ANALYTICAL AND NUMERICAL STUDY OF THE NOZZLE JET*

Grigore CICAN[1], Virgil STANCIU[2], Daniel CRUNTEANU[3]

In this article one presents a comparison of the flow parameters: velocity, temperature, Mach's number through a convergent nozzle and of the characteristic measures of the jet produced by it through the analytical method and the numerical analysis using ANSYS-Fluent software.

Keywords: nozzle, jet, flow, characteristic measures, analytical, Fluent.

1. Introduction

A free jet of gas is a stream of gas that enters without obstacles through a gas that has the same or different termophysical properties[1].

A jet of gas is characterized by three parts, namely[2]:

a. the core region where the velocity remains constant.

b. the transition region where the redistribution of velocity takes place, leading to stabilization of the boundary layer thickness. This zone is so small, that in calculus it can be equal to 0.

c. developed region that is the largest one and it exists only if there are movements in the jet.

The three regions of the free jet are presented below.

At the nozzle exit the jet tends nurturing such baseline characteristics in the initial phase of its development, the jet mixing consists of two layers (layers of swirling) separated by a potential core. At a certain distance from the nozzle exit the two layers are mixing together, and the jet becomes completely developed, turbulent flow, stationary (mean) self-similar and invariant [3]. Formation and maintenance consume swirl kinetic energy of the jet in the section. After the disappearance of the potential core, the energy needed for their maintenance is

[1] Ph. D. Student, Faculty of Aerospace Engineering,1-7 Polizu Street, 011061, University POLITEHNICA of Bucharest, Bucharest 1, Romania, e-mail: cicangrig@yahoo.com
[2] Professor, Faculty of Aerospace Engineering,1-7 Polizu Street, 011061, University POLITEHNICA of Bucharest, Bucharest 1, Romania
[3] Lecturer, University POLITEHNICA of Bucharest, Faculty of Aerospace Engineering,1-7 Polizu Street, 011061, Bucharest 1, Romania
The results presented in this article were obtained withthe support of Ministry of Labour, Family and Social Protection through Sectorial Operational Programme Human Resources Development 2007-2013, Contract no. POSDRU/107/1.5/S/76909.

taken from the energy contained in the gas stream. The consequence of consuming the gas's energy is reflected in the flow rate decrease . [1]

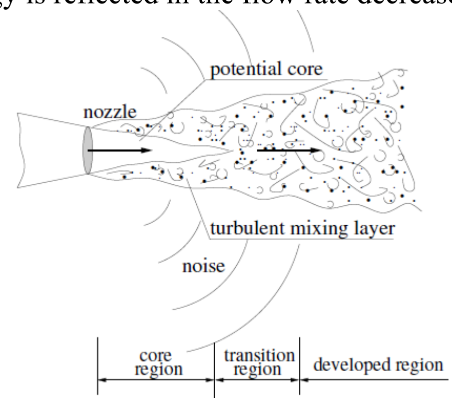


Fig. 1. Schematic of a turbulent jet; sources of noise. [4]

In this paper a computer program was developed in FORTRAN commercial software that resolved analytical the equations governing the flow through a convergent nozzle, and the equations defining characteristic measures of the jet produced by the nozzle. The Fortran results were compared with the ones obtained with the commercial software ANSYS-FLUENT in this way validating the analytical calculation of the simple theory.

2. Analytical calculation model for convergent nozzle and for isothermal jet

For the calculation of subsonic jet nozzle one takes into account the friction between fluid and walls and so, the relaxation process is irreversible adiabatic. The entropy of gas at the nozzle exit is greater than the initial entropy and the actual velocity of the exhaust nozzle section is inferior to the theoretical one therefore one introduces a stall coefficient [5].

Starting with the equations of energy, momentum and continuity one obtaines the formulas to be applied in nozzle parameters calculation.

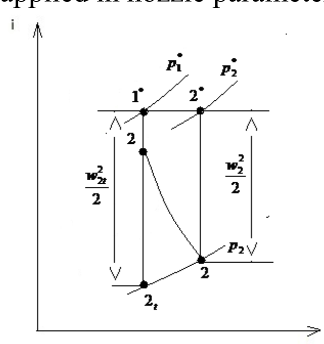


Fig. 2. Relaxation response diagram for the geometrical nozzle.

$$i^* = i + \frac{1}{2} \cdot v^2 = const. \tag{1}$$

$$\frac{T_2}{T_2^*} = 1 - \varphi^2 \left(1 - \left(\frac{p_2}{p_1^*} \right)^{\frac{k-1}{k}} \right) \tag{2}$$

$$\frac{p_2}{p_2^*} = \left(\frac{T_2}{T_2^*} \right)^{\frac{k}{k-1}} = \left(\left(1 - \varphi^2 \left(1 - \left(\frac{p_2}{p_1^*} \right)^{\frac{k-1}{k}} \right) \right) \right)^{\frac{k}{k-1}} \tag{3}$$

In order to calculate the jet characteristics one uses the formulas used in [1]:

Length of core region

$$x_0 = 0.96 \cdot \frac{R_2}{ct} - 0.29 \cdot \frac{R_2}{ct} \tag{4}$$

Velocity variation in axial section

$$v_x = v_2 \cdot \frac{0.96}{0.29 + \frac{ct \cdot x}{R_2}} \tag{5}$$

Velocity variation in radial section

$$v_y = v_x \left(1 - \left(\frac{y}{R_j} \right)^{\frac{3}{2}} \right)^2 \tag{6}$$

Table 1

Dimensionless velocity values w_y / w_x

$\frac{y}{R_j}$	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1
$\frac{w_y}{w_x}$	1	0.907	0.758	0.596	0.443	0.300	0.186	0.098	0.044	0.013	0

Flare angle of the jet is given by the following formula

$$tg\frac{\alpha}{2} = \frac{R_j}{x + 0.29 \cdot \frac{R_2}{ct}} \qquad (7)$$

In practice it was observed that the average flare angle for perfect gas jet has values between 7 and 14 degrees and the length of developed region until the axial velocity of the jet has a value of 0.25 m / s [6].

3. Geometrical model and computational grid

Using analytical results, the geometrical dimensions leads to mesh generation.
The parameters of the nozzle are:
Inlet diameter D1=0. 3256m
Exit diameter D2=0.2422m
Convergence angle: 5 degrees

The flow field length for the jet expansion is equivalent to 20 exit nozzle diameters in axial direction and 10 exit nozzle diameters in radial direction . The calculus grid was produced in GAMBIT by using only a structured mesh.

Knowing that the field is symmetric on axis, one considers only the computational domain comprising an angle of 36 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 500,000 nodes are used. Thus the entire area has a total of 5 million nodes.

In the figure below one observes how this mesh is formated and structured.

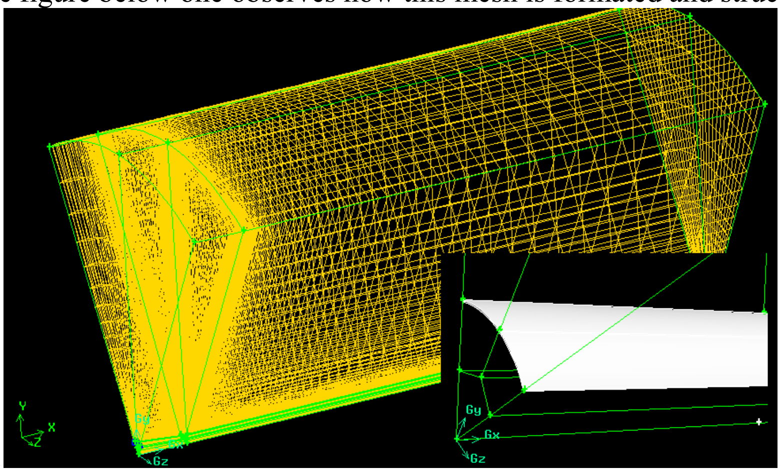


Fig. 3. Gambit structured Mesh and detail of the nozzle sector

The computational domain has a length on Ox axis and on Oy axis equivalent with 20 exit nozzle diameters and 10 exit nozzle diameters.

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

4. Boundary conditions

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.

One takes into consideration the following boundary conditions:

- pressure inlet; - pressure outlet

- at the left and right sides of computational domain, the rotational periodic boundary

-pressure far-field

For the nozzle inlet: the velocity of the fluid is 100 (m/s); the fluid temperature is 345 (K); the pressure of the fluid is 120215 (Pascal)

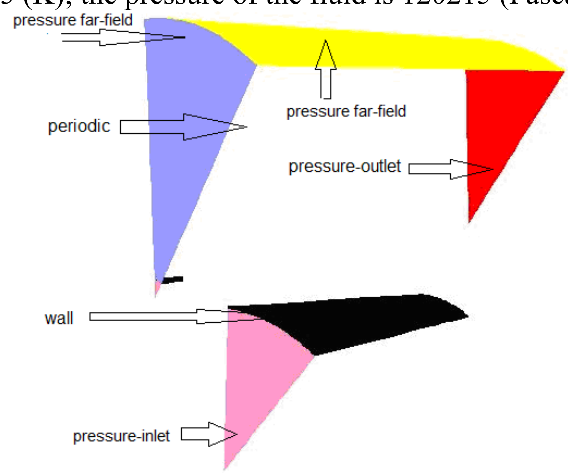


Fig. 4. The boundary conditions for the computational domain

5. Governing equations

During the calculus one considered that the fluid is compressible.

For a three-dimensional rotating Cartesian coordinate system, the unsteady Reynolds averaged

Navier-Stokes equations are presented below [7,8]:

-the continuity equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u_j}{\partial x_j}=0 \tag{8}$$

-the momentum equation:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho (u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} - \rho u_i' u_j'\right) \tag{9}$$

where τ_{ij} represents the viscous tension

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{10}$$

-the energy equation:

$$\frac{\partial}{\partial t}(\rho H) + \frac{\partial}{\partial x_j}\left(\rho u_j H + \rho u_j' H' - k\frac{\partial T}{\partial x_j}\right) = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j}\left(u_i \tau_{ij} + u_i' \tau_{ij}\right) \tag{11}$$

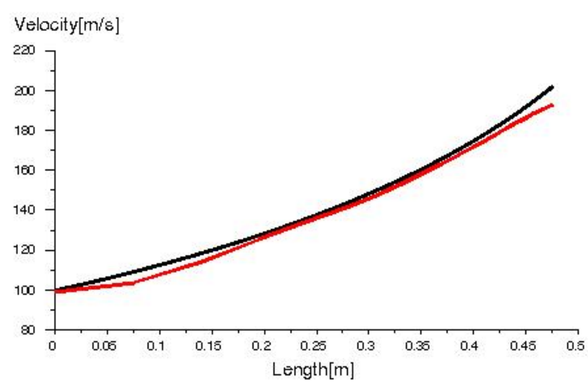
Following the calculus of the Reynolds Number that has a value of about 10 to the power -6 one considers that the flow is turbulent and the chosen model of turbulence is k-ε. The values of k and ε come from the transport equation of the turbulent kinetic energy and the turbulent dissipation.

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

The numerical simulations were performed with commercial CFD code Ansys Fluent [9]

6. Comparisons between analytic and numerical results

After computing the equations governing the flow in subsonic jet nozzles and jets produced by them in FORTRAN and simulation in similar conditions with the commercial software ANSYS-Fluent one presents the following comparisons.



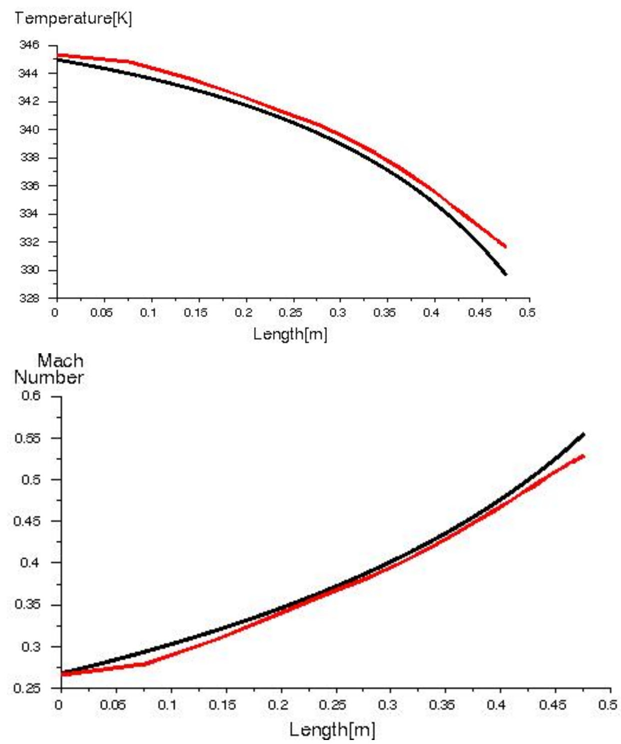


Fig. 5. Comparison between analytical (red) and numerical(Ansys - black) results for inside nozzle

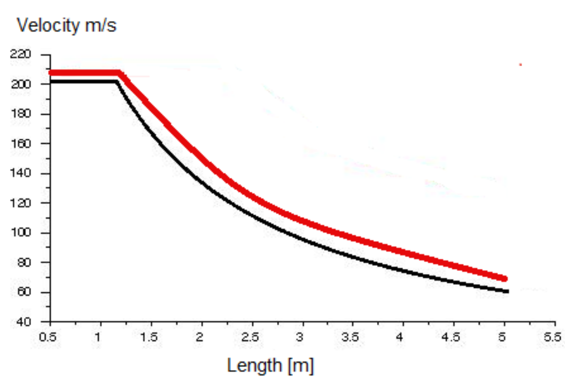


Fig. 6. Comparison between analytical (red) and numerical(Ansys - black) results for jet axial velocity

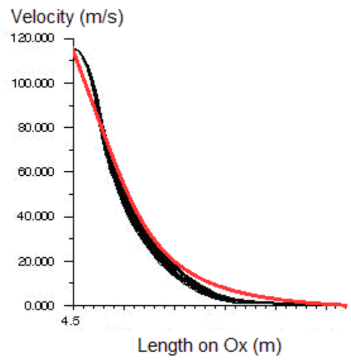


Fig. 7. Comparison between analytical (red) and numerical(Ansys - black) results for jet radial velocity

7. Conclusions

As a result of this study a comparative analysis of the main quantity characterizing a gas jet produced by a subsonic jet pipe was performed.

Visualizing these results one observes that the differences between the numerical and analytical study were quite small.

For a study that is not very detailed in the characteristic measurements of a jet , analytical study is much simpler because it requires less time to solve the equations that govern the flow than the process of achieving a similar geometry in GAMBIT and then for calculation with ANSYS-FLUENT commercial software.

In order to create the analytical program one considers unnecessary a powerfull computer to be taken into consideration, but the numerical study requires one.

REFERENCES

[1] *D. Stefanescu, M Marinesc, I. Ganea,* Technical thermo-gas dynamics, Technical Publishing, Bucharest, 1986

[2] *E. Badarau, M. Gruzescu*, Basics of modern acoustics, Romanian Academy Publishing, 1980

[3] *V. N. Constantinescu, S. Danaila, S. Galetuse*, Turbulent fluid dynamics, Romanian Academy Publishing, Bucharest, 2008

[4] *M. Mihailescu*, Computational Aeroacoustics Based on Large Eddy Simulation and Acoustic Analogies, PhD thesis, 2005

[5] *D. Stanciu, M. Marinescu,* Technical thermodynamics, Printech Publishing, 2002

[6] *I. Fulga, E. Strajescu,* Contributions to fluid dynamics in jetmills by a new aerodynamic model of supersonic flo, U.P.B. Sci. Bull., Series D, Vol. 71, Iss. 4, 2009

[7] *B. Gherman, V. Stanciu, V. Silivestru*, Comparison between one new jet engine solution, three spool engine and the turbofan, Aerospatial Conference, Bucharest, 2008

[8] *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

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