Acta Mechanica Slovaca 14 (1): 20-25, 2010
DOI: 10.2478/v10147-011-0003-2

Performance of Closed Type Impeller Designed for Easy Manufacturability

Mária ČARNOGURSKÁ * ***(SK)*** *maria.carnogurska@tuke.sk*

Peter GAŠPAROVIČ (SK) pgasparovic@gmail.com

BIOGRAPHICAL NOTES

prof. Ing. Mária Čarnogurská, CSc. She is a graduate of Faculty of Mechanical Engineering, Technical University in Kosice (TUKE). Currently works as a deputy of head at Department of Power Engineering at Faculty of Mechanical Engineering within a. m. university. Her long-term professional focus is within the area of power engineering. The research is concentrated on mathematical and physical modelling of processes and effects applied in power engineering, the flow of different media in energy machines and appliances as well as the exploitation of non-utilised heat energy in diverse industrial applications.

Ing. Peter Gašparovič, PhD. He graduated at the Air Force Academy in Kosice. During his eleven years' career in military positions, he was responsible for the preparation of aeronautical engineers. Currently, he is a freelance lecturer and consultant in aerodynamics. His scientific focus is on the numerical methods of computation of fluid dynamics and heat flow. He works on programming of the methods and on their applications to the aerodynamics of aircraft and wind turbines and to the problems of fluid-structure interaction.

KEY WORDS

Impeller, throttling characteristics, performance, numerical simulation

ABSTRACT

The impeller of the semi-open type from a refrigerator unit is redesigned to improve efficiency and to reduce the pressure pulsations of the front case. Therefore, the closed-type impeller with outer shroud is chosen. Dimensions are scaled according to Cordier diagram with rotational speed preserved. The resulting geometry is constrained by requirement of easy manufacturability by injection moulding process. These constrains don't permit to improve efficiency by redesigning impeller.

INTRODUCTION

The original impeller from refrigerator unit KGN is of the semi-open centrifugal type. The gap between the impeller and a front case is large compared to the height of impeller blades. This feature reduces pressure pulsations at the front case, but it also worsens energy efficiency of the impeller. Furthermore, energy efficiency is worsened by simple geometric shape with flat inner shroud.

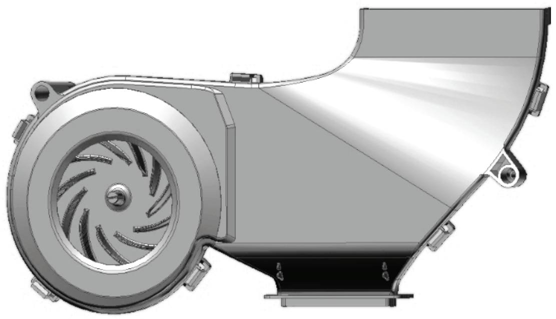

Fig. 1 *View of the original impeller KGN inserted in a case*

THE PERFORMANCE OF THE ORIGINAL IMPLLER

The performance of the original impeller, which relates to specific values of mass flow, is showed by flow-head curve (indicated by "KGN fan head" label in Fig. 2). The picture also contains system curve (indicated by label "system curve") that characterizes the pressure losses induced by flow in refrigerator unit, which were modelled in rectangle duct with dimensions 20x100x1400 mm. The curve was interpolated from CFD simulation data.

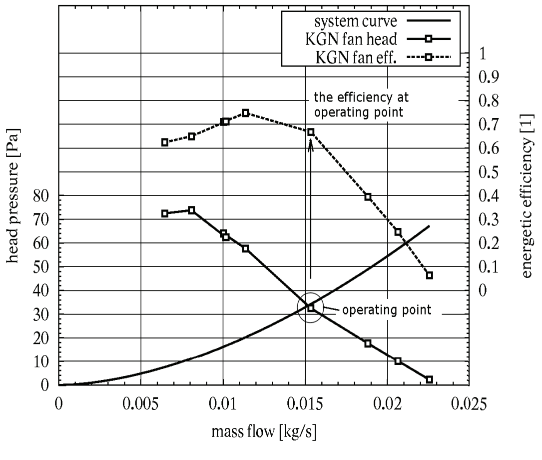

Fig. 2 *Flow-head performance of the original impeller*

Impeller's operating point lies in the part of flow-head curve where the efficiency of impeller approaches its maximum. This implies that the former impeller design is well-matched to flow conditions but, actually, says nothing whether the type of impeller is the optimal construction solution. To compare with other possible impeller designs, the performance must be always expressed in non-dimensional form. Non-dimensional form must also suit the design process, where diameter D and rotational speed n must be independent. Thus, each impeller can be independently expressed as a function of the required mass flow $\boldsymbol{Q_m}$ and difference of total pressures Δp. These conditions are fulfilled by (non-dimensional) specific diameter D_s and specific speed Ns in terms of formula:

$$D_S = D\frac{\left(\frac{\Delta p}{\rho}\right)^{1/4}}{\left(\frac{Q_m}{\rho}\right)^{1/2}}, \qquad N_S = n\frac{\left(\frac{Q_m}{\rho}\right)^{1/2}}{\left(\frac{\Delta p}{\rho}\right)^{3/4}} \tag{1}$$

where: *Q_m is mass flow (kg.s^{-1}),*
ρ is density of the working fluid (kg.m^{-3}),
D is outer diameter of impeller (m),
n is rotational speed (rad.s^{-1}),
Δp is difference of total head pressures (Pa).

Other definition of these numbers, based on volumetric flow and head height, can be found in engineering books (e. g. [1], [2]).

The performance of the impeller expressed in these numbers is plotted in Fig. 3. Apart from the flow-head curve, the figure contains another curve called Cordier diagram ([3], [1]).

All optimal flow machines operating at their best efficiency points lie very close to Cordier curve in this diagram (or nearby indispersion). The best efficiency point of the impeller from analysed body of KGN impeller doesn't lie on this curve therefore, from this point of view, it is possibly not optimal design structure.

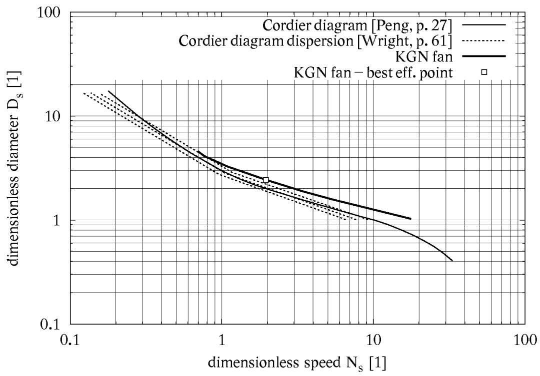

Fig. 3 *Performance of the original impeller compared with Cordier diagram*

REDESIGN OF IMPLLER

The process of impeller design is not straight and needs experience and iterations. Some basic rules can be found in [1] and [2], however the requirement of easy manufacturability doesn't permit to use them. In this article, a "quality" comparison of original and redesigned impeller (KGN-4) is performed, while preserving mass flow, head pressure and rotational speed. The basic geometry is dictated by manufacturability constraints. Parts of the duct profile in symmetry plane were chosen by heuristic way and the inlet and outlet angles of the vanes were estimated from the general laws of centrifugal machines (e.g. Euler's turbine equation). The analysed geometry is only initial approximation to the optimal design.

The first step of the design is the dimensioning based on Cordier diagram and the desired values of mass flow and head pressure that are dictated by system conditions. The Ds of the original impeller is slightly above Cordier diagram, therefore outer diameter of redesigned impeller should be reduced (from 0,12 m to 0,10 m).

The specific speed must be preserved also in redesigned otherwise the rotational speed would change. However construction type of impeller will be changed. The type of optimal flow machine impeller design, according to Cordier diagram, depends on specific speed N_s. This dependence is displayed in Fig. 4 (from [4]).

It is evident, that the specific speed of the original impeller ($N_s = 2{,}0$) lies in region of axial-radial impellers, therefore the original radial shape will be accordingly changed to reflect this fact. Moreover, the use of mixed flow impeller permits to experiment with the use of outer shroud of the impeller. The outer shroud prevents different pressures on sides of vanes to create pressure pulsations on front case. Easy manufacturability of impeller by injection moulding process imposes big constraints on possible shape. The inner diameter of the outer shroud must be larger than outer diameter of the inner shroud. The vanes of the impeller must be untwisted around radial axe. It means that their shape must be basically two-dimensional, with the only curvature in plane of rotation. This is severe limitation, because it means, that the trailing edge on inner (bottom) side of vanes must have the same angle as the leading edge on outer (top) side of vanes. It also causes the sweep angle of vanes (relative to the direction of flow) and their short length (in direction of the flow). Therefore, to preserve the solidity (pitch/chord), the number of

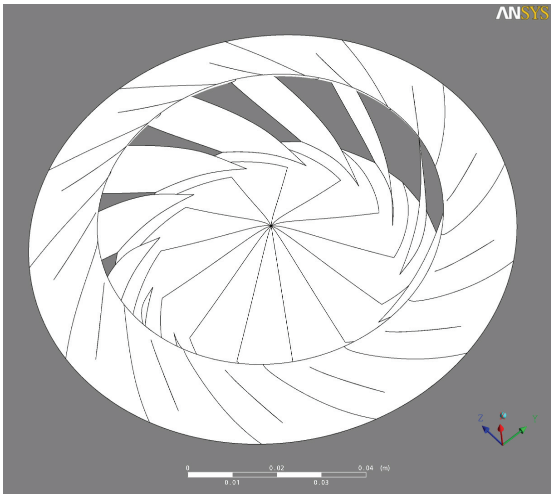

Fig. 5 *The shape of redesigned impeller KGN-4*

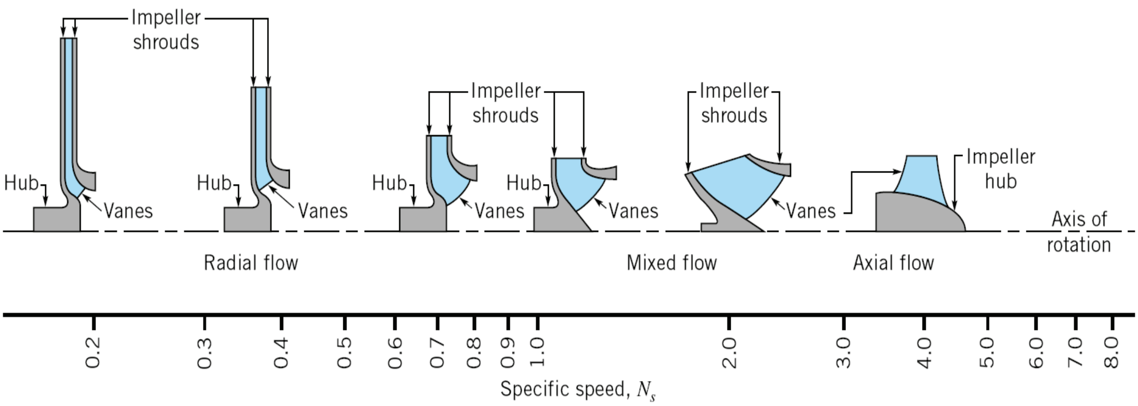

Fig. 4 *Dependence of optimal design type on specific speed (from Munson 2002)*

the blades is increased (from 11 to 13). The resulting impeller shape also with new shape of vanes (labelled as KGN-4) is in Fig. 5.

CFD SIMULATION OF REDESIGNED IMPELLER

The comparison of the original and redesigned impellers is performed only by the CFD. The CFD simulation constitutes third approach (beside analytical theory and experiment) in the study of the flow [6], and it often substitutes the role of classical experiment in the cases where all relevant flow phenomena can be resolved with sufficient level of details. As evidenced by the work [7], even more complicated case of impeller flow then the presented case are reliably modelled by present commercial CFD codes. The only prerequisite is to finely mesh the geometry in the area, where the strong boundary layer effects are expected - like the vanes in this case.

For simulation purposes of flow situation and gaining an information about "quality" of proposed impeller, the geometry of the CFD model is created as a 13th part of full circle (see Fig. 6). The cut-out segment contains one vane in the middle, tubular inlet and radial outlet. The geometry of surfaces is simplified, smooth, without any real clearances and edges. The vane is infinitely thin. The mesh was created as a structured, using HEXA module of the ICEM CFD software. The resulting mesh contains 153 000 hexahedral elements and 164 000 nodes. The boundary layer details on vanes and outer shroud are meshed, so *y+* approaches the value of 1.

The computation was performed by software ANSYS_CFX 11.0. The solver is based on Reynolds averaged Navier-Stokes equations. The fluid was assumed incompressible with constant density of 1,225 kg.m^{-3}. The turbulence was computed using k-omega SST model with gamma-theta model of boundary layer transition.

The boundary conditions were:

- *The speed of rotation: 1950 min^{-1}.*
- *Inlet: constant normal velocity (2,26 - 4,6 m.s^{-1}), turbulence intensity 5%.*
- *Outlet: static pressure (0 Pa).*

The solution in each case was stopped after 150–300 iterations, based on the monitor of head pressure during computation. Overall pressure distribution at minimum mass flow (Q_m = 0,005 kg.s^{-1}) of the air is found on Fig. 7.

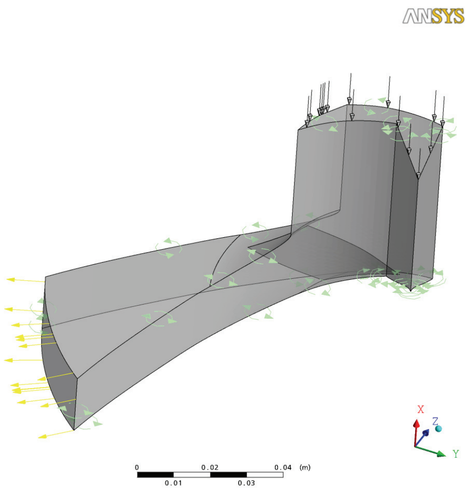

Fig. 6 *The geometry of redesigned impeller*

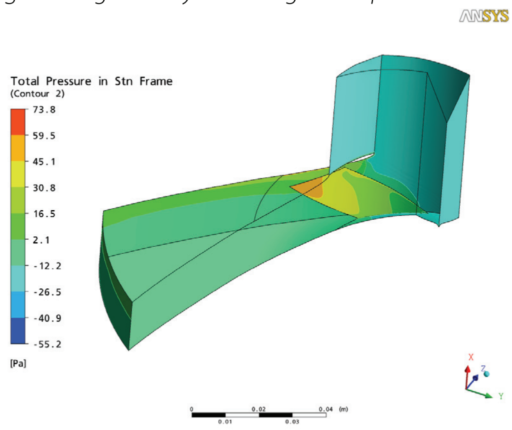

Fig. 7 *Computed total pressure at low flow*

RESULTS AND DISCUSSION

The resulting performance of the redesigned impeller is plotted in Fig. 8 together with performance of the original impeller. The optimum operating point of the new impeller is shifted to higher mass flow, and the head pressure is lower in total (progression shown in fig as "KGN-4 fan head"). This can be partially corrected by the change of vane angles, but the lack of vane area on largest radius is probably equally important cause of lower head. Solution process has shown that the efficiency of the redesigned impeller is not higher than of the original impeller. This is probably caused by severe limitations on vane geometry. The optimal shape of vanes needs to be twisted in three dimensions, to reflect the change of airflow on different radius and big sweep angle of leading and trailing edges. Especially at low mass flow it is evident, that sweep creates weak load of the inner part of the vane and this causes slow flow on the back case of the impeller (Fig. 9). The second possible cause of low efficiency is rather low Reynolds number of vanes (Re = ~9000).

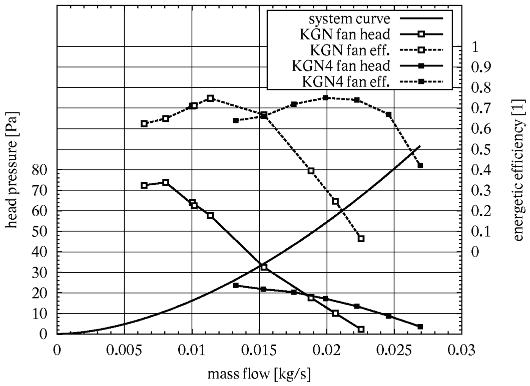

Fig. 8 *Comparison of performance of original (KGN) and redesigned (KGN-4) impeller*

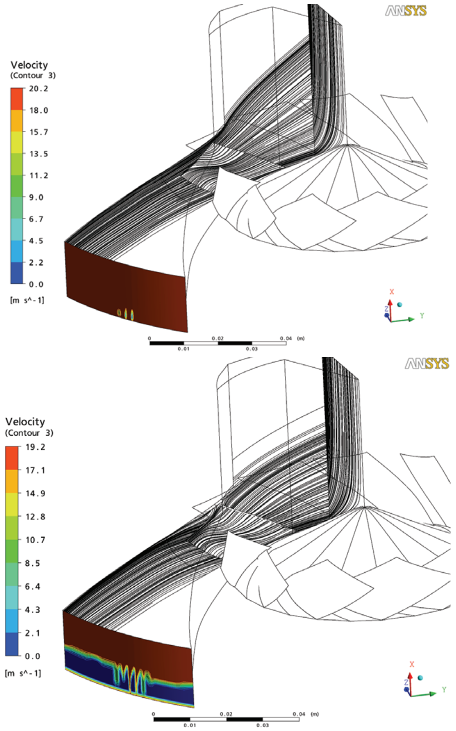

Fig. 9 *Comparison of streamlines at low and high mass flow (KGN-4 impeller)*

The display of best efficiency point of the new impeller in N_s - D_s graph (Fig. 10) only confirms that it is not the optimal design and it is as far from Cordier diagram as the old impeller.

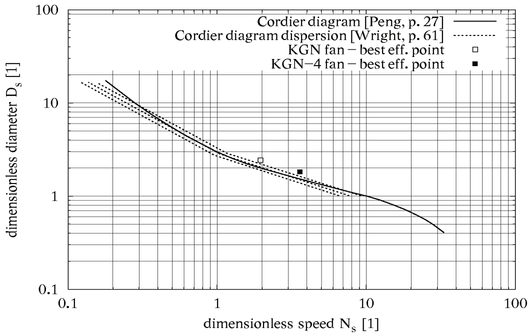

Fig. 10 *Comparison of best efficiency points of old and new impeller in N_s - D_s graph*

Therefore easy manufacturability of the impeller as well the covering box by injection moulding process impose too severe limitations to improve efficiency of the original impeller.

REFERENCES

[1] Wright, T.: Fluid Machinery: Performance, Analysis, and Design. CRC Press 1999, 376 p., ISBN 0849320151

[2] TGülich, J. F.: Centrifugal Pumps. 3rd edn., Springer 2008, 926 p., ISBN 3540736948

[3] Peng, W. W.: Fundamentals of turbomachinery. John Wiley and Sons 2007, 369 p., ISBN 0470124229

[4] Munson, B. R.: Fundamentals of Fluid Mechanics. 4th edn., John Wiley and Sons 2002, 816 p., ISBN 047144250X

[5] GAŠPAROVIČ, P., ČARNOGURSKÁ, M.: Aerodynamic Optimization of Centrifugal fan Casing using CFD. ACTA HYDRAULICA ET PNEUMATICA, 1/2008, ISSN 1336-7535, str. 8-12

[6] Anderson, J. D.: Computational Fluid Dynamics. McGraw-Hill 1995, 548 p., ISBN 0070016852

[7] GAŠPAROVIČ, P.: CFD modelovanie transsonického prúdenia v dúchadle leteckého motora. (PhD. thesis), Technická univerzita. Košice, 110 p., 2008