Sanda-Mariana BODEA, Iacob-Liviu SCURTU

AERODYNAMIC EVALUATION OF A MOTORCYCLE HELMET

Abstract: *Traditionally, the product design has been based on physical experiments using different prototypes. Nowadays, the evolution of the computers has an important effect on the product design, it is based on the computer software from the conception phase. This paper presents an aerodynamic evaluation of a motorcycle helmet model using CFD simulation methods. The beginning of the evaluation includes the current state of the art regarding the computational methods for evaluate the aerodynamic performance of the motorcycle helmet, followed by the mathematical approach applied in this paper establishment of the airflow regime. Another part of the paper presents the CAD design of the motorcycle helmet modelled in five variants and performing the CFD simulation at 33.34 m/s airflow velocity for each helmet variant. The resulted aerodynamic force and the aerodynamic coefficient for each simulation model are discussed in the third part of the paper. The end of the paper presents the conclusions of this aerodynamic study, highlighting the advantages and disadvantages of using the CFD simulation to establish the helmet aerodynamics.*

Key words: *CFD simulation, helmet design, frontal area, drag coefficient.*

1. INTRODUCTION

Lately, an important topic to study is the aerodynamic evaluation of the bike racing components. The key to the aerodynamic performance is represented by the low aerodynamic drag, that can be influenced by the biker helmet geometry. To fulfil a low drag coefficient, the helmet geometry is designed like a tear drop. In recent time the numerical study was developed to find the optimal shape of the motorcycle helmet to increase the aerodynamic performance, especially on the motorcycle racing.

The aerodynamic performance of the helmet body can be determined in two ways: in the first method the helmet prototype is introduced into a wind tunnel, where the air speed is imposed according to the study requirements, and in the second method the aerodynamic evaluation is solved using the software environment, developed for computational fluid dynamics (CFD) simulation.

Due to the possibility of creating several virtual study models that can be tested in different operating conditions the CFD simulation method are used. An advantage of CFD simulation method is given by the constant use of the simulation models, that can be tested in different operating conditions without additional cost. The weakness points of using the CFD simulation method can be highlighted by the additional training of the researcher and the purchase price of the software.

In addition of this study, an important numerical evaluation of a helmet has been done by using the Ansys Fluent software. The aerodynamic evaluation is done at 15°C temperature and a viscosity of $1.7894 \cdot 10^{-5}$ kg/m·s. The k-ω SST turbulence model is used in this evaluation at four velocity cases: 5.56/ m/s, 13.89 m/s, 27.78 and 41.67 m/s. The results show the pressure and velocity distribution; drag force and drag coefficient obtained for each simulation case [3].

Another important research was done by using the OpenFOAM software. The studied model was extracted from an example case presented in this software. The position of the motorcyclist is as in the speed race, in range of 20-27 m/s velocity. The graphical results show the velocity magnitude contour establishing the low velocity area, also is presented the turbulent energy visualisations contour. The low velocity magnitude of the air can be observed in the near motorbike area and biker [4]. For a better aerodynamics at motorcycle race is represented by the helmet design and the biker position. The experimental procedures are done into a wind tunnel by using four motorcycle models, at 0-50 m/s wind velocity range and 60 s duration test. At the end of the research is established that the position of the rider influences the aerodynamics performances in range of 1-8 % of drag coefficient [8].

To improve the aerodynamic capability a redesigned model of a helmet model has done using Pro-E software. The rear side of the helmet body has been modified, making a helmet geometry like a teardrop. In the end of this study following a specification are calculated the drag force and drag pressure [10].

In this paper a helmet design and aerodynamic study are performed by using CFD simulation method. The study was performed for five helmet shapes.

2. AERODYNAMIC STUDY

In this section are presented the mathematical aspects regarding this aerodynamic study. The start point of the aerodynamic study is given by the establishing the force that acts on the helmet body, the turbulence model, and the airflow regime.

2.1 Aerodynamic force on helmet shape

This research presents a comparative study of five helmet models. The first helmet model is considered the basic model, the other four deriving from the basic model. The comparation term between the helmet models has given by the aerodynamic drag and the drag coefficient obtained after CFD study. In Figure 1 the main forces that acts on the helmet model are presented.

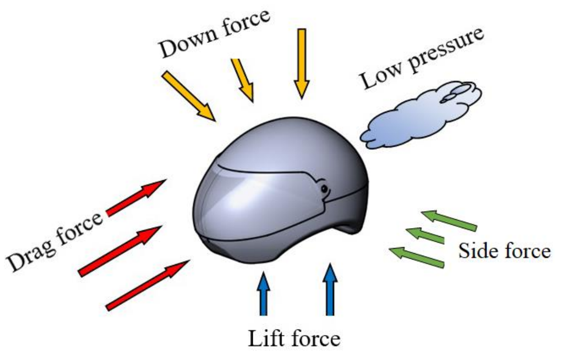


Figure 1 Aerodynamic force on the helmet surface.

The aerodynamic drag is the main force that acts on the helmet model during riding on the road [1], [2]. The aerodynamic drag force expression is presented in relation 1, where C_D represent the drag coefficient, A is frontal area of the helmet body, given in m^2, V airflow velocity, given in m/s and ρ is air density in kg/m^3.

$$F_D = \frac{1}{2}\rho \cdot C_D \cdot A \cdot V^2 \tag{1}$$

The expression of the drag coefficient is deducted from the drag force expression, as can be seen in relation 2:

$$C_D = \frac{2 \cdot F_D}{\rho \cdot A \cdot V^2} \tag{2}$$

The lift force is formed by the airflow that act in perpendicular to the forward direction. In relation 3 is presented the expression of this force.

$$F_y = \frac{1}{2}\rho \cdot C_L \cdot A \cdot V^2 \tag{3}$$

The deducted relation of lift coefficient - C_L is presented in relation 4:

$$C_L = \frac{2 \cdot F_y}{\rho \cdot A \cdot V^2} \tag{4}$$

In this study the lift force is neglected because in this study the helmet is placed on the generic motorcyclist head. The resulting values would not be relevant to establish the aerodynamic performance.

2.2 CFD model

The mathematical model uses are the Reynolds Averaged Navier Stokes (RANS) equations for incompressible fluid flow and the k-ω turbulence model. It is used this turbulence model because include a superior performance for complex boundary layer flows.

These equations are used in many areas of fluid mechanics to process modelling, such as: fluid flow through pipes, the movement of the ocean currents, aerodynamics of aircrafts or cars [11]. The RANS equations give the velocity of a particle, not the position of the particle. The expression of the continuity equation is presented in relation 5, [11].

$$\frac{\partial \overline{v}_j}{\partial x_j} = 0 \tag{5}$$

The expression of the momentum equation is presented in the next relations 6:

$$\frac{\partial}{\partial x_j}\left(\overline{v}_j \overline{v}_i\right) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial \overline{p}}{\partial x_j} \left[\mu\left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i}\right) + \mu_t\left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i}\right) - \frac{2}{3}\rho \overline{k}\,\delta_{ij}\right]. \tag{6}$$

Where, the mean of the airflow velocity is $\overline{v}_j$, the average of the air pressure is given by the coordinates are $x_{i,}$ $\overline{p}$ and turbulent viscosity is μ_t .The expression of the averaged turbulence kinetic energy is presented in relation 7:

$$\overline{k} = \frac{\overline{v_i^{'} \cdot v_i^{'}}}{2}. \tag{7}$$

2.3 Reynolds number

The Reynolds number is a dimensionless size that shows how much the fluid flow is affected by his inertial forces [5], [9]. The value of the Reynolds number can be calculated by using the relation 8:

$$R_e = \frac{\rho \cdot V \cdot L}{\mu} \tag{8}$$

Where, air density is ρ= 1.225 kg/m^3, air velocity V= 33.34 m/s, dynamic air viscosity $\mu=1{,}802 \cdot 10^5$ kg/m·s, and the length of the helmet L=0.261 m. To establish the flow regime around the helmet body the Reynolds number is calculated according to the relation 8. The calculated value of the Reynolds number is $1.56 \cdot 10^5$, value that indicates the turbulent flow regime, in case of external flow over sphere surface.

3. CAD MODEL

Using the advanced modelling techniques, the CAD model of the helmet is modelled in SolidWorks consisting of two parts: the outer shell and the visor shield [6], [7]. Due to the multiple possibilities of advanced modelling this software was chosen to design the helmet model.

3.1 Outer helmet design

The outer shell geometry of the helmet is generated by using a drafting technique called lofting. The sketch profile and guide of the helmet geometry is presented in Figure 2. Dimensions of the helmet model are chosen in according to a real human head.

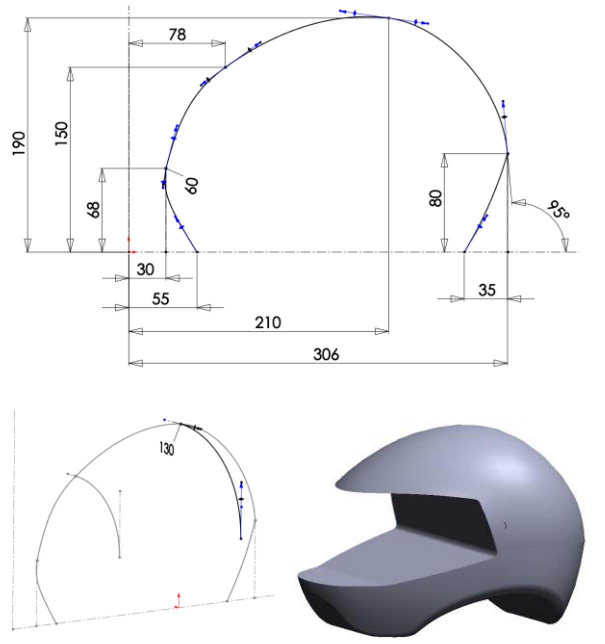
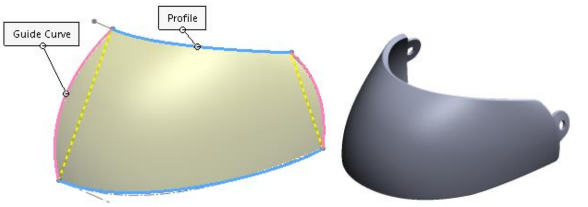


Figure 2 Sketch dimensions and shape of the outer helmet.

To keep a symmetry of the model, the outer shell is half modelled, then is mirrored. The final model of the outer helmet is presented in Figure 2.

3.2 Visor shield

The visor shield is modelled according to the helmet model to close the internal space.



a) Sketch profile and guide b) Final model

Figure 3 Visor shield model.

The sketch profile is lofted following the guide curves, as can be seen in Figure 3. The resulted surface is thickened to obtain the visor shield model.

3.3 Helmet model

The helmet model provided to CFD analysis is given by the assembly of the outer shell and the visor shield. The visor shield is mated with outer shell through the lateral support, which allows handling in the open or closed position.

The orthogonal projection and overall dimensions of the helmet are showed in Figure 4. The position of the helmet is chosen like as it positioned when the biker participates on the speed race.

Additional to the helmet model, the upper model of the motorcyclist's body was added, as can be seen in Figure 6.

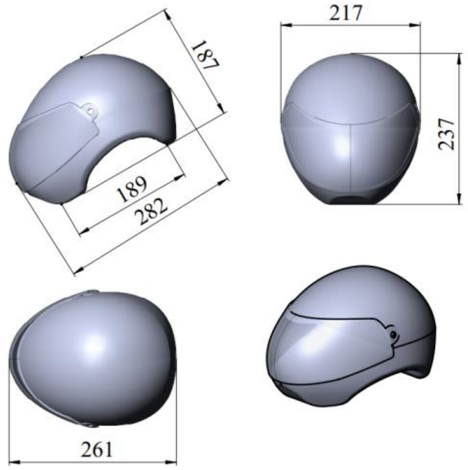


Figure 4 Overall dimensions of the helmet.

4. CFD APPROACH

In this section are presented the steps necessary to perform the CFD evaluation of the helmet in five geometrical configurations. In first case the helmet surface is smooth, as can be seen in Figure 4, in the second case on the helmet surface are made perpendicular protruding to the advance direction, in third case on the external surface of the helmet are made parallel grooves to the advance direction, in the fourth case on the helmet surface are generated the dimples. In the last configuration on the smooth base model are added in the rear side an air deflector composed from two elements.

To have a simulation environment more realistic, the generic body of the motorcyclist in ride position is added to the CFD study, as can be seen in Figure 6.

4.1 CFD setup

Following in this section the helmet model is added in FlowWorks environment. All airflow simulations to establish the aerodynamic drag and drag coefficient is performed at 33.34 m/s velocity.

To accuracy view the behaviour of the airflow around the helmet geometry the computational domain has a volume of 0,75 m^3 (width = 1 m, length =1.5 m and height=0.5 m).

The computational volume is meshed in 695180 cells, of which 31076 cells are in contact to the helmet surface. The adjacent air layers of the helmet geometry are denser to better view of the air streamline behaviour.

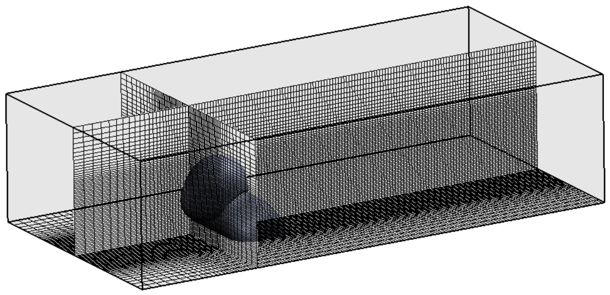


Figure 5 Computational domain and mesh grid distribution.

The computational domain and grid discretisation are presented in Figure 5.

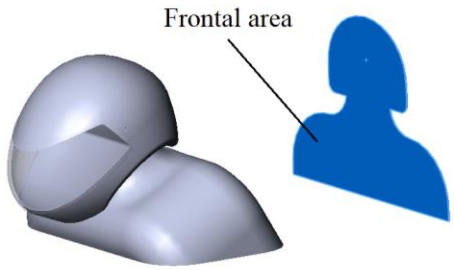


Figure 6 Frontal area of the helmet and the generic biker.

Drag coefficient for all simulation from this study are calculated according to relation 2, presented before. The frontal area of the helmet body, presented in Figure 6 is 0.703 m^2.

4.2 CFD results

In this section comparative results are presented by studding the air streamline behaviour around the helmet and motorcyclist generic body, the pressure distribution on the model surface and the drag force and coefficient obtained.

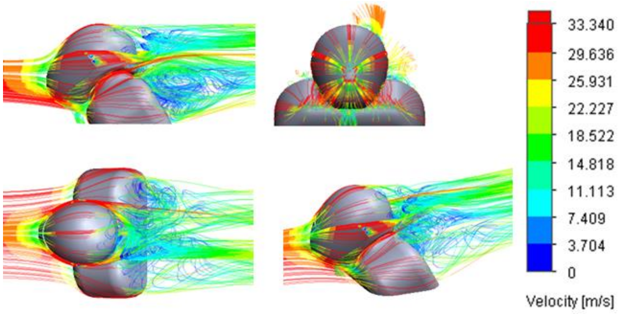


Figure 7 Airflow velocity distribution for the first simulation case.

To a better evaluation the distribution of the airflow and of the air pressure on the helmet and generic body are presented in orthogonal and axonometric projection for all design configuration of the helmet, like as in Figure 7.

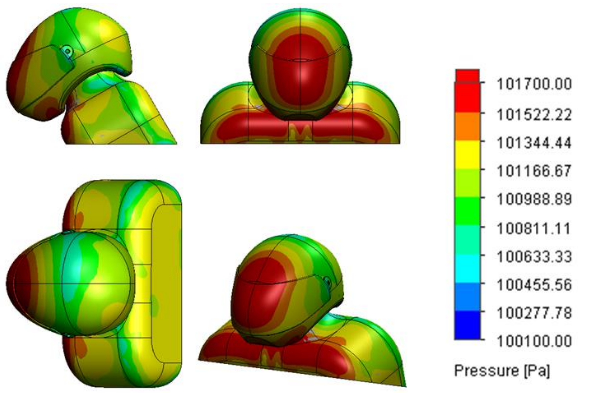


Figure 8 Airflow pressure distribution on the helmet surface for the first simulation case.

It can be observed that the airflow velocity decrease on the rear side of the helmet model generating a turbulent flow, the air velocity decreasing considerably.

The air pressure distribution on the helmet surface and the generic body are presented in Figure 8, where can be observed that the high pressure is concentrated on the frontal side of the helmet and motorcyclist body. The airflow velocity distribution resulted in second simulation case is presented in Figure 9.

Compared to the first simulation case results it can be observed that the turbulent zone created in the rear of helmet and generic body is smaller than the zone created at the smooth helmet model.

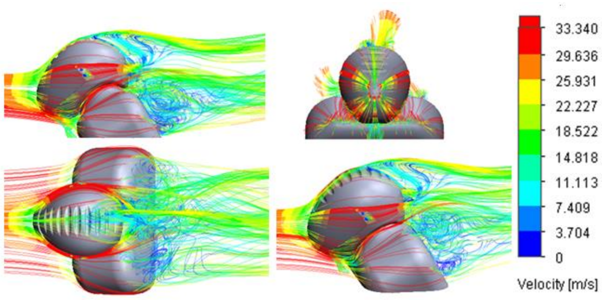


Figure 9 Airflow velocity distribution for the second simulation case.

The protruding created on the helmet surface have a positive airflow influence of an airflow. The air pressure distribution it also has a maximum intensity in the protrusions area, as can be seen in Figure 10.

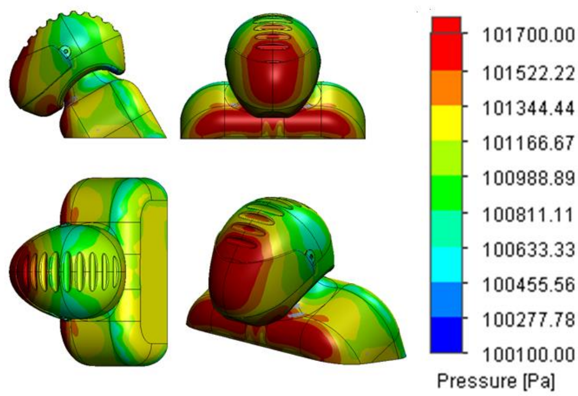


Figure 10 Airflow pressure distribution on the helmet surface for the second simulation case.

In third simulation case the helmet the grooves are parallel oriented to air flow. Because of this orientation the airflow is smoother than the previous simulations. In the rear side of the helmet, the low velocity volume is smaller, with an only turbulent volume behind the body motorcycle.

The airflow is oriented by the groove's geometry, maintaining the high velocity of the air around the helmet body. On the helmet surface are designed with nine grooves with 5 mm radius, filleted at 4 mm, as can be observed in Figure 12.

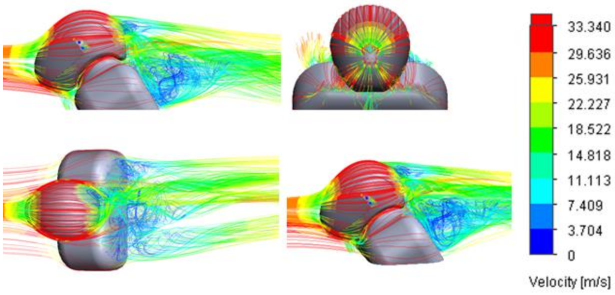


Figure 11 Airflow velocity distribution for the third simulation case.

Airflow velocity distribution for this case is presented in Figure 11. The pressure distribution is oriented in the frontal area of the helmet model and front of generic motorcycle body, as can be seen in Figure 12.

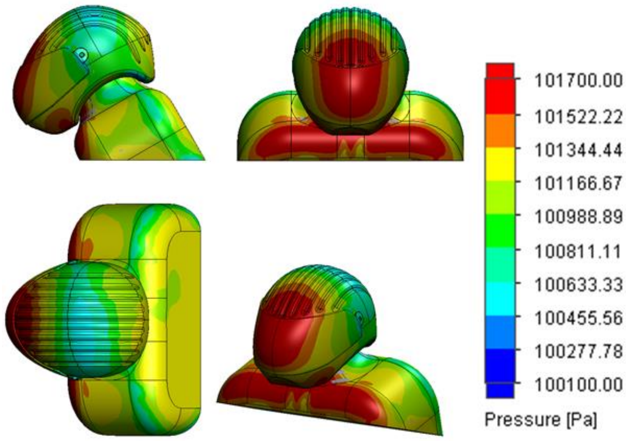


Figure 12 Airflow pressure distribution on the helmet surface for the second simulation case.

Dimpled helmet surface CFD simulation results are presented in fourth simulation case. In Figure 14 can be observed that the dimples are arranged in eleven rows placed on the helmet spherical surface. The distance between the rows dimples is 15 mm and the dimples radius is 8 mm.

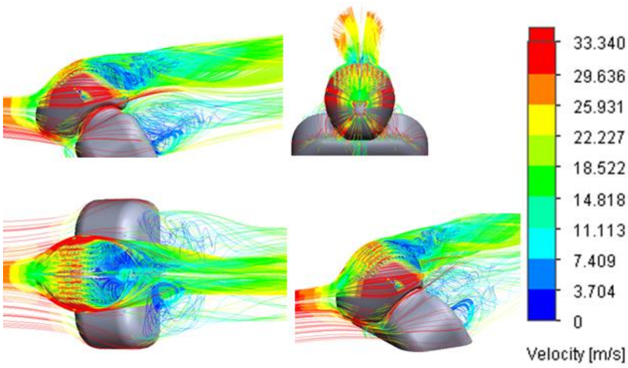


Figure 13 Airflow velocity distribution for the fourth simulation case.

In this case the turbulent area is formed in rear side close to the helmet surface. The air pressure distributions on the dimpled helmet surface are presented in Figure 14.

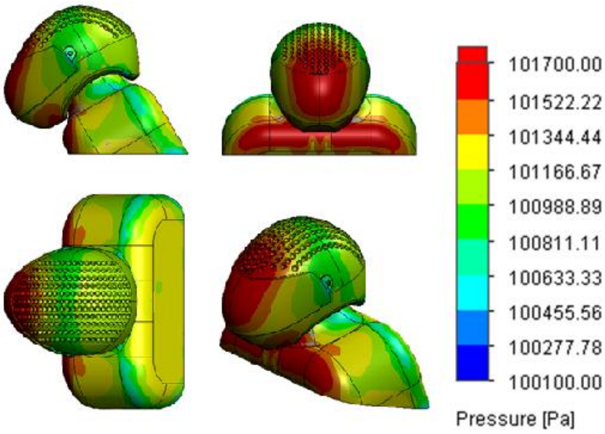


Figure 14 Airflow pressure distribution on the helmet surface for the second simulation case.

In the last simulation case on the rear side of the helmet are added two rib surfaces, presented in Figure 15.

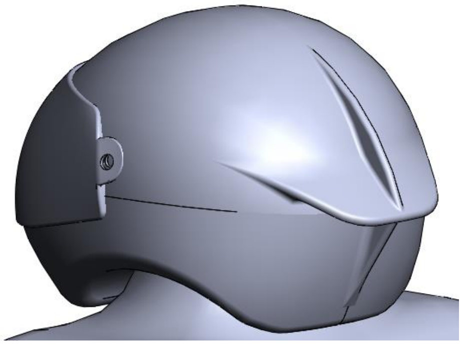


Figure 15 Rib surfaces added on rear side of the helmet.

The turbulent area is divided in two zone, first zone on rear side, close to the helmet and second largest turbulent zone caused by the generic body geometry, as can be viewed in Figure 16.

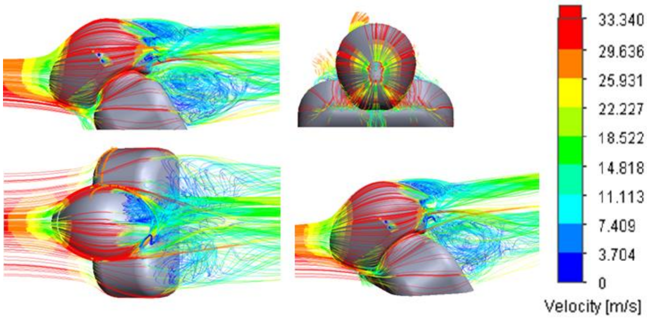


Figure 16 Airflow velocity distribution for the last simulation case.

The air pressure distribution on the helmet surface is presented in Figure 17 for last simulation case.

The calculated values of the drag force and drag coefficient are presented in table 1, for each simulation cases. Analysing the calculated results can be observed that the third and last simulation model have a better result.

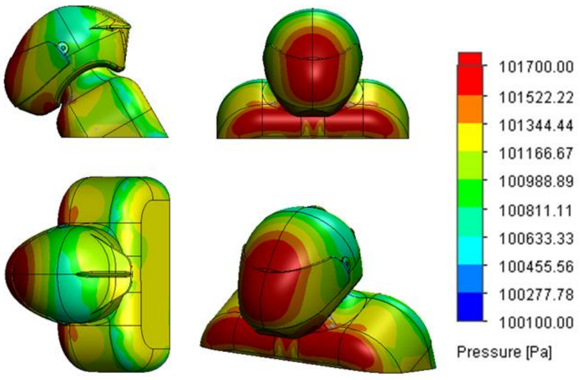


Figure 17 Airflow pressure distribution on the helmet surface for the second simulation case.

Table 1

Resulted values of drag coefficient and drag force.

Model	Drag force [N]	Drag coefficient
1	21.095	0.431
2	22.489	0.460
3	20.843	0.426
4	23.170	0.474
5	20.928	0.428

5. CONCLUSIONS

In this paper, the aerodynamic evaluation of the redesigned helmet model is studied to optimize its aerodynamic capability. To determine the optimal aerodynamic helmet model, five evaluation cases are performed. Current study can be a start point to manufacturing the helmet customized design having an aerodynamic shape. Comparing the real situation to a simulation it can be highlighted that in the real situation the air is fixed, and the motorcyclist have an initial velocity, while in the simulations the helmet and generic body are fixed, and the air have a velocity. Because of this, may occur some difference when is compared the obtained results in a wind tunnel whit the results obtained from numerical CFD simulations.

REFERENCES

[1] Bodea, S., Prodan, C., & Scurtu, I-L., (2018, October). *The Aerodynamic Study of a Body Truck.* In International Congress of Automotive and Transport Engineering, pp 73-79, Springer, Cham.

[2] Balcau, M. (2021). *Aerodynamic study of a car towing a motorcycle trailer.* Journal of Automotive Engineering, ISSN 2457 – 5275, Vol. 27-1, pp 31-38.

[3] Czyż, Z., Karpiński, P., & Sevdim, T. (2018). *Numerical analysis of the drag coefficient of a motorcycle helmet.* Applied Computer Science, 14(1).

[4] Dubey, P., Pramod, M. Y., Kumar, A. S., & Kannan, B. T. (2020, November). *Numerical simulation of flow over a racing motorbike using OpenFOAM.* In AIP Conference Proceedings (Vol. 2277, No. 1, p. 030029). AIP Publishing LLC.

[5] Jurco, A. N. (2021). *Study of the influence of roof luggage box on a vehicle aerodynamics.* Journal of Automotive Engineering, ISSN 2457 – 5275, Vol. 27-22, pp 15-38.

[6] Kiraly, A. (2019). *Optimizing the forms and dimensions of a field cultivator working knives*. Journal of Industrial Design and Engineering Graphics, 14(1), pp 71-74, Online ISSN 2344-4681.

[7] Kiraly,A., (2002) *Grafica Inginereasca*, Editura U.T.PRESS, ISBN 973-8335-35-3, Cluj-Napoca.

[8] Malerba, M., & Conti, P. (2017). *Influence of the rider seating position on motorcycle aerodynamic performance.* Int. J. Eng. Sci. Innov. Technol, 6(1), pp 90-97.

[9] Ljubomir, M., Dalibor, B., Andrija, B., & Hrvoje, K. (2018). *Aerodynamic Design of a Solar Road Vehicle.* International Journal of Automotive Technology, 19(6), pp 949-957.

[10] Sathish Gandhi, V. C., Kumaravelan, R., Ramesh, S., Venkatesan, M., & Siva Rama Krishnan, M. (2014). *Aerodynamic design and analysis of motorcycle helmet with anti-glare visor.* Tehnički glasnik, 8(1), pp 97-101.

[11] Scurtu, I. L. (2021). *Aerodynamic performance evaluation for a vehicle structure equipped with a bicycle rack.* Journal of Automotive Engineering, ISSN 2457 – 5275, Vol. 27-22, pp 5-14.

Authors:

Assoc. Prof. Ph.D. Sanda BODEA, Technical University of Cluj-Napoca, Faculty of Automotive, Mechatronics and Mechanical Engineering, Department of Automotive Engineering and Transports, E-mail: sanda.bodea@auto.utcluj.ro

Lecturer PhD. Eng. Iacob-Liviu SCURTU, Technical University of Cluj-Napoca, Faculty of Automotive, Mechatronics and Mechanical Engineering, Department of Automotive Engineering and Transports, E-mail: liviu.scurtu@auto.utcluj.ro