ISSN 2744-1741
Defense and Security Studies *Original Research*
Vol. 2, May 2021, pp.79-85
https://doi.org/10.37868/dss.v2.id172

Analysis of the influence of 5.56 mm projectile shape on drag coefficient using CFD

Tarik Šabanović[1]
[1] Mechanical Engineering Faculty, University of Sarajevo, Bosnia and Herzegovina

*Corresponding author: : tariksabanovic2203@gmail.com

Abstract
This paper describes the computer dynamics of fluids, which was used to determine the influence of certain geometrical characteristics of the projectile on the coefficient of drag force. The first section is an introduction and describes the projectile, the aerodynamic forces acting on the projectile with special reference to drag. The second section was reserved for a review of projectile parameters that affect drag, primarily the slenderness of the ogive and the frustum, and the shape of the ogive, and the angle of the frustum. This section also defines drag in more detail. The third section describes the mathematical model of fluid in supersonic flow and gives the equations for the mathematical model, used in simulations and software package Ansys fluent. This section was also reserved for describing the emergence of a physical model and the verification of the numerical simulation model. The fourth section presents and describes the model of CFD analysis of 5.56 mm projectiles: SS109, M855, L110, and M856, and the comparison of projectiles by geometry. The fifth section comprises the analysis of the results. In the sixth section, a conclusion is given.

Keywords: CFD, small caliber ammunition, aerodynamics, drag force

1. Introduction

Small caliber ammunition includes ammunition up to 12.7 mm caliber and is typical ammunition for weapons for personal protection. It is fired from direct fire weapons that have high initial velocity and flatter trajectory. Ammunition, in the general case, consists of a projectile and a propulsion group. Projectiles of small caliber ammunition generally do not contain explosive charges and fuse, and use only kinetic energy upon impact.
Due to the movement of the projectile through the air, the air environment is disturbed; the air is forced to move and the parameters of the state changes which requires a certain amount of kinetic energy.
The force with which the air acts on the projectile, which is the resultant of all forces acting on the surface of the projectile (namely the force of pressure acting normally on surface and friction force acting tangentially to the surface), and has a velocity in the opposite direction, is called the drag.
In aerodynamics, in order to avoid difficulties with dimensions, we work with dimensionless quantities [1]. Three reference quantities are used by which other quantities can be used to make it dimensionless and these are: reference length, reference area and reference pressure (dynamic pressure in undisturbed air flow). The force of drag that divided with the product of the reference area and the reference pressure gives a dimensionless quantity called the aerodynamic drag coefficient.
Empirical formulas were obtained on the basis of theoretical calculations and experimental tests which can be used to predict the pressure distribution, i.e. for the aerodynamic coefficient estimation, on a specific part of the projectile for a defined range of flow velocities. Based on these empirical expressions and the development of computer technology begins the era of computer programs for prediction of aerodynamic

coefficients based on the geometric characteristics of the projectile.
The rapid advancement of computer technology has introduced a new discipline called computational fluid dynamics (CFD - computational fluid dynamics) which represents the third method in aerodynamics completing the previous two, pure theory and pure experiment. Computer fluid dynamics is the art of approximating major partial differential equations which describe fluid flow by simple algebraic expressions [2]. The end result of the application the CFD on the projectile flow is a picture of the flow field around the projectile which allows for a detailed analysis of the influence of individual parameters on the pressure (velocity) distribution and thus on aerodynamic drag coefficient.

2. Influence of projectile shape on drag force coefficient

In addition to the flow velocity (represented through Mach number), the aerodynamic drag coefficient in large extent depends on the geometric characteristics of the projectile. Slenderness of the projectile implies the ratio of the length of the projectile to the reference diameter of the projectile. Figure 1. shows the dependence of the drag coefficient on the Mach number for the three shapes of projectiles. The upper curve represents the drag coefficient for a spherical projectile, the middle curve corresponds to the drag coefficient for the projectile without the rear cone and with front part made with small slenderness, and the lower curve is the drag coefficient of a modern projectile. In terms of projectile slenderness, the sphere has the lowest slenderness and the modern projectile has the greatest slenderness.

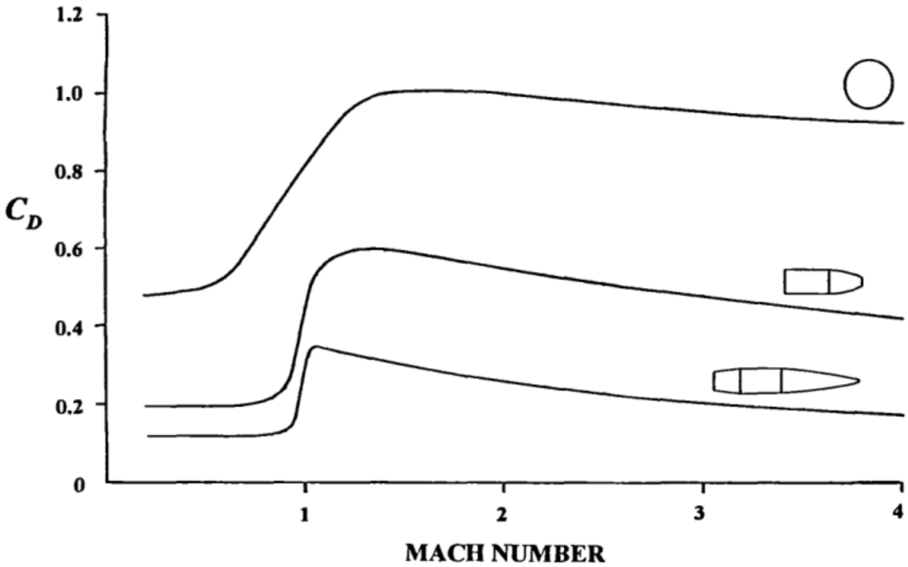

Figure 1. Coefficient of drag as a function of Mach number depending on the shape [3]

All three curves shown in Figure 1. have the same trend. In the subsonic area the drag coefficient is constant (middle and lower curve) or changes very little, in the transonic area rises sharply to its maximum value and begins to slowly decline. The trend of a slight decrease in the drag coefficient with an increase in the Mach number continues in the supersonic range. It can also be noticed that projectiles with higher slenderness have smaller drag coefficient and that their maximum value of the drag coefficient occurs for the values Mach's number closer to one.

3. Mathematical model

Aerodynamics is a theoretical and experimental science, and represents a branch of fluid mechanics. Theoretically approach is based on the analytical solution of mathematical models of air flow. An analytical solution gives a complete insight into the physics of a problem, and once determined the analytical solution is suitable for the analysis of the influence of individual parameters in the mathematical model. Most airflow problems are described by nonlinear partial differential equations, which do not have a general analytical solution. This is especially true for turbulent flow, which due to the stochastic nature of that flow cannot be described analytically. With the development of computers, conditions were created for the numerical solution of mathematics model. Each simulation is based on a mathematical model, which denotes a mathematical notation physical model.

The mathematical model includes the following assumptions:

- Air is a continuum.
- Air is considered a homogeneous mixture of gases.
- The physical properties of air are the same in all directions - the air is isotropic.
- Air is a single-phase fluid.
- Mass forces are neglected.

The continuum assumption implies that the density of the fluid is large enough that even the infinitesimally small element of the fluid contains a satisfactory number of particles so it is possible to specify average velocity and average kinetic energy. That way they can determine flow characteristics (velocity, pressure, temperature, etc.) at each fluid point [4].
Continuity behavior can be described by transport equations based on the basic laws of mass conservation, momentum, and energy. The equations derived from the given laws are presented in integral form for an arbitrarily selected part of the continuum, the volume *Ω* bounded by a closed area - the limit of the control volume *dΩ* (Figure 2). The surface element *dS* is defined by the unit vector of the normal $\vec{n}$ [4].

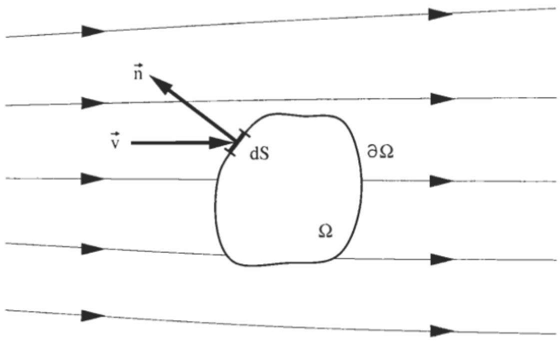

Figure 1. Control volume [4]

Law of mass conversation states:

$$\frac{\delta}{\delta t}\int_{\Omega} \rho d\Omega + \oint_{\delta\Omega} \rho(\vec{v} * \vec{n})\, dS = 0 \tag{1}$$

Law of momentum conservation implies:

$$\frac{\delta}{\delta t}\int_{\Omega} \rho\vec{v}\, d\Omega + \oint_{\delta\Omega} \rho\vec{v}(\vec{v} * \vec{n})\, dS = \int_{\Omega} \rho\vec{f_e}\, d\Omega - \oint_{\delta\Omega} \rho(\vec{n})\, dS + \oint_{\delta\Omega} (\bar{\bar{\tau}} * \vec{n})\, dS \tag{2}$$

Law of energy conservation states:

$$\frac{\delta}{\delta t}\int_{\Omega} \rho E\, d\Omega + \oint_{\delta\Omega} \rho E(\vec{v} * \vec{n})\, dS = \oint_{\delta\Omega} k(\nabla T * \vec{n})\, dS + \int_{\Omega} \left(\rho\vec{f_e} * \vec{v} + \dot{q_h}\right) d\Omega - \oint_{\delta\Omega} p(\vec{v} * \vec{n})\, dS + \oint_{\delta\Omega} (\bar{\bar{\tau}} * \vec{v})\, dS \tag{3}$$

Here v is velocity of airflow, p is pressure, ρ is density, T is temperature, E is total energy, and $\bar{\bar{\tau}}$ is stress tensor. The system of equations describing high-velocity flow cannot be solved analytically. To solve this system it is necessary to introduce a simplification or a problem or the equations. With the development of computers and computer fluid dynamics (CFD) numerical solution of equations describing the flow can be obtained.

4. Solving equations using CFD

Numerical solution of a mathematical model that describes the flow in the considered problem consists of three steps. In the first step, the area is discretized. Result of discretization of space is called a geometric grid. On the defined geometric grid it is necessary to discretize the partial differential equations of the mathematical model, respecting specific boundary conditions. The discretization of the equations is carried out by some of

the known methods (finite volume method, finite element method, finite difference method etc.). Result of discretization of partial differential equation on given geometric web is a system of algebraic equations. The nonlinear system of equations is solved iteratively by a procedure that involves solving a system of linear algebraic equations. A numerical solution is obtained, followed by its analysis, which includes a display of scalar, vector and tensor fields, integration of flow, force, moments, thermal flows, etc., and a diagram of the desired quantities.
One of the main prerequisites for using a CFD model is validation model that takes place in several steps [4]:

- Checking the program code.
- Comparison of the obtained results with the available experimental data (predicting measurement errors).
- Sensitivity analysis and parametric study.
- Application of different models, geometry and initial/boundary conditions.
- Reports on findings, model limitations, and parameter settings.

The aerodynamic drag coefficient of the sphere was used here to verify the flow model for which experimental data are available [3]. The physical model of this case, the sphere in air flow, is shown in Figure 3.

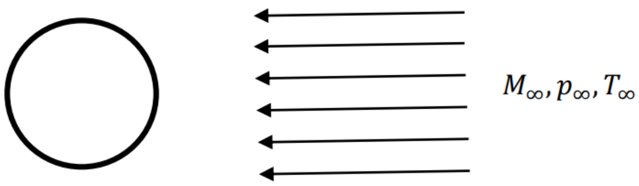

Figure 2. Sphere in air flow

The diameter of the sphere in the simulations was 2.54 mm and the flow was simulated at different velocities (2; 2.5; 2.64 and 3 Mach). Pressure and temperature of free air flow in all simulations were: p_∞ = 101325 Pa and T_∞ = 300 K.

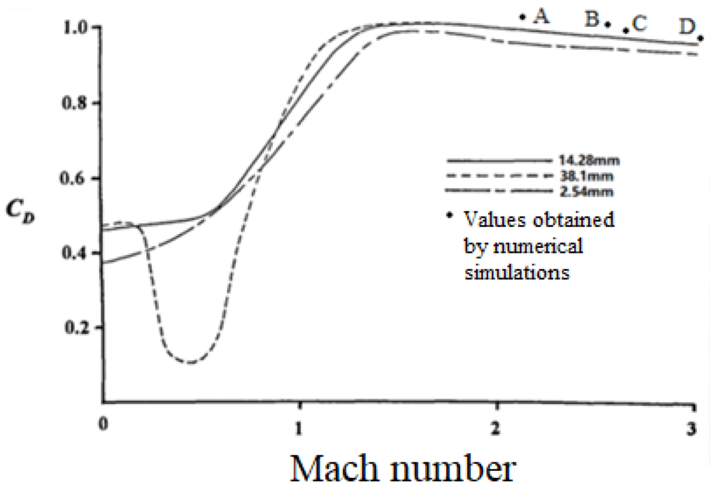

Figure 3. Drag coefficient as a function of Mach number for spheres with different diameters

Figure 4 shows a comparison of experimental and numerical results obtained. For free air flow velocity of Ma = 2.64 (point C) by numerical simulation the value of the coefficient of drag for the sphere, with diameter 2.54 mm, was Cd =0.95, which is about 1% higher than the value obtained experimentally. Deviation (relative difference) observed were as follows: for M = 2 the deviation was 3%, for M = 2.5 the deviation was 4%, for M = 2.64 the deviation was 5% and for M = 3 the deviation was 4%.
Comparing the values, very good agreement was observed between the experimental values and values determined using the Ansys Fluent software package.

5. Analysis of results

The flow for four 5.56 mm projectiles with different (supersonic) velocities was considered in the research. The following assumptions were adopted for all simulations:

- Working fluid is air, an ideal gas, which is modified in accordance with compressibility and changes in thermo-physical characteristics with temperature.
- Density and viscosity depend on temperature, and c_p and thermal conductivity are considered constant.
- The parameters of free air flow were: $p_\infty = 101325$ Pa and $T_\infty = 300$K.
- The flow around the projectile is considered compressible and turbulent.
- Discretization of the spatial domain was performed by non-uniform unstructured mesh.
- A "density-based solver" was used, developed for compressible high-speed flows.
- The equations were linearized in implicit form, i.e. for given variable, unknown in each cell was calculated using relations that include existing and values from adjacent cells.

The flow field around the projectile and the aerodynamic drag coefficient were determined by using FLUENT program for specified conditions.
Four small-caliber projectile were used for num. simulations; for these projectiles experimentally determined values of aerodynamic drag coefficient [6] were known. Models used were:

- 5.56 mm SS109,
- 5.56 mm M855,
- 5.56 mm L110,
- 5.56 mm, M856.

Simulations with free flow for each of the projectiles was performed for velocities in the range of 1.2 up to 3 Mach for the following values of Mach numbers: 1.2; 1.5; 1.7; 2; 2.5; 2.64 and 3.
Although all observed projectiles consist of a front part, a cylindrical part and a rear part, on projectiles SS109 and M855 the rear part have the shape of a bevelled cone and on projectiles L110 and M856 the rear part have the edged shape. For this reason, two groups will be formed in the analysis of results. The first group consists of projectiles with a conical rear part, SS109 and M855, and the second group consists of projectiles with a edged rear end, L110 and M856. The results of the simulations, in the form of the drag coefficient values, for the first group of projectiles are shown in Figure 5. In Figure 5. are also presented the results of experimental tests for projectiles SS109 and M855.

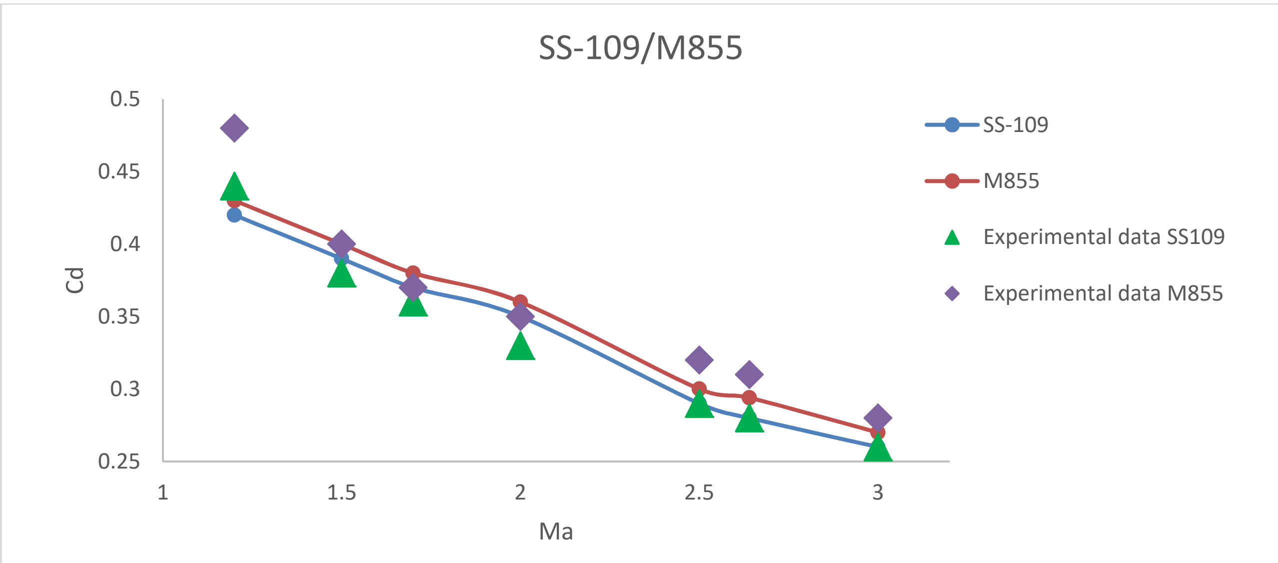

Figure 4. Drag coefficient vs Mach number for projectiles 5.56 mm, models SS109 and M855 [5]

From the figure 5. it can be seen that the agreement of experimental and numerical data in supersonics is very good and the maximum rel. error is 2%. Agreement in transonics is somewhat lower, with rel. error of 10%. It can be noticed that the 5.56 mm SS109 projectile has a lower drag coefficient than the 5.56 mm projectile M855 in the supersonic range. Observing the pressure distribution [6] around the projectiles 5.56 mm, SS109 and M855, shows that the SS109 projectile has a lower maximum pressure at the top of the front part in relation to the pressure at the top of the front part of the projectile 5.56 mm, M855; this also results in less drag on this part of the projectile. If we compare the pressure distribution at the rear for these two projectiles, it can be seen [6] that the M855 projectile has a lower bottom pressure than the SS109 projectile at the same velocity flow, which results in a higher coefficient of drag for this part and in accordance with the theoretical considerations 5.56 mm projectile SS109 has a slightly larger slenderness of the front part, significantly higher slenderness of the rear end and approximately the same overall slenderness in relation to the 5.56 mm M855 projectile; although it has larger angle of inclination of the rear cone, which is unfavorable, it can be stated that it is better aerodynamically shaped from a 5.56 mm projectile, M855, for flight at supersonic speeds.
Simulation results, drag coefficient vs Ma number, for the second group of projectiles, projectiles 5.56mm, L110 and M856, are shown in Figure 6. together with the experimental data.

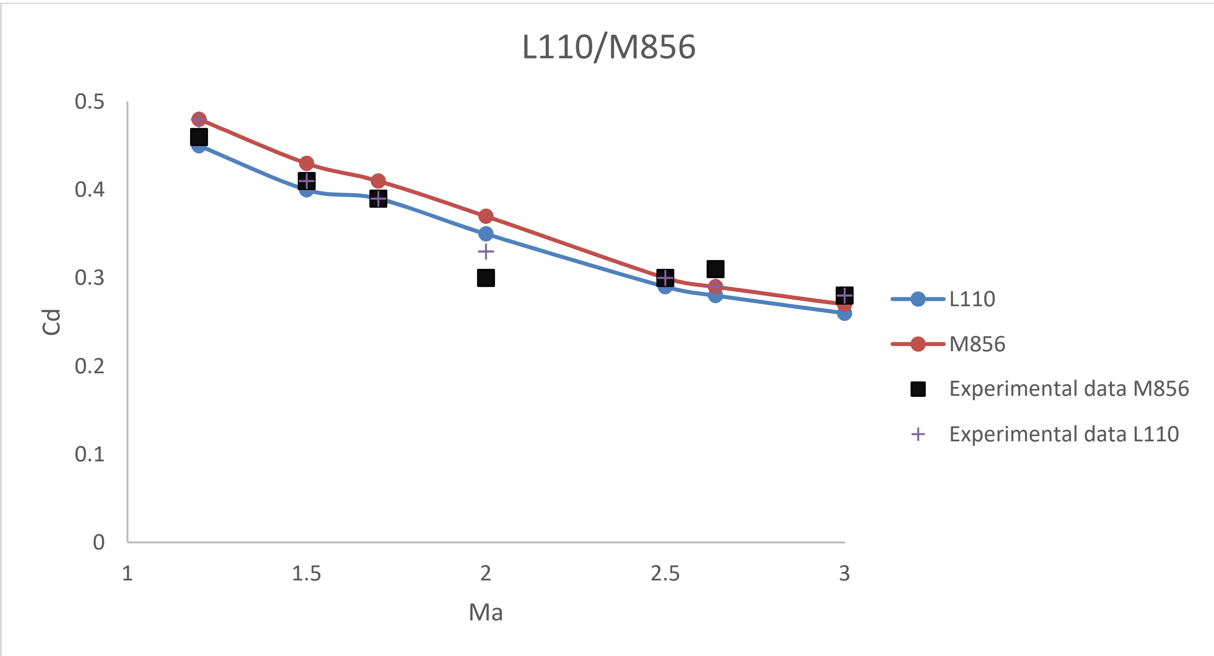

Figure 5. Drag coefficient vs. Mach number for 5.56 mm projectiles, models L110 and M856 [5]

On Figure 6. it can be seen that the agreement of the experimental with the numerical results is very good and that many points actually match. Observing the pressure distributions [6] flowing around the front of the 5.56 mm projectile L110, and around the front part of the projectile 5.56 mm M856, it can be noticed that the pressure at the top of the front of the projectile M856 is less than the pressure at the top of the projectile L110, which also results in less drag on this part. The front of the L110 projectile is also slimmer from the front part of the M856 projectile, resulting in a lower drag coefficient of the front part of the projectile. Considering the pressure distribution at the rear of the L110 projectile and at the rear projectile M856, a lower pressure is observed at the rear and at the bottom of projectile 5.56 mm L110. Although the M856 projectile has a greater slenderness of the rear and smaller coefficient of drag at the bottom, it can be concluded that the greatest influence on the total coefficient of drag in the supersonic area have the shape and slenderness of the front part of the projectile. Figure 6. shows that the 5.56 mm projectile L110 has a lower drag coefficient than projectile 5.56 mm M856 in the considered velocity range. In the range of velocities up to Mach 1.5, the trend of the L110 projectile drag curve is the same as the trend of the coefficient curve for projectile M856. At the flow rate M = 1.7, the value of the drag coefficient projectile L110 approaches the value of the drag coefficient of the projectile M856, where the rel. difference is less than 5%. With a further increase in the flow rate, this difference decreases and for M = 3 it is less than 4%.
More data from this research can be found in reference [6].

6. Conclusions

In this paper, the influence of projectile shape for 5.56 mm ammo on aerodynamic drag coefficient is analyzed at supersonic velocities, using computational fluid dynamics. During these analyses, the following was found:

- Shorter, blunt projectiles have a higher drag coefficient than slender projectiles.
- In the supersonic range, the length of the fornt part has a large influence on the drag coefficient and with increasing slenderness of the front of the projectile comes a significant reduction of the coefficient of drag.
- At higher (supersonic) velocities, the front of the projectile shaped like a secant has the lowest drag, less so than the conical and tangential front shapes.
- The rounded tip of the projectile will result in less drag than the blunt tip.
- Increasing the slenderness of the rear part of the projectile reduces the coefficient of drag and for supersonic velocities the optimal length of the rear part is between 0.5 and 1 caliber.
- The optimal value of the rear cone angle, at supersonic speeds, is 7.

In the Ansys FLUENT software package, based on a theoretical consideration of the supersonic projectile flow, a system of equations is chosen to describe the flow of air around projectiles. The domain in which the calculation is performed has been defined and boundary conditions have been set. From the offered software options for solving equations, the solving model was chosen and for each simulation the initial conditions were defined. Before num. simulations for projectiles, the model was verified based on the aerodynamic drag coefficients of the sphere for different flow velocities.

Analysis of the influence of projectile geometrical characteristics on the drag coefficient at zero yaw angle was performed based on the results of numerical simulations. They are simulated for 5.56 mm projectile, SS109, M855, L110 and M856 models. These projectiles have similar outer shape: front part in the shape of a secant, cylindrical part and end part. They differ in the slenderness of individual parts of the projectile, in the slenderness of the whole projectile, by the radii of the ogive, by the shape of the rear part of the projectile.

For each of the projectiles, 7 simulations were performed for Ma = 1.2; 1.5; 1.7; 2; 2.5; 2.64 and 3.

The greatest influence on the drag of the projectile has the shape and slenderness of the front part of the projectile; 5.56 mm SS109 and M855 projectiles have lower slenderness compared to projectiles 5.56 mm L110 and M856. Regardless of the slenderness and how the SS109 and L110 projectiles are different they show similar aerodynamic characteristics in terms of the influence of their shape on drag; subsequent analysis of the results concluded that these two projectiles in some parts of the supersonic regime have the same drag. The drag analysis showed that the SS109 and L110 projectiles have less drag compared to projectiles M855 and M856.

References

[1] S. Janković, Aerodynamics of projectiles, Faculty of Mechanical Engineering, Belgrade, 1979.

[2] Anderson, J., Modern compressible flow, McGraw-Hill Publishing, Singapore, 1990.

[3] McCoy, R., Modern exterior ballistics: The launch and flight dynamics of symmetric projectiles, Schiffer Publishing Ltd, 2nd edition, Pennsylvania, 2012.

[4] Blazek, J., Computational fluid dynamics: Principles and appications, Elsevier Science Ltd, Oxford, UK, 2001.

[5] Sabanovic T., “Analysis of influence of projectile shape on its drag coefficient for 5,56 mm ammunition using CFD”, Mechanical Engineering Faculty, University of Sarajevo, 2021.