Advances in Military Technology
Vol. 17, No. 2, 2022, pp. 397-410
ISSN 1802-2308, eISSN 2533-4123
DOI 10.3849/aimt.01743

CFD Simulation of the Gas-Operated Weapon Drive Applied to the UK-59 Machine Gun

V. D. Tien, S. Procházka, V. Horák*, R. Vítek, and B. T. Phan

University of Defence, Brno, Czech Republic

The manuscript was received on 8 June 2022 and was accepted
after revision for publication as research paper on 22 November 2022.

Abstract:

The flow characteristics of propellant gases inside the gas cylinder of the weapon gas-operated mechanism, applied to the UK-59 machine gun, are studied. Computational Fluid Dynamics (CFD) in ANSYS Fluent is used for the simulation of flow through the gas port from the barrel bore to the gas cylinder and back. The study's findings are presented along with the piston law of motion and thermodynamic quantities of gases inside the cylinder, specifically the pressure, temperature, and velocity distribution of gases in the cylinder for various piston displacements. The experiment was carried out to specify the pressure of the gases in the cylinder, as well as the displacement of the piston. The experimentally obtained values are compared to the simulation results with a very good agreement.

Keywords:

CFD simulation, ANSYS Fluent, gas-operated weapon, discharge coefficient, propellant gases, gas cylinder

1 Introduction

The gas drive system of gas-operated automatic guns works on the premise of utilizing the energy created by high-pressure propellant gases while firing to operate the action, see [1] for more details. The fluid flow in the gas cylinder is a complicated process characterized by a high pressure, temperature, and velocity of propellant gases during a short period of time. The propellant gases flow into the cylinder through the gas port as soon as the bullet passes through it during the shot. To demonstrate this process, Fig. 1 depicts the schematic of a gas-operated weapon.

* *Corresponding author: Department of Mechanical Engineering, University of Defence, Brno, Kounicova 156/65, CZ-662 10 Brno, Czech Republic. Phone: +420 973 44 26 16, E-mail: vladimir.horak@unob.cz. ORCID 0000-0003-1654-7441.*

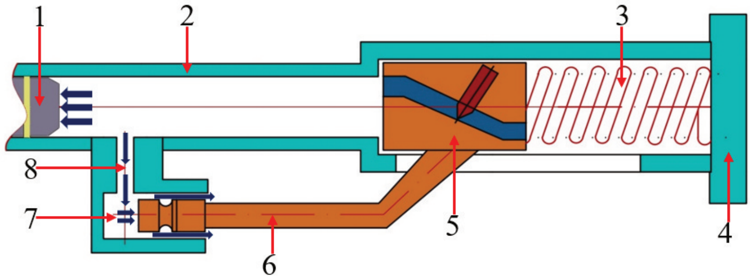

Fig. 1 Schema of the gas-operated automatic weapon drive: 1 – projectile; 2 – barrel walls; 3 – return spring; 4 – weapon casing; 5 – breech block carrier; 6 – piston; 7 – gas cylinder; 8 – gas port

The whole gas-operated automatic weapon drive can be considered as a process taking place in the thermodynamic system. The system consists of two chambers connected by the gas port. In the first chamber, i.e. in the space in barrel behind a projectile, propellant burns and produces propellant gases. The increase in the pressure of the propellant gases inside the first chamber causes the projectile to move and thus the chamber's volume increases. In the second chamber, i.e. in the gas cylinder, the incoming gas pressure increases and moves the piston, thus the chamber's volume also increases. Concurrently, some amount of propellant gases discharges from the cylinder to the surroundings through an annulus between the piston and cylinder.

This process takes place in two following periods. The first one is when the pressure of propellant gases in the barrel is higher than the pressure in the gas cylinder. During the second period, the pressure of propellant gases in the gas cylinder is higher than the pressure in the barrel. Then, the gas flows back from the cylinder to the barrel. The gas flow into the atmosphere through the annulus can be considered as a critical flow due to the great ratio of the pressure in the cylinder and the atmospheric pressure. The evolutions of the pressures in the cylinder and in the barrel are shown in Fig. 2.

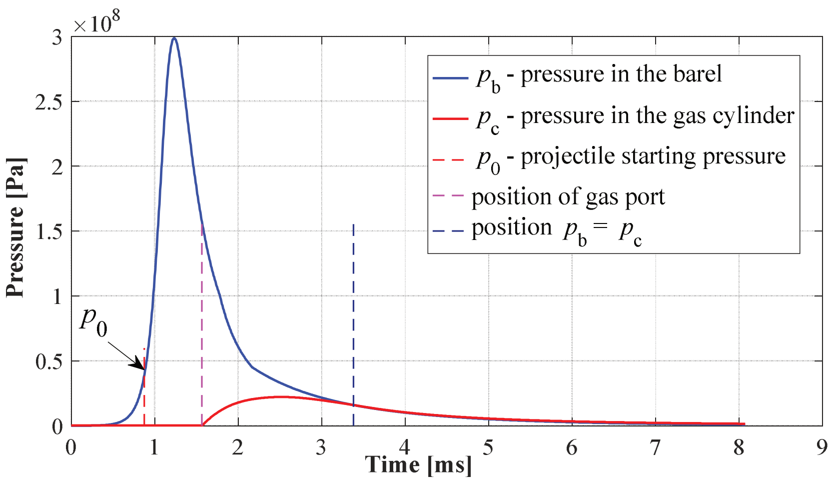

Fig. 2 Typical evolutions of gases pressure in the barrel and in the gas cylinder

To study the phenomena occurring in the gas cylinder, there are two different methods often used. The first one is based on thermodynamic laws, where a system of differential equations is derived to describe the thermodynamic mathematical model, which can be solved numerically. The obtained solutions consist of the average values of thermodynamic quantities such as the pressure, the temperature, the density, and the gas flow velocity in the system, see [2-4]. On the other hand, the second method covers an empirical approach, which is based on a range of experiments to measure pressure courses of propellant gases in the gas cylinder by piezoelectric pressure sensor and the measure of piston motion by high-speed camera, respectively, see in [5].

Recently, the computational fluid dynamics (CFD) has become a powerful numerical method to investigate thermodynamic quantities of gases in various parts of flowing systems. This method is often used for research and analysis of a gas flow through orifices and nozzles to determine the discharge coefficient of flowing gases by applying the CFD simulation, see e.g. [6-8].

In this study, we have used the CFD simulation in ANSYS Fluent software to examine thermodynamic quantities of gases in the cylinder and to determine the piston law of motion.

In addition, some experiments were carried out to measure the pressure course in the gas cylinder and to indicate the piston motion. Then, the results of numerical simulation were compared with the results of experiment to validate the proposed method.

2 CFD Simulation of Flow in Gas-Operated Mechanism

The gas-operated mechanism of the UK-59 machine gun consists of a gas cylinder, a piston, a gas block, and a regulator. The gas cylinder is connected to the barrel by a gas port. A schematic arrangement of the gas-operated mechanism of UK-59 machine gun is given in Fig. 3.

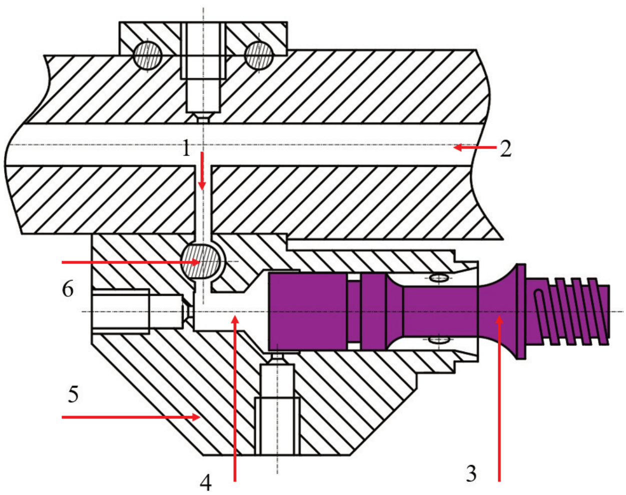

Fig. 3 The gas-operated mechanism of UK-59 machine gun: 1 – gas port; 2 – barrel; 3 – piston; 4 – gas cylinder; 5 – gas block; 6 – regulator

Geometric dimensions in millimeters of the gas block, the regulator and the piston are shown in Fig. 4.

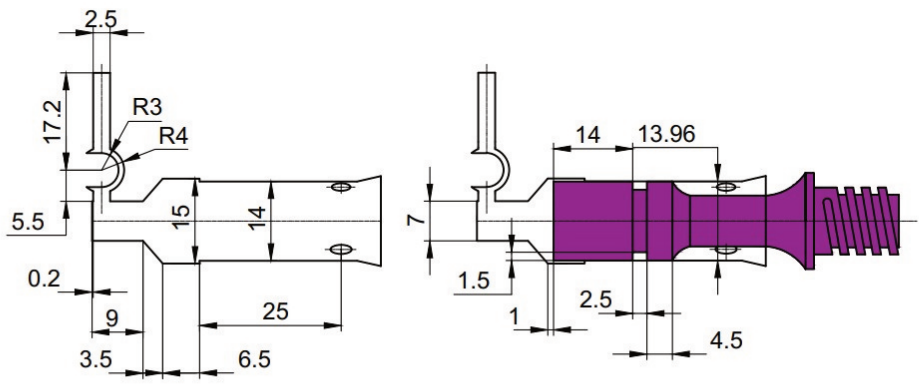

Fig. 4 Geometric dimensions for the UK-59 machine gun

A simplified 2D model of the gas-operated mechanism of UK-59 machine gun is sketched out in Fig. 5 based on their original 3D model.

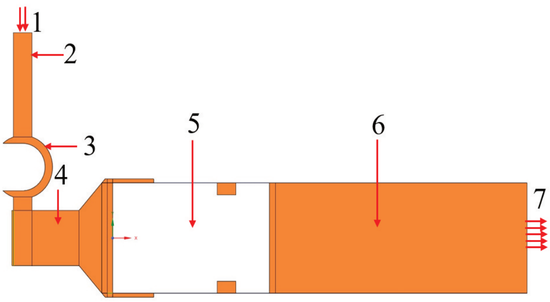

Fig. 5 CFD model of the gas-operated mechanism of UK-59 machine gun: 1 – inlet; 2 – gas port; 3 – gas vent of regulator; 4 – gas cylinder; 5 – piston; 6 – flow space behind the piston; 7 – outlet

Procedures for performing the simulation in ANSYS Fluent are as follows:

- the geometry model and the meshing process of the gas-operated mechanism were performed in ANSYS Workbench platform,
- the mesh quality has been checked and mesh quality criteria were fulfilled,
- the gas flow through the gas port represents a compressible flow,
- the solver type: density based solver,
- the order of discretization: second order upwind,
- the convergence criterion was 1×10^{-6}, this was applied for residual energy, for the others the convergence criterion was1×10^{-3},
- the SST k-omega turbulence model was used for the simulation, see [9].

The period of the numerical simulation lasts from the moment when the projectile has just passed the gas port until the piston has reached the position at the end of the gas drive phase, i.e. until the piston displacement has reached its position of $x_p = 30.5$ mm, where two exhaust holes with a diameter of 4 mm are drilled in the cylinder wall.

Thermodynamic quantities of inlet propellant gases are time-dependent functions, as shown in Figs 6 and 7. The CFD model input parameters were obtained from the interior ballistic calculation, which is often used at the Department of Weapons and Ammunition, of the University of Defence, and that is closely described in [10].

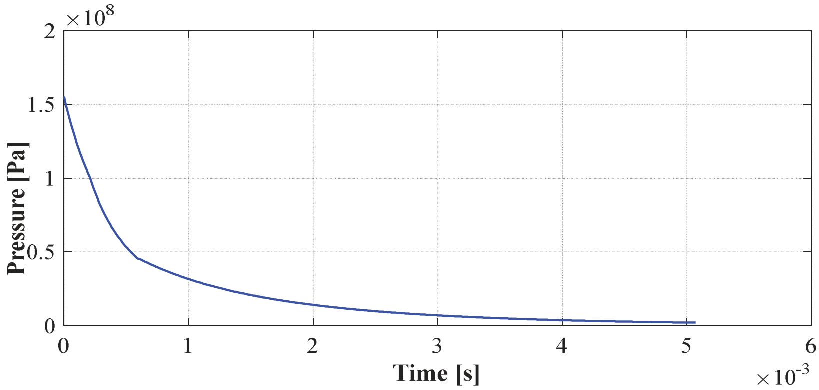

Fig. 6 Pressure of inlet propellant gases

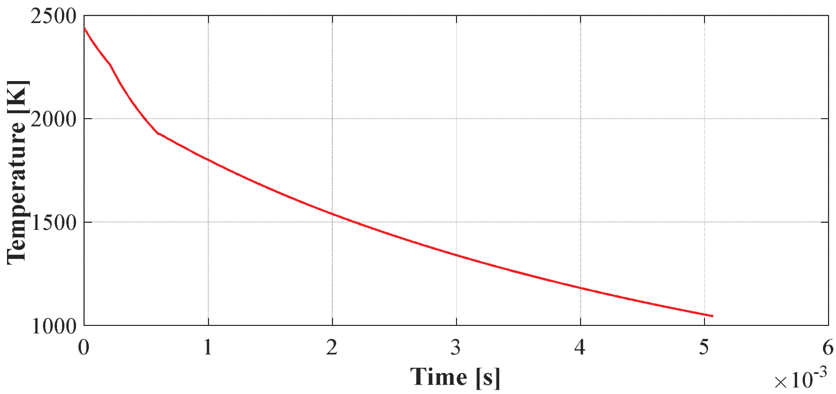

Fig. 7 Temperature of inlet propellant gases

Outputs of the CFD simulation are resulting thermodynamic quantities of propellant gases acting on the front of the piston, the thermodynamic quantities of gases in the cylinder and in the space behind the piston. The piston will move backwards due to the expansion of gas in the cylinder.

The law of the piston's motion (translational motion of its center of gravity) is defined by a UDF (user defined functions) file, which is compiled in the SDOF (six degree of freedom) solver in Fluent for the purposes of a dynamic mesh. Initial conditions are given by ambient meteorological conditions.

The UDF file is written in periods according to the UK-59 machine gun breech mechanism arrangement.

In the first period, for the piston displacement x_p from 0 mm to 14 mm:

- the mass of the piston linked to the breech mechanism is

$$\sum m_i = m_\mathrm{b} + \frac{1}{3} m_\mathrm{rs} \tag{1}$$

- the force acting on the piston linked to the breech mechanism is

$$\sum F_i = S_p\left(p_c - p_{atm}\right) - (F_{rs0} + C_{rs}x_p) - F_f \tag{2}$$

In the second period, for the piston displacement x_p from 14 mm to 24 mm:

- the mass of the piston linked to the breech mechanism is

$$\sum m_i = (m_b + \frac{1}{3}m_{rs} + m_{bi}) \tag{3}$$

- the force acting on the piston linked to the breech mechanism is

$$\sum F_i = S_p\left(p_c - p_{atm}\right) - (F_{rs0} + C_{rs}x_p) - F_f - F_{fob} \tag{4}$$

The last period is for the piston displacement x_p from 24 mm to the final position of 30.5 mm:

- the mass of the piston linked to the breech mechanism is

$$\sum m_i = m_b + \frac{1}{3}m_{rs} + m_{bi} + m_c \tag{5}$$

- the force acting on the piston linked to the breech mechanism is

$$\sum F_i = S_p\left(p_c - p_{atm}\right) - (F_{rs0} + C_{rs}x_p) - F_f - F_{exc} \tag{6}$$

where m_b – the mass of breech block carrier; m_{rs} – the mass of return spring; m_{bi}– mass of moveable of breech mechanism; m_c – the mass of cartridge case; S_p – the cross-sectional area of piston; p_c – the gas pressure in the gas cylinder; p_{atm} – the atmospheric pressure; F_{rs0} – the initial pre-stress of the return spring; C_{rs} – the return spring constant; x_p – the piston displacement; F_f – the friction force; F_{fob} – the friction force while unlocking breech block; F_{exc} – the cartridge case extraction force from cartridge chamber.

Specific values of the masses and forces for the piston linked to the breech mechanism of the UK-59 machine gun are given in [11].

3 Experimental Results and CFD Simulation

Experiments are carried out to measure the pressure evolution in the gas cylinder and the piston motion during a shot.

The pressure course is measured by the piezoelectric pressure sensor KISTLER Type 6215, which is located on the front of the gas cylinder head to measure the evolution of the gas pressure in the gas cylinder during the shot.

The high-speed camera FASTCAM SA-Z PHOTRON, which is used to measure the piston displacement, was placed on a tripod on the left side and perpendicular to the tested weapon at about 1 m. The view of the overall arrangement of the experimental apparatus is shown in Fig. 8.

The comparison of results of the CFD simulation in ANSYS Fluent and results of experiment for 5 repeated shots is shown in Figs 9 and 10. The evolution of pressure in the gas cylinder is given in Fig. 9 and the evolution of the piston displacement is given in Fig. 10.

It is obvious, from Figs 9 and 10 that the results of the propellant gases pressure course in the cylinder and the piston displacement from the CFD simulation correspond quite well with the results of experiment.

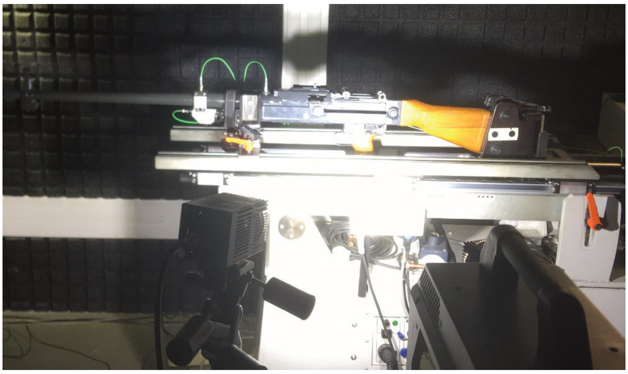

Fig. 8 View of experimental apparatus

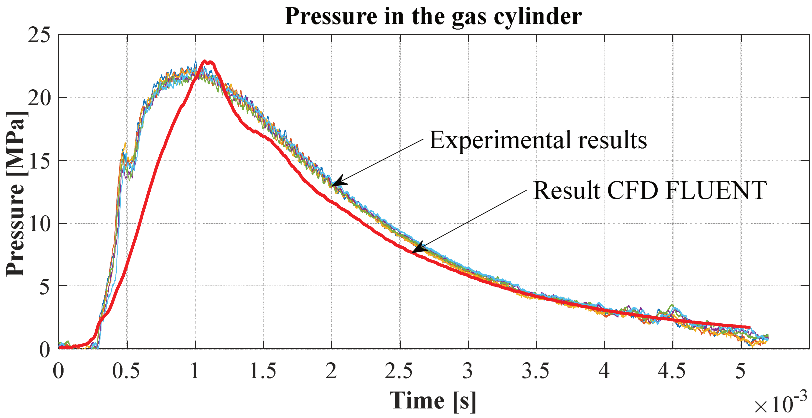

Fig. 9 Comparison of the evolution of the gas cylinder pressure by CFD simulation and experiment for UK-59 gas-operated mechanism

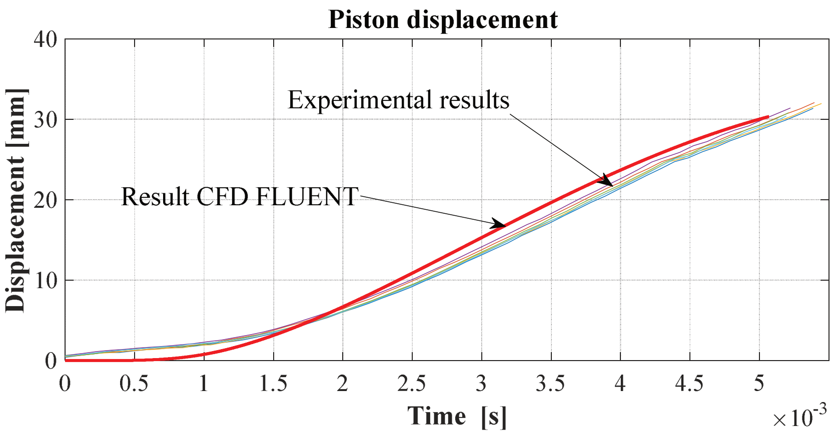

Fig. 10 Comparison of the piston displacement by CFD simulation and experiment for UK-59 gas-operated mechanism

Accuracy of the CFD simulation is influenced by the input thermodynamic quantities of propellant gases, assumptions, modelling simplifications in the CFD model, and also a time step size for transient simulation. Here, the time step of calculation was chosen as 10^{-6} s.

The CFD simulation provides us with results in various modes. Some results of the simulation, namely temperature and velocity distributions of propellant gases in the in gas-operated mechanism, are shown in Figs 11-14.

The presented results show us the temperature and velocity distribution of propellant gases flowing in the gas-operated mechanism during the piston motion in selected moments from 1 ms to 5.07 ms, when the piston displacement reaches its final position of $x_p = 30.5$ mm.

In the first phase of the gas-operated mechanism drive, when $p_b > p_c$, propellant gases flow into the cylinder from the barrel, thus the pressure in the cylinder increases rapidly. Then, the piston begins to move backwards and at the same time a part of gases flows from the cylinder into the atmosphere through the annulus, see Fig. 11.

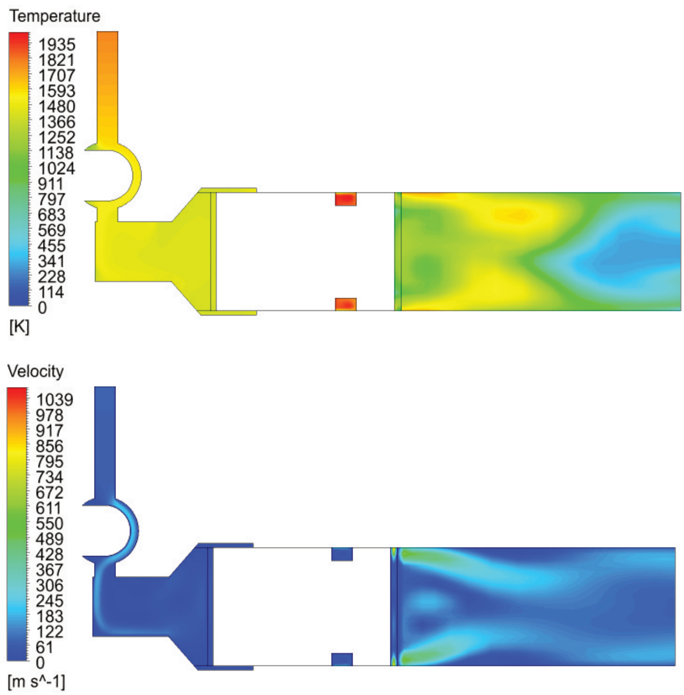

Fig. 11 Temperature and velocity distribution in gas-operated mechanism for time $t = 1$ ms *and piston displacement* $x_p = 0.75$ mm

Results of the CFD simulation at time $t = 1.5$ ms, corresponding to the piston displacement $x_p = 3.07$ mm, during the first phase of the gas-operated mechanism drive, are shown in Fig. 12.

It is obvious from Figs 11 and 12 that the flow velocity in gas port and regulator is closely connected with the piston displacement and the change in cylinder volume. Propellant gases flow through the vent of regulator very slowly. It means that the gas exchange between the barrel and the cylinder of the gas-operated mechanism is insignificant.

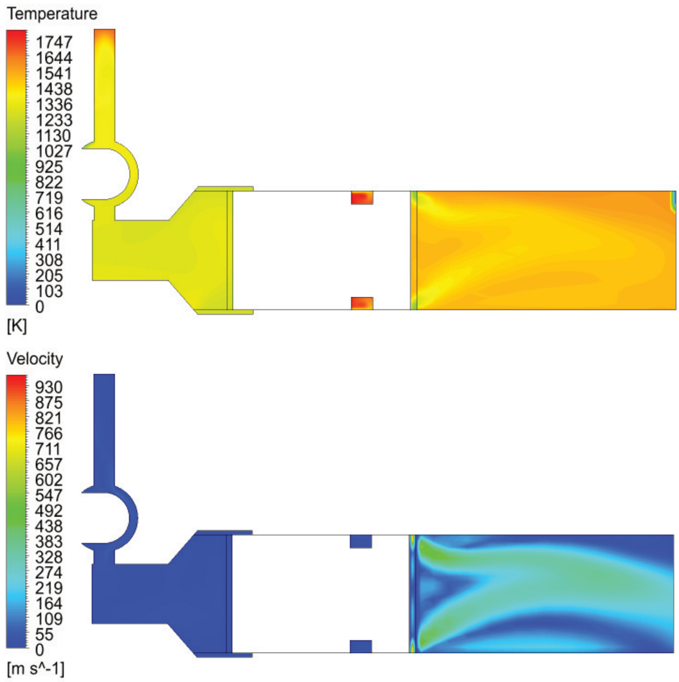

Fig. 12 Temperature and velocity distribution in gas-operated mechanism for time t = 1.5 ms *and piston displacement* x_p = 3.07 mm

Results of the CFD simulation in the second phase, when $p_b < p_c$, are shown in Figs 13 and 14 for two specific times. During this phase, the propellant gases flow from cylinder into the barrel and to the atmosphere. It causes that a certain amount of heat is transferred to the space behind the piston by the gas flow into the atmosphere.

It is noticed that the occurrence of a high velocity gas flow is in places of a small flow cross-section, e.g. in the outlet cross-section of the annulus between the piston and the cylinder, in the gas port and especially in the vent of regulator.

Velocity vectors and the pressure distribution in the vent of regulator at the moment when propellant gases flow from the barrel to the cylinder, i.e. when $p_b > p_c$, and at the moment when gases discharge from the cylinder back to the barrel, i.e. when $p_b < p_c$, are shown in Figs 15 and 16, respectively.

Additionally, velocity vectors of the propellant gases flow in the cylinder and in the annulus between the piston and the cylinder for $p_b > p_c$ are shown in Fig. 17. On the left side of Fig. 17, we can see vortices formed in the cylinder. On the right side of Fig. 17, it is noticed that a piston ring groove is designed on the piston and thus an enlarged space is created between the piston and the cylinder. The goal of this special design is to reduce gas leaks from the cylinder to the atmosphere. The mechanism of the gas loss reduction is based on the effect of gases vortices, which arise when the gases flow through this enlarged space and then act as a barrier to prevent the cylinder gases discharge to the atmosphere.

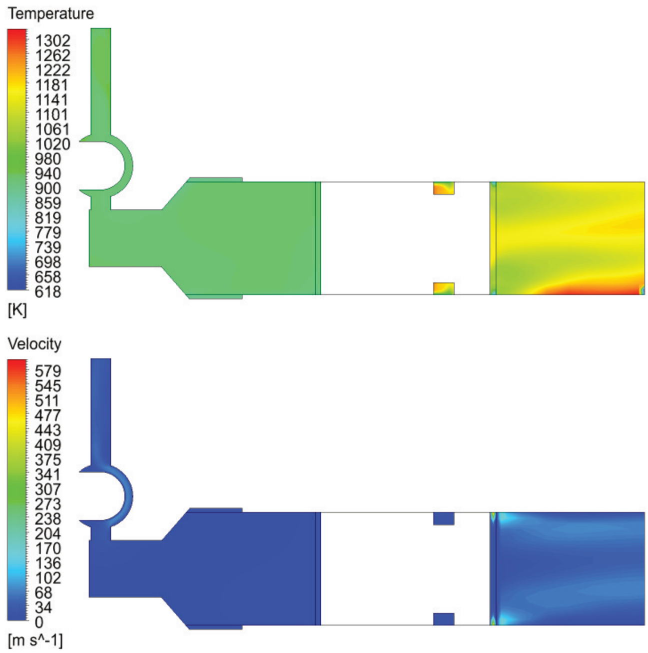

Fig. 13 Temperature and velocity distribution in gas-operated mechanism for time t = 3 ms *and piston displacement* x_p = 15.2 mm

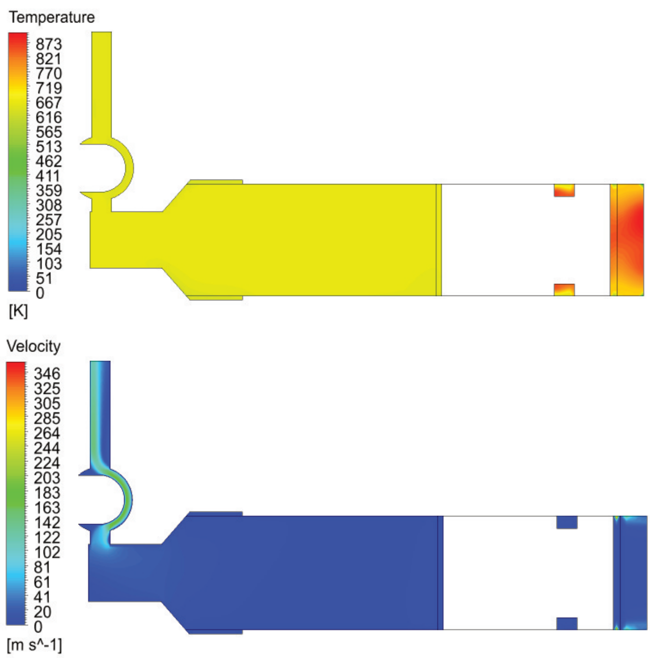

Fig. 14 Temperature and velocity distribution in gas-operated mechanism for time t = 5.07 ms *and piston displacement* x_p = 30.5 mm

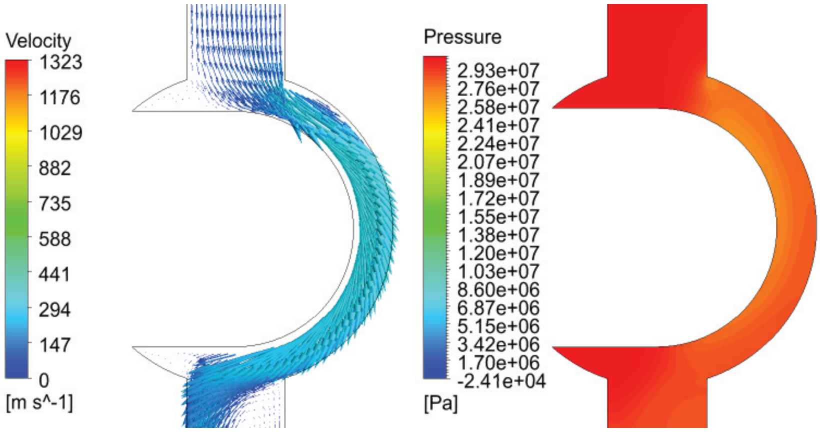

Fig. 15 Velocity vectors and pressure distribution in the vent of regulator for $p_b > p_c$

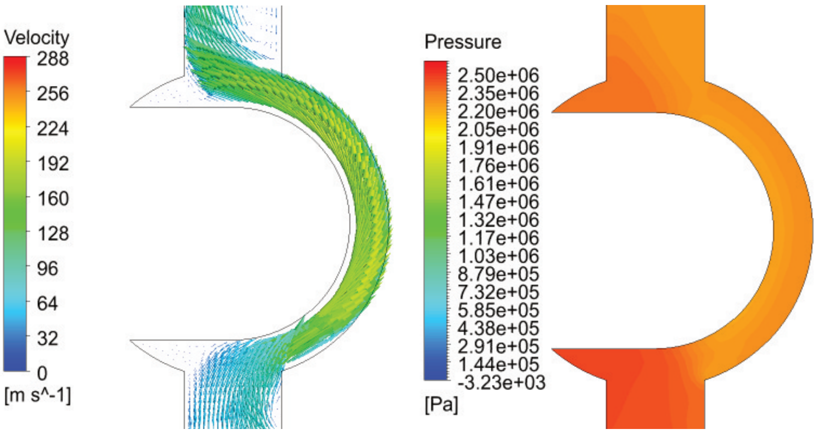

Fig. 16 Velocity vectors and pressure distribution in the vent of regulator for $p_b < p_c$

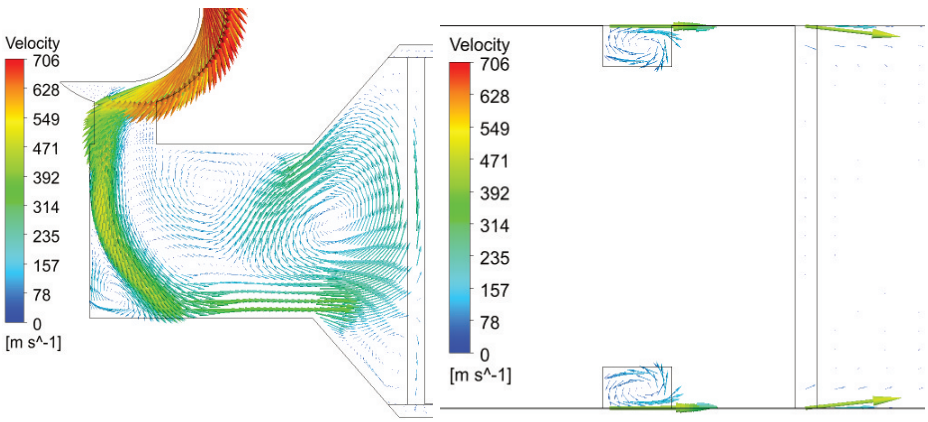

Fig. 17 Velocity vectors of the propellant gases flow in the cylinder and in the annulus between the piston and the cylinder for $p_b > p_c$

As it can be seen from the previous description, we can state that the CFD simulation of transient processes ongoing in the weapon gas-operated mechanism provides us with various modes of results. One of the simulation possibilities is also editing the video sequences of evolutions of thermodynamic quantities. The video showing the above simulation is available at https://www.youtube.com/watch?v=mspj7mDylFs.

By integrating the velocity and density distribution across any flow cross section, it is possible to obtain the propellant gases mass flow rates. The evolutions of the mass flow rates of gases flowing through the regulator vent and trough the annulus between the piston and the cylinder are shown in Fig. 18. Here, the negative sign is for the flow to the gas cylinder and the opposite flow has a positive sign.

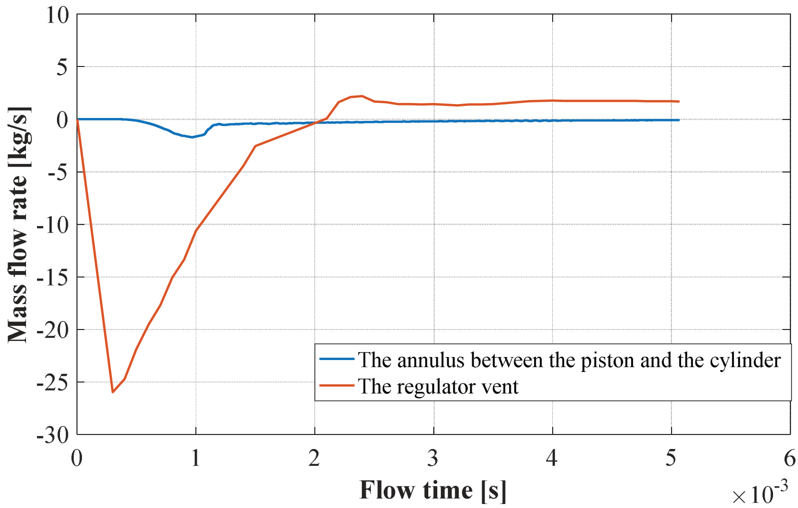

Fig. 18 Mass flow rate of gases flowing through the regulator vent and through the annulus between piston and cylinder versus time

This is a way that allows us to identify and quantify individual discharge losses during the operation of the weapon gas-operated mechanism. Presented mass flow rate is for the clearance between the piston and the cylinder of 0.02 mm.

4 Conclusion

The presented CFD simulation in ANSYS Fluent of processes connected with the gas-operated automatic weapon drive enables us to study and better understand complex phenomena of propellant gases flow in the gas port, the regulator vent, the gas cylinder, and the annulus between the piston and the cylinder of the weapon gas-operated mechanism. Here, the results of the simulation include thermodynamic quantities of gases in the system such as pressure, temperature, and velocity distribution. The interaction between propellant gases and moveable parts of the gas-operated mechanism determines the law of piston motion during the gas-drive phase.

A number of simulation results, specified for the UK-59 machine gun, were introduced and some of them has been validated by experiment with a good agreement.

The CFD simulation thus becomes a powerful tool in the design and development of automatic weapons. Various quantities and influences can be predicted and analyzed through the simulation, such as e.g. the prediction of effects due to changes and modifications of flow sections on gas flow characteristics. These parameters affect e.g. the law of piston motion or the discharge losses in gas cylinder. From the velocity and temperature distribution, the places potentially most at risk of erosion by flowing gases can be identified. This will then affect the choice of material and surface treatment of examined parts.

Acknowledgement

The authors would like to thank the Faculty of Military Technology, University of Defence, Brno, for the support by the Institutional Funding DZRO VAROPS.

References

[1] *Engineering Design Handbook: Guns Series. Automatic Weapons* [online]. Washington: AMC Headquarters, 1970 [viewed 2022-01-06]. Available from: https://apps.dtic.mil/sti/pdfs/AD0868578.pdf

[2] BALLA, J., L. POPELÍSKÝ and Z. KRIST. Theory of High Rate of Fire Automatic Weapon with together Bound Barrels and Breeches. *WSEAS Transactions on Applied and Theoretical Mechanics*, 2010, **5**(1), pp. 71-80. ISSN 1991-8747.

[3] JEVTIC, D., D. MICKOVIĆ, S. JARAMAZ, P. ELEK, M. MARKOVIC and S. ZIVKOVIC. Modelling of Gas Parameters in the Cylinder of the Automatic Gun during Firing. *Thermal Science*, 2020, **24**(6), pp. 4135-4215. DOI 10.2298/TSCI200118152J.

[4] HORÁK, V., L. DO DUC, R. VÍTEK, S. BEER and Q.H. MAI. Prediction of the Air Gun Performance. *Advances in Military Technology*, 2014, **9**(1), pp. 31-44. ISSN 1802-2308.

[5] TIEN, V.D., M. MACKO, S. PROCHÁZKA and V.V. BIEN. Mathematical Model of a Gas-Operated Machine Gun. *Advances in Military Technology*, 2022, **17**(1), pp. 63-77. DOI 10.3849/aimt.01449.

[6] KARTHIK, G.S., K.J.Y. KUMAR and V. SESHADRI. Prediction of Performance Characteristics of Orifice Plate Assembly for Non-Standard Conditions Using CFD. *International Journal of Engineering and Technical Research (IJETR)*, 2015, **3**(5), pp. 162-167. ISSN: 2321-0869.

[7] BAGASKARA, A. and M.A. MOELYADI. CFD Based Prediction of Discharge Coefficient of Sonic Nozzle with Surface Roughness. *Journal of Physics: Conference Series*, 2018, **1005**. DOI 10.1088/1742-6596/1005/1/012010.

[8] VU, D.T., S. PROCHÁZKA, Z. KRIST, H.B. LE and B.V. VO. CFD Simulation of Propellant Gas Flow Motion in Gas-Operated Systems. In: *2021 International Conference on Military Technologies (ICMT)*. Brno: IEEE, 2021, pp. 1-7. DOI 10.1109/ICMT52455.2021.9502808.

[9] DUTTA, T., K.P. SINHAMAHAPATRA and S.S. BANDYOPDHYAY. Comparison of Different Turbulence Models in Predicting the Temperature Separation in a Ranque-Hilsch Vortex Tube. *International Journal of Refrigeration*, 2010, **33**(4), pp. 783-792. ISSN 0140-7007.

[10] BEER, S., L. JEDLIČKA, and B. PLÍHAL. *Barrel Weapons Interior Ballistics* (in Czech). Brno: University of Defence, 2004. ISBN 978-80-85960-83-9.

[11] VU, D.T., S. PROCHÁZKA, Z. KRIST and V.B. VO. Influence of Gas Port on Forces and Their Impulses Acting in an Automatic Weapon. In: *2021 International Conference on Military Technologies (ICMT).* Brno: IEEE, 2021, pp. 1-8. DOI 10.1109/ICMT52455.2021.9502832.