

AERODYNAMIC CONSIDERATION ON IMPELLER, DIFFUSER AND VOLUTE FOR MEMS CENTRIFUGAL COMPRESSOR

Jyunichi Miwa^{1*}, Chun Hui Dou¹, Kazuki Sawai¹, Moriaki Namura¹ and Toshiyuki Toriyama¹

¹Department of Micro System Technology, Ritsumeikan University, Shiga, Japan

Abstract: This paper presents optimal aerodynamic consideration on impeller, diffuser and volute for a MEMS centrifugal compressor. It is well-known that aerodynamic matching of impeller, diffuser and volute controls total performance of centrifugal compressor. There are few systematic works concerning to aerodynamic matching of these elements for large-scale centrifugal compressors as well as MEMS counterparts. Two kinds of impellers, four kinds of diffusers, and three kinds of volutes are investigated by means of CFD analysis, in order to improve the adiabatic efficiency and to establish design guideline for the MEMS centrifugal compressor under constraint small-scale structure. As a result, the combination, to which highest adiabatic efficiency is expected, is impeller with splitter blade, low solidity tandem cascade diffuser and volute with the outlet radius of 16 mm.

Keywords: centrifugal compressor, aerodynamic matching, splitter blade

INTRODUCTION

We research aerodynamic elements of the MEMS turbomachinery. Especially, MEMS centrifugal compressor can be applied to the MEMS turbomachinery. Its performance is an important factor which greatly controls the total performance of the MEMS turbomachinery. Therefore, the development of an efficient MEMS centrifugal compressor is indispensable. As a pioneering work, development of centrifugal MEMS turbomachinery was successfully demonstrated by MIT group [1]. However, there are few works concerning to aerodynamic matching of impeller, diffuser and volute for large-scale centrifugal compressors as well as MEMS counterparts with micro-scale cascades [2].

Target values of total pressure ratio of 2.2 and adiabatic efficiency of 0.5 are specified for proposed MEMS centrifugal compressor, in order to apply it to the MEMS turbomachinery. Since the centrifugal compressor is fabricated by MEMS process, its structure is limited to two-dimensional extrusion. There is large design constraint comparing to the three-dimensional counterpart [1, 2]. Therefore, rigorous aerodynamic matching for impeller, diffuser and volute for the MEMS centrifugal compressor is carried out, in order to improve the adiabatic efficiency and reduce size under structural constraint.

DESIGN AND NUMERICAL ANALYSIS

Baseline design of centrifugal compressor

Figure 1 shows baseline design centrifugal compressor consists of impeller with six back-swept blades (ϕ 10 mm, blade height of 250 μm , back-swept angle of 75 degree), vaneless and six-blade diffusers and volute. The outlet absolute velocity can

be decelerated by using back-swept blade of the impeller. Moreover, a steady operation, where the choke and surge with the variation of mass flow do not occur easily, is expected [2]. The vaneless-zone, where the flow from the impeller is decelerated, is set in the inlet region of diffuser. Circular arc airfoils, which satisfy inlet and outlet flow angles, are adopted for blade. The logarithmic spiral-shaped volute corresponding to the free vortex flow is adopted.

As a result of CFD (CFX ver. 5.7.1) analysis for the baseline centrifugal compressor, boundary layer separation and total pressure loss were suggested on the blade suction surface at impeller outlet.

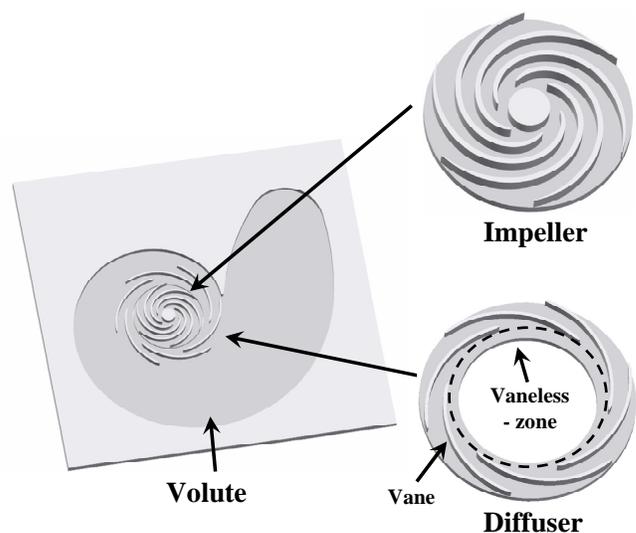


Figure 1. Schematic of baseline centrifugal compressor (impeller diameter: ϕ 10 mm, blade height: 250 μm).

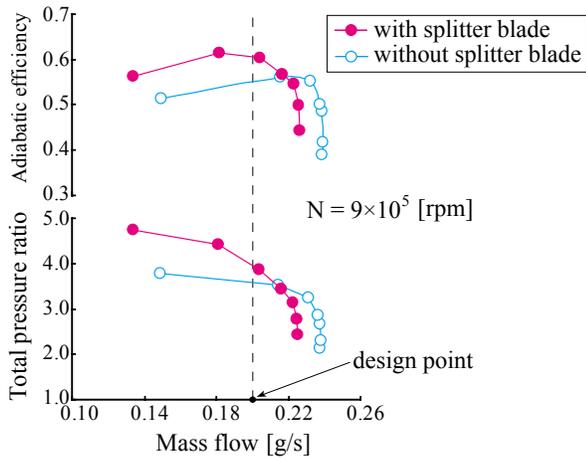


Figure 2. CFD performance prediction of centrifugal compressor (impeller outlet, $N = 9 \times 10^5$ rpm).

Impeller

It is considered that the boundary layer separation in the impeller is due to large pressure gradient between blades. In order to improve aerodynamic performance, six splitter blades are introduced between six full blades in the rear part. As a result of CFD analysis, inlet flow passage area can be kept in constant, while shorter blade pitch at trailing part reduces pressure gradient between blades. The boundary layer separation and total pressure loss are largely improved and adiabatic efficiency is increased (Fig. 2). In the following section, the splitter blades are adopted in optimal aerodynamic matching.

The adoption of the splitter blade can rectify the outlet flow, while the absolute velocity at the impeller outlet flow is increased. This introduces the total pressure loss by wall friction at diffuser inlet. Therefore, it is necessary to consider aerodynamic combination of impeller and diffuser.

Diffuser

Diffuser has the role which converts the dynamic pressure to the static pressure by flow diffusion. From the analytical result of the baseline design of the compressor, however, the total pressure has greatly lost in the conversion process. The reason is a viscous dissipation of the fluid by the friction due to long diffuser blade wall. In order to reduce the total pressure loss due to viscous dissipation by rapid deceleration of flow diffusion, effects of outlet radius of vaneless-zone and diffuser blade profile on aerodynamic performance are considered.

The roles of the vaneless-zone are to decelerate and rectify a non-uniform impeller outlet flow. For baseline design, in order to decelerate the absolute Mach number at impeller outlet, the outlet radius of the vaneless-zone is designed to be 6 mm. However,

too large vaneless-zone increases the frictional loss and causes the total pressure loss. Indeed, when the outlet radius of the vaneless-zone is decreased from 6 mm to 5.25 mm, the total pressure loss is decreased. It is noted that vaneless-zone radius of less than 5.25 mm generates a large boundary layer, and causes the total pressure loss in the volute due to insufficient rectification of flow mixture from the diffuser.

Aerodynamic performance of three kinds of diffuser shapes are considered for the improvement of the total pressure loss at diffuser. Each characteristics and design value are shown below. Moreover, the appropriate number of blades is decided by the CFD analysis. As a universal tendency, with increasing the number of blades, the frictional loss increases, the pressure gradient between blades decreases and the mixture friction of diffuser outlet flow does not occur. On the contrary, with decreasing the number of blades, the frictional loss decreases, the pressure gradient between blades increases, the boundary layer separation easily generates and the mixture friction of diffuser outlet flow occurs.

Thin straight blade diffuser [2]:

The main advantage of this diffuser is ease for design and fabrication. The shape is decided by the blade number, flow angle and blade thickness. However, disadvantages are relatively large blade number and fixed diffuser diverging angle. In the CFD analysis, flow angle of 5 degree and blade thickness of 250 μm are selected. As an empirical rule for large-scale thin straight blade diffuser, channel length – throat width ratio L/h_t of 6 - 7 gives relatively high adiabatic efficiency. Therefore, we choose 14 blades corresponding to $L/h_t = 6.8$ as baseline design. The total number of blades of 14, 16 and 18 ($L/h_t = 1.4, 6.8$ and 17.7, respectively) are evaluated. Change in the adiabatic efficiency and the pressure recovery with the variation of blade number is negligible small. Therefore, we choose 18 blades from the viewpoint of flow uniformity of diffuser outlet flow.

Channel diffuser [3]:

This diffuser adopts wedge type blade. The main advantages of this diffuser are reduction of channel number and end wall loss with keeping optimal diffuser diverging angle. The shape is decided by the blade number, flow angle and wedge angle. In the CFD analysis, flow angle of 5 degree and wedge angle of 10 degree are selected. The total number of blades of 12, 14, 20 and 24 are evaluated. As a universal tendency, the adiabatic efficiency and the

total pressure loss with the variation of blade number possess maxima and minima. The total number of blades of 20 is selected, because a maximum of the adiabatic efficiency is obtained.

Low solidity tandem cascade diffuser [4]:

The main advantage of this diffuser is large pressure recovery. USA35B is selected as blade profile. It has large operation region and stall stability against large incident flow angle. Solidities of front cascade of 0.35 and rear cascade of 0.69 are selected. The circumferential overlapping of tandem cascade is selected within 9 % of blade pitch angle ($2\pi/\text{blade number}$). The total number of blades of 8, 10 and 12 are evaluated. As a universal tendency, the adiabatic efficiency and the static pressure recovery coefficient increase with the increment of blade number. The total number of blades of 12 is selected, because maxima of the adiabatic efficiency and the static pressure recovery coefficient are obtained.

Volute

In baseline design, the logarithmic spiral-shaped volute with the outlet radius of 24.2 mm, which corresponds to the free vortex flow, is adopted. The outlet radius should be decreased for the miniaturization. Small radius causes the increase of the outlet flow velocity. However, the outlet radius is limited within 16 mm to keep target outlet absolute Mach number ($M < 0.15$) for the application of the MEMS turbomachinery element such as a micro combustor [5].

The fluid is assumed to be an incompressible flow ($M < 0.15$), and the outlet radius of the volute is decided by the following expression [2].

$$r_{out} = r_{in} \exp(\tan \alpha \cdot \theta), \tag{1}$$

where inlet radius r_{in} is 8 mm from the design value, θ is angle in the circumferential direction, and α is flow angle, respectively. It is understood the outlet radius can be reduced by reducing the flow angle from Eq. (1). However, a small flow angle generates flow friction and the impact along the volute end wall, and causes the secondary flow. CFD analysis is carried out in the case of volute outlet radius of 16, 20 and 24.2 mm, respectively. Consequently, the total pressure loss by the frictional loss takes a minimum for the outlet radius of 20 mm.

RESULTS AND DISCUSSION

The optimal shape in impeller, diffuser and volute is considered in the previous sections. However, the total performance of centrifugal compressor is varied

according to combination of each element [2]. Fig. 3 shows the combination of elements.

The diffuser locates between the impeller and the volute, and is important element controlling the pressure ratio and adiabatic efficiency of the compressor. Therefore, an optimal combination of these elements is precisely considered on a basis of four kinds of diffuser aerodynamic performance as previously evaluated (Fig. 3).

The combination of highest total pressure is the impeller with splitter blade, the thin straight blade diffuser, and the volute with outer radius of 16mm. In this case, the total pressure ratio of 2.92 is obtained (Table 1). However, poor adiabatic efficiency can not hold thermodynamic cycle for power plant application.

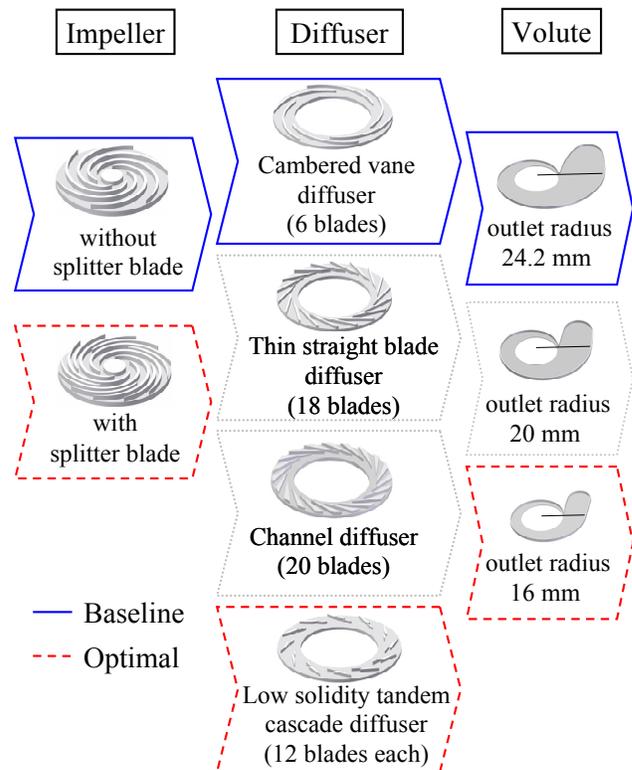


Figure 3. Combination of impeller, diffuser and volute.

Table 1: The results of aerodynamic matching of impeller, diffuser and volute.

		diffuser				
impeller	volute outlet		combered vane	thin straight blade	channel	low solidity tandem cascade
with splitter blade	16 mm	η	0.387	0.367	0.395	0.4
		PR	2.91	2.92	2.9	2.9
	20 mm	η	0.395	0.398	0.4	0.395
		PR	2.84	2.85	2.82	2.85
	24.2 mm	η	0.4	0.4	0.4	0.4
		PR	2.8	2.71	2.77	2.7

η : adiabatic efficiency, PR: total pressure ratio

The combination of highest adiabatic efficiency is the impeller with splitter blade, the low solidity tandem cascade diffuser and the volute with outer radius of 16 mm. In this case, the total pressure ratio of 2.9 and the adiabatic efficiency of 0.4 are obtained (Table 1).

In the low solidity tandem cascade diffuser, relatively slow flow on the pressure surface of front cascade passes through the blade gap between trailing edge of front cascade and leading edge of rear cascade. It is mixed with main stream in the rear cascade and passes toward downstream. As a consequence, uniform velocity profile in the rear cascade and large pressure recovery are realized. In the case of the low solidity tandem cascade diffuser, many geometrical design parameters can be specified comparing to the other type of diffusers. Therefore, we have good design prospect for improvement of the adiabatic efficiency.

FABRICATION

A proposed MEMS turbocharger consists of above designed compressor (Fig. 3), turbine and gas bearings. The final goal is to evaluate the total pressure ratio – mass flow map for centrifugal compressor by operating the MEMS turbocharger, and compare it with the CFD prediction (Fig. 2). Fig. 4 shows the schematic structure of MEMS turbocharger. Size of the main body structure, which consists of six silicon chips, is $50 \times 50 \times 3 \text{ mm}^3$. The structure of MEMS turbocharger is fabricated by bulk micromachining (Fig. 5). Target performance of the MEMS turbocharger is the rotational speed of 900,000 rpm and the total pressure ratio of 2.2. Fabrication was completed, and air turbine driven operation was confirmed.

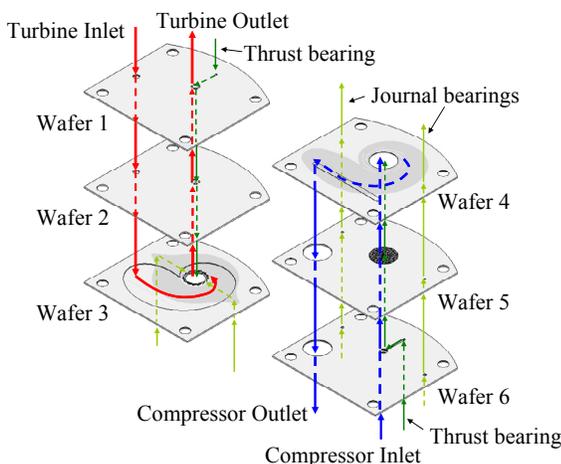


Figure 4. Schematic of MEMS turbocharger structure and gas passage. The structure consists of six silicon wafers has dimension of $50 \times 50 \times 3 \text{ mm}^3$.

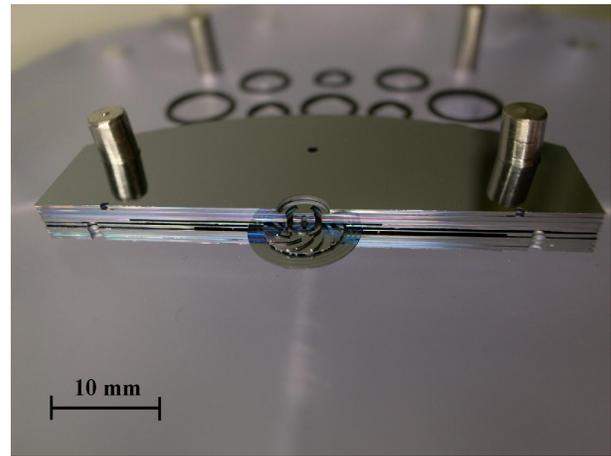


Figure 5. Cross-sectional view of MEMS turbocharger.

CONCLUSION

From a viewpoint of adiabatic efficiency, the appropriate matching among the combination is the impeller with splitter blade, the low solidity tandem cascade diffuser and the volute with outlet radius of 16 mm. In this combination, the total pressure ratio of 2.9 and the adiabatic efficiency of 0.4 are expected. However, in order to satisfy the target adiabatic efficiency, further aerodynamic consideration is necessary.

REFERENCES

- [1] N. Savoulides, S.A. Jacobson, C.J. Teo, J.M. Protz, M.A. Schmidt and A.H. Epstein, Fabrication and Testing of a High-Speed Microscale Turbocharger, *J. Microelectromech. Syst.*, **17**, pp. 1270-1282, 2008.
- [2] N.A. Cumpsty, Compressor Aerodynamics, Krieger Pub Co, Florida, 2004.
- [3] J.S. Kang and S.H. Kang, Steady and Unsteady Flow Phenomena in a Channel Diffuser of a Centrifugal Compressor, *JSME International Journal, Series B*, **47**, No.1, pp. 91-100, 2004.
- [4] Y. Senoo, H. Hayami and H. Ueki, Low Solidity Circular Tandem Cascade Diffusers of Centrifugal Blower, *Transactions of the Japan Society of Mechanical Engineers, B*, **49**, pp.578-583, 1983 (in Japanese).
- [5] A. Furutani, Y. Suzuki, S. Ozeki, C. Kim, N. Takano, N. Matsuzuka and T. Toriyama, Combustion of a Micro Can Type Combustor for MEMS Gas Turbine Engines, *Technical Digest of PowerMEMS 2008 (Sendai, Japan, 9-12 November 2008)* pp. 113-116.