Numerical simulations on increasing turbojet engines exhaust mixture ratio using fluidic chevrons

Adrian GRUZEA*,[a]

*Corresponding author
"POLITEHNICA" University of Bucharest, Faculty of Aerospace Engineering
Gh. Polizu Street 1-5, 011061, Bucharest, Romania
gruzea.adrian@gmail.com

DOI: 10.13111/2066-8201.2017.9.2.7

Received: 30 April 2017/ Accepted: 24 May 2017/ Published: June 2017

Abstracti: *This paper refers to some aspects regarding the terms "chevron" and "fluidic chevron" and to the process of increasing the jet engines exhaust mixing rate towards achieving noise reduction. One of the noise reduction methods consists in covering the high velocity main flow with a secondary one, having a much lower velocity, similar to the turbofan engines. The fluidic chevrons try to accomplish these requirements, being used just in particular moments of the flight. This study will be based on numerical simulations carried using the commercial software ANSYS. The geometry used will the based on the micro jet engine JetCat P80, equipping the turbines laboratory from the Faculty of Aerospace Engineering. A research based on the measured geometric, gasodynamic and cinematic parameters will be carried varying the mass flow and keeping the immersion angle constant. As a result of these simulations we'll observe the influence of the mentioned parameters on the jet's flow field.*

Key Words: *CFX, fluidic chevron, nozzle, jet engine*

1. INTRODUCTION

Nowadays, one of the biggest challenges faced by the aviation is the noise pollution which affects the personnel servicing the airport and more significantly the airport adjacent community.

Both worldwide and in the European Union, the aircraft jet noise reduction is set as a priority, thus the ACARE vision for the year 2020 is set to equip all the modern aircrafts with noise reduction systems to reduce the noise by 10dB compared to the year 2000 [1].

From an aicraft flightpath, the main flight phases that phonically disturbs the comunity are the takeoff and landing [2] figure 1, thus the biggest noise source for these phases is represented by the turbojet engines.

In regards to the turbojet noise produced by the aircraft engines, the most important source is the jet exhaust [3], figure 2.

---

[a] PhD. Student Eng.

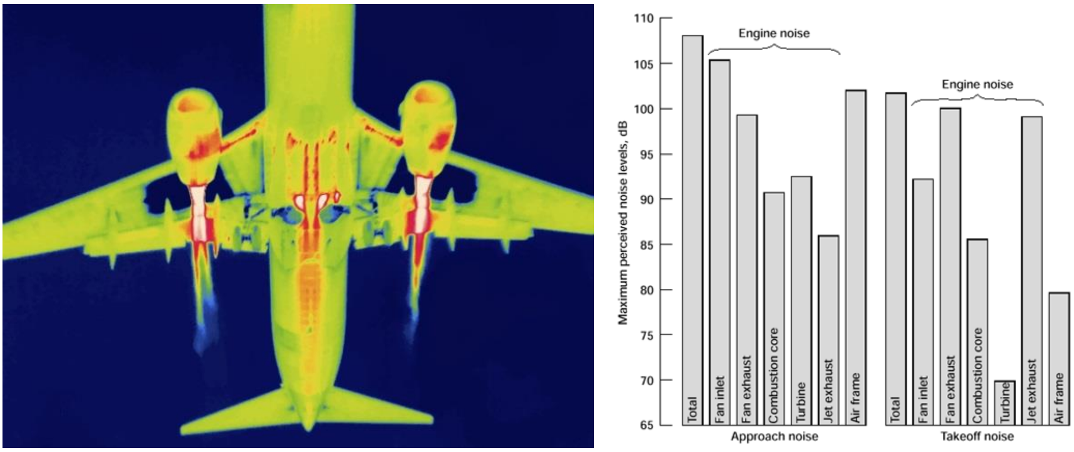

Figure 1. Aircraft noisemap and the main sources of noise

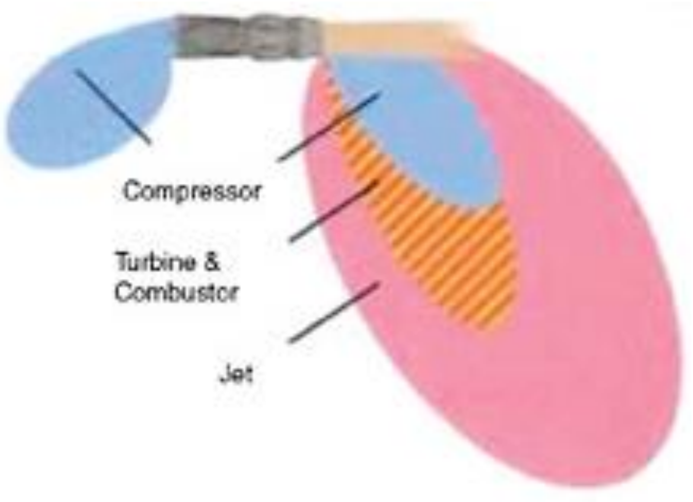

Figure 2. Turbojet noise map

It is important to understand the physics behind the described phenomenon. [4]

A free jet of gas is a stream of fluid that is projected into a surrounding medium, having either the same or different thermodynamic properties.

A jet of gas is characterized by three parts:

a. The core region, with near constant velocity
b. The transition region, viscous mixing spreads through the entire jet flow
c. Developed region, the flow velocity continues decreasing along all directions but the profiles of relative velocity becomes contant.

The three regions of the jet flow are presented below in figure 3. At the nozzle exit, the mixing jet consists of two turbulent main layers separated by a triangular core.

At a certain distance from the exit of the engine, the two layers mix together and the jet starts to develop.

Formation of the developed flow consumes energy, that is taken from the gas stream itself. The consequence of consuming the gas’s energy is reflected in the flow rate decrease.

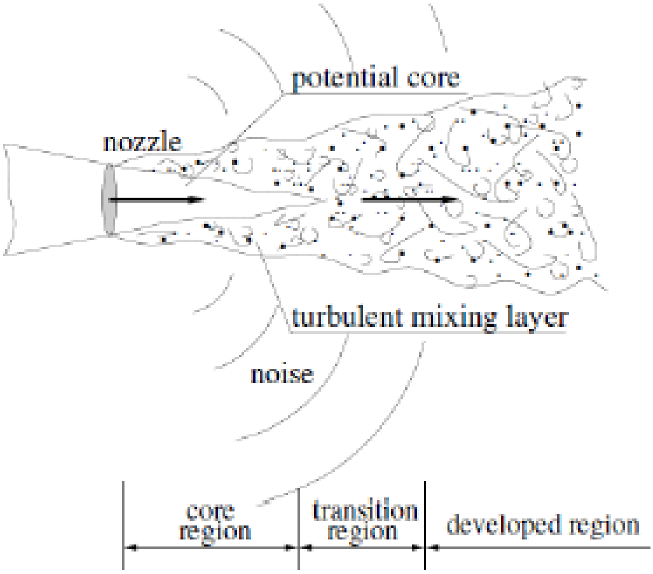

Figure 3. Schematic of a turbulent jet; sources of noise. [4]

Many methods are used to decrease the jet noise. A first step towards this direction was the introduction of high bypass ratio turbofans, that reduces the average velocity of the exhaust, and afterwards improving the solution by installing mechanical chevrons.

Currently, among the extensive research conducted towards noise reduction for the aircrafts, “The Chevrons” [5, 6] method is the most frequently used. The chevrons devices resemble the teeth of a saw and the noise reduction occures from the pelicular mixture of hot gases and air. This method of noise attenuation has the advantage of having relatively low thrust losses and their construction is very simple.

The use of the atenuators is justified just during takeoff and landing phases and when the aircraft is around inhabited zones, other than that these devices should be disabled to reduce the losses. From Lihil’s equation we can observe that the main parameter that defines the acoustic pressure along the flow direction is the velocity gradient along a perpendicular axis. These fluidic chevrons are playing this precise role of minimizing the vertical velocity gradient if we consider the jet flow to be horizontal. Fluidic chevrons could solve the problem caused by using the devices among the whole flight path of an aircraft, being used just during takeoff and landing phases or activated at will when necessary. Their schematic way of implementation can be observed in the picture below.

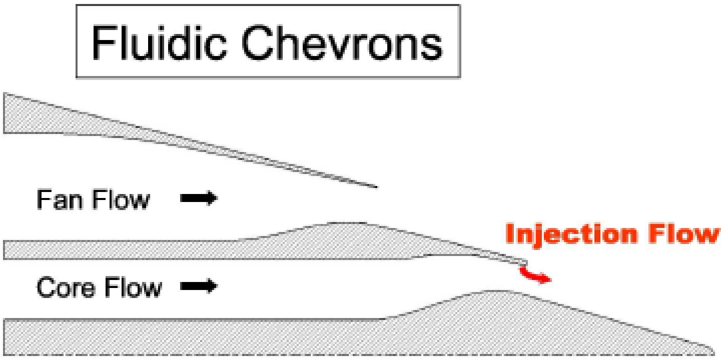

Figure 4. Fluidic chevrons for a by pass

The airflow that will be injected into the exhaust gases can be stored aboard the aircraft or can be bled out from the compressor area. However, it should be noted that, despite the high quantities or fluid used by the chevrons, it will add up to the exhaust jet. The figure No. 5 shows various constructive solutions.

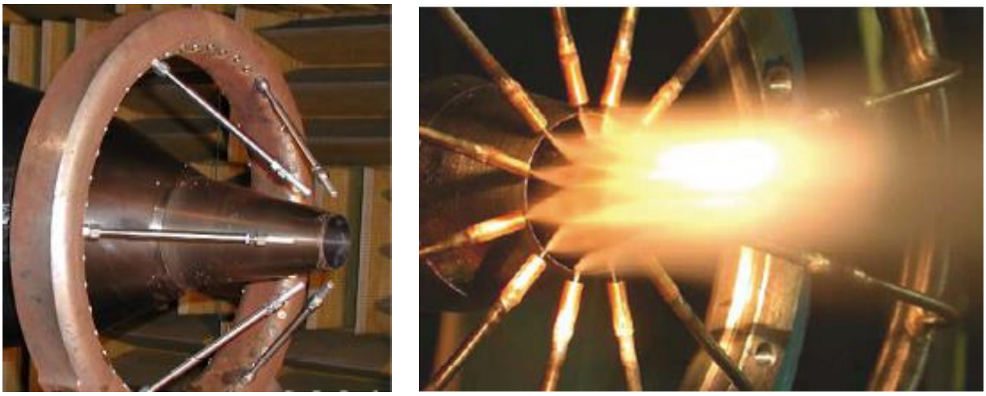

Figure 5. Image showing demonstrators with ways of injecting air for fluidic chevrons [7]

Studies regarding various working parameters of the fluidic chevrons for the purpose of noise reduction show a powerful dependence on the mass of the fluid injected as well as on its pressure [8]. In addition to all mentioned above, the amplitude of the parameters is highly dependent on the exhaust velocity, meaning it’s different for a turbojet than for a turbofan.

The fluidic chevrons have started to be widely regarded in studies and experiments for jet noise reduction, thus increasing the near future probability to be implemented on the future aircrafts. This paper will consist of a numerical methods study applied to fluidic chevrons evidencing the increase in jet mixture ratio. The simulations will be conducted on the micro turbojet engine, JetCat P80 geometry with gasodynamic parameters equivalent to about 77% of the max rpm.

2. CAD GEOMETRY AND MATHEMATICAL GRID

For this study we’re going to use the base geometry of the JetCat P80. The general setup is presented below, together with the test bench currently available in the turbines laboratory from the Faculty of Aerospace Engineering. The system description is detailed in the US Patent, Jet Cat instruction manual [9]. The parameters used for these simulations are gathered form tests conducted on this engine and are presented in the reference [10]. At a rate of 77% of the max rpm, the engine produces a mass flow of about 0.172kg/s at a temperature of 783,15 K, values used on the inlet of the nozzle for the simulations.

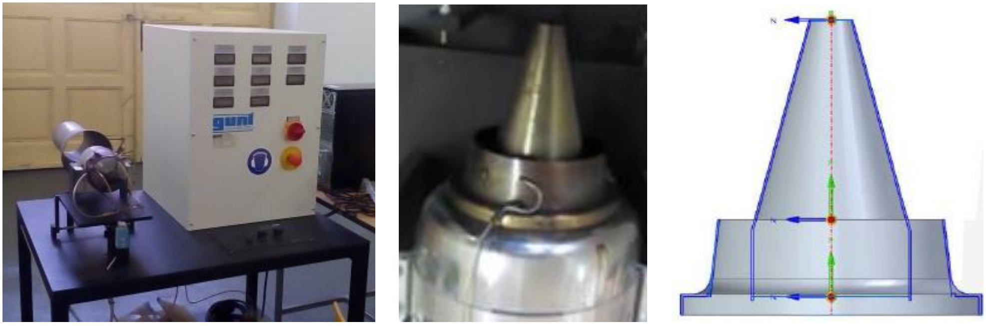

Figure 6. Micro turbojet engine test bench, the engine and the CAD geometry

In order to increase the jet mixture ratio, we suggest the particular case of using the fluidic chevrons.

The study will be performed at a constant immersion angle of 11deg, using constant number of chevrons and varying the mass flow through them. A number of 3 cases and the base model will be tested, having 12 chevrons and using a flow equivalent to 2%, 5% and 10% of the total mass flow through the engine at a temperature of 300K. The cases will be illustrated below in figure 7.

Figure 7. The CAD models of the 2 configurations: base model and 12 chevrons

Using the analytical results, the geometrical dimensions led to mesh generation.

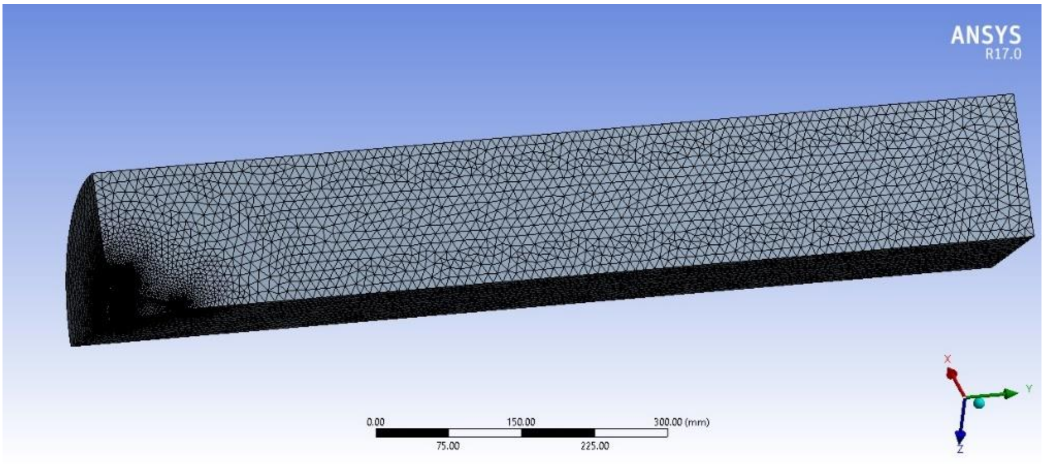

Figure 8. Meshing grid

The flow field length for the jet expansion is equivalent to 30 exit nozzle diameters on axial direction and 5 exit nozzle diameters on radial direction. The calculus grid was produced in commercial software by using an unstructured mesh.

Knowing that the field is axially symetric, we'll be considering only the computational domain comprising an angle of 90 degrees, so having a 3D domain.

In this way we could introduce more elements in the field related to the power of the computer. In this sector of the field 1062500 nodes are used. Thus, the entire area has a total of almost 4 million nodes.

The coordinate system: the Oy axis is directed downstream and the Ox axis radial.

The chosen working fluid was an ideal gas, the inlet parameters being the ones calculated for the combustion gases of the propellant chosen for this study.

The boundary conditions have an important role in the description of the behavior of complex flows.

Usually, these conditions are chosen so that the model can reproduce the real phenomenon.

We consider the following boundary conditions:

- Mass flow inlet + temperature;
- Opening;
- At the left and right sides of computational domain, the rotational periodic boundary;

Following the calculus of the Reynolds Number which has a value of about 10^{-6} we can corectly assume that the flow is turbulent so we choose k-ε as a turbulence model. The values of k and ε come from the transport equation of theturbulent kinetic energy and the turbulent dissipation.

In order to validate the CFX results, the parameter y + must have the values below 30 and above 30.

3. COMPUTATIONAL RESULT

The numerical simulations were performed with commercial software Ansys 17 [11]. The result, as it can be observed in the figures below, is the flowfield, when a flow of 2, 5 and 10% of the main jet was used through the fluidic chevrons.

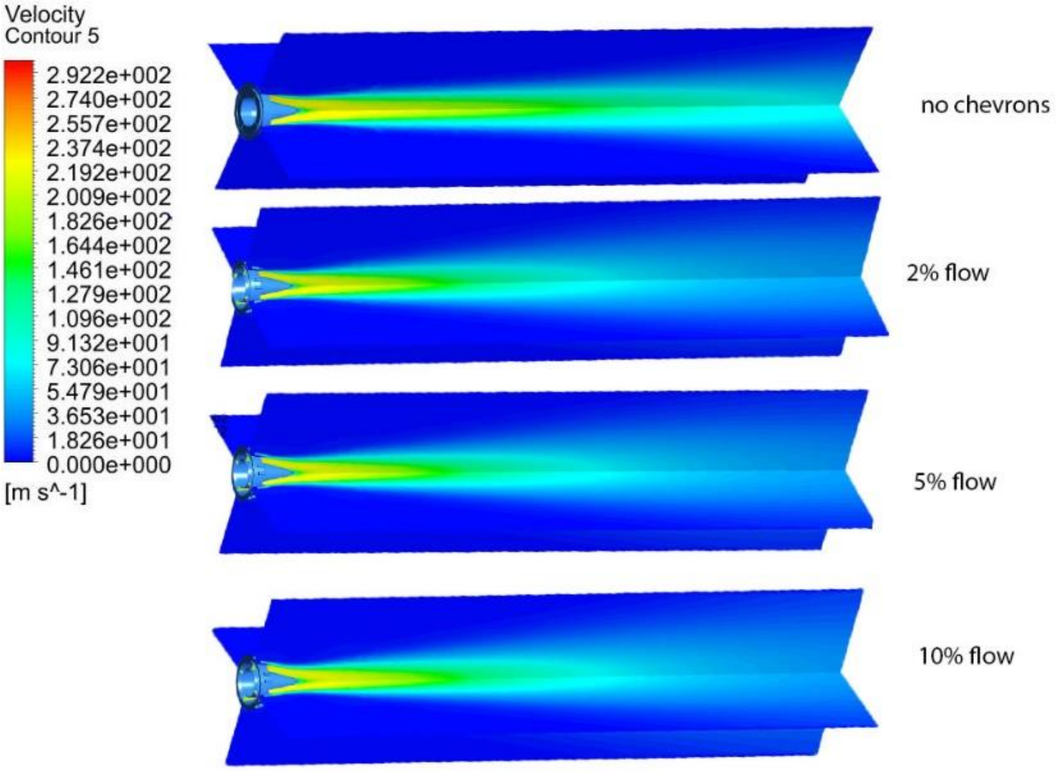

Figure 9. Velocity comparison

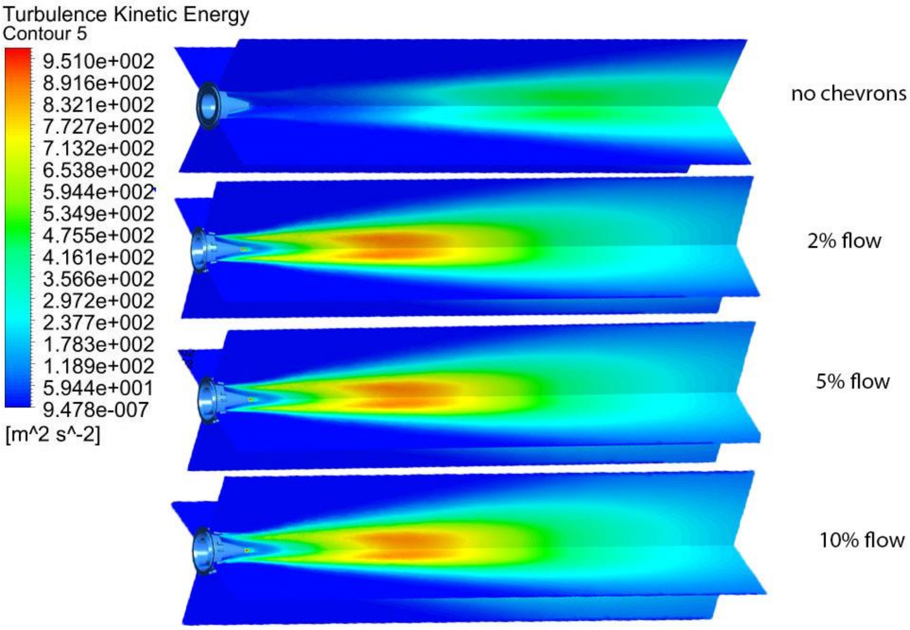

Figure 10. Turbulence kinetic energy comparison

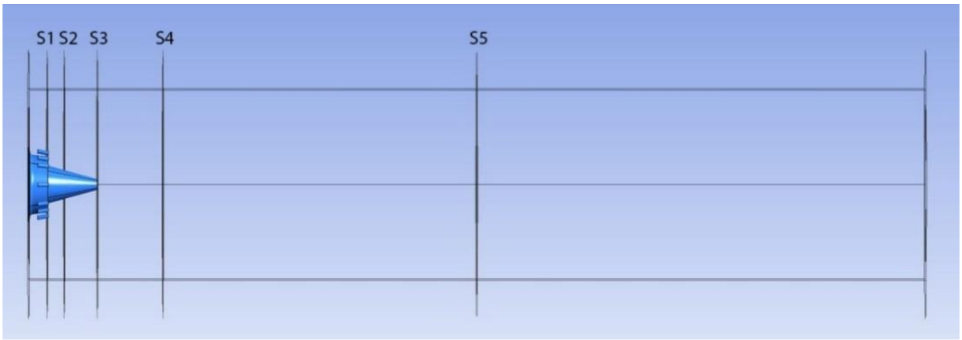

Figure 11. Flow sections definition

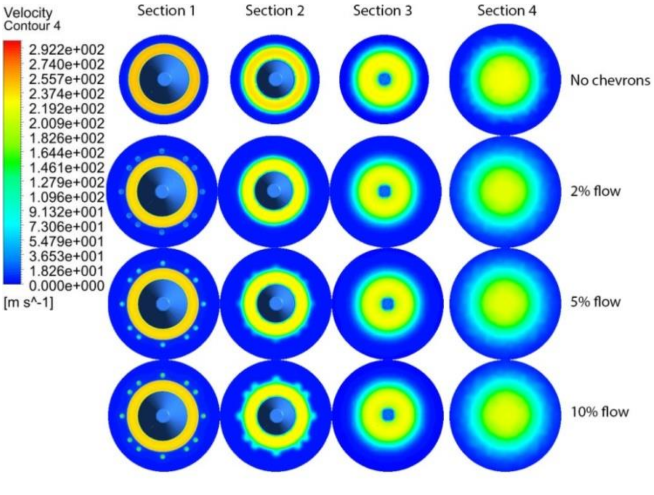

Figure 12. Velocity comparison through sections

The most significant difference can be observed in the developed region of the jet, so for a better comparison the legend was rescaled.

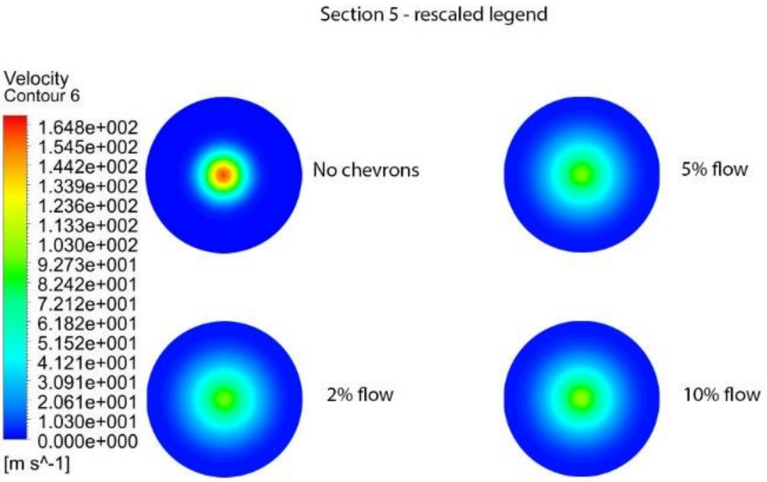

Figure 13. Section 5 rescaled legend

4. CONCLUSIONS AND FUTURE WORK

This study, in which a fix number of chevrons were used, points out the influence of the mass flow through the 12 chevrons on the exhaust gases. With the increase in the mass flow, as it can be observed in the figure 12, throw-out section 2, the hot gases get surrounded in the flow of the chevrons, increasing the mixture ratio of the flows and causing a decrease in velocity gradient. The effect is similar to the one generated by the mechanical chevrons but the advantage of this solution is that its action can be controlled along the flightpath.

The general conclusion is that the exhaust gases mixture ratio increases with the increase of the mass flow through the chevrons thus having a higher number of devices would create a continuous flow around the hot gases increasing the effect.

For future work, the influence of the number of the chevrons will be studied in detail, as well as their position around the nozzle and immersing angle relative to the flow path, considering both the flow field and the acoustic field.

REFERENCES

[1] * * * Aeronautics and Air Transport, Beyond Vision 2020 (Towards 2050) A Background Document from ACARE (The Advisory Council for Aeronautics Research in Europe).

[2] T. Raef , A. A. Elzahaby and M. K. Khalil, *Enhancement of Propulsion Performance through Jet Noise Reduction Technologies: A Review*, Proceedings of the 16 DV 102 th Int. AMME Conference, 27-29 May, 2014.

[3] * * *https://www.nap.edu/read/12928/chapter/7#57

[4] G. Cican, V. Stanciu, D. Crunteanu, Analytical and Numerical Study of the Nozzle Jet, *U.P.B. Sci. Bull., Series D*, Vol. **76**, Iss. 1, ISSN 1454-2358, 2014.

[5] V. Dippold III, *CFD Analyses and Jet-Noise Predictions of Chevron Nozzles With Vortex Stabilization*, Glenn Research Center, Cleveland, Ohio NASA/TM—2008-215150 AIAA–2008–0037, March 2008.

[6] J. Bridges and C. A. Brown, *Ohio Parametric Testing of Chevrons on Single Flow Hot Jets*, Glenn Research Center, Cleveland, NASA/TM—2004-213107 AIAA–2004–2824, 2004.

[7] C.J. Bargsten, M. T.Gibson, *NASA Innovation in Aeronautics: Select Technologies That Have Shaped Modern Aviation*, NASA/TM'2011.

[8] * * * US Patent 7159383.
[9] * * * US. Patent, “Jet Cat instruction manual”, 2008.
[10] G. Cican, V. Stanciu, V. Silivestru, R. M. Catana, *Fundamentals of Gas Turbine Engine Starting, Processes and Experiments,* Printech Publishing House, Bucharest 2016.
[11] * * * Ansys 15 Documentation-User’s Guide 2013