International Journal of Fluid Machinery and Systems
Vol. 7, No. 1, January-March 2014

DOI: http://dx.doi.org/10.5293/IJFMS.2014.7.1.028
ISSN (Online): 1882-9554

Original Paper (Invited)

Numerical simulation Analysis of Tip Clearance Flow in a Centrifugal Compressor

Shuiqing Zhou, Jun Wang, Chuanghua Wang and Ye Li

School of Energy and Power Engineering, Huazhong University of Science and Technology,
1037 Luoyu Road, Wuhan, 430074,China

Abstract

In order to research the relationship between the tip clearance and leakage flow of centrifugal compressor, a high speed centrifugal compressor was investigated by using CFD. A numerical study on the effect of four different rotor tip clearance sizes of centrifugal compressor, which were 0.5times, 1 times, 1.5times and 2.0times of the design tip clearance, was carried out. Efficiency and pressure ratio curves were obtained under different mass flow. The reasons of the clearance vortex and the factors of vortex size were analyzed. The result indicated that with the increase of tip clearance size, the performance of the compressor changed obviously, the performance parameters such as efficiency and pressure ratio tended to decrease obviously. While, the leakage flow does not always lead to leak vortex. The strength of the vortex increased with the tip clearance. The size of leak vortex was affected by the pressure difference between the suction side and the pressure side of blade tip.

Keywords: Leakage flow; Tip clearance; Pressure ratio; Pressure difference

1. Introduction

The tip clearance is one of the important source of losses centrifugal compressors. Due to the existence of the tip clearance, leakage flow in a centrifugal impeller have seriously affected the internal flow structure, energy transport and stability margin of work[1-2]. A large amount of studies were performed concerning the effect of this tip gap on the characteristics of centrifugal compressors and turbines. However the prediction of the characteristics of compressors and turbines remains the main objective for turbomachine manufacturer. Indeed, a better description of tip leakage flows can give valuable information to design high efficiency and more stable compressor. These last few years, the development of reliable numerical tools (Computational Fluid Dynamics) and high performance computers led to significant progress to study turbomachine flow problems.

Kirtley (1990) model assumed that leakage flow in the tip clearance only exist shear flow, and there were not momentum and energy loss during the process. In order to simulate the Vena-contracta effect in the tip clearance, the real clearance height will be designated for a lower value. The advantage of Kirtley method is very simple, but the top edge leaves must be right angle. A finite volume method on unstructured grid was applied to numerically study the influence of tip clearance on the internal flow fields and aerodynamic performance of an unshouded centrifugal compressor impeller(Zhang et al,2006). The numerical results showed that the aerodynamic efficiency, pressure ratio and toque all decreased as the tip clearance increased. Zhang had found that the scale of top blade clearance is closely relative to the strength of leakage flow and distribution of tail trace position in passage, the leakage flow and passage vortex interaction has seriously affected the flow field distribution of inner passage. To this day, reliable predictions of the leakage flow in the tip clearance of a centrifugal compressor must be regarded as a challenge. All mentioned experimental and numerical investigations delivered valuable contributions partly showing similar trends, partly contradicting. Still much work has to be done in the field of impeller-diffuser interaction.

Following a previous paper[3-4], the authors conducted further detailed leakage flow of the unshouded centrifugal compressor

This manuscript was present at The 5th International Symposium on Fluid Machinery and Fluids Engineering,October24 to 27, Jeju-Korea

impeller with three different tip clearances. The effect of clearance size on the overall performance of the stage compressor was discussed. In this paper, the authors study the causes of leakage vortex and the factors of the vortex size. At the same time, the internal flow phenomena in the centrifugal compressor was obtained by 3D flow numerical simulations.

Received August 31 2012; revised January 22 2013; accepted for publication February 28 2013: Review conducted by Prof. Hyung-Hee Cho. (Paper number O13018S)
Corresponding author: Jun Wang, Professor, wangjhust@163.com

This paper was presented at 5th International Symposium on Fluid Machinery and Fluids Engineering, October 24-27, 2012, Jeju, Korea.

2. Method

Fig.1 shows the model of centrifugal compressor impeller. The compressor stage is composed of a centrifugal impeller with 11 main blades, 11 splitter blades and a vaned diffuser with 18 blades. The tip clearance between the blade tip and the shroud is 0.7 mm, and is constant from impeller inlet to exit. Four different rotor tip clearance sizes of the compressor are 0.5 times, 1 times, 1.5 times and 2 times of the design tip clearance respectively. The main geometrical parameters are listed in Table 1.

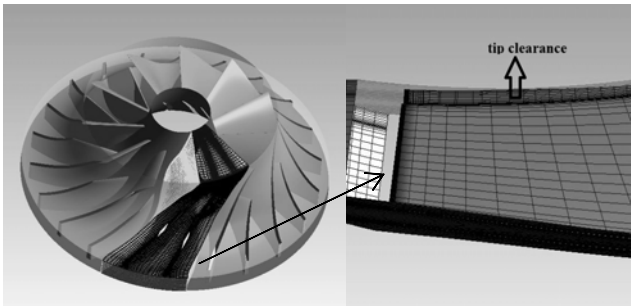


Fig. 1 Model of centrifugal compressor impeller

Table 1 Compressor stage main geometrical data

Impeller	
Blade number	11+11(splitter blade)
Inlet hub diameter	45mm
Inlet tip angle	25 deg
Impeller outside diameter	88mm
Outlet width of impeller	245mm
Exit backsweep angle	65 deg

2.1 Computational Grids

The computational Grid of impeller is showed in Fig.2.The computational domain is discretized with a multi-block approach, using an O-H meshing strategy for each passage of the compressor. The typical dimensions of a blade passage mesh are 85, 33 and 57 points, respectively in the axial, tangential and radial directions. An O-H mesh with 13 points in the radial direction is used to discretize the radial tip gap. To obtain a good balance between CPU cost and precision, a wall law approach is applied with a fixed wall cell size that corresponds to a mean normalized wall distance y+ of 15. The number of mesh points and the normalized wall distance y+ are kept constant but the dimension of the mesh cells inside the tip leakage is thus slightly increased in the case of the large tip gap.

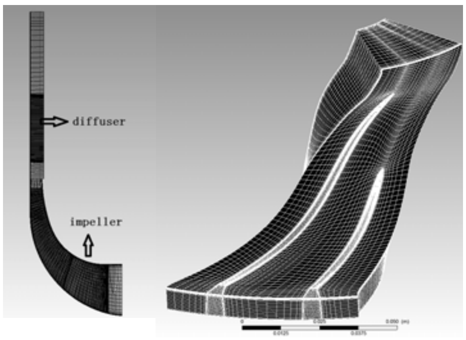


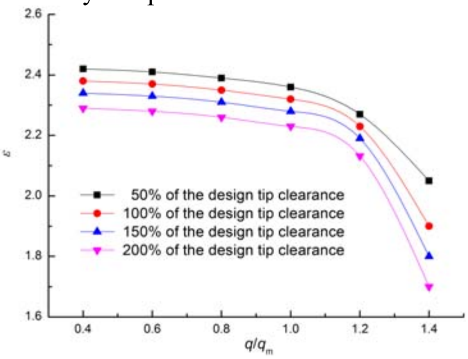
Fig. 2 Grid of impeller

2.2 Boundary conditions

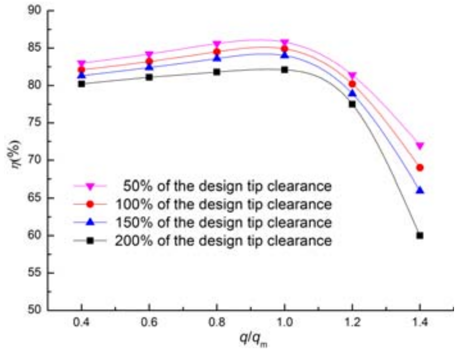
In CFD, RNG k -ε turbulent flow model is chosen. The N-S equation with finite volume method is solved and SIMPLE arithmetic is adopted. The boundary condition at compressor inlet is Total pressure-inlet. The boundary condition at compressor outlet is mass flow and its magnitude is computed from design point flow rate. The solid walls such as blade surface, hub, shroud and casing surface should satisfy the non-slip condition and are assumed to be adiabatic. All the solid walls of impeller are rotating and the rotating speed is the same as impeller. The operation pressure is 101325 Pa. The numerical calculations are carried out with a multiple frame of reference approach, whereby the impeller flow field is solved in a rotating frame and the casing in a fixed one(Wang, et al,2003).

3. Results and discussion

The total-to-total pressure ratio and the polytropic efficiency are plotted on Fig. 3 with respect to the mass flow. Fig.3(a) shows pressure ratio curve, while Fig.3(b) shows polytropic efficiency curve. The simulated results of differences between small and large tip clearance cases indicate that an increase of the tip gap leads to a reduction of the maximum pressure ratio (by 12.4%) and the maximum efficiency (by 10.1%). At nominal operating point, pressure ratio and efficiency are slightly decreased (respectively 7% and 3.6%). With the increase of tip clearance size, the performance of the compressor changed obviously, polytropic efficiency and pressure ratio tended to decrease.



(a) pressure ratio curve

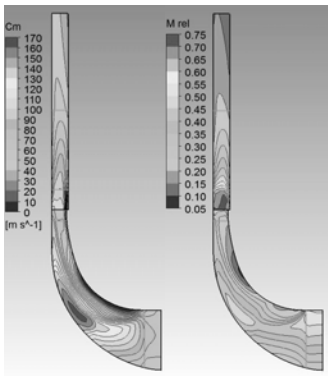


(b) polytropic efficiency curve

Fig.3 Performance curves of model compressor

The velocity in meridional surface of model compressor is shown in Fig. 4(a) under the design flow rate and 100% of the design tip clearance. A High-speed area is shown near the blade hub side. Correspond to this, Fig. 4(b) shows the distribution of relative Mach number. Because the tip leakage occurs, acceleration of the fluid nearly hub surface increases. Due to the existence of the tip clearance, a low-velocity region appears in the blade tip. The numerical model is also able to represent correctly the pressure and efficiency evolutions with respect to the mass flow.

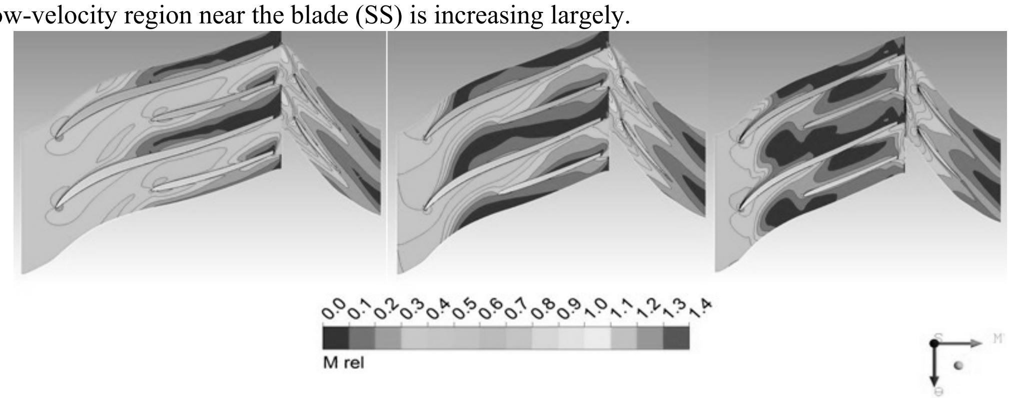
Since the cost reasons, it was not likely to product a lot of prototype with different tip clearance and test them in the reality. Using the numerical simulation technology , it was very meaningful and effective to research the leakage loss of tip clearance flow.



(a) velocity (b) relative Mach number

Fig.4 Flow simulated results of meridional surface

In order to investigate the reasons of this dramatic performance drop, a particular attention focus on the tip leakage flow phenomena. Four different rotor tip clearance sizes of the compressor are 0. 5 times, 1.0 times, 1.5 times and 2 times of the design tip clearance respectively. From them, we choose the minimum, maximum and design tip clearance cases as the main research object. Fig.5 shows a blade-to-blade view of the computed relative Mach number contours near the impeller shroud at about 95% span height, while Fig. 5 depicts an overview of tip leakage flow in the impeller-diffuser passage. Observation fig. 5(a)、(b)&(c), low-velocity region near the blade (SS) is increasing largely.



(a) 0.5times (b) 1.0times (c) 2.0times

Fig. 5 Relative Mach number contours

The low momentum regions seem to be formed mainly by tip leakage flows in fig.6. The aforementioned low-velocity region seems to be originated by both the tip leakage vortex .The boundary layer flow which undergoes a momentum loss through the shock, migrates toward the shroud, and is captured by the leakage flow and drawn in the mid-passage. Observation A and B in fig.6, obvious reflow occur in the tip clearance. While, the leakage flow would lead to leak vortex with the tip clearance increasing. Below S3 launched research, internal flow field could be monitored.

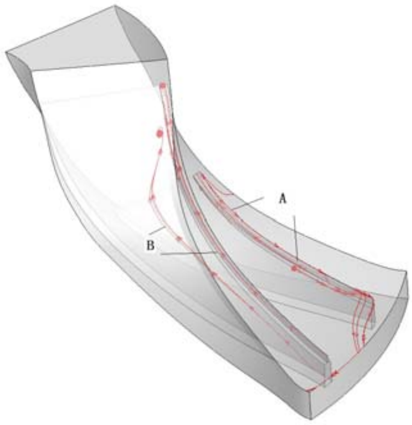


Fig. 6 Tip clearance flow

As expected the clearance model mostly influences the near shroud distributions: the low-velocity region generated by the blade tip leakage vortex, while the leakage vortex is more extended in the pitchwise direction and positioned closer to the splitter (PS) for the gridded tip gap. The high-velocity region near the shroud of the splitter (SS), due to the root of the shock wave, is more extended for the modeled than for the gridded tip gap.

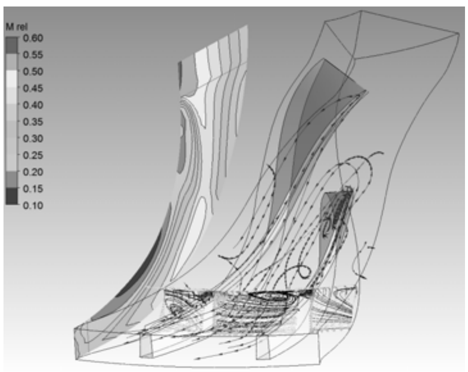
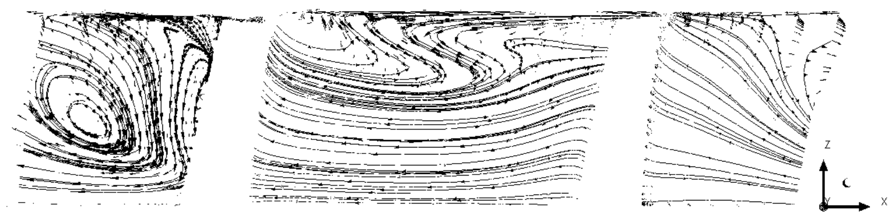


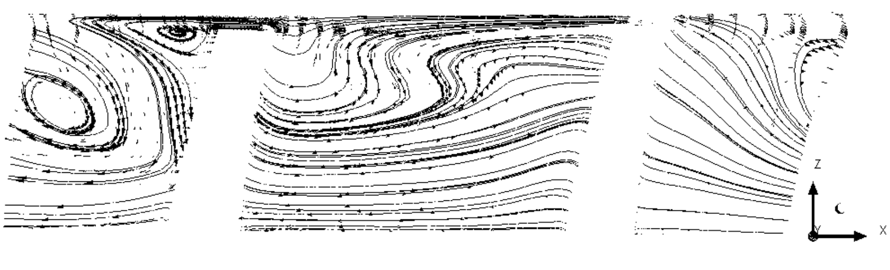
Fig. 7 Overview of 3-D steady flow

Fig. 7 depicts an overview of 3-D steady flow in model compressor. Tip leakage vortices, leakage, and secondary flows evolution in the impeller passage. The predicted relative Mach number distributions agree quite well with the tip leakage flow.

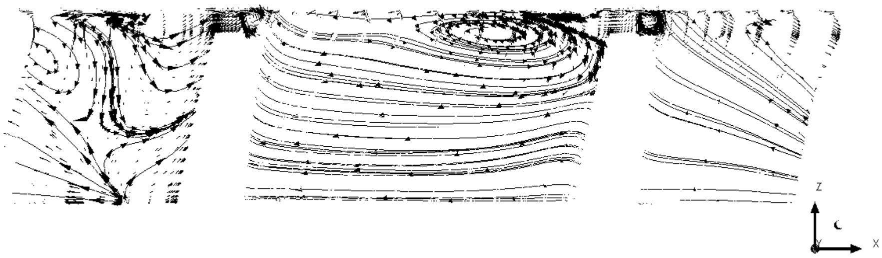
Starting from the blade SS, the tip leakage flow crosses the maximum velocity region at inlet of impeller and a low-velocity region due to a secondary flow coming from the SS of the blade. Finally, it encounters the vortex region originated from the splitter SS.



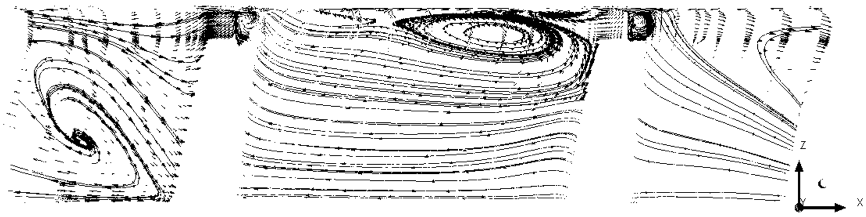
(a) 0.5times



(b) 1.0times



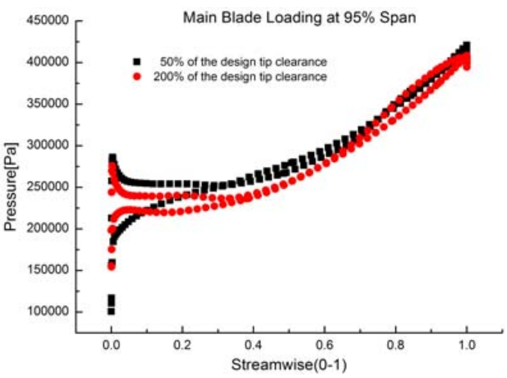
(c) 1.5times



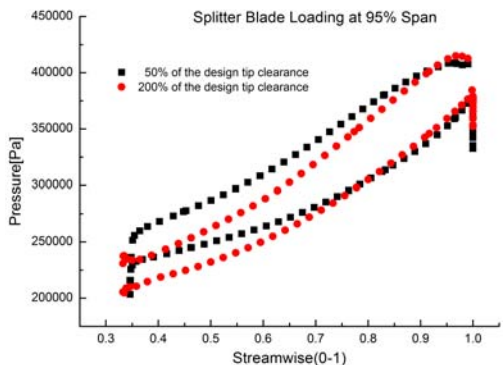
(d) 2.0times

Fig. 8 Tip leakage vortices, leakage, and secondary flows

As shown in Fig. 8, the flow phenomena of the tip leakage vortices, leakage, and secondary flows in S3 section (see in fig.7) is observed in cross sections along pitchwise direction. When the tip gap is 0.5 times of the design tip clearance, the mainly form of the flow in the cross section is secondary flow (See fig.7 (a)). With the increasing of tip clearance, clearance vortex gradually develop. And it occupy the most of the area in the blade tip, the performance of model compressor would seriously affect by leakage flow. It has been shown in the section that the impact of larger tip gap increases when the mass flow is reduced. At partial region, the flow blockage near the casing of the rotor leads to a dramatic drop in terms of mass flow, reducing also the main flow momentum. The consequence is that the tip leakage vortex tends to move closer to the rotor leading edge in fig.8 (b)、(c) ＆ (d).



(a) main blade



(b) splitter blade

Fig.9 Blade loading at 95% span

Fig.11 shows the blade loading distribution at 95% span near shroud in the model compressor. As shown in fig.9(a), the pressure difference with h/b ratios of 0.3，0.4 and 0.5 was a slightly lower than the one with h/b ratios of 0.7, 0.8 and 0.9. In fig.9(b) the pressure difference of splitter blade loading is larger than the one of main blade (See fig.9 (a)). So the leak vortex is originated at the leading edge of the blade (PS) and gradually extend to downstream along pitchwise direction .Because the pressure difference between the suction side and pressure side of the blade, it leads to leakage flow from PS to SS.

4. Conclusion

A numerical simulation has been presented to investigate the flow in a centrifugal compressor. The comparison with the different results of four kind tip clearances cases has shown the model is able to compute correctly the main characteristics of the flow, like pressure ratio and efficiency.

As expected, the tip gap modelization has an impact on the predicted flow. The predicted velocity, relative Mach number and static pressure distributions agree quite well with the tip leakage flow. Moreover, the stability limit and the tip leakage flow are well estimated by 3-D numerical simulation calculations. This study also show that a complex interaction exists between tip leakage vortices , leakage flow and secondary flows. When the tip gap is 0.5 time of the design tip clearance, the mainly form of the flow in the cross section is secondary flow. With the increasing of tip clearance, clearance vortex gradually develop. And it occupy the most of the area in the blade tip, the performance of model compressor would seriously affect by leakage flow.

From 3D flow numerical simulation, The tip leakage flow crosses the maximum velocity region at inlet of impeller and a low-velocity region due to a secondary flow coming from the SS of the blade. Finally, the pressure difference between the suction side and pressure side of the blade is the main reason for the clearance flow. With the pressure difference between PS and SS of blades increasing gradually , it can lead to leak vortex to extend along pitchwise direction.

References

[1] Higashimori, H., Hasagawa, K., Sumida, K., and Suita, T., 2004, “Detailed Flow Study of Mach Number 1.6 High Transonic Flow With a Shock Wave Pressure Ratio 11 Centrifugal Compressor Impeller,” ASME J. Turbomach,126(4), pp. 473-481.

[2] Liu, L. and Xu, Z., “Computational and experimental investigations of the steady and unsteady viscous flows in centrifugal compressor stages,” PhD thesis, Xi’an Jiaotong University, 1999.

[3] Wilke, I., Kau, H.P., 2004, “A Numerical Investigation of the Flow Mechanisms in a High Pressure Compressor Front Stage With Axial Slots,” Journal of Turbomachinery 126(3), 339–349.

[4] Hoshino, M., Ohki, H. and Yoshinaga, Y., “The effect of guide vane height on the diffuser flow in centrifugal compressors,” Trans. Jap. Soc. Mech. Engrs, 1985, 51(470)B, 3366–3369.

[5] Bonaiuti, D., Arnone, A., Ermini, M., and Baldassarre, L., 2006, “Analysis and Optimization of Transonic Centrifugal Compressor Impellers Using the Design of Experiments Technique,” ASME J. Turbomach., 128(4), pp. 786–797.

[6] Chen Hongxun, Guo Jiahong,” Numerical simulation of 3-D turbulent flow in the multi-intakes sump of the pump station[J],” Journal of Hydrodynamics, 2007, 19(1): 42-47.

[7] Nicholas P, Poul S L, and Christian B J, 2003, “Flow in a Centrifugal Pump Impeller at Design and Off-Design Conditions—Part I: Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) Measurements,” ASME Journal of Fluids Engineering, 125(1): 61-72.

[8] Ni Yongyan, Pan Zhongyong, and Li Hong, et al, 2006, “Application of Characters of Discharge Pressure Fluctuation to Monitor of Centrifugal Pump,” Drainage and Irrigation Machinery, 24(5): 40-43. (In Chinese)

[9] Wang H, Tsukamoto H, 2003, “Experimental and Numerical Study of Unsteady Flow in a Diffuser Pump at Off-Design Conditions,” ASME Journal of Fluids Engineering, 125(5): 767-778.

[10] Kirtley, K. R., Beach, T. A., and Adamczyk, J. J., 1990, “Numerical Analysis of Secondary Flow in a Two-Stage Turbine,” Proceedings 26th SAE, ASME,and ASEE, Joint Propulsion Conference, Orlando, FL, July 16–18, AIAA Paper No. 90-2356.