MECHANICS
Vol. 25 No. 2 2006

Krystyna Prync-Skotniczny*

# NUMERICAL ANALYSIS OF THE IMPACT OF CONICAL DIFFUSER GEOMETRY CHANGE ON VELOCITY DISTRIBUTION IN ITS OUTLET CROSS-SECTION

*SUMMARY*

*Paper presents the results of numerical calculations of air flow through a conical diffuser with divergence angle* $2\alpha = 70^o$ *and aspect ratio L/w = 0.82. Constructional changes were introduced into the diffuser in order to obtain a uniform velocity distribution in the outlet cross-section as well as to lower the pressure drop caused by the air flow. The changes consisted in installing two or three coaxial conical flow guides inside the diffuser. Three diffuser inlet velocities were investigated: 5.6 m/s, 6 m/s and 25 m/s. Computer modelling analyses were made using commercial ANSYS software. Calculation results were verified by comparison with laboratory experiment results carried out for the inlet velocity of 5.6 m/s and for two diffuser constructions – without any flow guides and with two conical flow guides installed.*

**Keywords***: conical diffuser, computer modelling*

*ANALIZA NUMERYCZNA WPŁYWU ZMIANY GEOMETRII DYFUZORA STOŻKOWEGO*
*NA ROZKŁAD PRĘDKOŚCI W JEGO PRZEKROJU WYLOTOWYM*

*W artykule przeprowadzono obliczenia numeryczne przepływu powietrza przez dyfuzor stożkowy o kącie rozwarcia* $2\alpha = 70^o$ *i smukłości L/w = 0,82. W celu uzyskania równomiernego rozkładu prędkości w przekroju wylotowym dyfuzora oraz zmniejszenia straty ciśnienia związanej z przepływem powietrza wprowadzono zmiany w konstrukcji dyfuzora poprzez współosiowe zainstalowanie w dyfuzorze dwóch lub trzech stożkowych kierownic. Prędkości na wlocie dyfuzora przyjmowały trzy wartości 5,6 m/s, 6 m/s i 25 m/s. Symulacje komputerowe wykonano w kodzie komercyjnym ANSYS. Weryfikację otrzymanych wyników uzyskano przez ich porównanie z wynikami badań laboratoryjnych dla prędkości wlotowej 5,6 m/s i dla dwóch przypadków konstrukcyjnych dyfuzora, tj. dyfuzora bez kierownic i dyfuzora z dwiema kierownicami.*

**Słowa kluczowe***: dyfuzor stożkowy, symulacje komputerowe*

## 1. INTRODUCTION

Conical diffusers with large divergence angle are often used as elements of ventilation systems, gas exhaust systems (catalytic afterburners, mufflers), filtration systems, etc.

Flow of viscous gas stream through such diffusers causes formation of regions of stream separation from diffuser wall and related blow-back areas (recirculation zones). Depending on gas velocity at the diffuser inlet, varying size vortices are created at diffuser walls. Vortex kinetic energy is a part of the main stream kinetic energy, therefore such flows result in significant energy losses.

In order to limit these losses diffuser geometry was modified by mounting coaxial conical flow guides inside the diffuser and impact of new designs on velocity distribution in diffuser outlet cross-section was investigated.

Earlier laboratory tests proved that installing flow guides causes partial smoothing of velocity profile in outlet cross-section as compared to non-modified diffuser [1].

Laboratory investigations are expensive and time-consuming, therefore testing multiple diffuser constructions and taking various flow parameters into account in order to optimise each design is difficult.

Such problem can be solved by numerical modelling using a modern tool, using one of well-known commercial applications, e.g. Ansys. In the first modelling step a theoretically best geometry could be found and only then, in the second step, modelling results could be verified experimentally.

This approach to finding a flow-optimal design requires providing the model with realistic boundary conditions. In order to define them, it is preferable to use best estimates based on laboratory tests.

## 2. GEOMETRICAL MODEL OF INVESTIGATED SYSTEM

Numerical calculations were done for the diffuser-pipeline geometry presented in Figure 1.

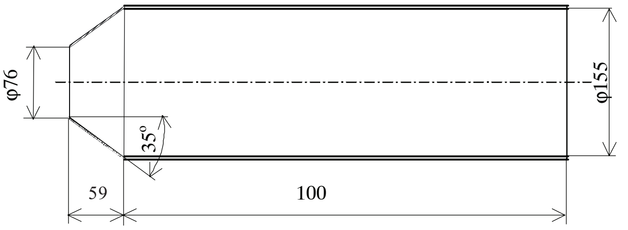

**Fig. 1.** Diffuser-pipeline system geometry

---

[*] AGH University of Science and Technology, Faculty of Mechanical Engineering and Robotics, Cracow; ghprync@cyf-kr.edu.pl

Modelling was done for steel conical diffuser with divergence angle $2\alpha = 70^{o}$ and aspect ratio $L/w = 0.82$.

Flow in a diffuser with such geometric parameters is a stream-type flow (Fig. 2).

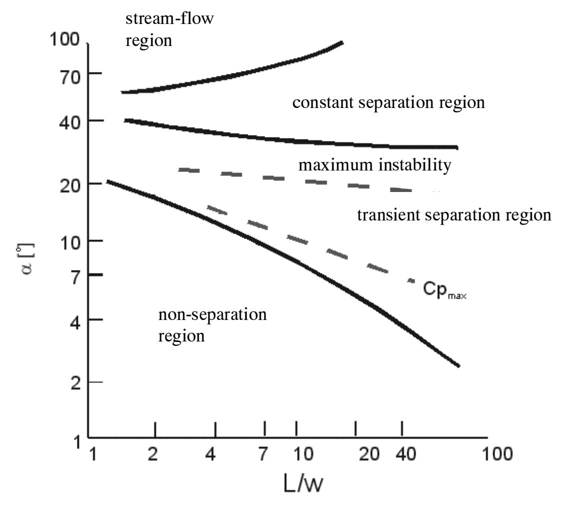

**Fig. 2.** Diagram of typical flow regions [2]

Despite the fact that investigated geometry diffusers are not presented in the cited diagram, they are used in equipment requiring compact design. An example of such application is the catalytic afterburner inlet diffuser [1].

The investigated diffuser was joined to a pipeline of 0.155 m diameter and length $L_1 = 1$ m. Calculations were done for three geometric cases of the diffuser – a sole diffuser, diffuser with two flow guides and diffuser with three flow guides. Flow guides had a shape of cut apex cone and divided the inlet and outlet diffuser areas into equal cross-section areas. Flow guides were mounted coaxially with the diffuser (Fig. 3). Flow guides' heights were equal to the diffuser height of 0.059 m. The remaining part of the system, the pipeline, remained unchanged in all investigated cases.

## 3. NUMERICAL MODEL OF THE INVESTIGATED SYSTEM

Numerical calculations were based on two-equation $k$–$\varepsilon$ turbulence model.

$$\rho\left(\frac{\partial k}{\partial t}+u_j\frac{\partial k}{\partial x_j}\right)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+ \\ +\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}-\rho \tag{1}$$

where eddy viscosity

$$\mu_t = \rho\, c_\mu \frac{k^2}{\varepsilon} \tag{2}$$

$$\rho\left(\frac{\partial \varepsilon}{\partial t}+u_j\frac{\partial \varepsilon}{\partial x_j}\right)=\frac{\partial}{\partial x_j}\left[\left(\mu_0+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+ \\ +c_1\frac{\varepsilon}{k}\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}+ \\ +c_4\frac{1}{\rho k}\left[\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}\right]-c_2\rho\frac{\varepsilon^2}{k} \tag{3}$$

and constants have the following values:

$$c_\mu = 0.09,\quad \sigma_k = 0.4,\quad \sigma_\varepsilon = 1.00,$$

$$c_1 = 1.44,\quad c_2 = 1.90,\quad c_4 = 0.00.$$

Flow in viscous and transient boundary sublayers was calculated basing on van Driest model.

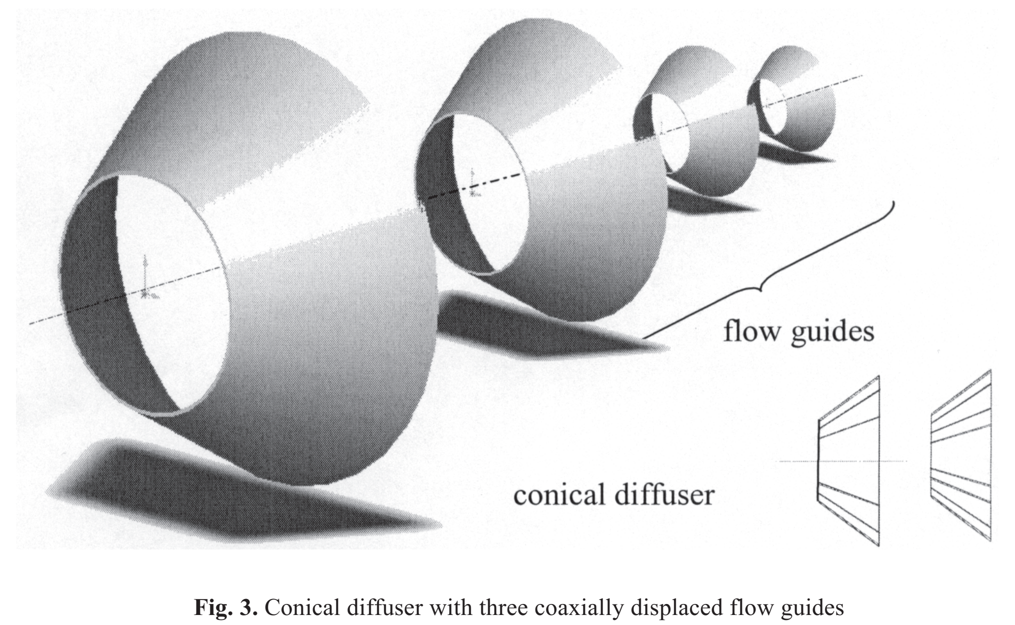

**Fig. 3.** Conical diffuser with three coaxially displaced flow guides

## 4. TWO-DIMENSIONAL SOLUTION OF THE DIFFUSER-PIPELINE SYSTEM

Due to geometrical symmetry of the diffuser-pipeline system the first stage of modelling was done in a 2D setup. Calculation mesh generated in Ansys software was made of triangular and quadrangular elements (Fig. 4).

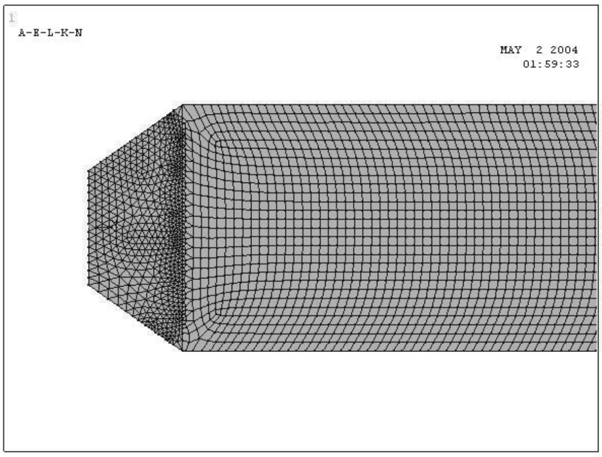

**Fig. 4.** Generated 2D model mesh

The diffuser-pipeline system was fed with air at 296 K. Air velocities at diffuser inlet were 5.6 m/s, 6 m/s and 25 m/s (5.6 m/s was an average velocity in the diffuser inlet cross-section obtained during laboratory tests). A no-slip, adiabatic and stabilised flow was assumed. Adopted boundary conditions are presented in Figure 5.

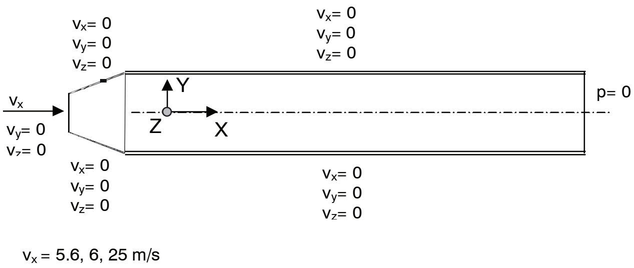

**Fig. 5.** Boundary conditions

## 5. 2D MODEL CALCULATION RESULTS

Numerical analysis resulted in data for flowing air velocity component values and modulus, pressures, energy dissipation and turbulence kinetic energy in the whole diffuser-pipeline system cross-section.

Graphs of velocity and pressure distributions in the diffuser outlet cross-section are presented in Figure 6. They cover all velocities of 5.6 m/s, 6 m/s and 25 m/s as well as the three investigated constructions, i.e. sole diffuser, diffuser with two flow guides and diffuser with three flow guides.

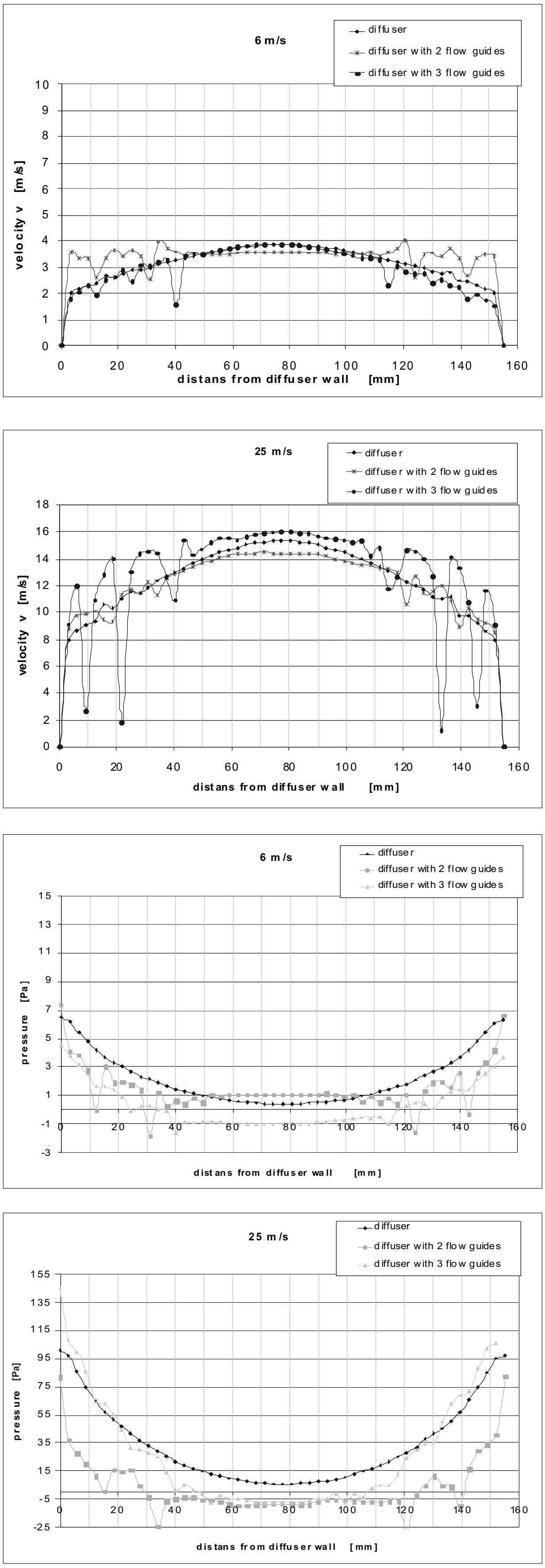

**Fig. 6.** Velocity and pressure distributions in diffuser outlet cross-section for 6 m/s and 25 m/s velocities in the inlet cross-section

Figure 7 presents velocity maps in the diffuser and pipeline longitudinal cross-section for the diffuser inlet velocity of 25 m/s and for three diffuser constructions, i.e. sole diffuser, diffuser with two flow guides and diffuser with three flow guides.

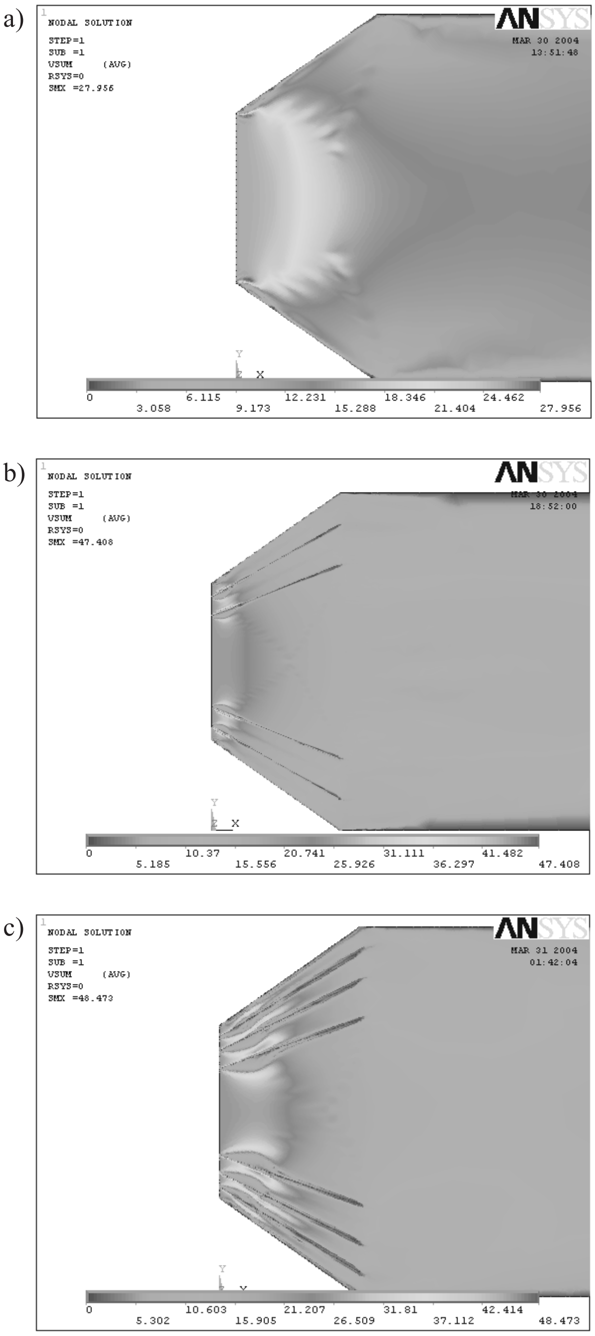

**Fig. 7.** Velocity maps in the diffuser and pipeline longitudinal cross-section for the diffuser inlet velocity of 25 m/s: a) sole diffuser; b) diffuser with two flow guides; c) diffuser with three flow guides

Figure 8 presents pressure maps in the diffuser and pipeline longitudinal cross-section for the diffuser inlet velocity of 25 m/s and for three diffuser constructions, i.e. sole diffuser, diffuser with two flow guides and diffuser with three flow guides.

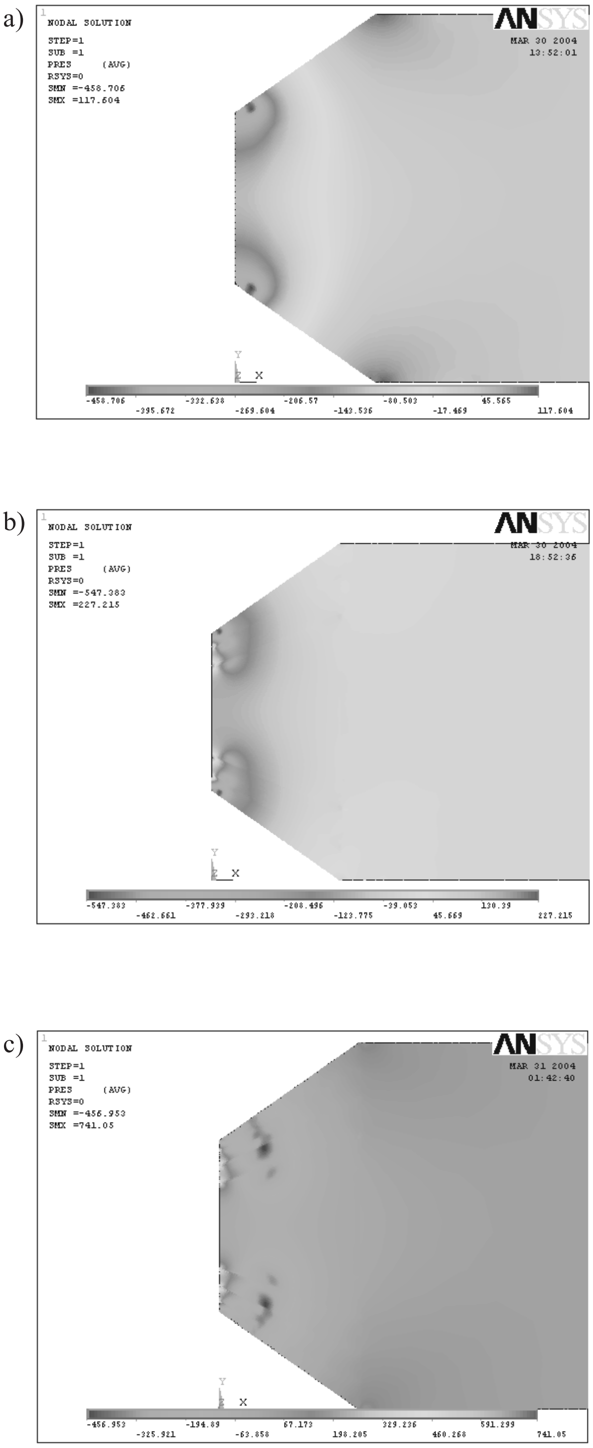

**Fig. 8.** Pressure maps in the diffuser and pipeline longitudinal cross-section for the diffuser inlet velocity of 25 m/s: a) sole diffuser; b) diffuser with two flow guides; c) diffuser with three flow guides

Additionally air stream flow through a diffuser was modelled for inlet cross-section velocity of 5.6 m/s in order to compare numerical results with laboratory tests results. The same boundary conditions as in case of discussed 6 m/s and 25 m/s velocities were used. Calculation results are presented in Figure 9 as velocity, energy dissipation and turbulence kinetic energy maps.

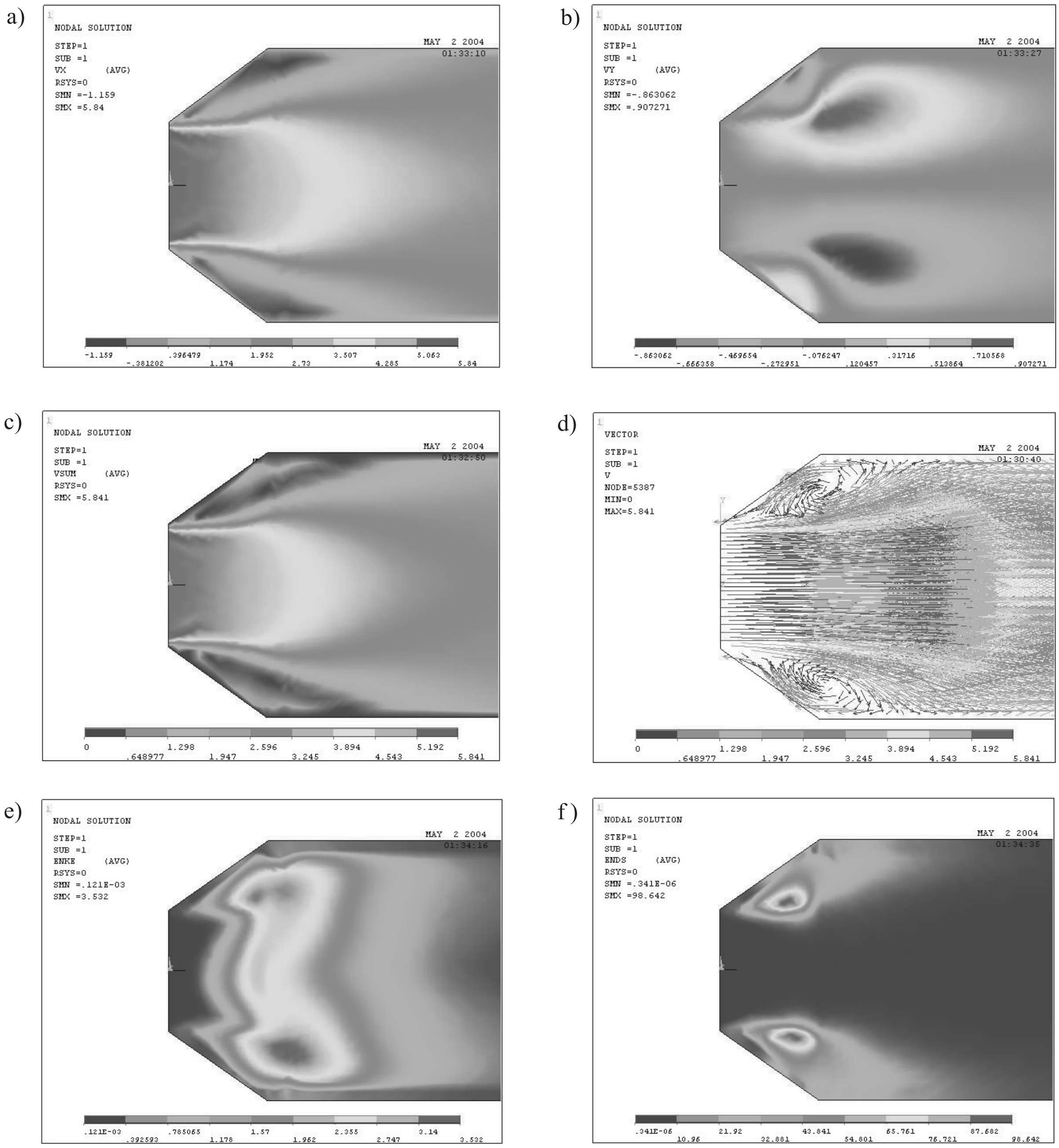

**Fig. 9.** Maps obtained from modelling of a 2D diffuser-pipeline system for diffuser inlet velocity of 5.6 m/s: a) *X* axis direction velocity component map; b) *Y* axis direction velocity component map; c) velocity modulus map; d) velocity modulus vector map; e) kinetic energy dissipation map; f) kinetic energy map

## 6. THREE-DIMENSIONAL SOLUTION OF THE DIFFUSER-PIPELINE SYSTEM

Using a 2D model in flow analysis problems is an oversimplification in most cases. Investigation results may sometimes be deprived of data inevitable for proper analysis of the phenomenon. Further analyses were therefore made using a 3D numerical model, made for inlet velocity of 5.6 m/s, and their results were compared with experimental data.

Figure 10 presents a 3D, non-structural mesh for the diffuser and pipeline generated in Ansys software.

Selected results of calculations based on the 3D model are shown in Figure 11. Presented velocity maps for three model cross-sections indicate the existence of an asymmetric velocity distribution in the pipeline. This information could be missed when only solving the problem using a 2D model.

In order to verify numerical results, the diffuser-pipeline system was experimentally tested in two design versions, one being a sole diffuser and the other – diffuser with two flow guides.

Experiments were run in a measuring setup of diffuser-pipeline construction presented in Figure 12.

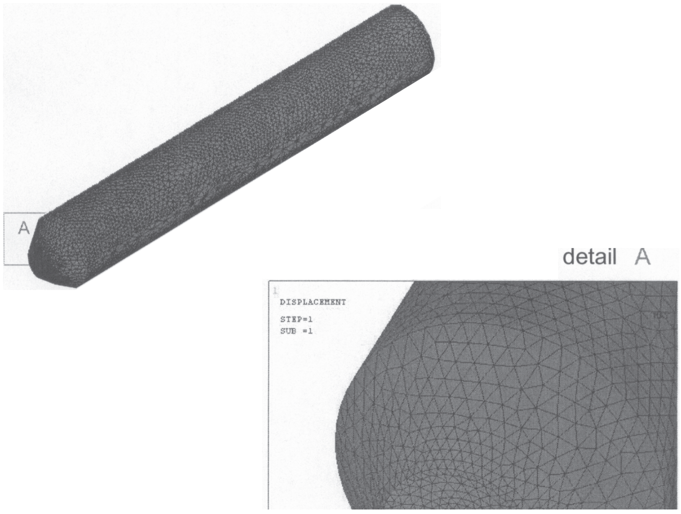

**Fig. 10.** Non-structural mesh in 3D model

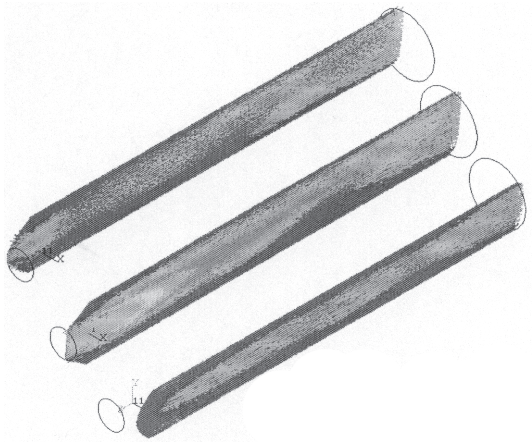

**Fig. 11.** 3D model velocity maps in three longitudinal cross-sections of diffuser-pipeline system for inlet velocity of 5.6 m/s

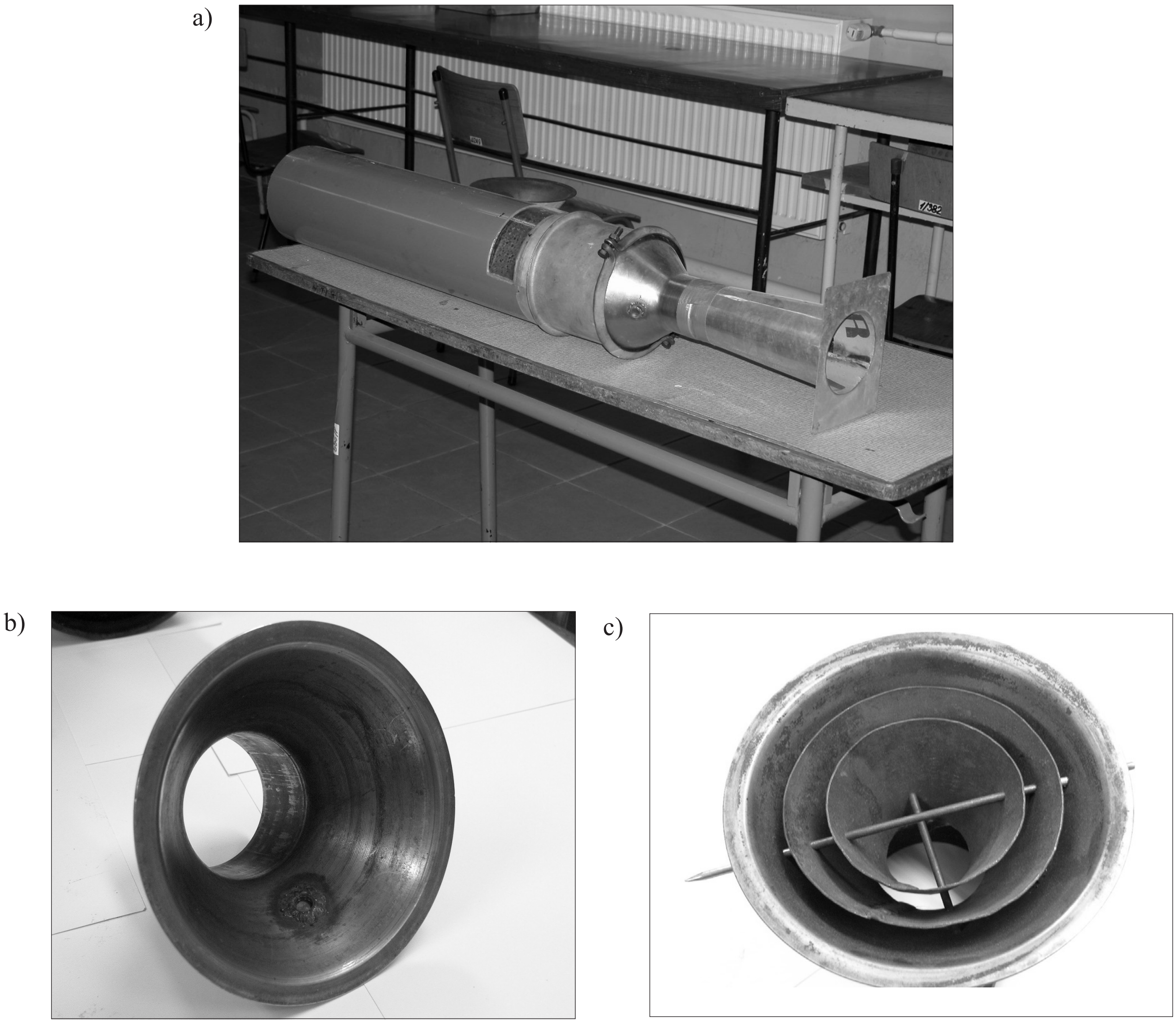

**Fig. 12.** Measurement section of the diffuser-pipeline system (a); diffuser (b); diffuser with two flow guides (c)

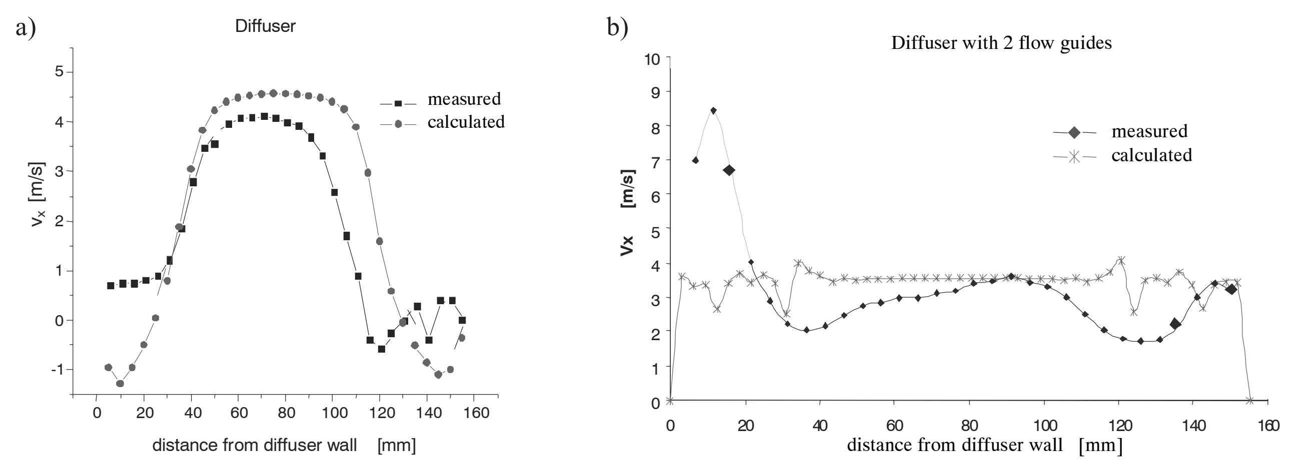

**Fig. 13.** Comparison of laboratory test results with computed values. Distribution of velocity component in X axis direction: a) diffuser; b) diffuser with two flow guides

Velocity distribution measurement was made using a HSA-4 semiconducting anemometer.

Modelling results were compared with laboratory measurements. Distributions of X component of velocity vector in diffuser and diffuser with two flow guides outlet cross-sections were considered. Results are presented in Figure 13; similarity of velocity graphs can easily be observed, however numerical calculations graphs are more symmetrical.

## 7. SUMMARY

The paper presents the results of numerical modelling using Ansys software of air flow through the diffuser-pipeline system in three constructional cases of the diffuser part: diffuser without flow guides, diffuser with two flow guides and diffuser with three flow guides. Three different flow velocities in diffuser inlet cross-section were investigated: 5.6 m/s, 6 m/s and 25 m/s. In order to verify numerical calculations laboratory tests were done for the average inlet velocity of 5.6 m/s and two constructional cases:

1) diffuser without flow guides,
2) diffuser with two flow guides.

Analysis of laboratory experiments and numerical calculations indicates the complexity of flow mechanisms in the investigated diffuser-pipeline system.

Results obtained by numerical analysis confirmed earlier laboratory test results, showing that of the three constructional cases, i.e. sole diffuser, diffuser with two flow guides and diffuser with three flow guides, the best flow characteristics is obtained for the diffuser with two flow guides. Introducing flow guides in diffuser construction causes limiting the size of vortices, which suggests lower energy dissipation (Fig. 7) than in case of the sole diffuser (Fig. 9).

In case of using the diffuser without flow guides, separation of the boundary layer occurs right after the inlet cross-section (stream flow, Fig. 2).

Using a two-dimensional model for calculation of turbulent flow phenomena can result in significant errors due to limiting a three-dimensional turbulence phenomenon to a 2D-constrained system.

Results obtained in laboratory experiment confirmed the validity of adopting the $k$–$\varepsilon$ model. An average discrepancy of 10% observed between laboratory and numerical results can be caused by using idealised boundary conditions (even velocity distribution in diffuser inlet cross-section, stationary flow characteristics), as well as disturbing the flow of air stream by the measurement probe and elements positioning the flow guides, visible in Figure 12c, which is inevitable during laboratory experiments.

The results obtained in this research form a basis for broader investigation using more advanced techniques of measuring a three-dimensional turbulent flow fluctuation in order to provide more precise data for numerical calculations.

## References

[1] Gumuła S., Prync-Skotniczny K., Skotniczny P.: *Korekta konstrukcji katalizatora silnika spalinowego*. Zeszyty Naukowe Politechniki Śląskiej, Seria: Inżynieria Środowiska, z. 41, 1999

[2] White F.M.: *Fluid Mechanics*. 3rd ed., Singapore, McGraw-Hill Book Co. 1994, ISBN 0-07-113774-2