Acoustic control study of turbofan nozzles with triangular chevrons

Grigore CICAN*[,1,a], Virgil STANCIU[1,b], Daniel-Eugeniu CRUNTEANU[1,c]

*Corresponding author
*[,1] "POLITEHNICA" University of Bucharest, Faculty of Aerospace Engineering
Gh. Polizu Street 1-5, 011061, Bucharest, Romania
cicangrig@yahoo.com*, vvirgilstanciu@yahoo.com, crunti_dani2005@yahoo.com

DOI: 10.13111/2066-8201.2014.6.1.2

Abstract*: This paper has a small part dealing with the notion of chevron and the process that helps reducing the noise pollution. Based on the gas dynamics and the geometrical parameters of the turbofan jet engine a model of CFD data processing is created. In this process the influence of chevrons on acoustic wave intensity produced by the jet is observed by analyzing this process. A series of tests have been made on 10 si 20 lobed chevrons. The combination between them and the 7 resulting cases have been studied, namely the triangular chevrons in order to settle the influence of the geometrical parameters on the flow and on the jet acoustics. Finally the contribution of the chevrons in noise pollution reduction has been highlighted.*

Key words*: numerical simulation, CFD, Gambit, noise control, chevron, acoustic, intensity level.*

1. INTRODUCTION

Presently, the noise pollution is one of the most important problems for the people who live nearby the airports. Plane noise pollution has an undesired effect on human health for the people working in the airport and for those who are living in the nearby zones.

For an airplane one of the most important noise source are the engines and so the engine noise has become a problem since the jet engine was used for commercial lines. The most important noise is that produced by the jet and this will be the topic of this study.

The first step in acoustics development was made by James Lighthill [1], [2] that published the "acoustic analogy" theory in 1952. This analogy is one of the first theories about aerodynamic noise generation by jet turbulence.

The chevrons were introduced the fist time by Alex D. Young, G.M.Lilley (the author of Lilley's equation) and R.Westley in 1953.

By definition, the chevrons are dynamic gas equipment, that by initiating the vortical flow, helps the mixture of the two flows that have different velocities and decreases the resultant noise of those flows interaction. The chevrons work by initialising the vortex flows on axes that are parallel with the new flow direction [3].

The chevrons are gas dynamics devices that by initialising a vortex flow help the pelicular mixing of two different velocity fluxes, through that reducing the noise caused by the fluxes interaction [4].

[a] PhD. Student, eng.
[b] Professor, PhD, eng.
[c] Lecturer, PhD, eng.

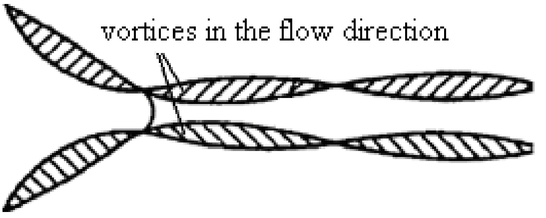


Fig. 1 Flow arround the chevron [4]

The physical basis of this type of noise generation is Kelvin-Helmholtz turbulence that occurs as a result of the velocity difference between the two fluxes.

The figure above presents schematically the chevron function in vortex flow intialisation. This study investigates the possibility of using the triangular chevrons for the tubofan jet engines to reduce noise pollution. The paper also includes a case study with an engine equipped with triangular chevrons highlighting the CFD study and noise pollution effects on the propulsive force.

2. GEOMETRICAL MODEL AND COMPUTATIONAL GRID

The nozzle's geometry is presented in Fig. 2. This is a reaction nozzle [5] that has the following geometrical parameters and that contains, in addition, triangular chevrons.

The parameters of the nozzle are:

Air is considered in this case the working fluid. The Input data are calculated using the computer program of the engine cycle. Turbofan CFM 56-3 specifications [7] were taken into account as input data.

So for the second output stream after stream these data were considered:

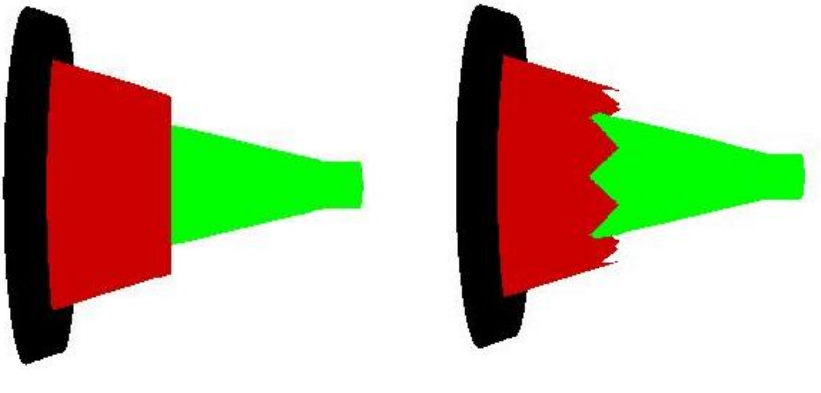
-speed = 305 m/s
-temperature = 350 K
-direction of jet exit = perpendicular to inlet

For the exit of the main stream flow:

-speed = 465 m/s
-temperature = 721 K
-direction of jet exit = perpendicular to inlet

To simulate the fact that there are no obstacles around the aircraft, around the nozzle in our case, the side walls and top wall outlets are considered with these characteristics:

-temperature = 288 K
-the considered pressure is the atmospheric one.



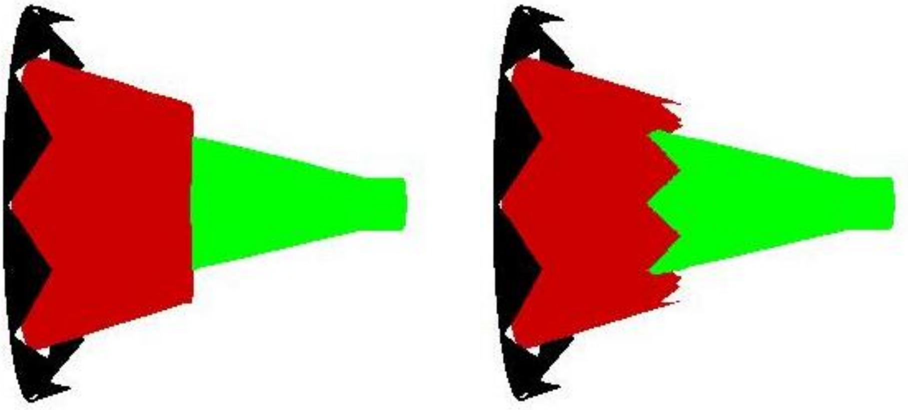


Fig. 2 The reaction nozzle's geometry

The calculus grid was produced in GAMBIT by using only a structured mesh.

The calculus domain has a periodical flow for each chevron and only a sector that belongs to a chevron was taken into consideration. With these details it was possible to introduce more elements in that zone of the flow.

For one sector one uses 415000 nodes and also note that for the case of a 36 degrees sector 4.15 million nodes are required for the entire domain.

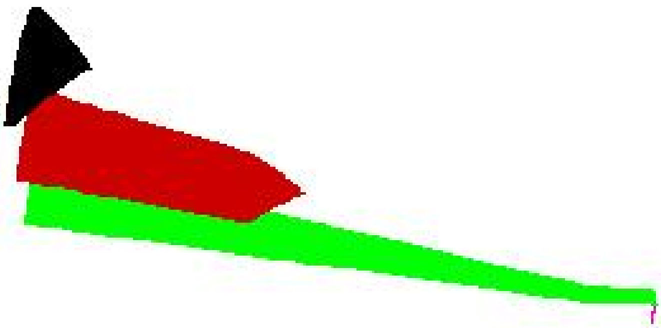


Fig. 3 The image of a sector of the nozzle with chevrons

The computational domain has a length on the Ox axis which is equivalent to 10 meters and on the Oy axis, equivalent to 1.5 meters.

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

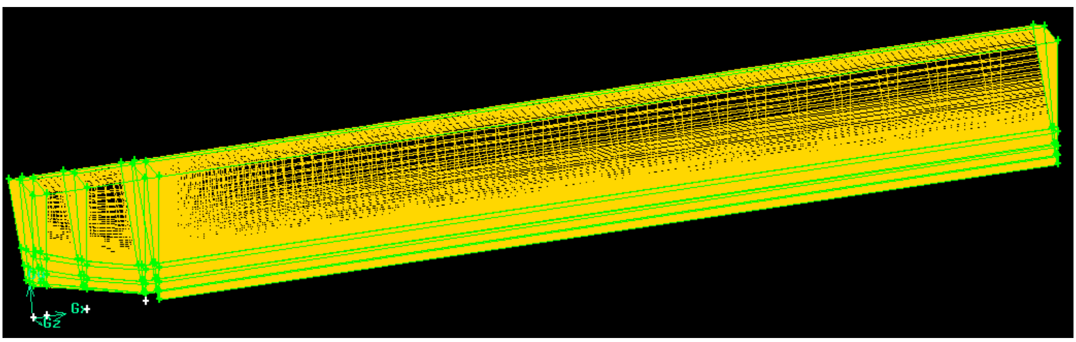


Fig. 4 The image of a sector calculus domain in GAMBIT

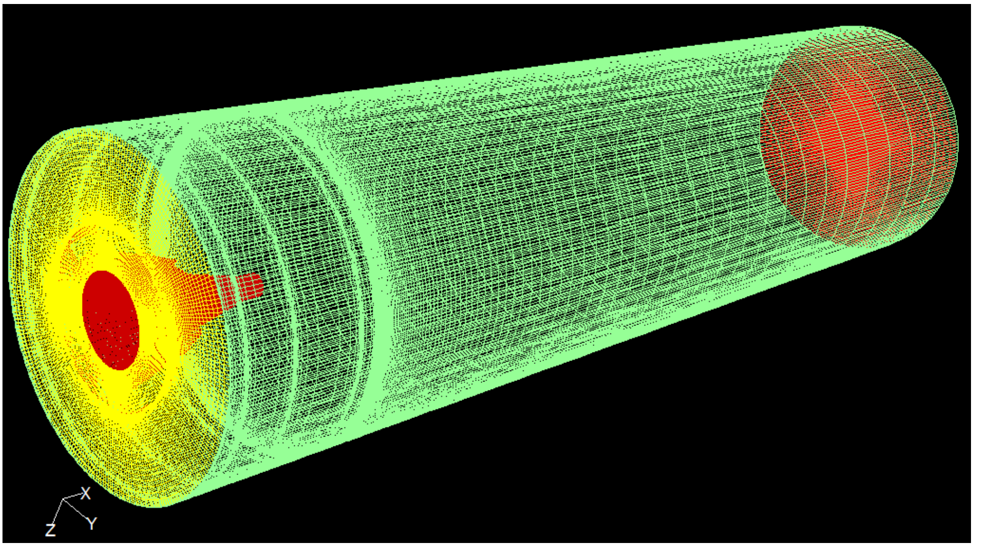


Fig. 5 The image of calculus domain

The chevrons height is calculated with the following formula [8]:

$$h = \frac{\pi D}{2N} \tag{1}$$

where D is the exit nozzle diameter, and N the number of chevrons.

In this paper there are multiple geometries studied as a result of the variation of number of chevrons for the two fluxes.

3. BOUNDARY CONDITIONS

One takes into consideration the following boundary conditions:

- far-field pressure
- outlet pressure
- on the left and right sides of the computational domain, the rotational periodic boundary conditions are imposed.

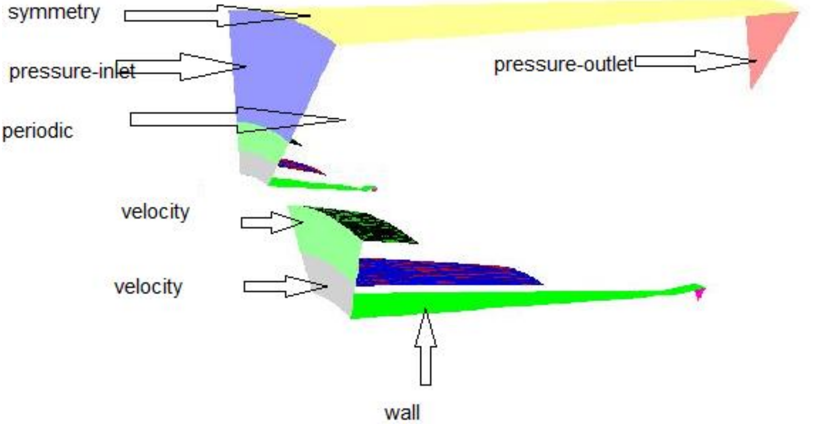


Fig. 6 The boundary conditions for the computational domain

4. GOVERNING EQUATIONS

During the calculus we assume that the fluid is compressible. For a three-dimensional rotating Cartesian coordinate system, the unsteady Reynolds-averaged Navier-Stokes equations are presented below [6]:

1. The continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0 \tag{2}$$

2. The momentum equation:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho (u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} - \rho u_i' u_j'\right) \tag{3}$$

where

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{4}$$

3. The energy equation:

$$\frac{\partial}{\partial t}(\rho H) + \frac{\partial}{\partial x_j}\left(\rho u_j H + \rho u_j' H' - k \frac{\partial T}{\partial x_j} \right) = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j}\left(u_i \tau_{ij} + u_i' \tau_{ij} \right) \tag{5}$$

Following the calculus of the Reynolds Number that has a value of about 10 to the power -6 we assume that the flow is turbulent and the chosen model of turbulence is k-ε. The values of k and ε come from the transport equation of the turbulent kinetic energy and the turbulent dissipation.

The value of the dimensionless velocity at the nozzle wall is in the right domain 1-30 and bigger than 30 [10].

5. COMPUTATIONAL RESULTS

The numerical simulations were performed with commercial CFD code Ansys Fluent [9], and the results obtained are presented as follows:

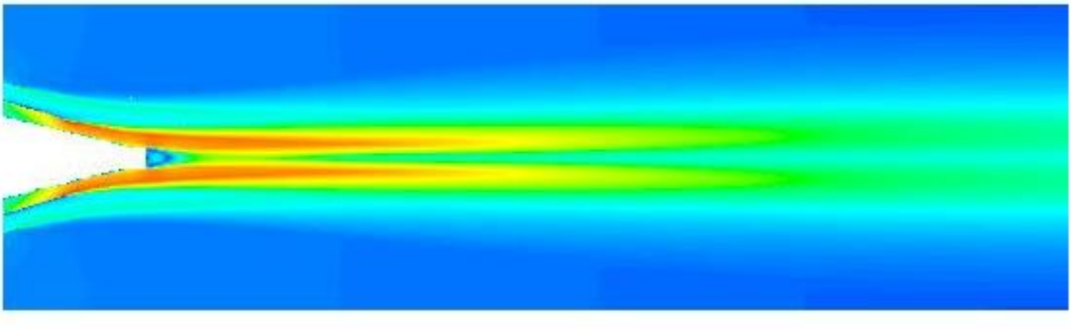


Fig. 7 Velocity distribution along the Ox axis for the nozzle

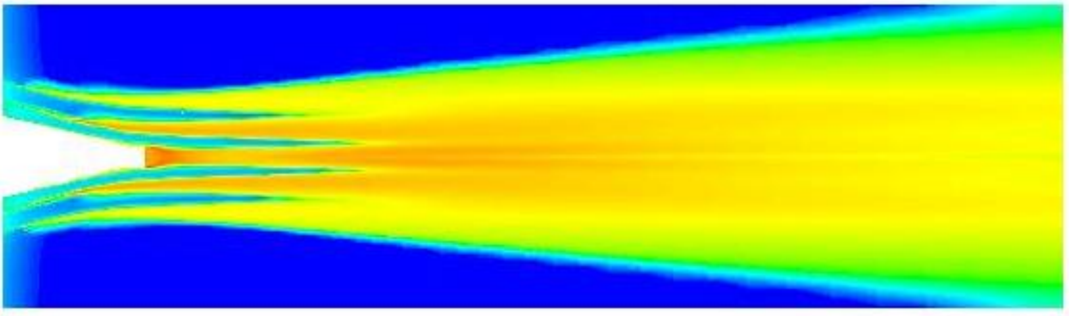


Fig. 8 Acoustic distribution along the Ox axis for the nozzle

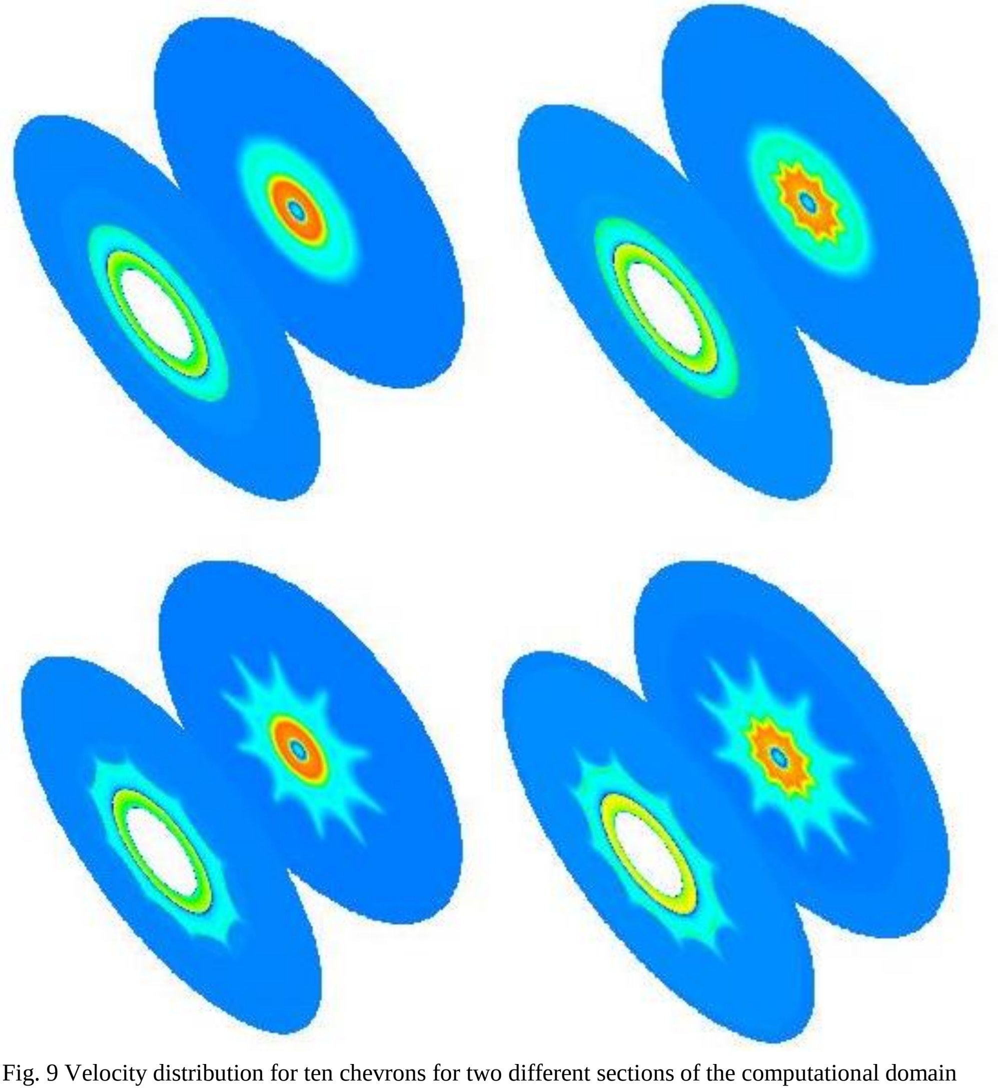


Fig. 9 Velocity distribution for ten chevrons for two different sections of the computational domain at 0.2 and 1.5 meters

The variation of the acoustic variation and speed are presented bellow.
Please note that the values presented on the Ox axis are the following:
"0" corresponds to the nozzle without chevrons

"1" corresponds to the nozzle with triangular chevrons on primary flow
"2" corresponds to the nozzle with triangular chevrons on secondary flow
"3" corresponds to the nozzle with triangular chevrons on primary and secondary flow

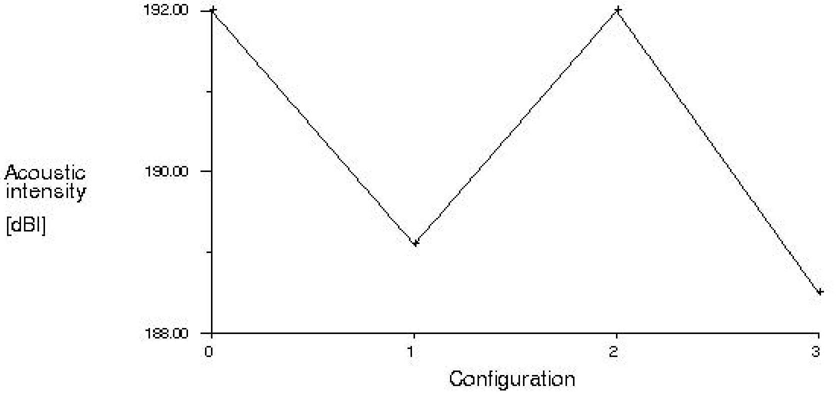


Fig. 10 Acoustic intensity level for ten chevrons cases

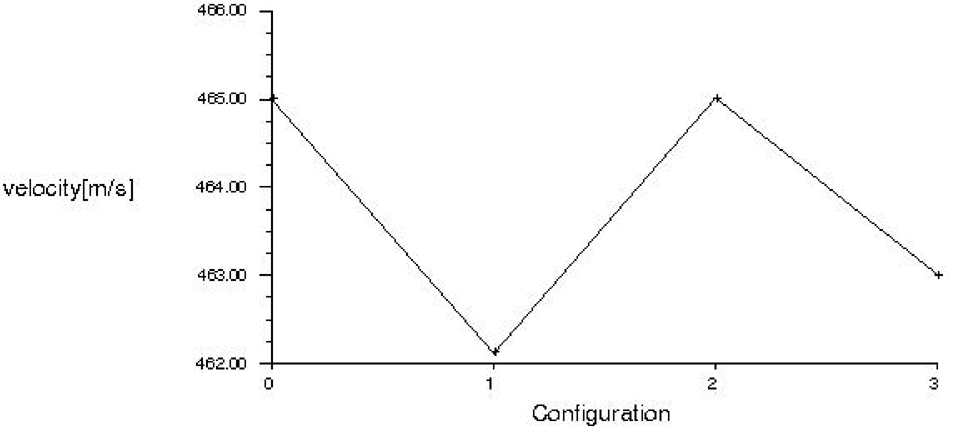


Fig. 11 The evacuation velocity for ten chevrons cases

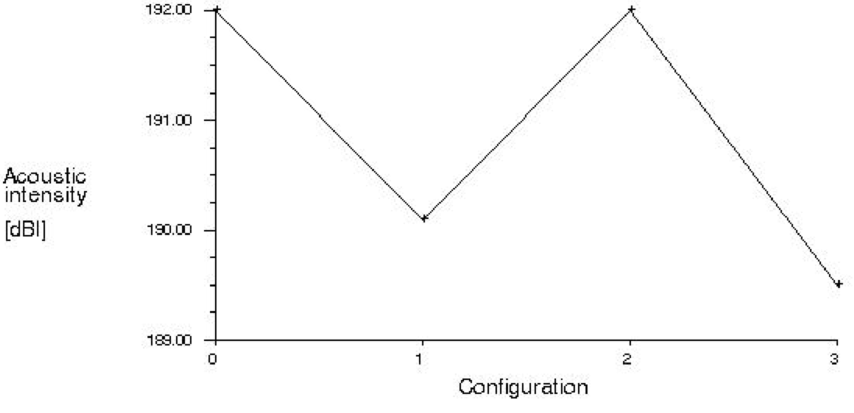


Fig. 12 Acoustic intensity level for twenty chevrons cases

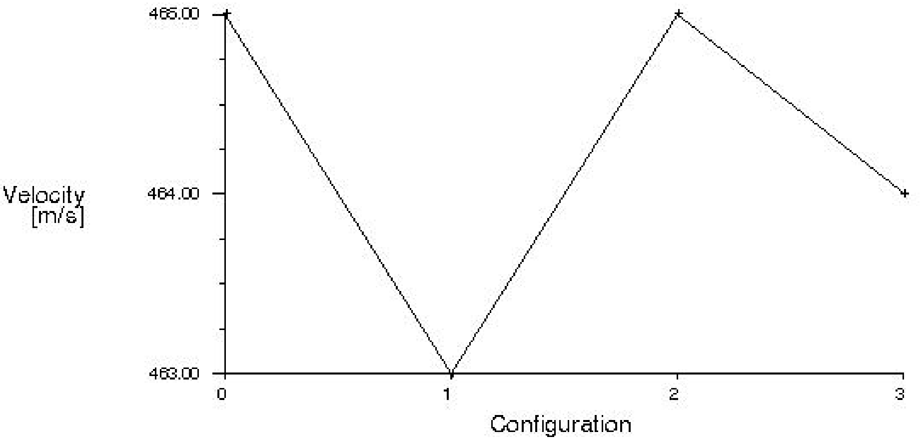


Fig. 13 The evacuation velocity for twenty chevrons cases

6. CONCLUSIONS

As a result of the numerical simulation we can conclude that the triangular chevron helps to reduce the acoustic power by some dB., results that can be compared with the ones obtained from some other experiments [11].

Following the completion of the 7 cases using chevrons it was found that increasing their number will results in noise decrease and also in exit velocity, thus, if reducing the 10 chevrons the acoustic intensity decreases but there is a greater loss of velocity.

Using the chevrons in the flow geometry for both the primary and the secondary flow we obtain the best results.

REFERENCES

[1] M. J. Lighthill, *On Sound Generated Aerodynamically: I. General Theory*, Proceedings of the Royal Society of London, Series A, Vol. **211**, pp. 564-587, 1952.

[2] M. J. Lighthill, *On Sound Generated Aerodynamically: II. Turbulence as a Source of Sound*, Proceedings of the Royal Society of London, Series A, Vol. **222**, pp. 1-32, 1954.

[3] D. Youn, *Jet noise suppression means*, United State Patent Office, No. US3153319, oct.20 1964 pp 7.

[4] R. L. Balzer, *Segment mixing device having chevrons for exhaust noise reduction in jet engines*, United State Patent, No. US6612106, sept. 2, 2003, pp4.

[5] *** https://cwcs.cfm56.com/.

[6] M. Niculescu, S. Danaila, An extraction of the dominant rotor-stator interaction modes by the use of pod tehchnique in a low pressure centrifugal compressor, *U.P.B. Sci. Bull.*, Series D, Vol. **73**, Iss.1, 2011, ISSN 1454-2358.

[7] B. Gherman, V. Stanciu, V. Silivestru, *Comparatia unei noi solutii de motor, motorul tripluflux, cu motorul dubluflux*, Aerospatial Conference, Bucharest, 2008.

[8] Mathews et al., *Tabbed nozzle for jet noise suppression*, United States Patent Office, No. US6314721, nov.13, 2001, pp 8.

[9] *** Ansys 12 Documentation-User's Guide 2004.

[10] V. N. Constantinescu, S. Danaila, S. Galetuse, *Dinamica fluidelor in regim turbulent*, ISBN: 978-973-27-1694-6, Editura Academiei Romane 2008.

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