CFD ANALYSIS OF COMPRESSIBLE FLOWS THROUGH NOZZLES AND DIFFUSERS

Kalin KRUMOV[1], Nina PENKOVA[2], Penka ZLATEVA[3]

Mathematical models of compressible adiabatic gas flows in nozzles and diffusors are composed. They were applied for numerical simulation of airflows through converging and diverging channels at subsonic inlet velocities. The models were validated on the base of well-known thermodynamic dependencies for computation of the outlet velocities and temperatures. They are bases for further development in order to investigate supersonic and transient flows at of different geometries of nozzles and diffusers.

Keywords: nozzles, diffusers, compressible gases, finite volume method, numerical simulation.

1. Introduction

Compressible gas jets at nozzles and diffusers are used at different energy converting units as steam and gas turbines, compressor installations, jet and rocket engines, gas ejecting devices, etc. The prediction of the gas behavior is essential for their correct design. The thermodynamically methods for computation of the parameters of the compressible gas flows at nozzles and diffusors are based on the assumptions below [1-4].

1) The flows are accepted as adiabatic. Actually, the changes of the velocity and the pressure cause higher influence on the temperature in comparison to the heat exchange with the environment. Therefore, the heat transfer through the channel walls can be neglected at the computation methods [1]. The ideal gas relation for adiabatic processes describes the changes of the gas flow parameters along the diverging and the converging channels:

$$\frac{dv}{v} + \frac{1}{k}\frac{dp}{p} = 0 \tag{1}$$

where v = specific volume of the gas, m^3/kg; p = absolute pressure, Pa; k = adiabatic index.

2) The processes are assumed as steady state: the gas parameters in the channel do not change over the time and the mass flow of the gas $\dot{m}$ is constant:

---

[1] Sen. assistant prof., Faculty of Metallurgy and Material Science, University of Chemical Technology and Metallurgy, Bulgaria, e-mail: kkrumov@uctm.edu
[2] Assoc. prof., Faculty of Metallurgy and Material Science, University of Chemical Technology and Metallurgy, Bulgaria, e-mail: nina@uctm.edu
[3] Assoc. prof., Department of Thermal Engineering, Technical University of Varna, Bulgaria, e-mail: pzlateva@tu-varna.bg

$$\dot{m} = \frac{fw}{v} = const \tag{2}$$

where f = channel cross section, m^2; w = average velocity for the section, m/s.

3) The relation between the kinetic energy of the gases and their parameters is obtained on the base of the first law of the thermodynamics:

$$dh = -d\left(\frac{w^2}{2}\right) = vdp \tag{3}$$

The gas velocity is usually expressed through the Mach number Ma [1, 3]:

$$Ma = \frac{w}{a} \tag{4}$$

where a is the local speed of the sound in the gas media:

$$a = \sqrt{kpv} = \sqrt{kRT}\ , \tag{5}$$

computed via the specific gas constant R, J/(kgK) and the absolute temperature T.

In principle, all gases are compressible. However, the gas flows are treated as incompressible when the Mach number is less than 0.3 as the density change with the velocity is about 5% at this condition [3]. The dependence between the pressure, Mach number and the cross section change is obtained by joint consideration of the equations above [1]:

$$\frac{df}{f} = \frac{1 - Ma^2}{kMa^2}\frac{dp}{p} \tag{6}$$

The purpose of the nozzles is to reduce the pressure (dp < 0) and to increase the flow rate of the gas. Therefore the channel is:

- converging ($df < 0$) at subsonic inlet velocities (Ma < 1);

- diverging ($df > 0$) at supersonic inlet velocities (Ma > 1).

In the case of diffusor, the velocity is decreasing and the pressure is increasing (dp > 0). Therefore, the channel has to be:

- diverging ($df > 0$) at a subsonic inlet velocity ($Ma < 1$);

- converging (df < 0) at a supersonic inlet velocity ($Ma > 1$).

These relations are illustrated on Fig. 1

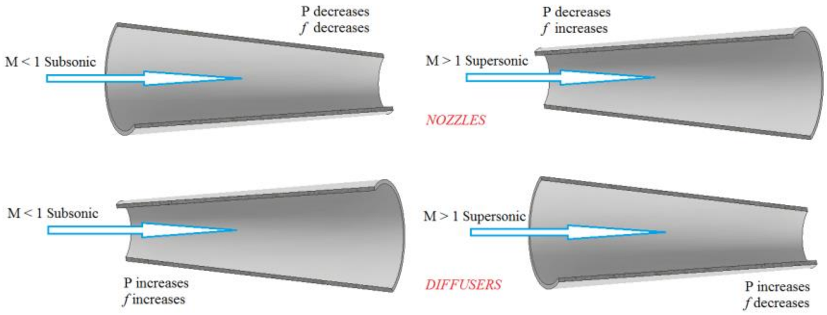


Fig. 1. Nozzles and diffusers

The outlet velocity and temperature at converging nozzles are computed by the equations below, based on (3) and the ideal gas relations [1]:

$$w_2 = \sqrt{2\frac{k}{k-1}RT_1\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]} \tag{7}$$

$$T_2 = T_1\left(\frac{p_1}{p_2}\right)^{\frac{1-k}{k}} \tag{8}$$

All dependences above include area average velocities and temperatures of the gas flows for the cross section of the channels. The computational fluid dynamics (CFD) and the advanced software ANSYS for its implementation allows obtaining of detail information of compressible flows at nozzles and diffusors [5 - 7], taking into account the exact geometries of the channels and the features of the boundary layers. Such numerical simulations and investigations of compressible gas flows at high velocity and pressure gradients have been published in the last decades [8 - 10]. Two dimensional fluid flow fields in De Laval nozzles, obtained via finite volume methods are presented in [8 and 9]. The results have good agreement with the thermodynamically predictions and demonstrate the power of the CFD at such problems. However, the use of this approach for three-dimensional objects without axis symmetry is still an actual scientific challenge [10]. The calibration of the turbulence and boundary layer models, the adjustment of the computational procedures and generation of proper meshes are important and open tasks especially at supersonic speeds and transient flows where shock waves are expected.

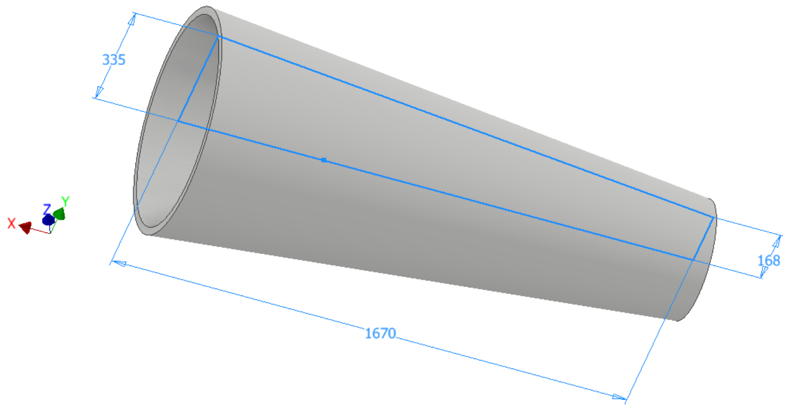
Results of numerical simulations of airflows at converging and diverging channels with simple geometries are presented in this paper. They were implemented in order to develop, calibrate and validate models of compressible gas flows at nozzles and diffusors, suitable for analysis and sizing of complex geometry channels, as this, shown on Fig. 2.

Fig. 2. De-Laval nozzle -a part of a rocket engine

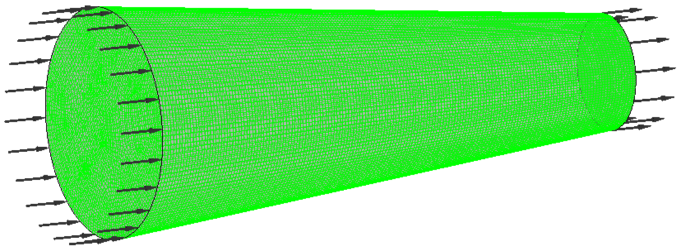
2. Conception for modeling and numerical simulation

The system of partial differential equations, describing the compressible gas flows in nozzles and diffusors includes continuity equation, momentum and energy equations at compressible cases and *k-ε* turbulence model. The last is suitable to model high-velocities flows according to [10]. The flow parameters in the boundary layer are computed using log-law based wall functions [5]. The change of the density and the temperature with the velocity and the pressure are obtained by the ideal gas equation for the subsequent gas. The specific heat capacity, viscosity and thermal conductivity can be used as constant if the temperature change of the gas is less than 100 K. The equations are solved numerically by finite volume technics at boundary conditions, including adiabatic smoot walls, inlet temperature, inlet pressure or velocity and ambient pressure on the outlet. The outlet is accepted as opening boundary to the ambient environment.

For the purpose of the present studies one three dimensional geometrical model can be used as a nozzle and a diffusor, changing the boundary conditions of the inlets and outlets according to Fig. 1. Such gas domain of a converging channel, using as nozzle of an ejector unit in [11] was generated. It was discretized by a mesh, containing 595894 nodes and 580272 elements (Fig. 3). The mesh is fine throughout the gas domain to ensure satisfactory convergence at the iterative computational procedures. The walls of the channel are not modeled as they are accepted as adiabatic.



(a)



(b)

Fig. 3. Dimensions, mm (a) and finite elements mesh (b)

Two numerical simulations of compressible airflows in the cases of a converging (nozzle) and diverging channel (diffusor) are implemented using ANSYS/CFX software, changing the positions of the inlet and outlet. The boundary conditions and some of the results are summarized in Table 1. The adiabatic index and the specific gas constant of the air are accepted k=1.4 and R=287 J/(kgK). The inlet velocities are subsonic at the accepted temperature.

Table 1

Boundary conditions for simulation of the airflow in a nozzle			
Relative inlet pressure	p_1	kPa	50
Inlet temperature	T_1	K	523
Relative ambient pressure at the outlet (opening boundary)	p_2	kPa	0
Results			
Average outlet velocity at the CFD analysis	w_2	m/s	342
Average outlet temperature at the CFD analysis	T_2	K	467
Local speed of sound on the outlet according to (4)	a	m/s	459
Outlet velocity according to (7)	w_2	m/s	339
Outlet temperature according to (8)	T_2	K	466

Table 2

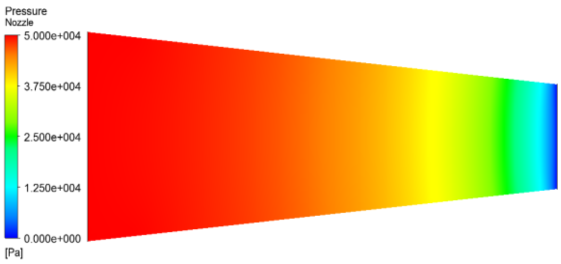
Boundary conditions for simulation of the airflow in a diffusor			
Inlet velocity	w_1	m/s	150
Inlet temperature	T_1	K	523
Relative ambient pressure at the outlet (opening boundary)	p_2	kPa	0
Results			
Average outlet velocity at the CFD analysis	w_2	m/s	36
Relative outlet pressure at the CFD analysis	p_2	kPa	6.13
Average outlet temperature at the CFD analysis	T_2	K	533
Outlet velocity according to (2)	w_2	m/s	36

3. Results and discussion

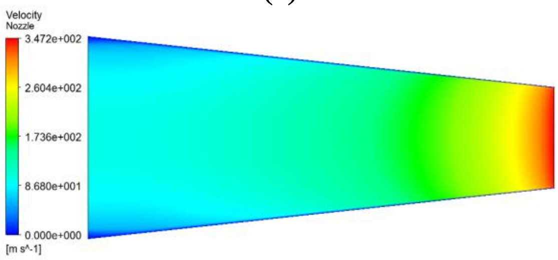
The pressure, temperature and velocity fields are shown on Fig. 4 and 5.

The changes of the pressures and the velocities along the nozzle and the diffusor are according to expectations. The influence of the boundary layer on the velocity and temperature field is negligible at the case of the nozzle: the difference between the maximal velocity (347 m/s) in the center and the area average velocity (342 m/s) on the outlet is about 1% (Fig 4 b). This is not so in the diffuser – the thickness of the dynamic and the thermal boundary layers increase with the decreasing of the velocity along the channel (Fig. 5 b and c).

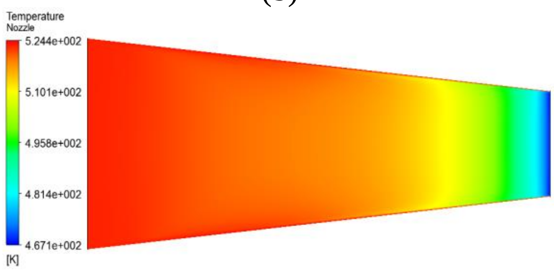
The validation of the model is done comparing the outlet velocity and temperature, obtained by the CFD modeling and the thermodynamically dependencies. The differences between the subsequent outlet velocities and temperatures are 0.9 % and 0.2 % respectively in the case of the nozzle (Table 1). The Mach number on the outlet (0.8) obtained at the simulation completely coincides with the same, obtained by the thermodynamic theory. These differences are smaller in the case of the diffusor.



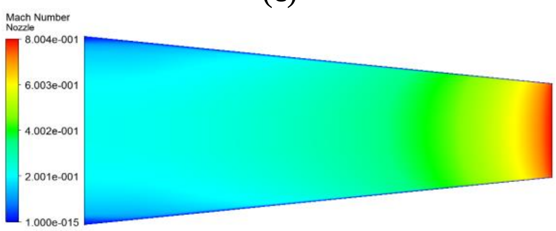
(a)



(b)

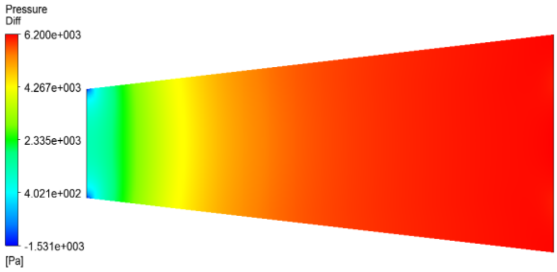


(c)

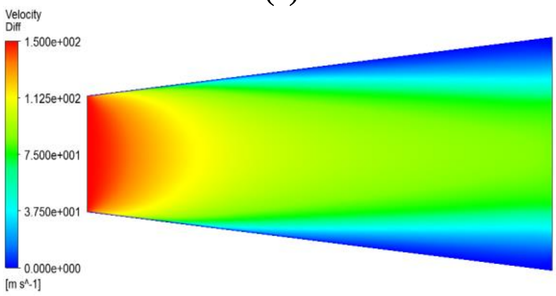


(d)

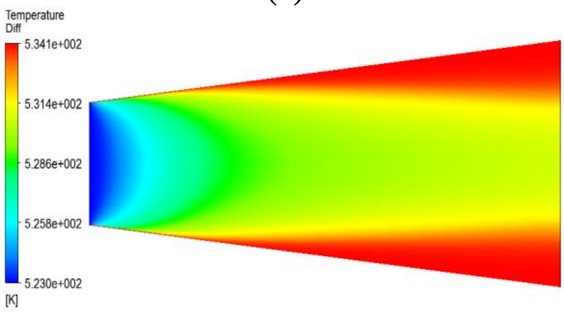
Fig. 4. Relative pressure (a), velocity (b), temperature (c) and Mach number (d), obtained at the CFD simulation of airflow through a nozzle



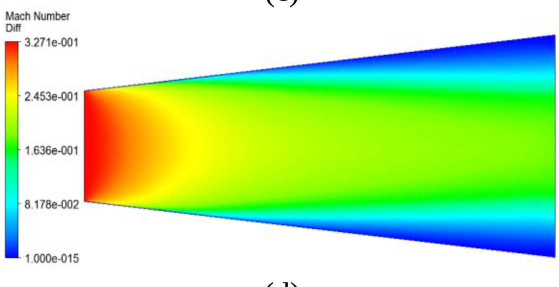
(a)



(b)



(c)



(d)

Fig. 5. Relative pressure (a), velocity (b), temperature (c) and Mach number (d) fields, obtained at the CFD simulation of air flow through a diffusor

Additional simulations are implemented at supersonic inlet velocities. However, unsatisfactory convergences at the computation of the pressure and the velocity are established even at high number of iterations. That impose the use of pseudo transient process to achieve the steady state fields starting from suitable initial conditions and using enough small time steps for proper Courant number.

6. Conclusions

The modeling and CFD techniques allow obtaining of detail information about the 3D fluid flow and temperature fields in compressible gas domains at nozzles and diffusors. They are easy for application for subsonic velocities and difficult for stable solutions at supersonic ones.

A conception for numerical simulation of compressible airflows at subsonic velocities at converging and diverting channels is proposed and tested. The models have good agreement with the thermodynamics of the nozzles and diffusors. They are open for future development for supersonic and transient flows in complex geometry channels concerning the computational methods and technics.

Acknowledgments

The investigations in that paper have been implemented with the financial support of National Programme “Young scientist and postdoctoral students”, funded by Bulgarian Ministry of Education and Science.

R E F E R E N C E S

[1]. *G. V. Wylen, R. Sonntag and C. Borgnakke*, Fundamentals of classical thermodynamics Fourth edition, John Wiley and Sons Inc, New York, 1994.
[2]. *B. K. Sultanian*, Gas Turbines: Internal Flow Systems Modeling, Cambridge University Press, 2018.
[3]. *P. G. Tucker*, Advanced Computational Fluid and Aerodynamics, Cambridge University Press, 2016.
[4]. *B. K. Sultanian*, The Finite Element Method with Heat Transfer and Fluid Mechanics Applications, Cambridge University Press, 2014.
[5]. ANSYS Release 16 - © SAS IP, Inc, 2016.
[6]. *B. E. Launder and D. B. Spalding*, “The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering”, **vol. 3**, issue 2, 1974.
[7]. *W. L. Oberkampf and T. G. Trucano*, “Verification and validation in computational fluid dynamics”, Progress in Aerospace Sciences, **vol. 38**, 2002, pp. 209–272
[8]. *M. S. Patel, S. D. Mane and M. Raman*, Concepts and CFD analysis of De-Laval nozzle, International Journal of Mechanical Engineering and Technology, IJMET, **vol. 7**, issue 5, 2016, pp. 221–240

[9]. *N. D. Deshpande, S. S. Vidwans, P. R. Mahale, R. S. Joshi and K. R. Jagtap,* “Theoretical & CFD analysis of De Laval nozzle”, International Journal of Mechanical and Production Engineering, IJMPE, **vol. 2**, issue 4, 2014, pp. 33–36

[10]. *F. Popescu, R. A. Mahu, N. A. Antonescu and I. V. Ion*, “CFD prediction of combustion in a swirl combustor”, IOP Conf. Ser.: Mater. Sci. Eng, **vol. 444**, 2018, 082009

[11]. *P.V. Levchenko,* Calculation of furnaces and dryers of Silicate industry. Publishing house Higher school, 1968 (in Russian).