CFD SIMULATION OF FLOW REGIMES IN THE SUPERSONIC WIND TUNNEL OF THE FACULTY OF AERONAUTICS

Lubomir Stanko – Peter Gasparovic

Three cases of the working regime of the supersonic wind tunnel of the Aeronautical Faculty are simulated by using CFD (computational fluid dynamics) program ANSYS CFX. CFD simulation results of the wind tunnel are in agreement with existing technical documentation and experiences with the wind tunnel's operation and provide detailed information about inlet airflow speeds, required strength of the models in the test section and requirements of the vacuum pump. It is concluded, that CFD program is able to describe the airflow in Laval nozzle with supersonic speeds, according with published theory of quasi-one-dimensional and two-dimensional airflow.

K e y w o r d s: Wind tunnel, aerodynamic tunnel, Laval nozzle, Mach number, shock wave;

1 INTRODUCTION

Supersonic airspeeds brought many new problems, which had to be overcome to make the supersonic flight possible and safe. Transonic and supersonic aerodynamics is still an object of research, using different methods of simulations, for example flight simulations, computer simulations, or using wind tunnels. Intention of this paper is to investigate the wind tunnel flow regimes by using CFD methods. The goal is to determine effects of wind tunnel control elements on flow regimes, safety, vacuum pump load, and to determine the requirement for strength design of models.

2 INTRODUCTION TO COMPRESSIBLE FLUID DYNAMICS

We consider gasses, which act according physical laws of Boyle-Mariott, Gay-Lusaac and Joule. The first two laws are connected with state equation of perfect gasses. Joule's law tells us that the internal energy of gas depends only on its temperature. Pressure, density and absolute gas temperature or a perfect gasses are bound together with state equation of perfect gasses. Compression of the flexible gas consumes mechanical work, which appears as the heating. If we don't feed, or discharge heat from the outside, each change in gas density corresponds exactly with some change in its temperature. With steady flow is the mass of the gas, which passes through any cross-section [m^3] of current tube, constant: $\rho.S.v$ = const. The final shape of the continuity equation for high speeds: $\rho.S.p = \rho.S.p$ Bernoulli's principle is the fundamental theory of hydrodynamics and gas dynamics and is expressed as law of conservation of energy: At steady flow the total energy of perfect gas along the streamline is constant, this means, that an increase in kinetic energy of the gas, causes potential energy decreasing and vice versa. The final shape of the Bernoulli's principle:

$$\frac{1}{2}.\rho.v_1^2 + p_1 = \frac{1}{2}.\rho.v_2^2 + p_2$$

As seen from fig.1, decrease of cross-section from S_1 to S_2 ($S_1 > S_2$) will cause a speed increase $v_1 < v_2$ and static pressure decrease $p_1 > p_2$.
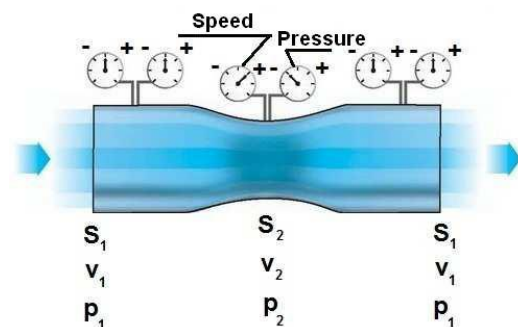

Fig.1 Effect of increasing speed into static pressure decrease

Equation for speed of sound: $c = 20{,}05.\sqrt{T}$

The speed of the sound depends only on the temperature of the gas and it is independent of the pressure or density of the gas, because the ratio is determined by the absolute temperature of the gas. Mach number of the gas flow is defined as a ratio of speed of the gas flow and speed of the sound by this equation: $M = \frac{v}{c} [1]$

This numerical ratio is dimensionless.

2 VISCOUS FLOW TROUGH A LAVAL NOZZLE

The purpose of a Laval nozzle is to create supersonic flow on its outlet. Theodor Meyer computed Laval nozzles, using general equation of inviscid flow by applying two-dimensional Taylor expansions to pregiven velocity distributions on the nozzle centreline, but this is applicable only, when the total pressure ratio across the nozzle exceeds a certain minimum value. For smaller ratios, the shock wave will move upstream to a smaller cross-sectional area due to an entropy increase with given total pressure ratio. The position of the shock wave in the Laval nozzle is determined by the total pressure ratio, that produces the flow trough the nozzle. By decreasing of total pressure ratio an entirely subsonic „venturi flow" is obtained, in which the maximum velocity occurs at the nozzle throat. William Emmons determined by using so-called relaxation method, that the transition from the pure venture flow to the „Meyer flow" begins with supersonic pockets, forming on the walls at the throat, each of them with a certain pressure ratio, terminated by a shock wave. With the pressure ratio increase the two pockets and their shock waves grow to the centerline. At this point the two

sonic lines and the shock waves meet. Then a single shock wave moves downstream and a single sonic line spans the throat area.

3 NUMERICAL SIMULATION OF TWO-DIMENSIONAL LAVAL NOZZLE

In order to evaluate whether the ANSYS CFX is appropriate for our demands, it is confronted with the generally accepted theoretic results. At first, a simple two-dimensional Laval Nozzle without boundary layer was simulated. After model geometry creation and definition of all physical parameters in the sub-program CFX-Pre, the simulation itself is run in the sub-program CFX-Solver Manager. The outlet speed was gradually changed from 20m.s^{-1} to 160m.s^{-1} in order to find Mach number, as close as possible to M=1 at the throat of the nozzle. When the simulation is completed, the results are acquired in the sub-program CFX-Post.
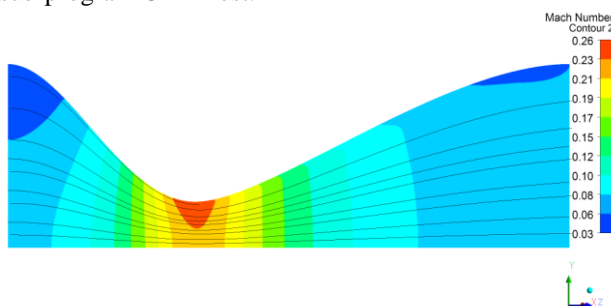
Fig.2 Mach number in the Laval nozzle with outlet speed 20 m.s^{-1} and M$_{max}$=0,23at the centerline of the nozzle
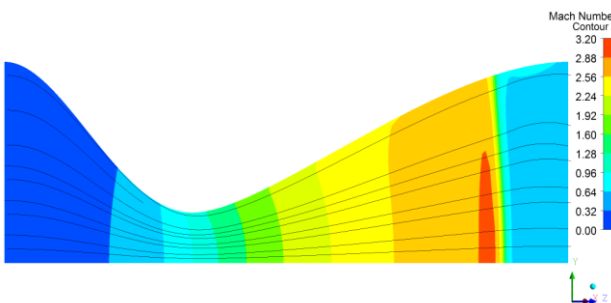

Fig.3 Mach number in the Laval nozzle with outlet speed 160 m.s^{-1} and M$_{max}$=2,926 at the centerline of the nozzle
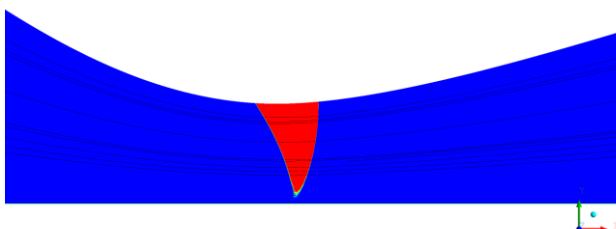

Fig.4 Detailed view to the throat of the nozzle with M$_{max}$=0,9992 at the centerline of the nozzle, blue color is for M<1, red color is for M>1

Especially important are the graphs, describing Mach number and pressure change along the nozzle. The closest outlet speed for achieving the Mach number of 1.0 at the throat, is 48,955 m.s^{-1}.
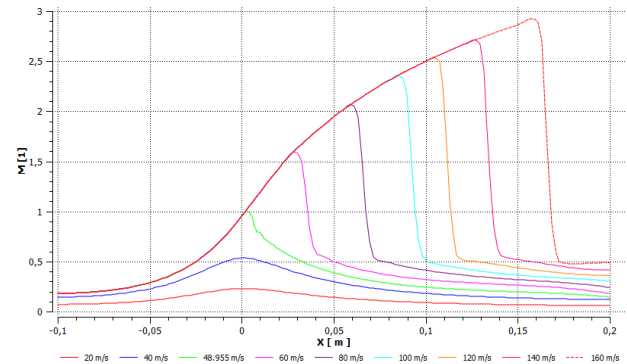

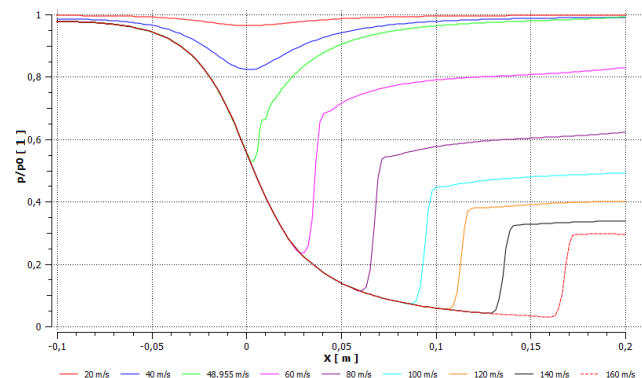**Fig.5 Mach number change along the nozzle, 0 is at the throat**

Fig.6 Pressure change along the nozzle, 0 is at the throat

Shown graphs correspond with a theory so the ANSYS CFX is appropriate for the wind tunnel simulations.

4 MODEL OF THE SUPERSONIC WIND TUNNEL

The first step of the creation process is geometrical model of the wind tunnel. The shape and dimensions are based on measurements of the real wind tunnel, using templates, photos and laser planes. Geometrical model itself was created in CREO Parametric 2.0 program. Geometry is made as thin-walled, and it is sufficient to use only upper-half of the wind tunnel, because the model is symmetrical and physical characteristics of the airflow permit this simplification. Three models of the wind tunnel were created for various control elements settings.
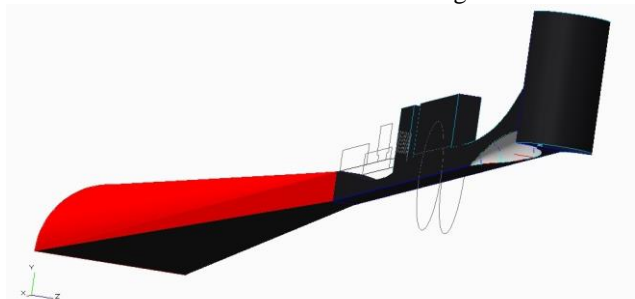

Fig.7 Geometry of the wind tunnel, drawn in CREO Parametric 2.0

Final models as a STEP file are saved and exported them into ICEM CFD program, in which a mesh

model was created. To start creating a mesh model it is needed to create blocks for better mesh associating into geometry. At the inlet of the tunnel is created spherical surface, which carries an information about the surrounding atmosphere. After model mesh creating it is needed to check mesh at the surface and inside the model and to modify blocks, if needed.
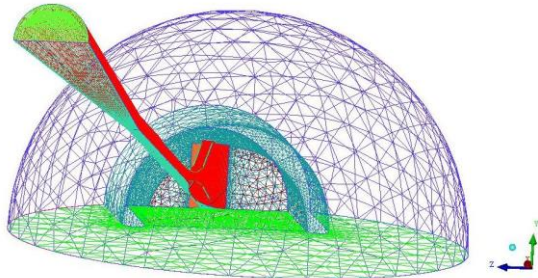

Fig.8 Resultant tunnel mesh

ANSYS CFX was chosen as a program solver and created export as a CFX5 file.

5 NUMERICAL SIMULATION OF THE WIND TUNNEL

In sub-program CFX-Pre all created mesh was imported, defined atmospheric conditions and ordered all parts of the tunnel into domains. Before define run it is needed to set solver control and output control.

5.1 Model with closed throat

The first model has a spindle, moved into full forward position and the rear throttle is down. This model does not have a boundary layer simulated. Together 5 simulations for different outlet mass flow are made in order to find volume flow as close to operating point $(0,58 m^3.s^{-1})$, as possible. These outlet mass flow values are chosen: $0,09 kg.s^{-1}$; $0,10 kg.s^{-1}$; $0,105 kg.s^{-1}$; $0,107 kg.s^{-1}$; $0,11 kg.s^{-1}$; The closest mass flow to needed volume flow is $0,107 kg.s^{-1}$, which has $0,66 m^3.s^{-1}$.
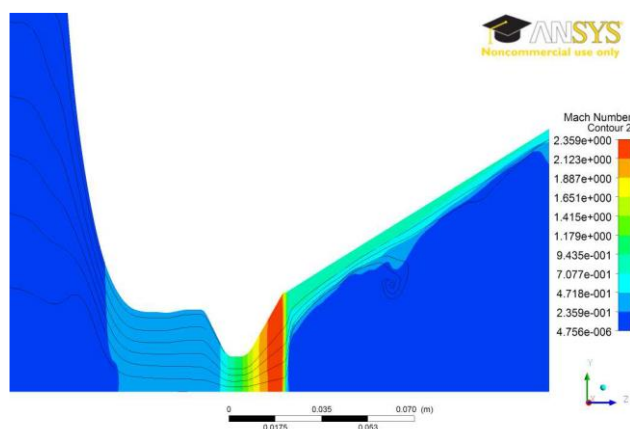

Fig.9 Mach number in closed throat, $Q_m=0,107 kg.s^{-1}$ and M=0,264 in the test section

The next parameters are measured for the wind tunnel in the test section with $Q_m=0,107$ $kg.s^{-1}$: Temperature: 287,132 K; Density: 1,17 $kg.m^3$; Pressure:
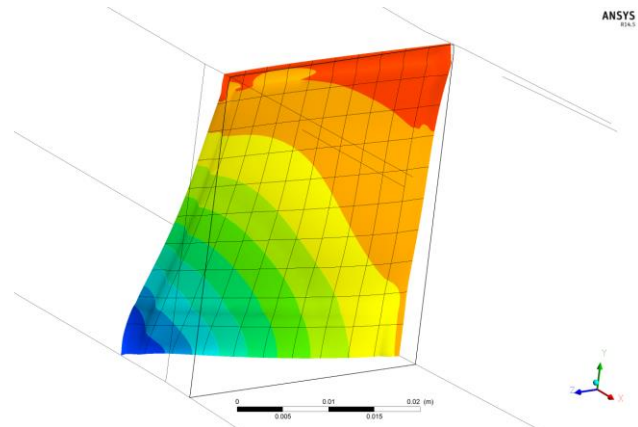
-4864,05 Pa; Velocity: 89,845 $m.s^3$; Mach number: 0,264;

Fig.10 Shock wave in the tunnel with closed throat, $Q_m=0,107$ kg.s^{-1} and M=0,264 in the test section
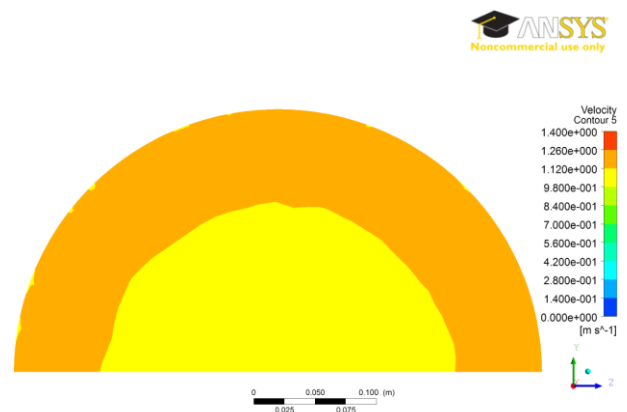

Fig.11 Inlet speed into the wind tunnel with closed throat and M=0,264 in the test section

5.2 Model with open throat

To increase Mach number in the test section it is needed to gradually open the rear throttle. The following outlet mass flow is chosen: $0,25$ $kg.s^{-1}$; $0,26$ $kg.s^{-1}$; $0,27$ $kg.s^{-1}$; $0,28$ $kg.s^{-1}$; $0,29$ $kg.s^{-1}$; The closest mass flow to needed volume flow is $0,27 kg.s^{-1}$, which has $0,529 m^3.s^{-1}$.
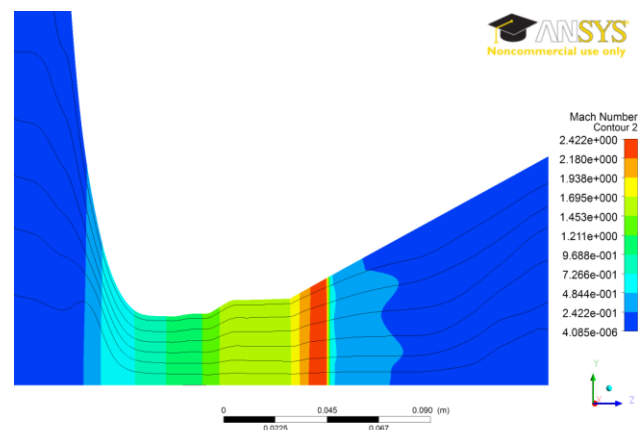

Fig.12 Mach number in open throat, $Q_m=0,27 kg.s^{-1}$ and M=1,154 in the test section

This parameters are determined in the test section with $Q_m=0,27$ kg.s^{-1} and without a boundary layer: Temperature: 229,859 K; Density: 0,67 kg.m^3; Pressure: -57087,5 Pa; Velocity: 350,884 m.s^3; Mach number: 1,154;
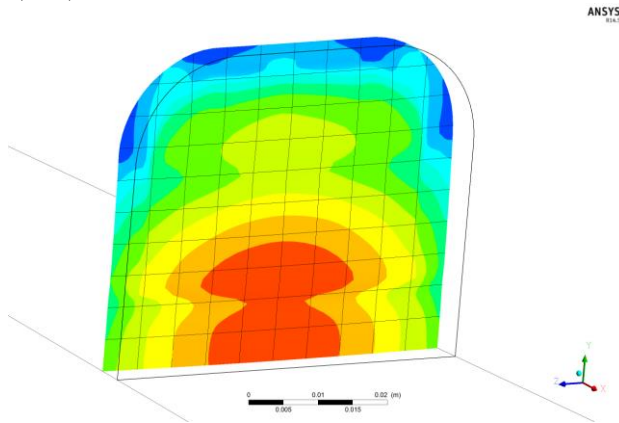

Fig.13 Shock wave in the tunnel with open throat, $Q_m=0,27$ kg.s^{-1} and M=1,154 in the test section
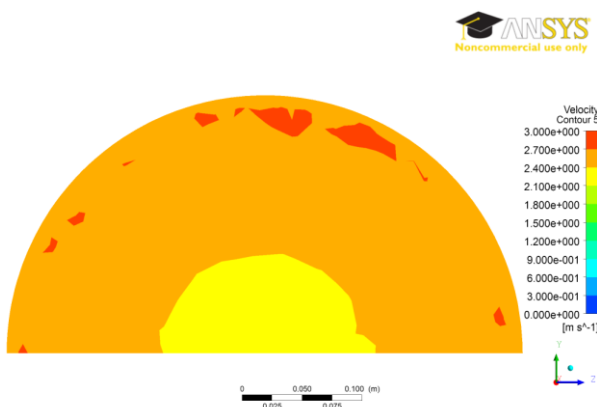

Fig.14 Inlet speed into the wind tunnel with open throat and M=1,154 in the test section

5.3 Model with open throat and with a boundary layer

School computers do not have a sufficient read access memory (RAM) to simulate a boundary layer. They have only 4 GB RAM and 32-bit operating system. For a boundary layer simulating it is needed to have at least 8 GB RAM and 64-bit operating system. One model with a boundary layer was simulated with a boundary layer, using a computer with sufficient RAM.

This model has an outlet mass flow 0,27 kg.s^{-1} with a boundary layer and following parameters in the test section are determined: Temperature: 231,022 K; Density: 0,662 kg.m^3; Pressure: -57413,4 Pa; Velocity: 345,263 m.s^3; Mach number: 1,134;
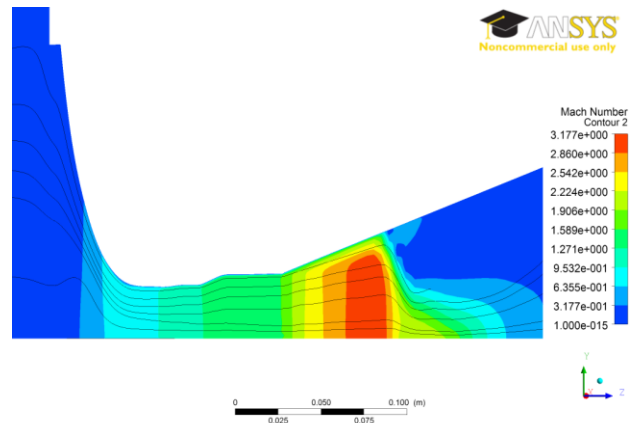

Fig.15 Mach number in open throat, $Q_m=0,27$kg.s^{-1} and M=1,134 in the test section with a boundary layer
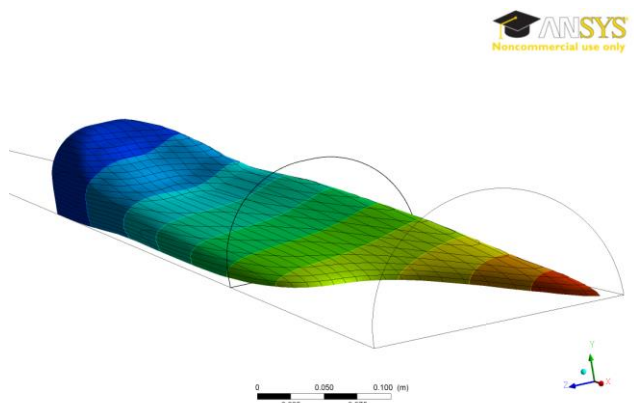

Fig.16 Shock wave in the tunnel with open throat, $Q_m=0,27$ kg.s^{-1} and M=0,264 in the test section with a boundary layer
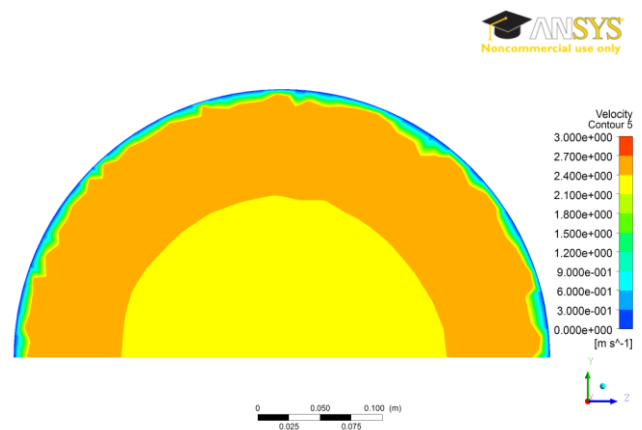

Fig.17 Inlet speed into the wind tunnel with open throat and M=1,134 in the test section with a boundary layer

5.4 Model with open throat and with a spindle in full rearward position

To obtain maximum velocity in the test section it is needed to move a spindle gradually to full rearward position to a place, where is a throat thinner. The narrowest cross-section in this case will be in front of the test section. However, a throat in the narrowest cross-section is too narrow to be able to adapt the existing mesh topology into a cross-section geometry without degenerating of its elements inside a mesh. This created

an elements with inappropriate angles and negative volumes. Possibly, it is needed to make a different blocking order.
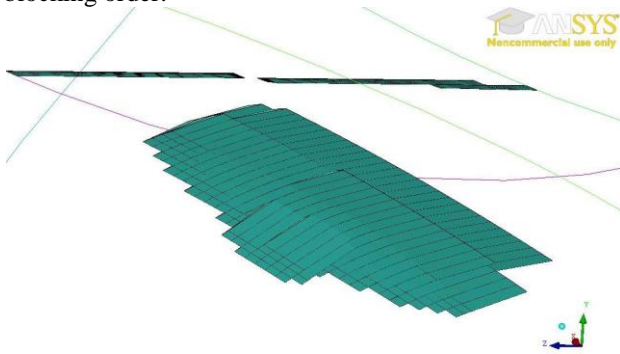

Fig.17 Degenerated elements inside a mesh of the wind tunnel with open throat and a spindle in full rearward position

6 CONCLUSION

For mechanical design of tested models it is needed to assume a maximum density 1,175 kg.m^{-3} and maximum differential pressure 87 435,6 Pa. Vacuum pump must create a negative pressure between 42 797 Pa and 69 861 Pa. Maximum simulated negative pressure is 57 413,4 Pa. Using a rear throttle position change in the test section in the real wind tunnel can create a Mach number between M=0,3 and M=1,1. The resulting Mach number is 0,264 with a closed throat and 1,134 with open throat and a boundary layer. The wind tunnel with open throat and with a spindle in full rearward position failed to simulate in a given time. Probably, it is needed to use a different blocking order. Inlet speed into the wind tunnel is approximately 3m.s-1. This means, that a work nearby inlet is safe. 32 bit operating system with 4 GB RAM is insufficient, because the operating system can use only 2 GB of RAM. For this wind tunnel simulations it is appropriate to use a computer with 64 bit operating system and 8 GB RAM.

BIBLIOGRAPHY

[1] ANDERSON, John: Fundamentals of aerodynamics, third edition. McGraw-Hill : 2001. 912 p. ISBN 0-07-237335-0.
[2] HOŠEK, Josef: Aerodynamika vysokých rýchlostí. Naše vojsko: 1949. 616 p.
[3] RAMM, Heinrich: Fluid Dynamics for the Study of Transonic Flow. Oxford University Press: 1990. 211p. ISBN 0-19-506097-0.
[4] ANSYS CFX Solver Modeling Guide. Ansys, Inc. 2009. 498p.

AUTHOR(S)´ ADDRESS(ES)

Stanko Ľubomír, Bc.
Vyšná Šebastová 290, 080 06 Prešov
lubo.stanko@azet.sk

Peter Gašparovič, Ing., PhD.
Department of Aerodynamics and Simulations
Aeronautical faculty, TUKE
Rampova 7, 041 21 Kosice, Slovak Republic
peter.gasparovic@tuke.sk