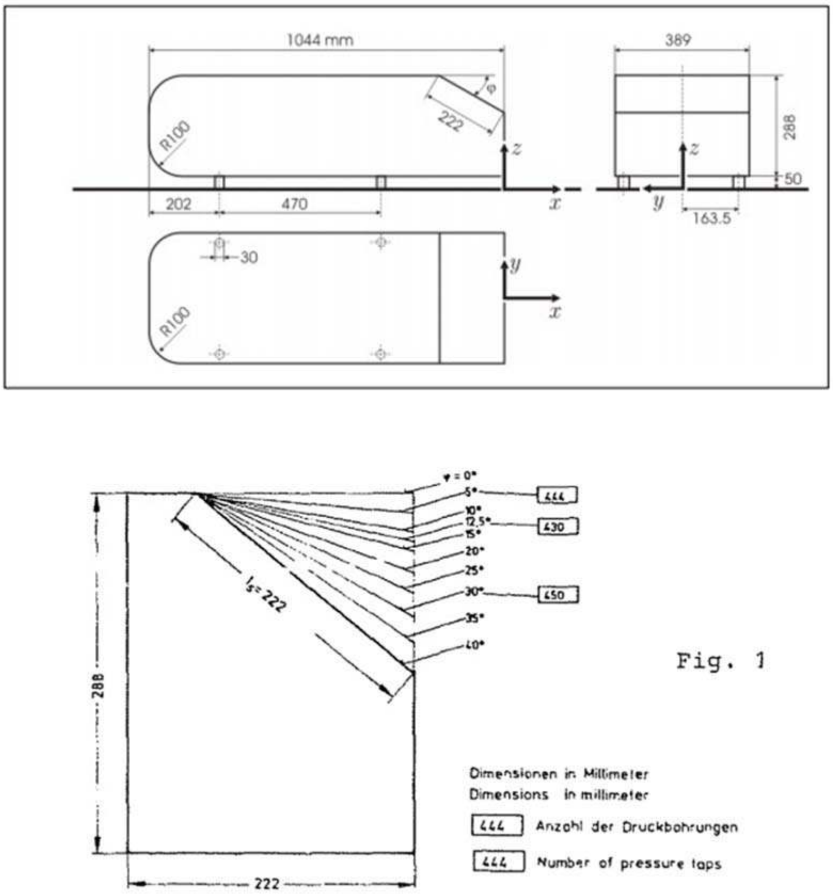

Figure 1.1. Schematic of Ahmed Body Dimensions and Slant Angle Variations (Ahmed et al., 1984)

Three major components of Ahmed body are; front cylindrical surface, longitudinal part and angled rear window. The main flow separation occurs at the angled rear window area with the air comes from top, bottom and side areas. These separated and unformed shear layers after the end of the vehicle tends to each other because of a stagnation zone occurs. This zone created by layers is called wake zone. In Figure 1.2 can be seen drag coefficient changement according to slant angle. In the beginning of slant angled area flow starts to separate and end of the slant area flow reform to separating bubble shape. To see effect of slant angle on vehicle aerodynamic in the range of 0º to 40º angle values are studied by Ahmed et al., 1984. According to this study two critical angle value is detected which they are 12.5º and 30º. Investigating the wake structure of body shows a significant increment detected between the values 12.5º and 30º (Lienhart and Becker et al., 2003). If the angle value closes to 30 º, C-pillar vortices increase because of pressure drop on slant area with the effect of separating bubble. Over the 30 º slant angle the C-pillar become less strength because flow cannot interact with slant angle surface comparing to previous angle values. Therefor maximum drag coefficient value is at 30 º and over this values tend to decrease (Vino et al., 2005).

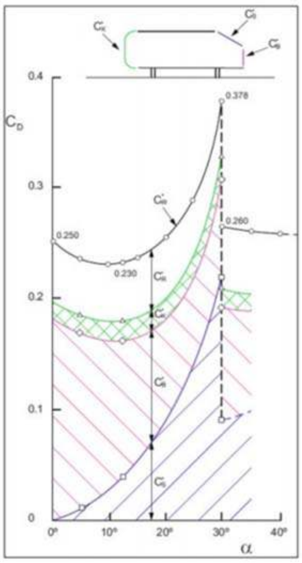

Figure 1.2. Ahmed body Cd variation with the change of slant angle(α) (Ahmed et al., 1984)

In this study slant angle value is not variable parameter and choose 35 º. The significant characteristics flow of Ahmed body with flow separation and recirculation areas in the wake can be seen from Figure 1.3. As mentioned wake geometry part, pressure drop on wake region area is one of the main reasons of strong C-pillar vortices and high, drag coefficient (Guilmineau et al., 2011). There are some methods to eliminate the sudden pressure drop on wake region and this method are divided into active and passive methods depend on whether they use energy or not (Altaf et al., 2014).

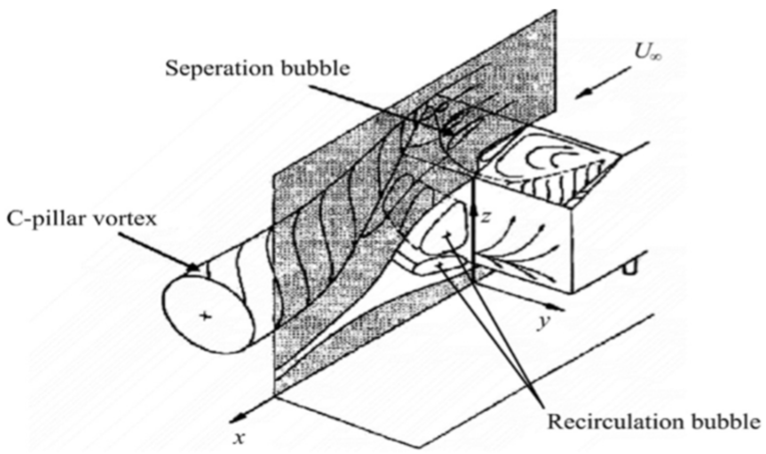

Figure 1.3. Averaged near wake structure of Ahmed model of α= 12.5-30 (Ahmed et al., 1984)

Active control methods generally use an internal power energy by synchronized with the vehicle's electrical distribution system. These active methods work in harmony with more than one engineering systems like electromagnetic, mechanic, electric, piezoelectric and acoustic. In the contrary of passive methods, active methods mostly use moveable wings, walls, vortex generators, covers or grills in order to control flow with depends on vehicle velocity. Of course design, size and weight parameters very important because they can negatively effect when joined in the vehicle system. As can be understood from all this, active drag reduction methods are more complicated and costlier so difficult to perform and work on. In aeronautics there are some validated control methods for years. As can be seen from current sport cars this method were being used with same working principle but as to cause the opposite effect. Bayraktar et al., 2001 investigates effect of Reynolds number on Ahmed body drag and lift coefficient. Reynolds number range was selected between 2.2 to 13.2 million. Results of the study maximum drag coefficient improvement was taken 3.5 % and maximum lift coefficient was taken as 2 %. Also it was understood high Reynolds number has no effect on drag coefficient.

Generally passive control methods include grooves or add on operations to vehicle surface. They are divided into two categories how it affects the flow. The first category includes from barriers directly mounted on vehicle surface and the second category consists of barriers located upper or lower position of vehicle.

Main reference of this study Ahmed et al. 1984 was made all experiment with 60 m/s wind velocity. This velocity equals to Reynolds number of 4.29 million based on vehicle length. Even Ahmed body is simple and available for CFD analysis, each turbulence model is not capable of solve rear wake turbulence zone. That is why some literature studies aim to investigate predicting situation of turbulence models for different conditions. For example a study was made by Krajnović et al., 2004 implemented Large-Eddy simulation model flow around the Ahmed body. Large-Eddy Simulation (LES) is a CFD method which it solves large flow wakes with using Navier Stokes equations directly. But LES is effective medium scale of Reynolds number it is capable of directly solution instead of to modelling. That is why Krajnović et al., 2004 studied at low Reynolds number ($2x10^5$) on 25° slant angle Ahmed body with the 9.6 and 16.5 million cells. Also the results of this study validated and well predicted comparing to reference studies.

In another study Kapadia et al., 2003 studied a different CFD modelling technique as Detached Eddy Simulation (DES) with cell number of 1.74 million. 25° and 35° slant angle Ahmed body models analyzed and results showed 5%

approximate value was taken according to experimental results of Ahmed at al., 1984.

Despite the DES and LES techniques have perfect estimating capable of flow, they can not used every time due to the computer requirements and mucher solution time. In this situation Reynolds averaged Navier Stokes (RANS) equation based turbulence models are good alternative methods and very common in automotive industry. Ahmed body with 25° analyzed by Braun et al., 2001 with the Realizable k-ε model and 2.3 million cell number. Despite the RANS models not very good to predict flow separation characteristics, this study was gived acceptable proximity values.

Nicolas et al., 2015 made an active flow control study on 25 º slant angle Ahmed body model with using steady micro jet actuators. In this study multiple actuator variations were determined according to critical vorticity areas and aimed to see effect of vehicle aerodynamic. Using with the most effective configurations maximum drag reduction value was achieved as 13 % but when implementing different combinations as simultaneously the maximum drag reduction was achieved additional %2 values.

In another active flow control by Wanbing et al., 2018 with using new unsteady jet inspired from synthetic jet to reduce aerodynamic drag. Ahmed body 25-degree slant angle is serviced to implement analyzing method Large eddy simulation. Flow control devices are distributed on rear window of Ahmed body. Effect of this devices and flow characteristics saw significantly after analysis. %13.6 maximum drag reduction value was a demonstration providing positive effect on pressure distribution on wake area.

Active flow control can be benefit using hybrid flow control systems. Airfoil Ra5243 is a common analysis tool to improve aerodynamic and studied Bedri et al., 2011. The main aim of this study was reduce strength pressure wake shock and wave drag. Hybrid system contained 2D single bump, individual jet actuator and implementing simultaneously this two control device. The simulation results show that after optimization studies 3.94 % decrease in total drag and 5.03 % increase in lift using different combinations.

To create smaller wake flap or deflectors, splitter plates, boat tails, vortex generators and non-smooth surfaces one of the passive drag reduction methods. Implementing vortex generator on Ahmed body 25-degree slant angle on 20 m/s free stream velocity provide 11.7 % maximum drag reduction. Vortex generators are positioned on top of the Ahmed body surface with determined distance to control flow and prevent early separation (Krajnovi´c et al.,2004). Another study different height deflectors were implemented on the side and leading edges of 25 slant angle Ahmed Body. Analsys in this study have been done with Ansys CFD commercial software.

Design engineers struggles to minimize design limitations and make more similar like ideal streamlined shape. But some factors like space for passenger, luggage and engine compartment cause being different from desired shape. Front corner roundness and flow separation area that occurs end of the bluff body are very effective parameters for determining drag coefficient number. As seen in Figure 2 Ahmed body car model has a cornered rear surface and making elliptical surface was one of the combinations to see effect of drag number. Engineers have studied to find new modifications named add-on devices because of limitations on body design. These devices are classified as two types; active drag reduction and passive drag literature study shows aerodynamic efficiency of an airfoil can be improved with addition a single bump at 10 % and CD 6% can be lower than a smooth air foil (Strachan et al., 2007). With another article results shows that increasing dimple depth which it causes directly increasing surface roughness increase the drag coefficient values (Vino et al., 2005). The term of turbulent air related with Reynolds number, when Reynolds number is high it causes to consume extra energy during movement from wake flow separation area and drag coefficient number is correct proportion with wake area.

So of course it is not easy to implement this passive drag reduction technique to commercial vehicles when evaluated in terms of productability side. In this study base Ahmed body will be investigated to compare flow characteristics and wake region characteristics. During CFD simulations poly-hex core meshed CFD setups were solved with k-w SST turbulence model.

Active drag reductions are more complicated and costlier so difficult to perform and work on. Flap or deflectors (Mohammed et al., 2010, Lee et al., 2010, Fourrie et al., 2011), splitter plates (Gillie´ron et al., 2011), boat tails (Khaligi et al., 2001), vortex generators (Modi et al., 1995), non-smooth surfaces (Chear et al., 2015), and linking tunnels (B. Mohammadikalakooa et al.,) are one of the techniques passive drag reduction.

Non-smooth surfaces are very effective technique for reducing drag coefficient and make proper flow separation rear surface of Ahmed body. Dimpled surfaces are good example for non-smooth surfaces. Dimpled surfaces on a golf ball which it provides longer distance travel if compared a non-dimpled surface ball are inspired from engineers. Because of longer distance travel is providing lower turbulent air on flow separation area with the aid of dimpled surfaces. In Figure 1.4 wake characteristics of golf balls compared with dimple surface and without dimple.

This study investigates effects of passive drag reduction techniques using a generic car model which is named Ahmed body (Ahmed et al., 1984). The final object of this analyzes is to evaluate changement drag coefficient and see effect of variable parameters on vehicle aerodynamic after the implementing different techniques and their combinations.

Furthermore, reducing drag coefficient an automobile improves performance of the vehicle and optimizes fuel efficiency so the main purpose of the study is to reduce energy losses and inefficiencies by making aerodynamics improvements on road vehicle model Ahmed body with angle of 35°.

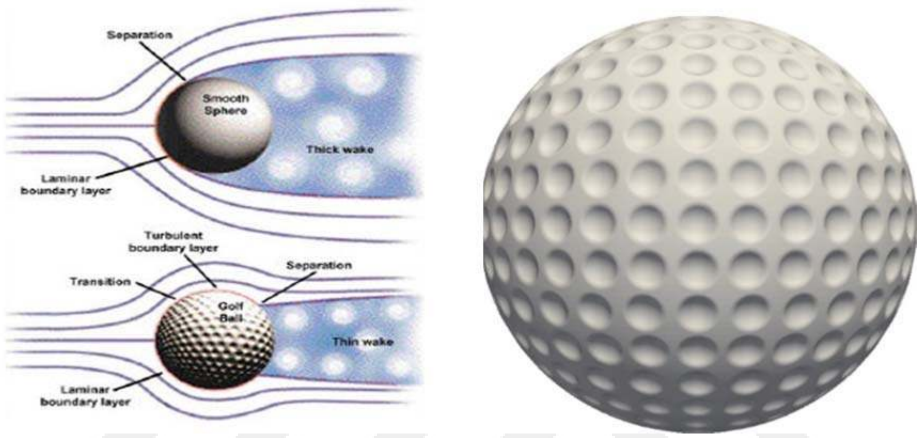

Figure 1.4. Dimpled surfaces on a golf ball (Chear et al., 2015)

2. NUMERICAL APPROACH

2.1. Cad Model

As seen on Figure 1 dimensions of model; length L = 1044 mm; width W = 389 mm; height H = 288 mm; length of slant back E = 222 mm. Also, diameter of the cyclinders are 30 mm and height values are 50 mm from origin. CATIA software was used creating 3D models of Ahmed body. In simulations dimple surface passive drag reduction technique was used and in this study dimple surfaces were implemented on Ahmed body rear window area. Dimple characteristics has defined two different ways with same formula in literature studies. They are dimple depth ratio and surface roughness.

$$Dimple\ depth\ ratio = \frac{k}{d} \qquad (1)$$

This passive drag reduction method was also used from Chear et al., 2015 according to formula k defines dimple depth value and d defines diameter value. The second way is named as surface roughness value instead of dimple depth ratio.

$$\mathcal{E} = \frac{\mathrm{k}}{\mathrm{d}} \qquad (2)$$

There are 15 different dimpled model used in simulations and they have 3 diffirent diameters. Also each model has different depth values in its own group. It can be seen from Figure 2.1 and Table 2.1 dimple surfaces implemented with the same configurations on Ahmed body rear window are in all models. Also according to dimple dimensions' model's surface roughness values generally vary from each other. Additionally, to prevent sharp surface between rear window and dimple 0.5 mm fillet radius was applied.

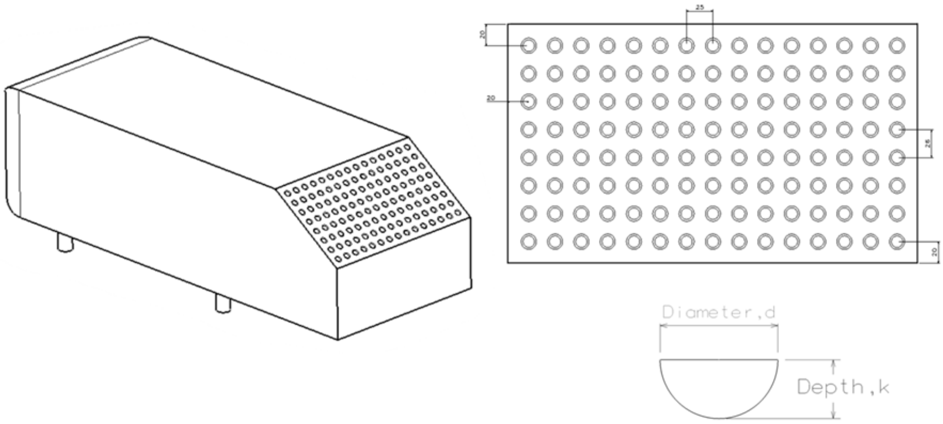

Figure 2.1. Ahmed body with dimple surfaces and dimensions in mm

Table 2.1. Diameter and depth values for each model

Model Number	#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12	#13	#14	#15
Dimple Diameter (mm)	10	10	10	10	10	15	15	15	15	15	20	20	20	20	20
Dimple Depth (mm)	1	2	3	4	5	1	2	3	4	5	1	2	3	4	5
Surface Roughness (Ɛ)	0,1	0,2	0,3	0,4	0,5	0,06	0,13	0,2	0,26	0,33	0,05	0,1	0,15	0,2	0,25

2.2. Mesh Generation and Mesh Independence

CFD simulations depend on multiple parameters. For example, in case of creating too fine or too coarse grid structure, it is possible to over-resolve or under-resolve solution. Choosing small element size completely on flow region causes a highly increment on the solution time and computational source requirements. In this case, flow region separated in 3 different sizes was created, in which the mesh density increases from the flow region wall to the vehicle body wall. During generating meshes for CFD simulations, the Watertight Geometry workflow in Ansys Fluent was used. The workflow is much simple and by simplifying mesh generation it provides to complete all stages in one window. Also thanks to new poly-hexcore meshing algorithm, it is possible improve mesh quality and solution time.

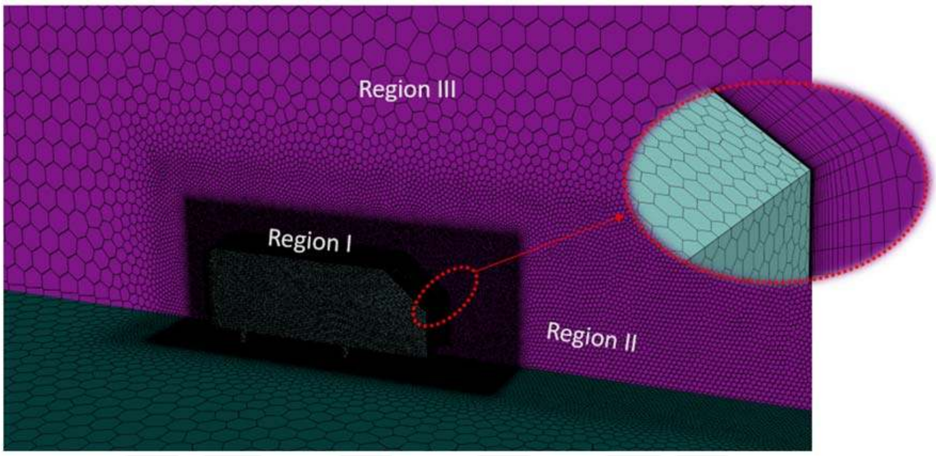

Figure 2.2. View of grid regions with 3 different mesh densities

To make strategic mesh refinement, volume of poly hex-core elements were divided with 3 different mesh density (Figure 2.2). The closest region to vehicle walls have smallest mesh sizes for the purpose of catch flow characteristic around the vehicle. Y^{+}~1 values was chosed as a essential requirement when solving viscous sublayer with k-ω SST to take advantages of this model. Also 24 prism layers were used to increase predicting possibility of full boundary layer profile and effect of the adverse gradient forces. Additionally, in numerical simulations mesh independence study is very important to select available and sensible number of mesh cells. In this study as can be seen from Table 2.2, each region was analysed 3 different setup which they are a Mesh 1,Mesh 2, and Mesh 3. When evaluated according to results of mesh indepedence study (Figure 2.3) , it was found that flow solutions are almost independent from grid size after the grid level of Mesh 2.

Table 2.2. Mesh Sizing Strategy

	Mesh 1	Mesh 2	Mesh 3
Element size of region I (mm)	20	7,5	5
Element size of region II (mm)	40	17,5	12,5
Global element size (region III) (mm)	175	100	75
First prism layer height (mm)	0,04	0,04	0,04
Number of prism layer	24	24	24
Growth rate	1,2	1,2	1,2
Volume mesh type	Poly-hexcore	Poly-hexcore	Poly-hexcore
Total mesh count ($x10^6$)	1,1	2,4	5,5

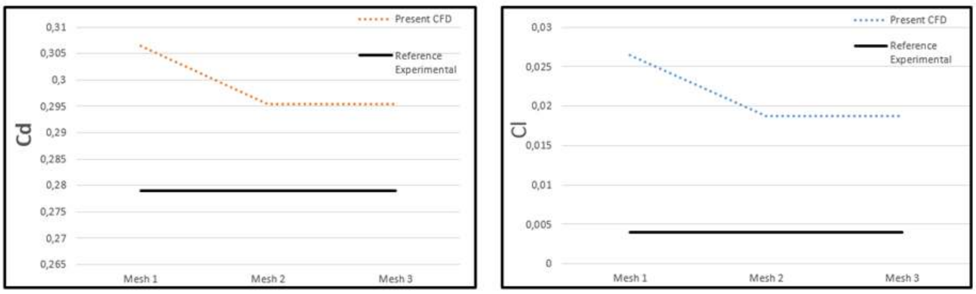

Figure 2.3. Mesh independence study

2.3. Turbulence Model

RANS numerical method was selected for numerical simulations in this study since it is is adventageous in terms of computational cost and comparing with Despite Detached Eddy Simulations (DES), Large Eddy Simulations (LES), and Unsteady Reynolds-Averaged Navier-Stokes (URANS).

It is very important to select appropriate turbulence model for current study. Because advantages of the eight RANS turbulence modes may differ for different conditions.

So the main aim of the selecting k-ω Shear Stress Transport (SST) (Menter et al., 1992) provides better estimation one of the RANS models and it has a better estimation ability in adverse pressure gradients. This SST k-ω model which is used widely in aerodynamic studies, combines of ability resolving k-ω with near the walls and k-Ɛ with free shear layers far from surfaces.

2.4. Boundary Conditions, Solver Settings and Post Processing

To provide different element sizes on specific areas body of influence mesh insert technique was used. Also, near of the Ahmed body was meshed finer, away from model was meshed coarser. Air has constant properties and assumed incompressible was adopted as a simulation fluid. The air velocity has been set to 40 m/s with a Reynolds number of Re = $2{,}86\ x\ 10^6$ was based length of the Ahmed body and turbulence intensity value was selected 0.5%. Also the pressure outlet value was set 0 Bar with 5 % turbulence intensity. These turbulent intensity values were selected for comparison experimental results of Meile et al. in 40 m/s study. The experimental results of Ahmed et al. at Re = $4{,}29\ x\ 10^6$ are included in this study just for comparison. In preparing solution setup 'Double Precision' was selected to increase accuracy of solution and. Parallel processing options was chosed instead of serial and number of processes was selected according to specifications solver computer.

The spatial discretisation part was 2nd order for the pressure, and 2nd order upwind for the momentum. Additionally, if it selected 'Pseudo Transient' option optimizing solution control values can provide more relaxed solution convergence. The residual absolute citeria value was 10^{-5} which this value effect of convergence accuracy.

The boundary conditions and flow inlet parameters used during numerical solution were shown from Table 2.3.

The whole enclosure body includes of three boxes to apply different element size. As can be seen from Figure 2.4 if model length is called L; length of main box is 15.5L, length of the between inlet and model is 2.5L. Also, the width is

3L and height is 1.5L to provide condition of free stream condition with zero normal gradients.

A quasi-three-dimensional model (quasi-3D) was used during creating computational domain and they were analyzed by dividing them in two. These dividing operation provided a reduction number of elements and analyzing time.

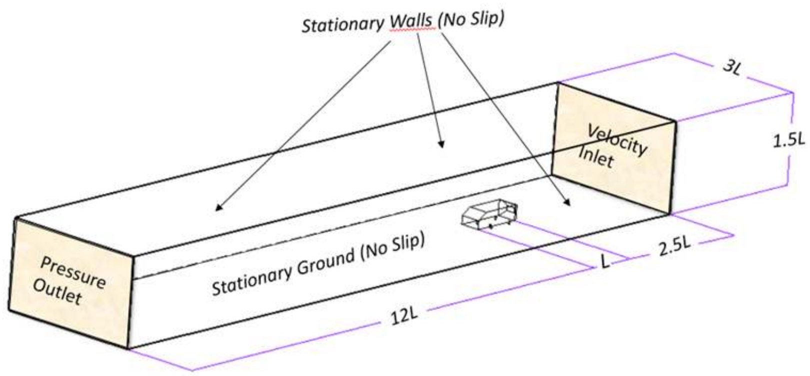

Figure 2.4. Dimensions and boundary conditions of the computational domain

Table 2.3. Boundary condition setup for numerical analysis

Boundary Condition Parameters	Boundary Condition Type	Quantity		V	40 m/s
Inlet	Velocity inlet	40 m/s		Re	$2{,}86 \ x \ 10^{6}$
Outlet	Pressure outlet	1 atm		ρ	$1.225 \ kg/m^{3}$
Walls(Ahmed body)	Stationary wall with no-slip condition	–		μ	$1.789 \ x \ 10^{-5} \ kg/ms$
Walls(Enclosure)	Stationary wall with no-slip condition	–		I	≤ 0.25 %

There are some procedures to obtain stabilized solution and realistic comparative with experimental data's. Geometry creation is first step of this procedure list. Because changeable parameter like dimensions or shape of the drag reduction surface is directly affect CFD results. Mesh creation is the second important pre-processing step after creating geometry. Especially face and volume mesh quality improvements are very important to take convergence during simulations. Additionally, solver step includes critical parameters like turbulence model, y+ value, number of layers, boundary conditions and convergence criteria. According to aim of the study some parameters may be essential to achieve reliable results. In some situations, it is better to select appropriate values otherwise solution time may lead longer. Validation and verification is the last step of CFD simulation procedure.

Drag, lift and residual monitors should be compare with experimental and reference studies. The approaching percent may give information about the completion of a reliable simulation procedure. CFD simulation procedure of these study can be seen from Figure 2.5. The results obtained during this study are outcome of selections and improvements following this procedure.

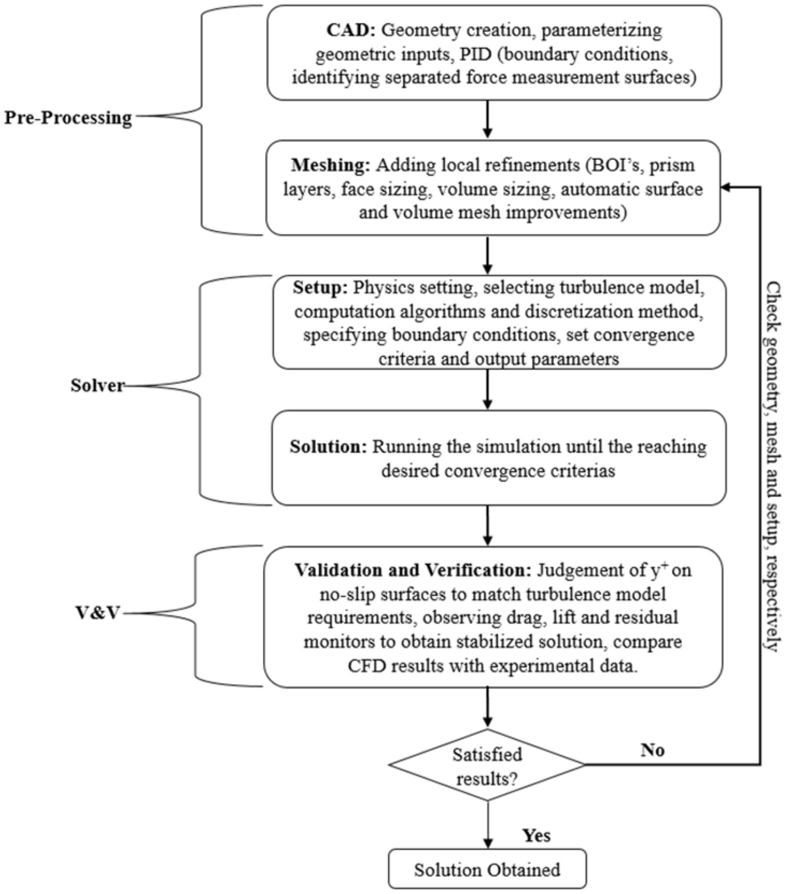

Figure 2.5. Procedures for CFD simulations

3. RESULTS, VALIDATIONS AND EVALUATION

3.1. Validation

Numerical analysis results in this study were compared in Table 4. References studies both includes numerical results (Thabit et al., 2018, Braun et al., 2001) and experimental results (Ahmed et al.,1984 Meile et al., 2011). Reference studies selected similar boundary settings as possible with the current study but depending enclosure volume, turbulence model, Reynolds number or total mesh number parameters the deviation value changes with the reference studies. One of the main experimental reference is Ahmed et al. made all tests with 60 m/s air-stream. This is corresponding a Reynold s number o 4.29 million based on body length. This Reynolds number is approximately 1.87 million higher than the current study. The second experimental reference study by Meile at al., 2011 investigated effect of slant angle with using 25 ° and 35 °. Analysis were made velocity range 10 m/s ≤ U∞ ≤ 40 m/s to see effect of Reynolds number but our concern for this study the maximum velocity 40 m/s which it corresponds Re = $2.784 x 10^6$. When compared with this two experimental study deviation values for Cd coefficients are 12,68% (Ahmed et al.) and 6,59% (Meile et al.). The main reason of deviation difference is wind velocity is highest at Ahmed et al. study. First numerical study by Emmanuel et al. [32] studied with two different slant angle which they are 25 ° and 35 °. Also to see effect of turbulence model each slant angle analysed with different models like Spalart-Allmaras , SST K – ω , Rij – ωIP , Rij – ωSSG , EASM and ASM . With using SST K – ω turbulence model our current study deviating value is 2%. The second numerical studied by Thabit et al., 2018. In this study realizable k-ε model with 2nd order was applied and taking a experimental study was referenced with 40 ° slant angle Ahmed body. To evaluate effect of different slant angles on Ahmed body, 10-40 ° slant angle models were analyzed. When compared the result of 35 ° slant angle model drag coefficient value deviating value 0,16%. This variation can be acceptable and Cd results of this current study is in a good agreement with experimental results.

3.2. Evaluation of Force Coefficients

Table 3.1. Numerical studies results comparision with references studies

	Thabit et al., 2018	Emmanuel et al.,	Ahmed et al., 1984	Meile et al., 2011	Base	10D1H	10D2H	10D3H	10D4H	10D5H
C_D	0,2949	0,2895	0,258	0,276	0,29548	0,29145	0,2864	0,28634	0,2865	0,28786
C_L	0,0205	n.a	n.a	0,004	0,018	0,026	0,0685	0,0518	0,018	0,0546
	15D1H	15D2H	15D3H	15D4H	15D5H	20D1H	20D2H	20D3H	20D4H	20D5H
C_D	0,2862	0,28585	0,2856	0,28595	0,286	0,2855	0,28524	0,28513	0,28517	0,28549
C_L	0,062	0,064	0,0619	0,0501	0,096	0,0686	0,0547	0,0487	0,0359	0,017

According to numerical results in the Table 3.1 all dimple models provide drag reduction when compared with reference base Ahmed body model. Nevertheless, if the dimple depth increases after a certain value, drag value is effected negatively. The biggest drag reduction was taken 3.5 % with 20D3H according to base model value 0.29458. The lowest drag reduction was taken 2.5 % with 10D5H comparing to base value. It can be seen from Table 3.1 increasing dimple diameter provide positive effect to decrease drag value but dimple depth factor causes negative effect after a point.

3.3. Comparison of Wake Flow Structures

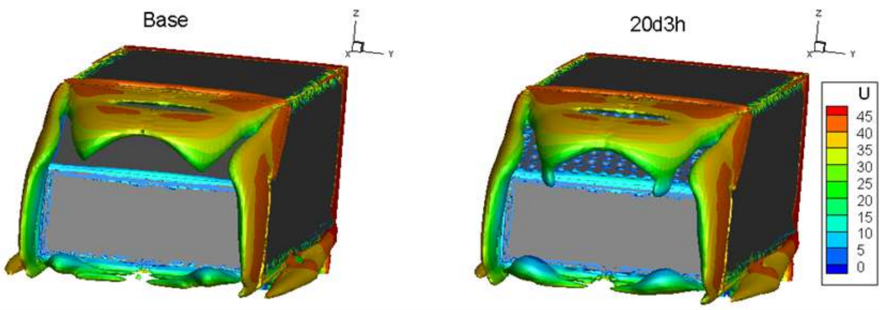

Figure 3.1. Iso-surfaces of Q criterion demonstration for numerical analysis

In this part of the study, qualitative flow representations are carried out between the optimum model with minimum drag and base model. According to Figure 3.1 Q criterion can be seen, which is one of the vortex demonstration methods in fluid dynamics. By definition, the Q criterion defines vortices as areas where the vorticity magnitude is greater than the magnitude of the rate of strain. For this two model it is possible to see upper, lower and longitudinal flow separate from rear of the body. The obvious difference is lower and upper vortex tend to interact each other when compared with base model. This interaction results in smaller wake region on rear area and this provides pressure recovery at the rear of the body.

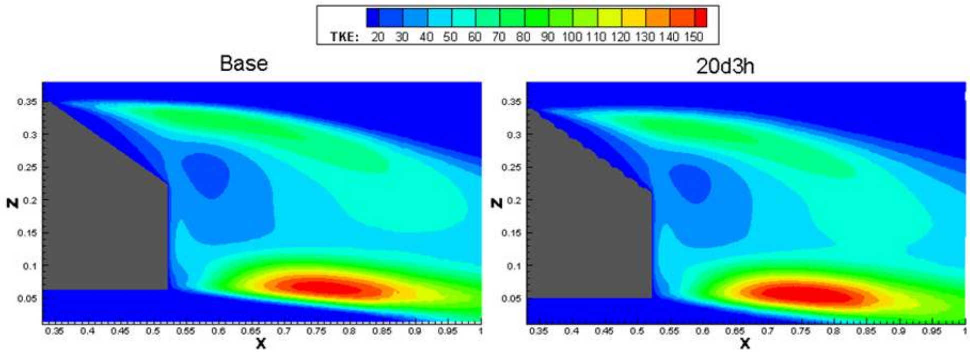

Figure 3.2. TKE contours in the symmetry plane

Turbulent kinetic energy (TKE) is the qualitative measure of energy dissipation and loss for a given flow. It can be measured as the root mean square of the fluctuation in the flow velocity. TKE contours demonstrate the higher energy loses areas because of higher drag. TKE contours in the Figure 3.2, supports previous comments about these two models. As can be seen, the lower and upper vortex tend to interact rear of the body. The green flow vortexes break off clearly in the base model but the same vortexes interact with each other on 20d3h model. This indicates the flow is tending to regular and uniform flow regime..

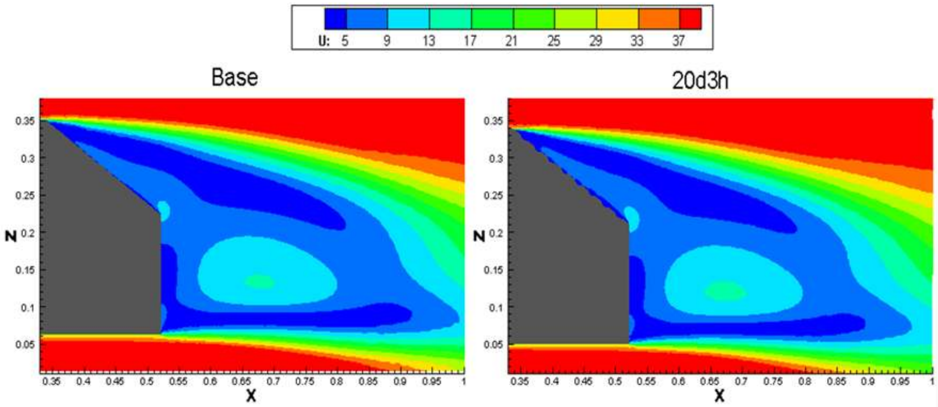

Figure 3.3. Velocity magnitude (u) contours in the symmetry plane

A regular and narrower wake is essential in terms of aerodynamic performance for a vehicle. Ahmed body rear vortex is almost independent from front and longitudinal surfaces. That is why, in this study drag reduction surfaces effect only wake zone of body. For the base model, it is possible to see bigger vortex creation on rear but on the optimized model using dimple surfaces eliminate rear vortex creation and provide pressure and velocity recovery. At the same time Figure 3.3 shows velocity magnitude contours are shown in symmetry plane. According to velocity distribution of the rear area and it can be understood that dimple surfaces reduce power of the vortex lines. When compared end of the two contours it can be seen dark blue velocity is smaller and terminates before according to base model. Also size of the green elliptical region in the middle of the contours bigger than comparing to base model. When this informations evaluated, it can be seen that there is a velocity recovery with using dimple surfaces.

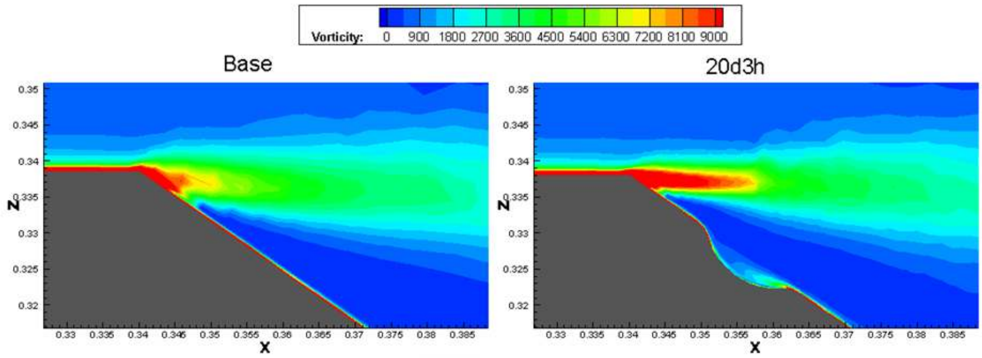

Figure 3.4. Vorticity contours in rear separation zone

One of the main aim of the using dimple surfaces is eliminating or delaying flow separation from rear surface. The vorticity contours are represented to see in terms of cohesion effect between flow particles and vehicle surface. In Figure 3.4, since there is an elongation during separation rear corner of the vehicle it can say that an improvement has been gain in terms of attaching to surface.

Table 3.2.. Locations of the planes

	Location		Location
X_1	x/L=0.1	Y_1	z/H=0.574
X_2	x/L=0.2	Y_2	z/H=1
X_3	x/L=0.3	Symmetry	y/W=0

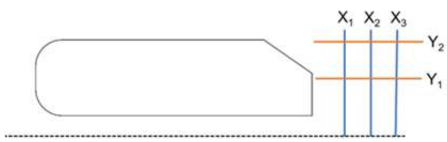

Figure 3.5. Demonstration of planes

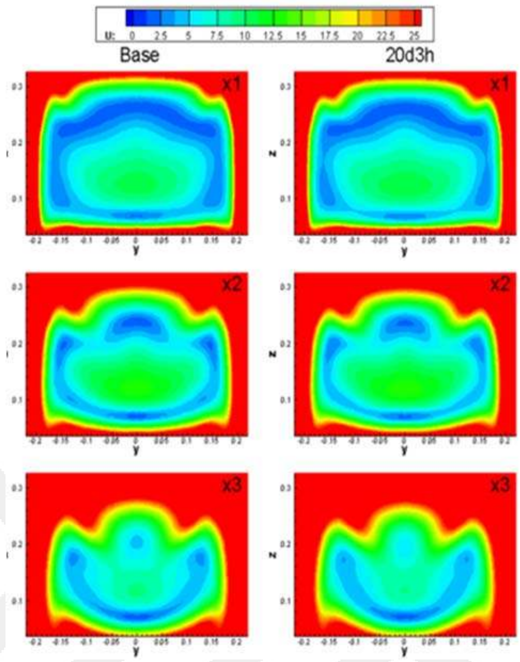

Figure 3.6. Velocity magnitude contours with vertical planes (X_1 , X_2 , X_3)

In Figure 3.6, normalized velocity contours demonstrated with using X_1 , X_2 and X_3 planes. This planes cut the vehicle from top to bottom also exact locations can be found Figure 3.5 and Table 3.2 Naturally as the distance increases from vehicle surface, size and value of velocity contours are reducing. But when evaluated same distance level of planes, the optimized model' contours have smaller velocity regions according to base model's.

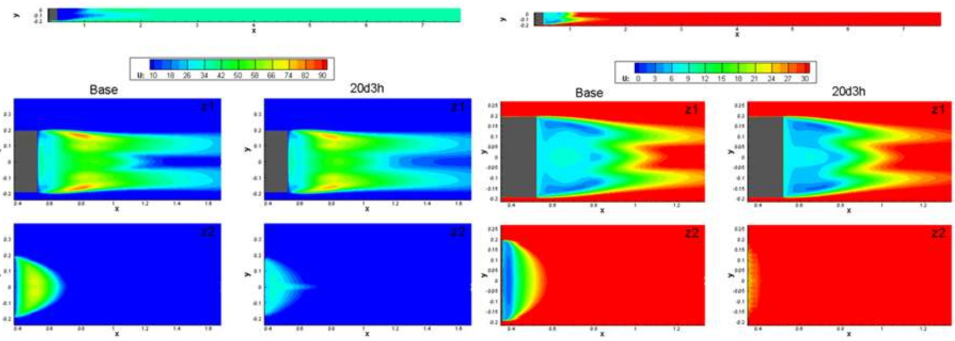

Figure 3.7. Velocity magnitude contours with horizontal planes (Z_1 , Z_2)

In Figure 16, velocity magnitude contours illustrated with two horizontal plane. Z_2 plane locates upper side and Z_1 plane locates lower side of the vehicle. It can be seen from figure, near of the vehicle the velocity values smaller comparing to flow contour separating from vehicle. But with the optimized 20d3h model, wake region is narrowing. Especially Z_2 contours shows that there is a remarkable velocity recover with 20d3h model which this recovering directly related decreasing drag value and improving rear flow structure.

4. CONCLUSIONS

In this study a drag reduction study is represented to improve aerodynamic characteristic of a generic car model's rear wake zone. The rear window of the 35° slant angle of Ahmed body is optimized with variable parameters in each model. This study shows similar results with the main reference study (Chear et al., 2015) in passive drag reduction methods. These passive methods reduce pressure gradients by creating friction on flow boundary layer. Inferences from this study can be summarized like below:

- During creation dimple surfaces parameters can effect whether decrease drag coefficient or not.
- Although increasing dimple diameter effects positive, the bigger dimple depth causes to increase drag coefficient value after a point.
- Despite optimized models provide C_{D} value reducing, there is no consistent variance on C_{L} values between optimized models and base model.

In the light of the information's in this study it can say that dimple surfaces reduce vortex and turbulent intensity on the wake. Especially flow separation zones are very critical to achieve aim of the aerodynamic studies but in the future studies can be investigate effect of optimized dimple surface with implementing different zones of car like bottom, top, side and front surfaces and their combinations. Besides all this, considering that the drag reduction values are very low compared to vehicle size, it creates a discussion subject whether this study is feasible or not.

Also it should be take into account in terms of commercial side because changing vehicle part shapes effect production conditions and it is important for the manufacturers whether it will be preferred visually or not.

REFERENCES

C.K Chear , S. S Dol (2015) World Academy of Science, Engineering and Technology International Journal of Mechanical and Mechatronics Engineering Vol:9, No:1

Abd-Alla, G. H. (2002). Using exhaust gas recirculation in internal combustion engines: a review. In *Energy Conversion and Management* (Vol. 43). www.elsevier.com/locate/enconman

Altaf, A., And, A. A. O., & Asrar, W. (2014). Revıew Of Passıve Drag Reductıon Technıques For Bluff Road Vehıcles. In *Iıum Engineering Journal* (Vol. 15, Issue 1).

Author, W., Ahmed, S. R., Ramm, G., & Faltin, G. (1984). Some Salient Features of the Time-Averaged Ground Vehicle. In *Transactions* (Vol. 93). https://about.jstor.org/terms

Bayraktar, I., Landman, D., & Baysal, O. (n.d.). *Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics*.

Chowdhury, H., Loganathan, B., Wang, Y., Mustary, I., & Alam, F. (2016). A Study of Dimple Characteristics on Golf Ball Drag. *Procedia Engineering*, *147*, 87–91. https://doi.org/10.1016/j.proeng.2016.06.194

Fuss, F. K. (2011). The effect of surface skewness on the super/postcritical coefficient of drag of roughened cylinders. *Procedia Engineering*, *13*, 284–289. https://doi.org/10.1016/j.proeng.2011.05.086

Gilliéron, P., & Kourta, A. (2010). Aerodynamic drag reduction by vertical splitter plates. *Experiments in Fluids*, *48*(1), 1–16. https://doi.org/10.1007/s00348-009-0705-7

Howell, J., Sherwin, C., Passmore, M., & le Good, G. (2002). Aerodynamic drag of a compact SUV as measured on-road and in the wind tunnel. *SAE Technical Papers*. https://doi.org/10.4271/2002-01-0529

Kapadia, S., Roy, S., & Wurtzler, K. (n.d.). *Detached Eddy Simulation Over a Reference Ahmed Car Model*.

Khalighi, B., Zhang, S., Koromilas, C., Balkanyi, S. R., Bernal, L. P., Iaccarino, G., & Moin, P. (2001). Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device. *SAE Technical Papers*. https://doi.org/10.4271/2001-01-1042

Krajnović, S., & Davidson, L. (2004). Large-eddy simulation of the flow around simplified car model. *SAE Technical Papers*. https://doi.org/10.4271/2004-01-0227

Leroyer, A., Wackers, J., Queutey, P., Guilmineau, E., Wackers, J., & Queutey, P. (n.d.). *Numerical strategies to speed up CFD computations with free surface Numerical strategies to speed up CFD computations with free surface-Application to the dynamic equilibrium of hulls*. https://doi.org/10.1016/j.oceaneng.2011.09.006ï

Lienhart, H., Stoots, C., & Becker, S. (n.d.). *Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)*.

Mccallen, R., Couch, R., Hsu, J., Browand, F., Hammache, M., Leonard, A., Brady, M., Salari, K., Rutledge, W., Ross, J., Storms, B., Heineck, J. T., Driver, D., Bell, J., & Zilliac, G. (n.d.). *Progress in Reducing Aerodynamic Drag for Higher Efficiency of Heavy Duty Trucks (Class 7-8)*.

McNally, J. W., Alvi, F. S., Mazellier, N., & Kourta, A. (2015). Active flow control on an ahmed body - An experimental study. *53rd AIAA Aerospace Sciences Meeting*. https://doi.org/10.2514/6.2015-0825

Meile, W., Brenn, G., Reppenhagen, A., Lechner, B., & Fuchs, A. (2011). Experiments and numerical simulations on the aerodynamics of the Ahmed body. In *www.cfdl.issres.net* (Vol. 3, Issue 1). www.cfdl.issres.net

Menter, F. R. (n.d.). *Improved Two-Equation k-Turbulence Models for Aerodynamic Flows*.

Modi, V., Hill, Ss., & Yokomizo, T. (1995). *Drag reduction of trucks through boundary-layer control*.

Mohamed-Kassim, Z., & Filippone, A. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. *Transportation Research Part D: Transport and Environment*, *15*(5), 275–284. https://doi.org/10.1016/j.trd.2010.02.010

Mohammadikalakoo, B., Schito, P., & Mani, M. (2020). Passive flow control on Ahmed body by rear linking tunnels. *Journal of Wind Engineering and Industrial Aerodynamics*, *205*. https://doi.org/10.1016/j.jweia.2020.104330

Peter, J. E. V., & Dwight, R. P. (2010). Numerical sensitivity analysis for aerodynamic optimization: A survey of approaches. In *Computers and Fluids* (Vol. 39, Issue 3, pp. 373–391). https://doi.org/10.1016/j.compfluid.2009.09.013

Society of Automotive Engineers. (2003). *Emissions : advanced catalyst and substrates, measurement and testing, and diesel gaseous emissions.* SAE International.

Strachan, R. K., Knowles, K., & Lawson, N. J. (2007). The vortex structure behind an Ahmed reference model in the presence of a moving ground plane. *Experiments in Fluids*, *42*(5), 659–669. https://doi.org/10.1007/s00348-007-0270-x

Thabet, S., & Thabit, T. H. (2018). CFD Simulation of the Air Flow around a Car Model (Ahmed Body). *International Journal of Scientific and Research Publications (IJSRP)*, *8*(7). https://doi.org/10.29322/ijsrp.8.7.2018.p7979

Third IMO GHG Study 2014 Executive Summary and Final Report. (2015). www.imo.org

Vehicle Aerodynamics Drag Reduction by Surface Dimples. (n.d.).

Vino, G., Watkins, S., Mousley, P., Watmuff, J., & Prasad, S. (2005a). Flow structures in the near-wake of the Ahmed model. *Journal of Fluids and Structures*, *20*(5), 673–695. https://doi.org/10.1016/j.jfluidstructs.2005.03.006

Vino, G., Watkins, S., Mousley, P., Watmuff, J., & Prasad, S. (2005b). Flow structures in the near-wake of the Ahmed model. *Journal of Fluids and Structures*, *20*(5), 673–695. https://doi.org/10.1016/j.jfluidstructs.2005.03.006

Wang, B. xin, Yang, Z. gang, & Zhu, H. (2019). Active flow control on the 25°Ahmed body using a new unsteady jet. *International Journal of Heat and Fluid Flow*, *79*. https://doi.org/10.1016/j.ijheatfluidflow.2019.108459

Yagiz, B., Kandil, O., & Pehlivanoglu, Y. V. (2012). Drag minimization using active and passive flow control techniques. *Aerospace Science and Technology*, *17*(1), 21–31. https://doi.org/10.1016/j.ast.2011.03.003

Zhang, J., He, K., Xiong, X., Wang, J., & Gao, G. (2017). Numerical simulation with a DES approach for a high-speed train subjected to the crosswind. *Journal of Applied Fluid Mechanics*, *10*(5), 1329–1342. https://doi.org/10.18869/acadpub.jafm.73.242.27566

CURRICULUM VITAE

Veli Kemal AYDIN, graduated primary school in 2010 and graduated from Göksu Anatolian High School 2014. Then, he started his BSc degree in 2014 at Cukurova University, Automotive Engineering Department. He started his MSc education at the same department of Cukurova University in 2020. He has been working as a Design Engineer since 2019.