TECHNICAL TRANSACTIONS | **CZASOPISMO TECHNICZNE**
MECHANICS | MECHANIKA

1-M/2015

PIOTR CYKLIS*, PRZEMYSŁAW MŁYNARCZYK*

NOZZLE SUPPRESSED PULSATING FLOW CFD SIMULATION ISSUES

PROBLEMY SYMULACJI CFD PULSUJĄCEGO PRZEPŁYWU TŁUMIONEGO PRZEZ DYSZĘ

A b s t r a c t

Pressure pulsations in volumetric compressor manifolds are one of the most important problems in compressor operation. These problems occur not only in huge compressor systems such as those used in natural gas piping in gas mines or national transport systems, but also in small refrigeration compressors found in domestic applications. Nowadays, systems require a new approach since in all applications, variable revolution speed compressors are introduced. Mufflers designed in a conventional way on the basis of the Helmholtz theory only have good pressure pulsation damping action within the designed frequency range. In the case of revolution speed change, the reaction of the damper designed according to the Helmholtz theory may be insufficient. Therefore, any innovative ideas for pressure attenuation is welcomed by the compressor industry. One of the possibilities to attenuate pressure pulsations over a wide range of frequencies is the introduction of specially shaped nozzles in the gas duct flow directly after the compressor outlet chamber. It is obvious that the nozzle attenuates pressure and flow pulsations due to energy dissipation, but at the same time, it also raises the requirement for the pumping power of the compressor. The estimation of nozzle pulsation attenuation may be assessed using CFD simulation. In the paper, the influence of the time step and viscous models choices have been shown. The differences between viscous and inviscid gas models have been shown.

Keywords: *CFD simulations*, *Pressure pulsations damping*, *nozzle gas flow*

S t r e s z c z e n i e

Pulsacje ciśnienia w instalacjach sprężarek wyporowych są jednym z najważniejszych problemów w eksploatacji sprężarek. Problem ten pojawia się nie tylko w dużych systemach sprężarkowych, jak na przykład w sprężarkach gazu ziemnego w kopalniach i rurociągach transportowych, ale również w małych sprężarkach chłodniczych w zastosowaniach domowych. Aktualnie te systemy wymagają nowego podejścia, jako że we wszystkich zastosowaniach wprowadzane są sprężarki o zmiennych prędkościach obrotowych. Tłumiki projektowane zgodnie z teorią Helmholtza mają dobre wskaźniki tłumienia pulsacji tylko w projektowym zakresie częstotliwości. W przypadku zmian prędkości obrotowej sprężarki działanie tłumika opartego na teorii Helmholtza może być niewystarczające. Dlatego każda innowacyjna technika tłumienia pulsacji ciśnienia jest oczekiwana przez przemysł sprężarkowy. Jedną z możliwości tłumienia pulsacji w szerokim zakresie jest zastosowanie zwężek kształtowych wprowadzonych bezpośrednio na tłoczeniu sprężarki. Oczywiście zwężka taka ogranicza pulsacje, powodując jednak równocześnie zwiększenie mocy potrzebnej do sprężania czynnika. Ocena efektywności tłumienia pulsacji ciśnienia może być oceniona na podstawie wyników symulacji CFD. W pracy pokazano wpływ modelu gazu lepkiego i doboru kroku czasowego na wyniki symulacji. Przedstawiono także różnice w wynikach dla gazu ściśliwego i nieściśliwego

Słowa kluczowe: *symulacje CFD*, *pulsacje ciśnienia*, *przepływ gazu w dyszy*

* Prof. Ph.D. D.Sc. Eng. Piotr Cyklis, M.Sc. Eng. Przemysław Młynarczyk, Faculty of Mechanical Engineering, Cracow University of Technology.

Nomenclature

ξ – damping coefficient

1. Introduction

Pressure pulsation attenuation in volumetric compressor manifolds is still one of the most difficult problems to solve in volumetric compressor manifolds.

The pulsating flow causes the following problems:

- system vibration which cause system damage,
- noise which is very unwelcome in small refrigerant compressor systems,
- interaction between the frequency of valve oscillations and pulsation frequency which affects the dynamic performance of valves causing dynamic leaks or premature wear,
- dynamic boost or weakening which causes a problem witch directly impact on the power of compression.

Standard pressure attenuators have many disadvantages as volume dampers, especially in the case of variable revolution speed, therefore, finding other solutions is very desirable for the compressor industry. The modelling of pressure pulsation attenuation is widely analysed in many papers dealing with problems in periodically working machine installations like compressors, pumps or engines. Various numerical methods used for calculating transmission loss in pipelines, mufflers and silencer systems are described in different studies. In [1, 5], comparison of experimental results and Helmholtz model results of pressure pulsations in existing installations are discussed. The author [1] shows that the error of the conventional Helmholtz method may in some cases reach 90% and after introducing a new transmittance matrix method, significant improvements have been achieved. The Helmholtz model method has been applied by the authors [2, 4, 9, 10] to simulate and analyse different acoustic systems.

There are many published investigations concerning car mufflers. The theory is similar to the volumetric compressor mufflers theory. In [4] the transfer matrix of the muffler is calculated and compared to the CFD simulations and test rig experiments. In paper [8], an algorithm for the efficient acoustic analysis of silencers of any general geometry with a transfer matrix is shown. In [7], a three-dimensional finite element approach for predicting the transmission loss in mufflers and silencers is presented. In paper [11], the effect of roughness and the distribution of holes in the long concentric perforated resonator were studied. The main difference between car and compressor mufflers is their size. The car muffler always has a free outlet and the compressor muffler works in the manifold therefore the manifold reaction has to be considered.

Paper [6] describes a CFD simulation of a single pipe excited with a single disturbance. The response, which is periodic with a constant frequency, is characterized by a certain degree of damping. The paper shows that the analysis of pressure pulsation damping by different elements is important. However there is still an issue how the CFD simulation results can be applied for Helmholtz zero dimensional model.

2. Investigated muffling elements

In this paper, passive choking elements mounted in the compressor manifold are investigated as pulsation attenuators. The possibility of passive damping of the pressure pulsations using specially shaped nozzles placed in the gas duct flow directly after the compressor outlet chamber has been analysed. Arbitrary chosen nozzle shapes have been prepared for experimental analysis of pressure pulsation damping. In Fig. 1, examples of nozzle geometries are shown. Three main nozzle profiles in different configurations and size(Venturi orifice, Venturi nozzle and hyperboloidal nozzle)were chosen as most promising for pressure attenuation with low flow restriction.

The key element of this investigation is the assessment of the influence of the nozzle on pulsation on the basis of computer simulation. This method was proposed in [2]. In our method, each manifold element may be characterised by its transmittance. Transmittance describes the response of the element to flow excitation for upward and downward flow.

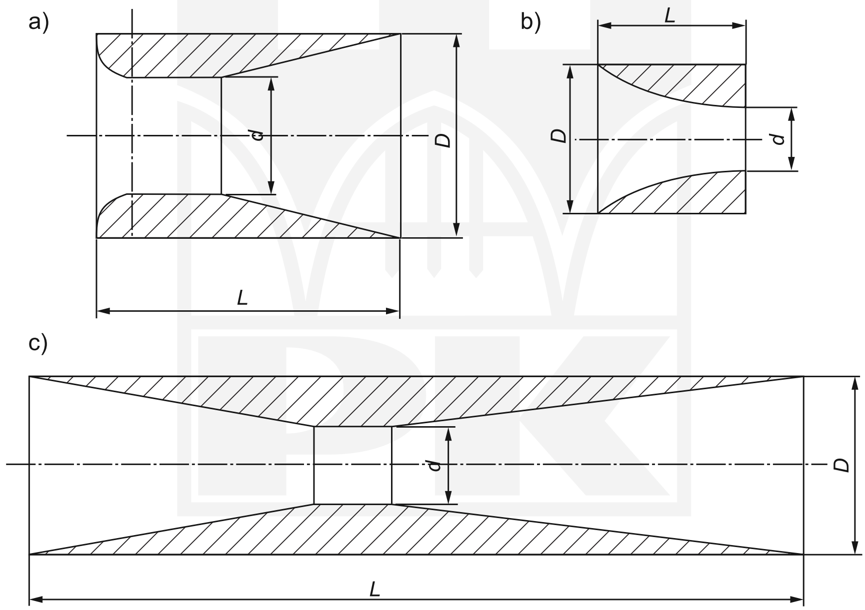


Fig. 1. Shape and main dimensions of the a) Venturinozzle, b) Hiperboloidal nozzle, c) Venturi orifice

In the case of a manifold element, there are two physical phenomena – pressure and flow pulsations which may be used in calculation as excitation or response. There are two ways to calculate transmittances – experimental [1] or theoretical based on the CFD simulation [2]. The concept of the method is as follows: for a considered element of a manifold, a full multi-dimensional CFD non-linear simulation is carried out, solving the Navier-Stokes set of equations numerically together with the necessary closing models, i.e. gas state model, turbulence model, boundary conditions. The obtained results are averaged at the inlet and outlet of the element in question, then a complex transformation of the results is performed so that the transmittances consistent with the generalized form of matrices are calculated. In this way, the advantages of both methods can be combined – the Helmholtz model possibility of analysis of geometrically complex installations and the possibility of introducing real geometry of any element, without priori simplifications.

Using CFD, it is convenient to put the closed end with the closing impedance $Z_k = \propto$ and $M_2 = 0$ or an open end with $Z_k = 0$ and $P_2 = 0$ as a boundary condition. Therefore, the CFD simulation with impulse flow excitation using CFD methods is critical to assessing the nozzle element influence on pressure pulsations.

3. Simulation results

Several simulation problems were studied in order to find out the best possible simulation method. The FULENT software package was used with several simulation parameters. First, the inviscid model was applied, then the Spalart-Allmaras (S-A) and finally, the Reynolds stress model (RSM). For inviscid simulation,a1mmdefault mesh was used, for SA and RSM, three boundary layers were introduced as shown in Fig. 2.

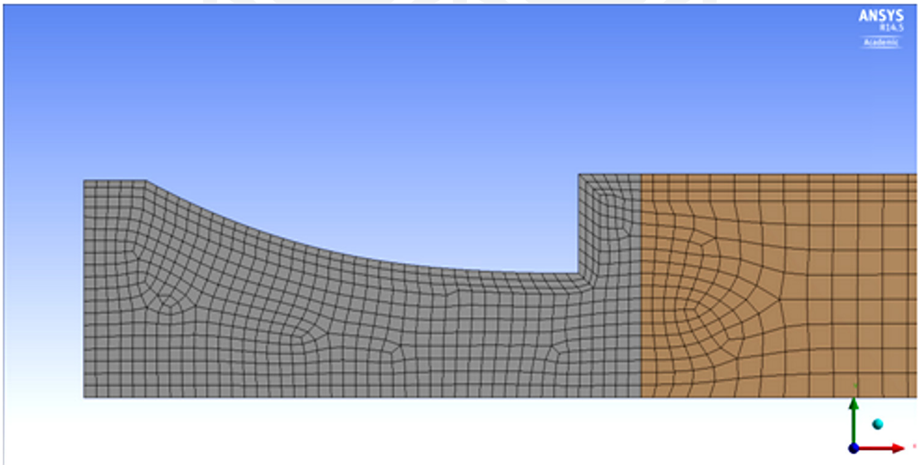


Fig. 2. Mesh with three boundary layers for S-A and RSM models

Boundary conditions:

- At the inlet impulse excitation of the 0.1 [kg/s] peak mass inflow is introduced. The impulse excitation means in numerical application that its duration is equal to one time step. The mass flow in all other time steps is zero.
- Pressure outlet where the pressure at the outlet is defined as the arithmetical average between the pressure outside the domain and the last cell inside the domain.
- Wall (also for closed end elements) where tangential stresses are included in the momentum conservation equation. Velocity at the wall is equal to zero.

The ideal gas isentropic flow model has been applied. The flow is turbulent due to unsteady excitation and high peak velocity. Mach number approximately 0.46.

Results have been obtained in 2D mode using axial symmetry.

The results were spatially averaged at the inlet and outlet to obtain one dimensional flow and pressure pulsations. For closed elements for both direction flows, the pressure pulsation is the response and for open elements mass flow rate for impulse inflow excitation. Examples of this flow are shown in Fig. 3.

It is clearly visible that the inviscid flow model application results in the highest amplitudes of pressure and flow pulsations. The RSM and S-A model gave similar results.

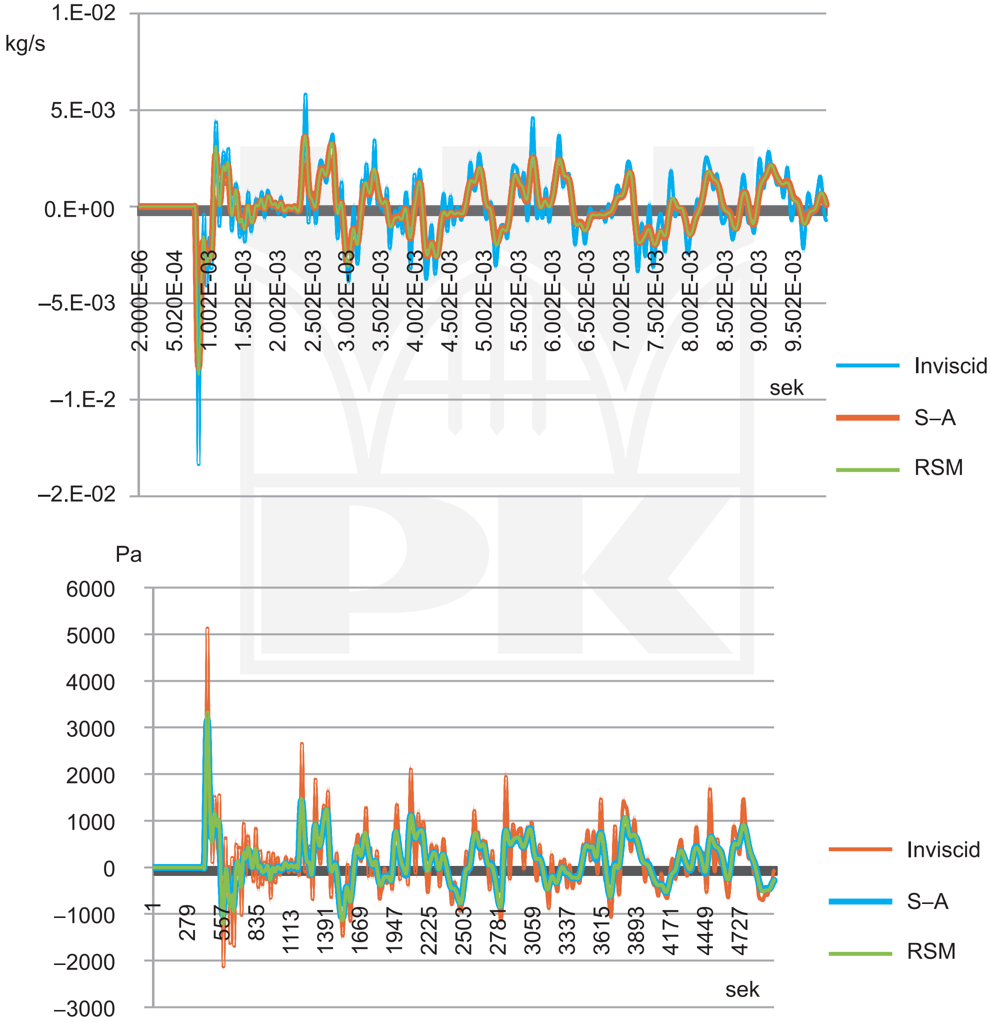


Fig. 3. Outlet pulsations at the outlet of open and closed element with the hyperboloidal nozzle *fi* = 20 [mm]

In Fig. 4, the comparison of the time step influence on the results has been presented. As can be seen, there is nearly no influence of the time step on the pulsation simulation results. The fixed time step was applied. As a result from this investigation, a fixed time step of 2e-6 sec was selected for future simulation.

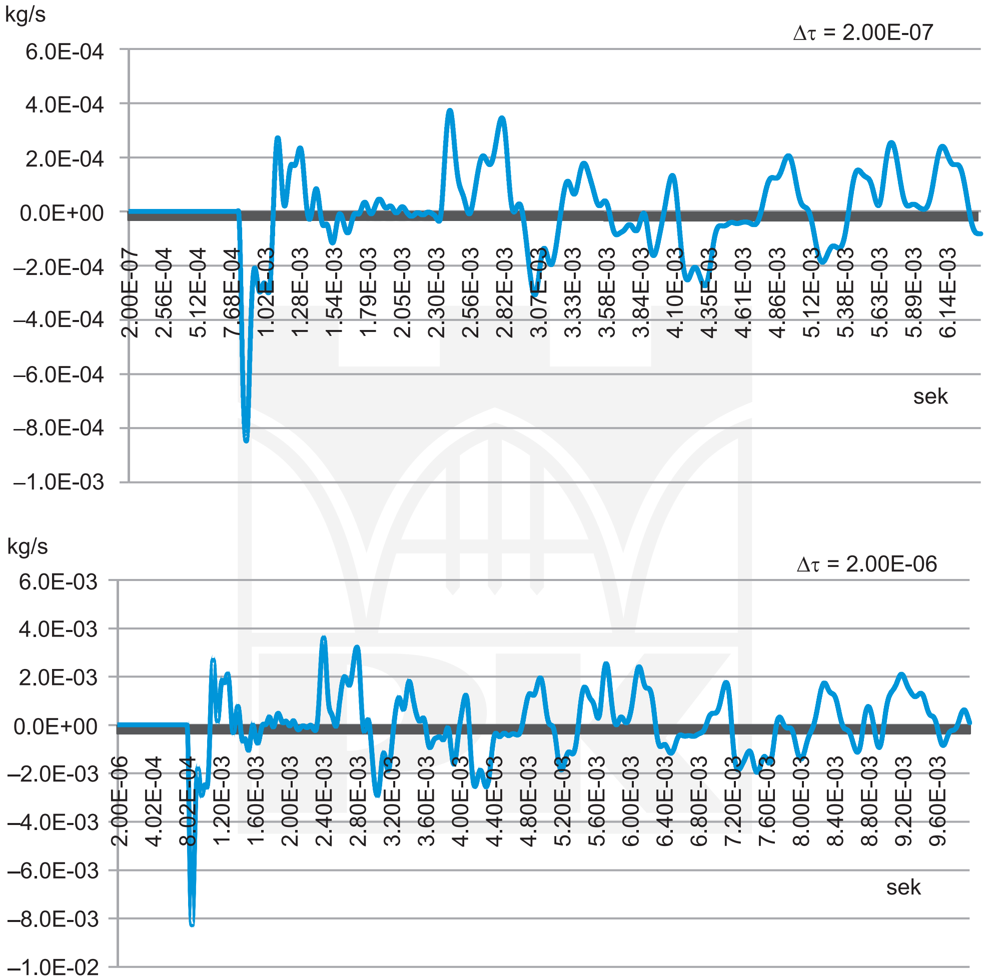
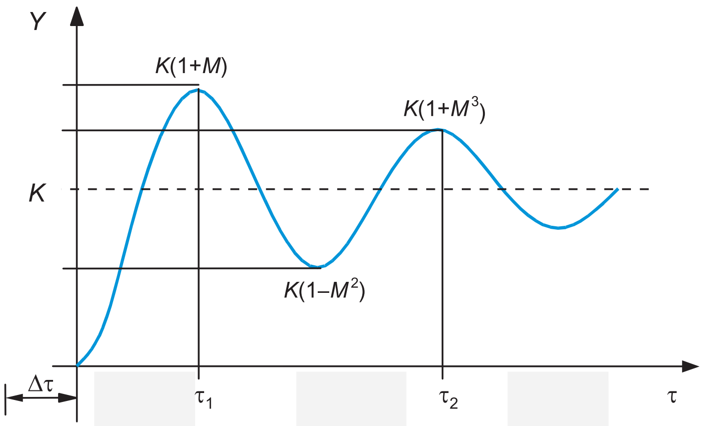


Fig. 4. Comparison of the pressure pulsation simulation for different time steps

4. Comparison of different viscosity approaches

The method for the parameter estimation of generalised transmittances is shown in Fig. 5. The damping coefficient ξ, free frequency ω, delay time $\Delta\tau$, and amplification coefficient K can be estimated by analysing pulsation curves shown in Fig 4. The problem requires the decomposition of the function for each free frequency. The method assumes linearity as the concept of transmittance requires. However, the linearization is used on the 'a posteriori' simulation results, and CFD simulation is not linear in general.



$$\xi = \frac{-\log M}{\sqrt{\pi^2 + (\log M)^2}}, \quad \omega = \frac{2}{T}\sqrt{\pi^2 + (\log M)^2}$$

Fig. 5. Method of estimation of generalised transmittance parameters

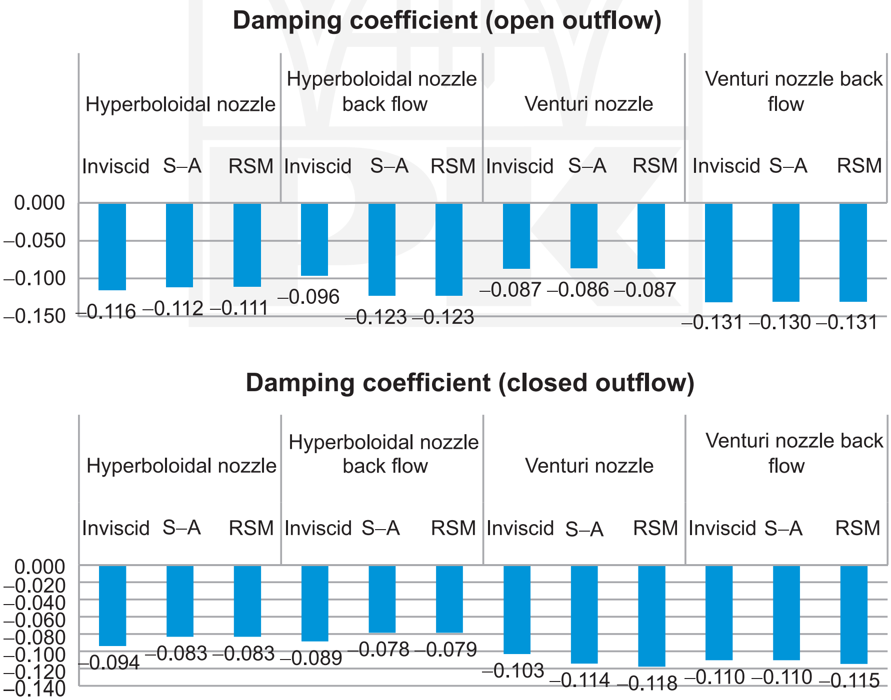
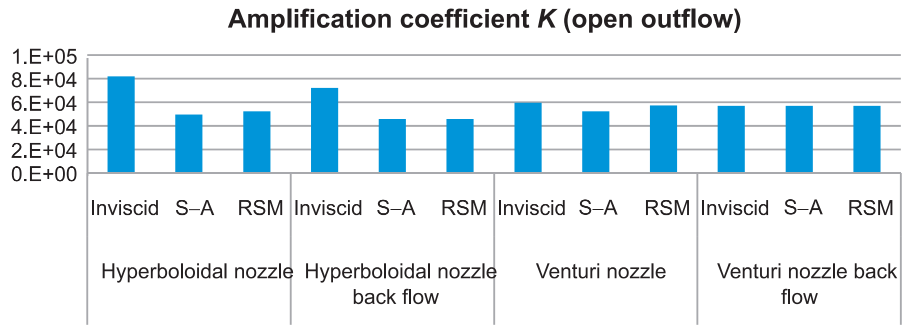


Fig. 6. Comparison of damping coefficient calculations for different cases (dimensionless)



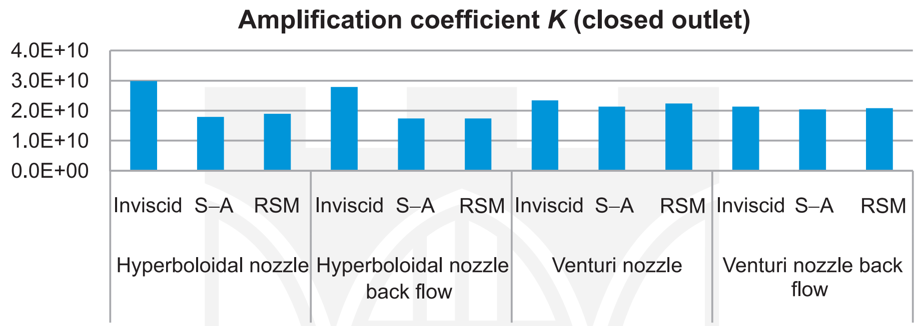


Fig. 7. Comparison of amplification coefficient K calculations for different cases (dimensionless)

In Fig. 6 and 7, only damping and amplification coefficients have been shown. All other parameters have been calculated, but due to the lack of space, they will not be shown here. However, the comparison is significant and may be used for all transmittance parameters. The comparison made for onward and backward flows, for three viscosity approaches (inviscid, RSM and S-A), for the open and closed ends of the pipe containing the investigated nozzle. Only two shapes have been shown (the Venuri and hyperboloidal nozzle)as in our experiments, those two showed the most promising results. Other shapes and dimensions have also been investigated. As may be expected, for most cases inviscid simulation gave a higher K value (amplification coefficient). It has been expected that the damping coefficient for inviscid simulation (absolute value) will be much lower than for viscous flow, but this is true only in some cases. For the hyperboloidal nozzle, it is even higher than for the viscous flow simulation. This is as a result of a high K coefficient, which causes a high output for the first pulsation wave.

5. Conclusions

The concept of introducing shaped nozzles for pressure pulsation attenuation is presented in this paper. The main problem is to estimate which nozzle shape is more effective for pressure pulsation damping with the lowest gas flow chocking. This can be assessed experimentally, but using this approach, shape and dimension optimisation is strongly limited. That is why the CFD simulation can be applied. The application of the CFD code leads to the generalised transmittances for a shaped nozzle, however, important questions arise, concerning simulation models and parameters. In this paper, the influence of the time step and viscous models is demonstrated. The proper time step choice is shown. The viscous gas model showed significant difference when comparing with the inviscid model. The viscosity models RSM or S-A give similar results. The choice for turbulence model depends upon simulation time.

References

[1] Cyklis P., *Experimental identification of the transmittance matrix for any element of the pulsating gas manifold*, Journal of Sound and Vibration, 244, 2001, 859-870.

[2] Cyklis P., *Transmittance estimation for any element of volumetric compressor manifold using CFD simulation*, The Archive of Mechanical Engineering, No. 2, Vol. LVI, 2009.

[3] Georges S.N.Y., Jordan R., Thieme F.A., Bento Coelho J.L., Arenas J.P., *Muffler Modelling by Transfer Matrix Method and Experimental Verification*, ABCM, Vol. XXVII, No. 2, 132-140.

[4] Andersen K.S., *Analysing Muffler Performance Using the Transfer Matrix Method*, COMSOL Conference, Hannover 2008.

[5] Ma Y.-C., Min O.-K., *Pressure calculation in a compressor cylinder by a modified new Helmholtz modelling*, Journal of Sound and Vibration, 243, 2001, 775-796.

[6] Sekavcnik M., Ogorevc T., Skerget L., *CFD analysis of the dynamic behaviour of a pipe system*, Forsh Ingenieurwes, 70, 2006, 139-144.

[7] Mehidizadeh O.Z., Paraschivoiu M., *A three-dimensional finite element approach for predicting the transmission loss in mufflers and silencers with no mean flow*, Applied Acoustics, 66, 2005, 902-918.

[8] Dowling J.F., Peat K.S., *An algorithm for the efficient acoustic analysis of silencers of any general geometry*, Applied Acoustics, 65, 2003, 211-227.

[9] Liu G., Li S., Li Y., Chen H., *Vibration analysis of pipelines with arbitrary branches by absorbing transfer matrix method*, Journal of Sound and Vibration, 332, 2013, 6519-6536.

[10] Huang Z., Jiang W., *Analysis of source models for two-dimensional acoustic systems using the transfer matrix method*, Journal of Sound and Vibration, 306, 2007, 215-226.

[11] Lee S.-H., Ih J.-G., *Effect of non-uniform perforation in the long concentric resonator on transmission loss and back pressure*, Journal of Sound and Vibration, 311, 2008, 208-296.