

PAPER • OPEN ACCESS

CFD-calculation method for vane diffusers of a centrifugal compressor stage

To cite this article: E P Petukhov *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **604** 012051

View the [article online](#) for updates and enhancements.

CFD-calculation method for vane diffusers of a centrifugal compressor stage

E P Petukhov¹, Yu B Galerkin² and A F Rekstin²

¹Supercomputer Center ‘Polytechnic’ Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia

²R&D Laboratory “Gas dynamics of turbo machines” Peter the Great St.Petersburg Polytechnic University, Polytechnical st. 29, St.Petersburg, Russia

Email: yuri_galerkin@mail.ru

Abstract. Centrifugal process compressors consume huge amount of energy. The gas dynamic design methods are based on physical and numerical experiments. The methods improvement is an actual task. The theory of subsonic axial compressors is based on the results of wind tunnel tests of straight blade cascades. The modern trend is to substitute physical experiments with CFD calculations. There is positive experience of vaneless diffuser virtual wind tunnel test results (CFD calculations) that are inserted in the math model for compressor characteristic simulation. CFD calculations of vane diffusers also can be used to improve math model of a centrifugal stage but the calculation technology is more complicated. The Authors have investigated the problem and propose rational solution.

1. Introduction

Improving the gas-dynamic design of industrial centrifugal compressors is an important task. The methods of computational gas dynamics are successfully used to search for optimal solutions and gas-dynamic calculations of stator part of a stage [1, 2, 3, 4]. Vane diffusers (VD) in the stages with high loading factor impellers give a noticeable increase in efficiency. The work [5] presents the positive experience of vaneless diffuser CFD calculations for the improvement of the compressor characteristics math model. The aim of this work is to propose a CFD calculation technique suitable for an extensive computational experiment with vane diffusers. The influence of the configuration and size of the computational domain, the size of the grid elements, the boundary conditions, the choice of the turbulence model has been studied. It should be said that the classical theory of blade cascades of axial compressors is entirely based on aerodynamic tests. Therefore, a computational experiment with vane diffusers, isolated from the impeller, can be considered a method justified by the successful practice of axial compressors design.

2. Object of computational research. Aerodynamic characteristics

Test object in a virtual wind tunnel is presented on Figure 1. The object of calculation is a vane cascade limited by radiuses r_3 , r_4 and a preceding vaneless space started at r_2 . This space imitates vaneless space between an impeller and a diffuser in a real stage. The object of calculation is preceded and followed by vaneless spaces: sections “1-2” and “4-5”. At the inlet and exit sections “1” and “5” the flow must be practically uniform.



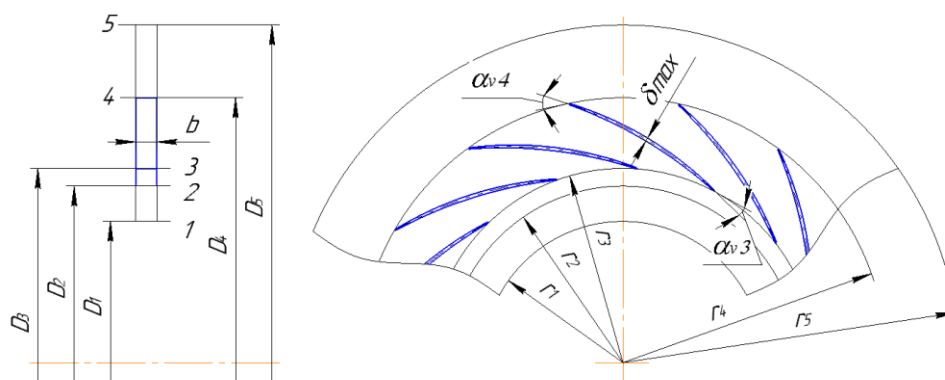


Figure 1. Test object in a virtual wind tunnel. Left – meridional cross section. Right – radial cross section

In the real centrifugal compressor stage between the exit from the impeller and the blades of the diffuser there is a small vaneless space – sections “2-3”. This space is included in the test object. In spaces preceding (section “1-2”) and following (section “4-5”) the test object, the flow behaves as in a vaneless diffuser. The flow circulation Γ is constant if there is no friction on boundary walls:

$$\Gamma = c_u \times r = \text{const}, \quad (1)$$

where c_u -circumferential velocity component, r - distance from the axis.

Rotating flow moves in vaneless spaces with velocity decrease and pressure increase. The bigger is the distance from boundaries to the test objects the bigger is the difference in parameters on inlet boundaries “1” and “2”, and exit boundaries “4” and “5”. Table 1 shows the gas-dynamic characteristics of the vane diffuser used in the practice of analysis, calculations and modeling. The formulae contain flow parameters that must be measured in the course of numerical experiment.

Table 1. VD gas-dynamic characteristics.

Name	Formula (compressible flow)	Formula (incompressible flow)
Efficiency	$\eta_d = \frac{\ln \frac{p_4}{p_2}}{\frac{k}{k-1} \ln \frac{T_4}{T_2}}$	$\eta_{dnc} = \frac{p_4 - p_2}{\rho \cdot 0,5 c_2^2 (1 - \dot{c}_o^2)}$
Flow deceleration	$\dot{c}_d = \frac{c_4}{c_2}$	the same
Loss coefficient	$\zeta_d = \frac{1 - \eta_d}{1 - \dot{c}_d^2}$	$\zeta_o = \frac{p_2^* - p_4^*}{\rho \cdot 0,5 c_2^2}$
Recovery coefficient	$\xi_d = 1 - \dot{c}_d^2 - \zeta_d$	the same
Drag angle	$\Delta \alpha_4 = \alpha_{v4} - \alpha_4$	the same
Flow turn	$\varepsilon = \alpha_4 - \alpha_3$	the same

In the table 1: p – pressure, k – isentropic coefficient, T – temperature, c - flow velocity, ρ - gas density, α_v - vane angle, α - flow angle.

The coefficients listed in the table are used in research analysis. To calculate the flow parameters at the exit from the diffuser, it is sufficient to have the values of any two coefficients. In engineering design methods the main dimensions of the flow path are determined on the basis of average values of pressure, temperature and flow velocity. The flow parameters obtained with CFD simulation in sections "2" and "4" must be correctly averaged. The correct averaging method [6]:

-average velocity at a VD exit:

$$c_{4av} = \left(\frac{\sum \Delta f \rho_4 c_4^3 \sin \alpha_4}{\bar{m}} \right)^{1/3}, \quad (2)$$

-average temperature at a VD exit:

$$T_{4av} = \frac{\sum \Delta f \rho_4 c_4 \sin \alpha_4 T_4}{\bar{m}}, \quad (3)$$

-average static pressure a VD exit:

$$p_{4av} = \frac{\sum \Delta f \rho_4 c_4 \sin \alpha_4 p_4}{\bar{m}}, \quad (4)$$

-average total pressure a VD exit:

$$p_{4av}^* = \frac{\sum \Delta f \rho_4 c_4 \sin \alpha_4 p_4^*}{\bar{m}}, \quad (5)$$

where: $\bar{m}$ - mass flow rate, Δf - surface element of a control section "4", surface of the control section $F_4 = 2\pi r_4 b = \sum \Delta f$.

3. Calculation method

Flow calculations are performed with the Ansys CFX software package. The stationary Reynolds-averaged Navier-Stokes equations (RANS) [6] are solved numerically. Steady state compressible flow simulations are carried out using standard turbulence models k - ϵ or k - ω SST [6]. The total quantities of pressure and temperature and flow direction are imposed at inlet boundary (section "1") and mass flow rate condition is set at outlet (section "5"). Inlet flow direction defines incidence angle $i_3 = \alpha_{v2} - \alpha_2$. Mass flow rate is based on inlet velocity magnitude of 170.5 m/s. The solution convergence criterion is taken of residuals value 10^{-6} .

4. 2D or 3D calculation of a VD cascade

Flow parameters before and after axial compressor blade cascades in a wind tunnel are measured on a mean surface of long blades. Influence of boundary walls is negligible. The flow character is two-dimensional (2D). Two-dimensional way of VD cascades calculation could economy a lot of computer time.

The Figure 2 presents important information on flow behavior in VD. The flow deceleration ratio is $\dot{c}_d \leq 0.40$, therefore flow separation is inevitable at design regime too.

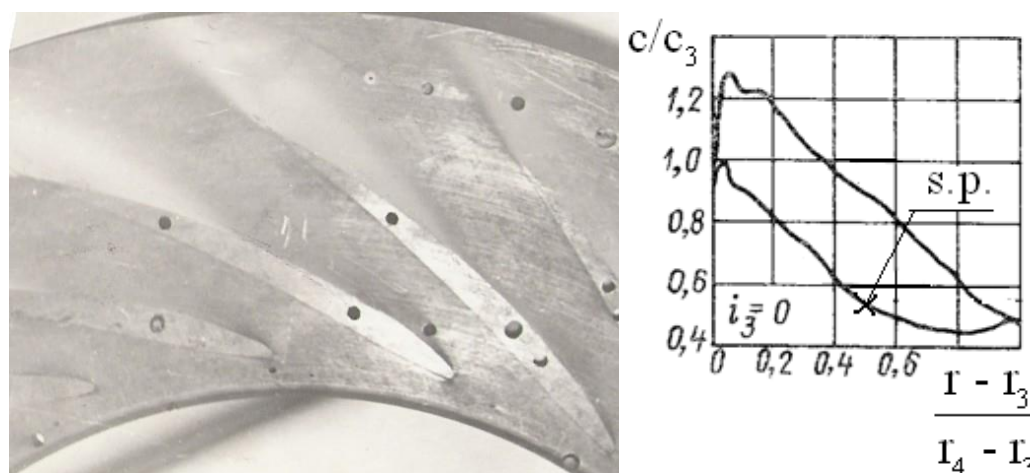


Figure 2. Left - experimental low energy zones visualization in VD. Right - the velocity diagram of inviscid flow of the VD vane and a separation point (s.p.) position in VD [6]

The separation zones on the boundary walls of the tested model stage are visualized by powder sprayed on a flow part. The powder stuck to surfaces at separation zones, where shear stresses are low. The important fact is that the separation occurs on the convex pressure side of vanes. Figure 2 (right) shows the position of the separation point on the velocity diagram. The flow velocity at a pressure side of a vane is comparatively low. Therefore separation losses are minimal.

The possibility of 2D calculation was investigated for VD with dimensions $D_2 = 350$ mm, $D_3 = 385$ mm, $D_4 = 525$ mm, $D_5 = 875$ mm, $b = 21$ mm, diffuser vane angles $\alpha_{v3} = 15^\circ$, $\alpha_{v4} = 30^\circ$, number of vanes $z_v = 15$. Calculation was made for one vane channel, turbulence model $k-\epsilon$.

The flow structure with inviscid boundary surfaces (2D flow) is shown in Figure 3 (left). The flow structure with viscous boundary surfaces (3D flow) is shown in Figure 3 (right).

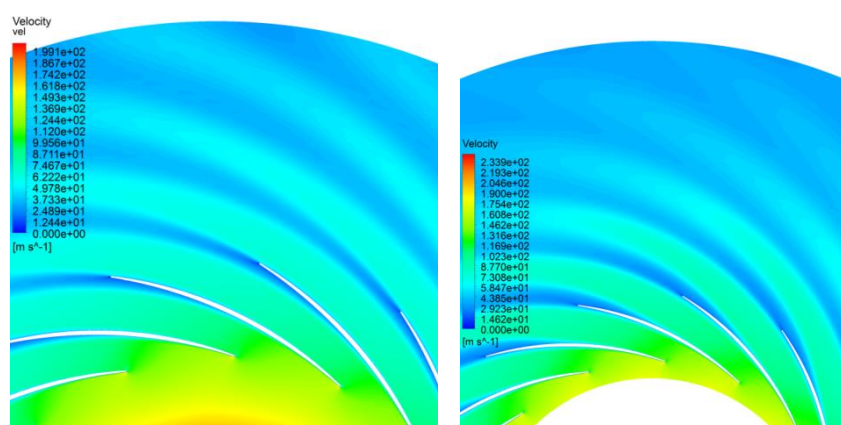


Figure 3. Flow velocity in VD by two-dimensional (left) and three-dimensional (right) CFD calculation.

CFD-calculation of 2D flow on Figure 2 (left) demonstrated flow separation on a suction side of vanes. It contradicts to flow visualization in a real VD – Figure 2 (left). To the contrary, 3D calculation with viscous flow on the boundary walls simulates real flow quite satisfactory. In this case, the flow pattern corresponds to the visualized flow pattern in the real diffuser. The tests of VD in a virtual wind tunnel must be three-dimensional. Accordingly, the condition of gas adhesion to the bounding surfaces is made between sections “2” and “4”. There is no gas adhesion on the boundary surfaces between the

“1-2” and “4-5” sections (Figure 1). Flow moves without resistance in these parts of the computational domain.

5. The position of the final section "5"

The flow pattern in the vaneless space “4” –“5” on Figure 3 above demonstrates the mixing process. The mixing is not completed at the $D_5/D_2 = 3$. Figure 4 demonstrates velocity field in this section.

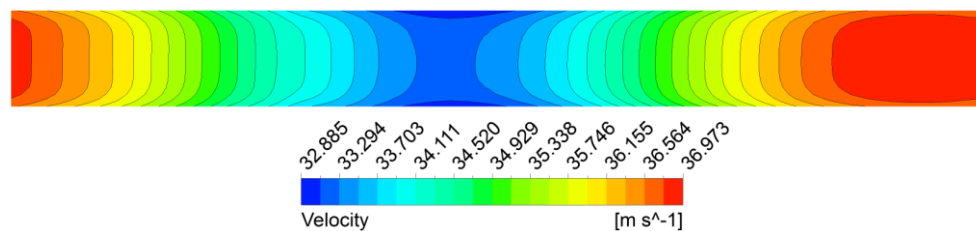


Figure 4. The velocity field in the exit section with $D_5/D_2 = 3$ (the sector corresponds to one vane channel).

The flow is not completely uniform even with such a large distance from the vane cascade. But the engineering approach does not require complete uniformity. Figure 5 shows how the total pressure loss coefficient changes over the radius behind the diffuser vanes.

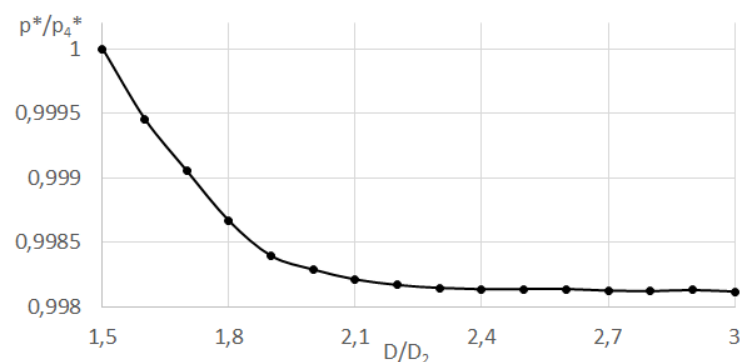


Figure 5. The total pressure loss coefficient versus radial distance of the vane cascade

In vaneless space “4”-“5” with no shear stresses on walls the drop of total pressure demonstrates the mixing process. When $D_5/D_2 \geq 2.5$ the mixing is practically complete. The tests of VD in the virtual wind tunnel will be made in a three-dimensional mode with $D_5/D_2 = 2.5$ ($D_5 = 1.67 D_4$).

6. The position of the inlet boundary - section "1"

The vanes' load leads to flow disturbance ahead of a vane cascade. The flow non-uniformity slackens on the distance of a cascade quickly. It is an attractive idea to place the inlet boundary to the section “2” where a real diffuser starts. The flow pattern in the boundary section with $D_1 = D_2$ is presented on Figure 6. In this case the boundary is on the radial distance 10% from a vane's leading edge.

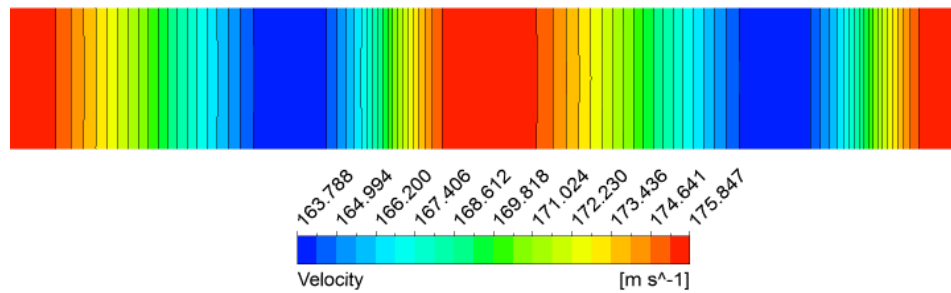


Figure 6. Velocity field in section "1" when $D_1 = D_2$

The flow non-uniformity in the case is rather high. The difference of maximum and minimum velocity is 6.9%.

In case when the section "1" is displaced by 20% of the vane's leading edge radial distance, the flow is practically uniform. The difference of maximum and minimum velocity is 0.4%. At the diffuser inlet – section "2" – the difference of maximum and minimum velocity is diminished to 3.4%.

The efficiency characteristics were calculated with two inlet boundary positions $D_1 = 0.8 D_2$ and $D_1 = D_2$ – Figure 7. The efficiency is presented as a function in an incidence angle $\eta_d = f(i_3)$, where $i_3 = \alpha_{v2} - a_2$.

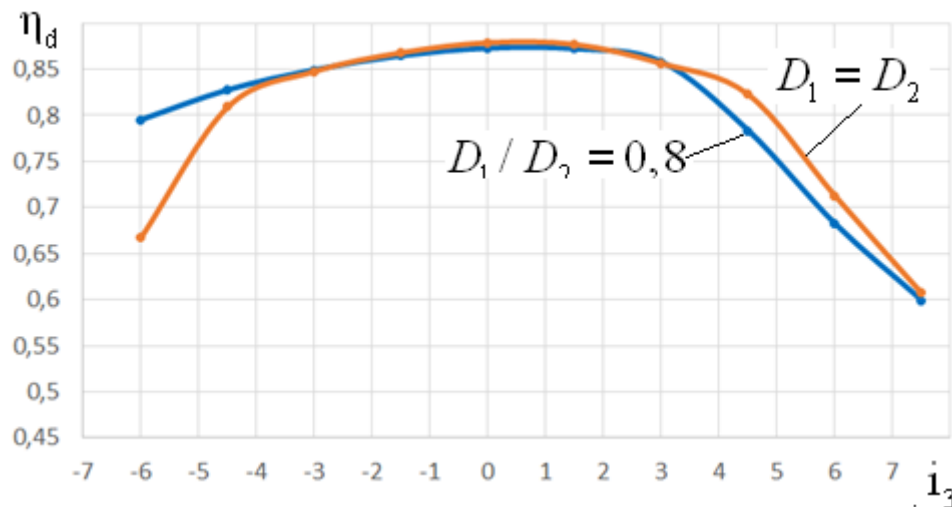


Figure 7. The diffuser efficiency at different positions of the inlet boundary

There are no shear stresses on walls between sections the "1" - "2". The difference of characteristics is visible. The position of the inlet boundary $D_1/D_2 = 0.8$ is correct.

7. Network dependence study

Two grids were constructed for one VD vane channel, consisting of 256512 (coarse grid) and 843136 (fine grid) elements.

VD that was represented by 1 and 15 vane channels were calculated with the coarse grid. One VD channel was calculated with the fine grid. Efficiency performance was calculated as $\eta_d = f(i_3)$ with $i_3 = -6 \dots 7.5^\circ$ and is presented on Figure 8.

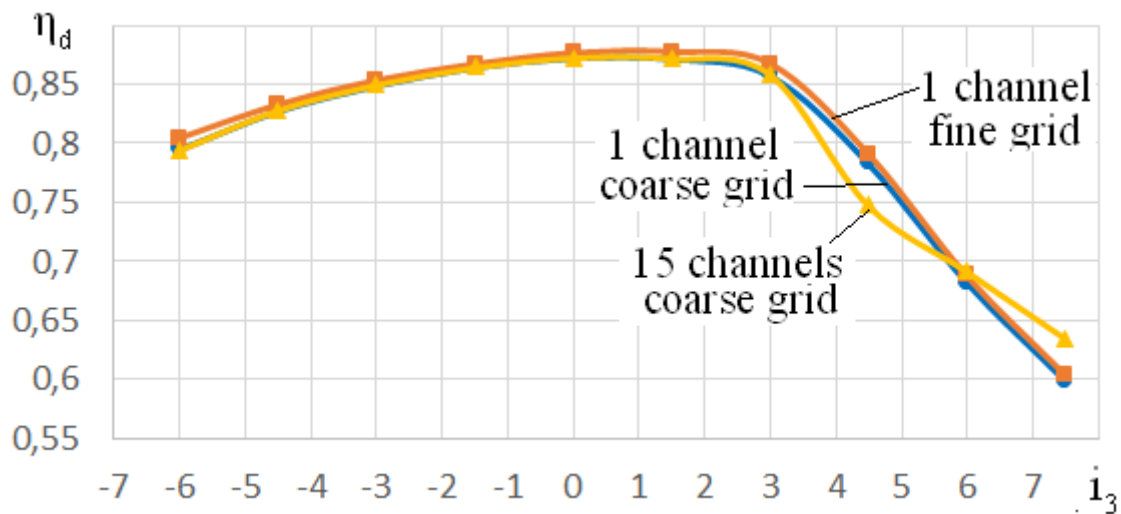


Figure 8. Efficiency characteristic of VD with different grids and vane channels

The proximity of the results makes it possible to use moderately detailed (coarse) grids and one vane channel in mass calculations.

8. Problems of flow non-uniformity at the VD exit (section "4")

As it was mentioned above to calculate the energy characteristics of VD, it is necessary to average the calculated parameters of gas by 3rd degree of velocity. The ANSYS CFX program provides averaging by surface area (1st degree) or mass flow rate (2nd degree) only. These two methods of averaging are applied to a VD exit (section "4"). There are low-energy zones of the separated flow there. The values of the parameters differ greatly along the section "4" surface.

Table 2 presents the flow parameters in section "4" at the exit of VD and in section "5" - the end of the computational domain. The averaging method is indicated as "MFR" - averaging over mass flow, "AR" - averaging over area of the section.

Table 2. Two ways of flow parameters averaging in the sections after VD.

Section «4»									
c_4	c_4	α_4	α_4	c_{r4}	c_{r4}	c_{u4}	c_{u4}	$\alpha_4 = \arctg \frac{c_{r4}}{c_{u4}}$	$\alpha_4 = \arctg \frac{c_{r4}}{c_{u4}}$
MFR	AR	MFR	AR	MFR	AR	MFR	AR	MFR	AR
45.285	36.091	38.35	33.67	28.037	21.894	35.483	28.323	38.314	37.7
Section «5»									
c_5	c_5	α_5	α_5	c_{r5}	c_{r5}	c_{u5}	c_{u5}	$\alpha_5 = \arctg \frac{c_{r5}}{c_{u5}}$	$\alpha_5 = \arctg \frac{c_{r5}}{c_{u5}}$
MFR	AR	MFR	AR	MFR	AR	MFR	AR	MFR	AR
24.615	24.533	30.913	31	12.578	12.569	21.137	21.047	30.75	30.84

The difference of velocity and flow angle in the section "4" calculated with two ways of averaging is beyond the acceptable calculation inaccuracy. In the section "5" the flow is practically uniform. The way of averaging does not influence results. To make correct simulation of VD characteristics the next way of processing is proposed.

In the vaneless space between sections “4” – “5” flow is incompressible practically as $M_{c4} \approx 0.16 - 0.18$. Viscosity does not influence flow as there are no shear stresses on boundary walls. Flow behaves there like in an ideal vaneless diffuser. It moves with constant angle $\alpha = \alpha_5$ and velocity $c = c_5 \frac{r_5}{r}$. Altogether with calculated flow parameters in the sections “2” and “5” the VD coefficients and angles in Table 1 above are easily calculated.

9. Influence of turbulence model

Two-parameter model $k-\varepsilon$ is widely used in engineering practice for more than forty years [7]. It was compared with $k-\omega$ SST model. This model is a combination of $k-\varepsilon$ and $k-\omega$ models, providing a combination of their best qualities. Figure 9 shows the efficiency characteristics of VD calculated by both models.

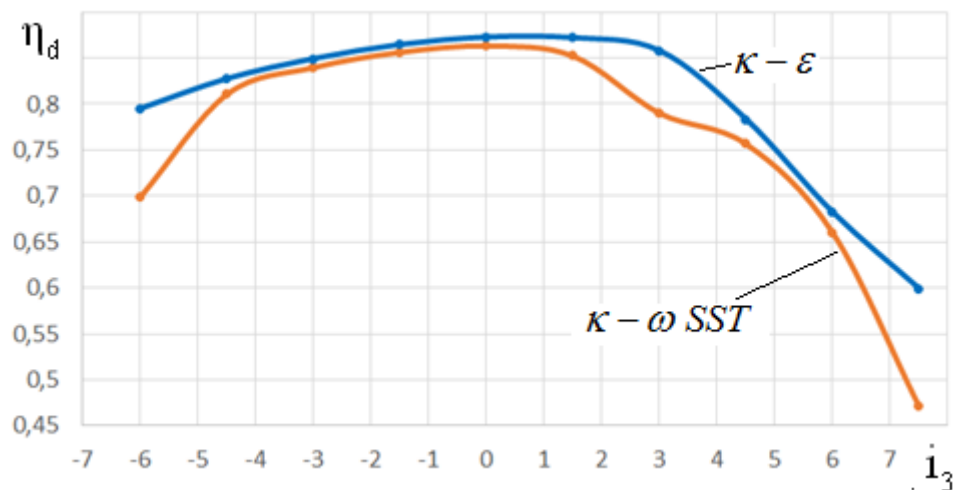


Figure 9. Efficiency characteristic of VD with two turbulence models

The characteristic calculated by $k-\varepsilon$ model is logical and demonstrates higher efficiency at all incidence angles. The $k-\omega$ SST model demonstrates iteration problem at $i_3 = 20^\circ$. But the analysis of flow pattern has shown that $k-\omega$ SST model simulates flow separation zones that are close to experimental flow visualization [6]. The model $k-\varepsilon$ underestimates flow separation in some cases.

The additional research was undertaken to study the effect of the turbulence model on the development of separation in a diffuser. The book [8] provides experimental data on the flow structure and pressure losses of straight diffusers. The graphs in Figure 10 show the flow zones in the diffusers without flow separation (I) and with separation (II).

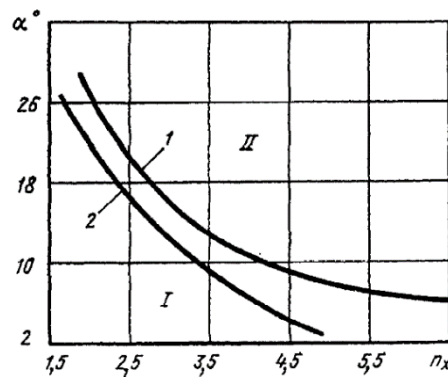


Figure 10. Flow patterns in straight diffusers [8].
 α - angle between walls, n_x - exit/inlet area ratio

In the Figure 10: $n_x = F_2 / F_1$ - exit /inlet area ratio, α - walls divergence angle. Separation is inevitable above line 1 (flow enters a diffuser with no boundary layer) or line 2 (flow enters a diffuser with boundary layer).

The VD with separated flow visualization on Figure 2 has $\alpha = 17^\circ$ and $n_x = 3.71$. The straight diffuser with these parameters is separated anyway in accordance with Figure 11. The flow in the straight diffuser with the same $\alpha = 17^\circ$ and $n_x = 3.71$ was simulated by k- ϵ and k- ω SST models at two values of similarity criteria $M_{cl} = 0.25$, $Re = c_1 d_h / \nu = 156800$ and $M_{cl} = 0.5$, $Re = c_1 d_h / \nu = 368580$. Results are presented on Figure 11.

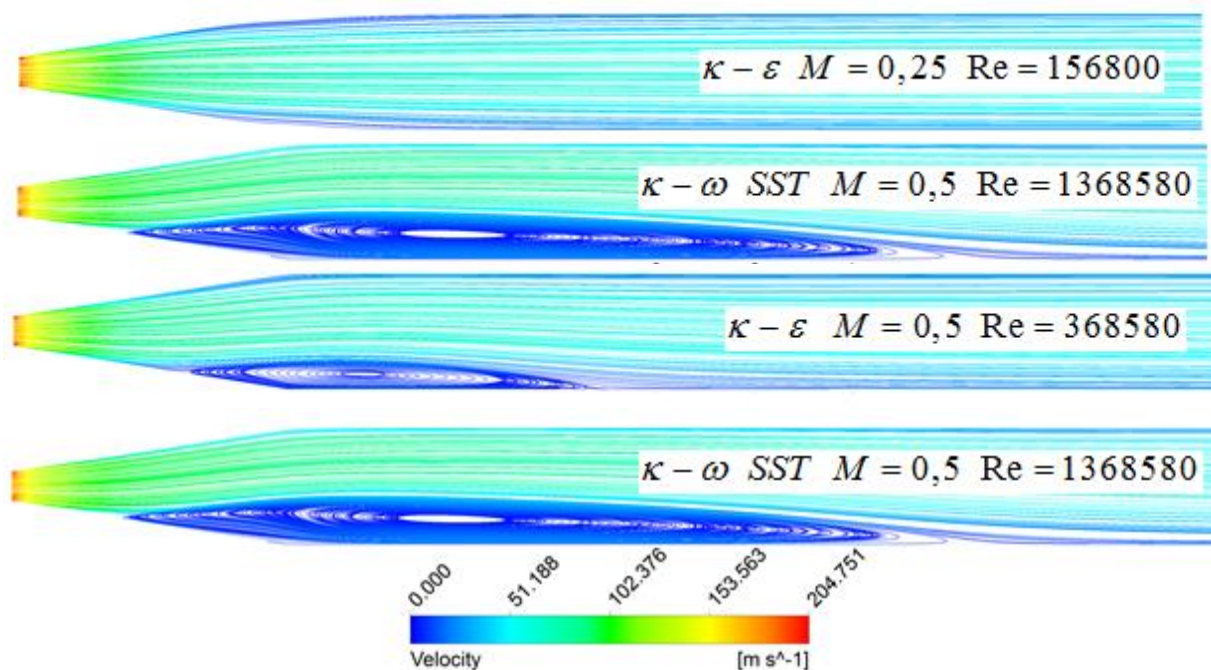


Figure 11. CFD-simulation of the flow in the straight diffuser (flow enters with no boundary layer)

In accordance with [7] the flow in the diffuser on Figure 11 must be separated unconditionally in all cases. But k- ϵ model underestimates separation at $M_{cl} = 0.25$ while k- ω SST model predicts separation realistically. This turbulence model is recommended for vane diffusers study.

10. Results

The recommended CFD technology of VD study: Ansys CFX software package, steady flow, $k-\omega$ SST turbulence model, one channel with approximate 260000 element grid, the inlet boundary is situated at $r_1 = 0,8r_2$, the exit boundary is situated at $r_5 = 1,75r_4$. The parameters calculated at the exit boundary "5" are recalculated to the vane cascade exit "4" meaning that shear stresses are absent on walls in the channel "4 – 5". One point of VD characteristic calculation takes 20-30 minutes on 28-core computer node (x2 Intel Xeon E5-2697v3) of SPbPU supercomputer system.

11. Conclusion

The presented study of CFD-calculation has demonstrated that test method of a VD circular cascade in a virtual wind tunnel is not as simple as of a straight cascade of an axial compressor. Unexpected was the result of VD two-dimensional calculation with flow separation on the suction side of vanes, which contradicts the experimental data. 3D calculation with boundary walls solved the problem of compliance with experiments. Another unexpected result is a comparison of turbulence models. When calculating the stages and compressors, both models $k-\varepsilon$ and $k-\omega$ SST give practically similar results. The presented analysis showed that the $k-\varepsilon$ model gives incorrect results when calculating diffusers with an intense flow separation. Other aspects of VD simulation were studied and proper recommendations were formulated. The next step of the work is the massive CFD-experiment with great number of VD of different parameters. The final aim is to offer the math model of a vane diffuser and optimal designs of VD for stages with different parameters.

Acknowledgments

The presented scientific research corresponds to the program of the National technological initiative center "New production technologies" SPbPU and contributes to the formation of competencies in new production technologies in part of power engineering.

References

- [1] Marenina L 2016 CFD wind tunnel tests of centrifugal stage return channel vane cascades. *Compressor technology and pneumatics*. № 3. – p. 27-35.
- [2] Galerkin Yu and Solovieva O. 2014 Improvement of vaneless diffuser calculations based on CFD experiment. Part 1. *Compressor technology and pneumatics*. № 3. – p. 35-41.
- [3] Galerkin Yu and Solovieva O 2014 Improvement of vaneless diffuser calculations based on CFD experiment. Part 2. *Compressor technology and pneumatics*. № 4. – p. 15-21.
- [4] Soldatova K. 2017 CFD study of leakage flows in shroud cavities of a compressor impeller. *International Conference on Compressors and their Systems*. London. City University. - UK. - DOI: 10.1088/1757-899X/232/1/012045.
- [5] Rekstin A.F., Drozdov A.A., Solovyeva O.A., Galerkin Y.B. 2018 Two mathematical models centrifugal compressor stage vaneless diffuser comparison. *Oil and Gas Engineering (OGE-2018)*. AIP Conf. Proc. 2007. <https://doi.org/10.1063/1.5051896>
- [6] Galerkin Yu 2010 *Turbocompressors*. LTD information and publishing center KHT, Moscow p. 596
- [7] Wilcox D 2006 *Turbulence Modeling for CFD* 3rd edition, DCW Industries, Inc., La Canada CA
- [8] Idelchik I 1983 *Aerohydrodynamics of technological devices: (supply, branch and distribution of a stream on devices section)* M.: mechanical engineering. -. - p. 350