



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 4, April 2023

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

 9940 572 462

 6381 907 438

 ijirset@gmail.com

 www.ijirset.com

Aerodynamic Studies in the Static Components of A Centrifugal Compressor Stage

S. Mariappan¹, S. Marimuthu², F. Rubus Robinson³, A. Antony Smolin Renis⁴ and S. John Mathew⁵

Department of Mechanical Engineering, St. Mother Theresa Engineering College, Vagaikkulam, Thoothukudi, India

ABSTRACT: Aerodynamic studies in the static components of a centrifugal compressor stage were conducted using the CFD solver FLUENT. A typical centrifugal compressor stage geometry having a flow coefficient of 0.053 was chosen for the simulation study. The study is confined to the static components of the centrifugal compressor stage like crossover bend (1800 U-bend), a radial cascade of return channel vanes and exit ducting (900 L-turn). The aerodynamic performance is reported in terms of total pressure loss coefficient, static pressure recovery coefficient, return channel vane surface static pressure distribution and stage exit swirl angle distribution. The simulated flow through the static components covered five different operating conditions of the actual centrifugal compressor stage namely, design point (with 100% flow rate) and off-design operating conditions (with 70%, 80%, 110% and 120% flow rates). The standard k- ϵ model was used with standard wall functions to predict the turbulence. A minimum total pressure loss coefficient was observed near the 80% flow rate when the average flow angle at U-bend inlet was 240. The static pressure recovery was seen to be better with 70%, 80% and 100% flow rates. The swirl angle distribution at the stage exit was observed to be satisfactory.

KEYWORDS: Return channel vanes, total pressure loss coefficient, static pressure recovery coefficient, vane surface static pressure coefficient, swirl angle.

I. INTRODUCTION

The development of modern centrifugal compressors demands extensive optimization of all the flow path components. In a multistage centrifugal compressor the pressurized fluid from one stage is passed on to the subsequent stage through a 1800 circumferential U-bend, a cascade of de-swirl vanes known as “return channel vanes” and L-turn ducting with a 900 bend for connection to the subsequent stage. The function of return channel vanes is to guide the swirling flow leaving the diffuser to the eye of the subsequent impeller stage with near zero inlet swirl. The flow through U-bend and its interaction with downstream row of de-swirl vanes presents a complex fluid dynamics problem. Aungier (2000, 1993) and Ludtke (2004) presented some design guide lines for the design of return channel passages of a centrifugal compressor stage.

Srinivasa Reddy et.al (2010) reported the experimental and numerical studies in the crossover system of a typical centrifugal compressor stage with two different return channel vane configurations namely RCV1 and RCV2. In their study the static components of the centrifugal compressor stage like crossover bend, cascade of return channel vanes and the exit L-turn ducting together was referred as “crossover system”. They conducted experimental investigations in a static model of 1:1 size simulating the actual flow conditions at the design and off-design operating conditions. The numerical studies were conducted by simulating the same flow conditions for the purpose of comparison of the results. A good agreement between the experimental and numerical results was reported on the performance of the crossover system in terms of total pressure loss coefficient, static pressure recovery coefficient and vane surface pressure distribution. RCV1 configuration was observed to give better performance than RCV2. Simon and Rothstein (1983) conducted experimental investigations on a static test rig with three different geometries of return channel vanes. They reported about the nature of flow taking place through the return vane channel vanes and emphasized the need to describe the flow with the aid of simplified calculation models. In a similar fashion Inoue and Koizumi (1986) conducted experimental investigations on static test rig and reported the presence of secondary flow in U-turn and exit L-turn sections. They concluded that two dimensional blades are adequate for the De-swirl vanes. They also concluded that the losses in deswirl vanes are mainly due to the separation on vane surface. Lenke and Simon (2000) conducted CFD studies and showed that the deceleration of the flow introduces large separations and recirculation which



decreases the efficiency. They also demonstrated about the qualitative agreement between the CFD results and experimental data. The inverse design and optimization of a multistage radial compressor stage consisting of a vaneless diffuser, cross over bend and return channel was presented by Veress and Braembussche (2004). They also studied the impact of vane lean on secondary flows and showed performance improvements with negative lean. Oh et. al (2005) solved the U-bend problem using FLUENT solver. It is shown that the wake/jet flow coming into the inlet of the U turn bend develops the secondary flow in the downstream side. An inverse method to design a circular cascade for a return channel of a centrifugal turbo machine, whose vane height varies in the radial direction, was developed by Toshiaki Kanemoto and Tomitaro Toyokura (1983) using singularity method. They also developed a circular cascade model and tested its performance experimentally. They concluded that the flow pattern through the cascade is very complicated owing to the interference between the secondary flow on the end wall and the flow separation on the vane surface.

In the present study the aerodynamic performance of the static components like U-bend, cascade of return channel vanes and exit L-turn ducting with a modified return channel vane configuration namely RCV3 is presented. The aerodynamic performance of RCV3 is compared with that of RCV2 and RCV3 reported by Srinivasa Reddy et.al (2010). The numerical solution with RCV3 configuration is obtained with the same operating conditions used in the study of RCV1 and RCV2. The computational details are described in the following section.

Modeling Details

The model is created using GAMBIT preprocessor with three different blocks namely crossover bend, return channel vane and exit L-turn ducting. Three dimensional sector models are used as the flow passages are axi-symmetric. This procedure also minimized the computer memory requirement and allowed grid refinement in critical regions. The crossover bend and the L-turn ducting were meshed with structured hexahedral volumes. The return channel vane sector was meshed with unstructured hex/wedge volumes. Boundary layers were created on the vane surface to resolve the flow conditions at critical regions where flow separation is likely occur. The three blocks were coupled with “interface” feature available in the program. The meridional view of the flow path is shown Figure 1. The meshed geometry of the three blocks is shown in Figure 2. Grid independence studies were carried out to determine the optimum grid size. The details of grid independence studies are shown in Table 1. The grid generated with an interval spacing of 0.002 m is used in all simulations as it was established as the optimum grid interval size. The design details of return channel vane configuration RCV3 is shown in Table2. The flow path width at various locations is shown in Table 3.

Table 1: Grid independence studies

SNo.	Grid interval size,metres	Total number of elements	Total pressure loss coefficient
1	0.003	72610	0.50185
2	0.0025	131950	0.45027
3	0.002	251460	0.4259
4	0.0015	497232	0.4230

Table2:FlowPathDimensions

Location	Flow path width
U-bend inlet	24.5 mm
U-bend exit	20.5 mm
L-turn exit	47.5 mm

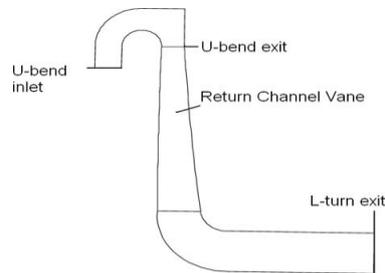


Figure 1: Meridional View of the Flow Path

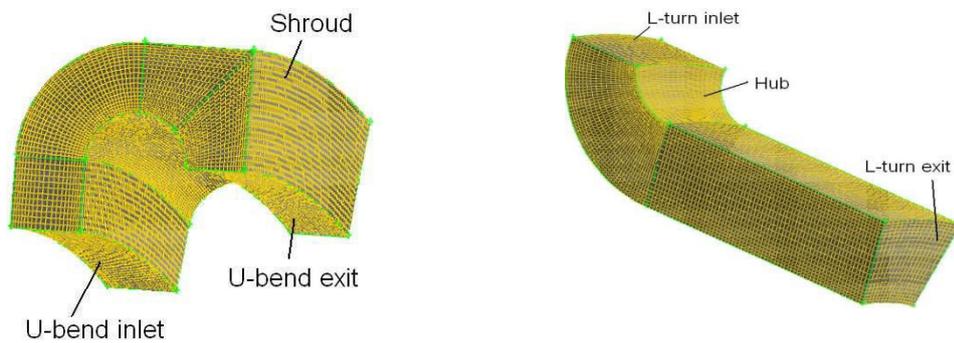


Figure 2: Meshed geometrical components and the combined model

Table 3: Vane Parameters

Parameter	Value
Inlet flow angle	25°
Exit flow angle	82°
Chord length	178 mm

Solver Settings

Pressure based solver with implicit formulation under 3-D steady flow conditions with absolute velocity formulation is chosen for the present study. Oh et.al, (2005) conducted numerical studies of U-bend in return channel systems of multistage centrifugal compressors. They used Reynolds stress turbulence model and also two equation $k - \epsilon$ turbulence model. In their study $k - \epsilon$ model predicted the turbulence closer to the experimental observations. In the present study the $k - \epsilon$ model is used to predict the turbulence. Standard wall functions were used to capture the flow phenomenon near the walls. At the inlet section “Total Pressure” was specified along with flow component directions. Also the density of air is specified. At the exit section of 90° bend the “static pressure” with “radial equilibrium pressure distribution” option with target mass flow rate is used as outlet boundary condition. The solution is assumed to converge when the maximum residual values are equal to $1e-06$. The operating conditions (the average flow angle at U-bend inlet and the corresponding mass flow rate) at which the simulations were carried out were taken based on the experimental investigations reported by Siva Reddy et.al (2003).

II. RESULTS AND DISCUSSION

Figure 3(a) shows the variation of meridional velocity from hub to shroud at U-bend exit for RCV3 configuration. Almost uniform velocities are observed from hub to shroud at all operating conditions. Higher magnitudes of meridional velocities are observed at this location than at U-bend inlet at the given operating conditions. The increase in meridional velocity from inlet to outlet of U-bend may be attributed to the narrowing of flow path width at the exit of U-bend. The distribution of circumferentially averaged flow angle distribution from hub to shroud for RCV3 configuration is shown in Figure 3(b). A very uniform distribution in flow angle is observed at all the studied operating conditions. The magnitude of average flow angle is seen to be higher at U-bend exit than at U-bend inlet at the given operating conditions. The increase in flow angle may also be attributed to the decreased flow path width at U-bend exit. The decreased flow path width causes the meridional component to increase and hence the increase in flow angle.

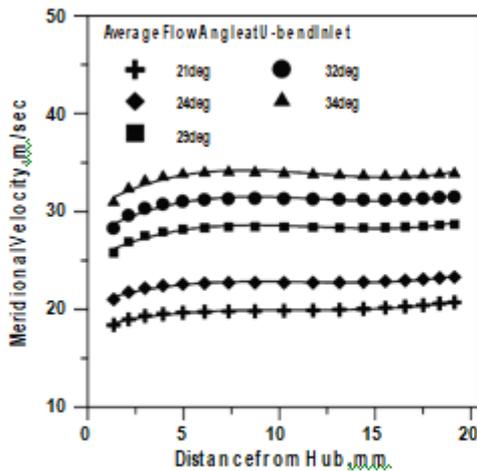


Figure3 (a): Meridional Velocity Distribution at U-bend exit

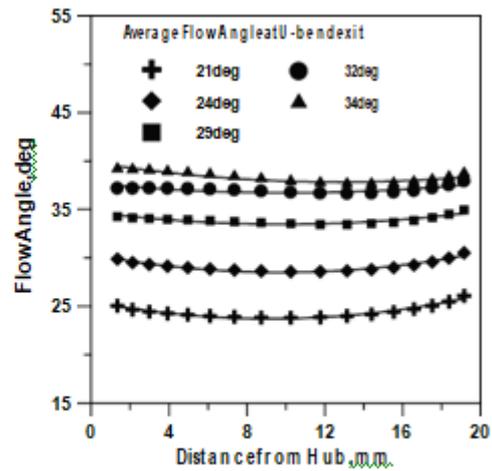


Figure3 (b): Flow Angle Distribution at U-bend exit

III. CONCLUSIONS

A numerical study was conducted to study the aerodynamic performance of the static components of centrifugal compressor stage with a vane configuration namely RCV3. A very uniform velocity and flow angle distributions were observed at the U-bend exit. The flow separations phenomenon that was seen in RCV1 and RCV2 configurations is eliminated with RCV3 configuration. The meridional velocity and swirl angle distributions are observed to be influenced by the return channel vane trailing edge. The stage exit swirl angle distribution is observed vary between 800 and 1020. The present study is found to be helpful in understanding the nature of flow through return channel passages of a centrifugal compressor stage

REFERENCES

- [1] Aungier, R. H (1993), Aerodynamic Design and Analysis of Vaneless Diffusers and Return Channels, Paper No. 93-GT-101, ASME, New York.
- [2] Aungier, R. H, 2000, Centrifugal Compressors, ASME Press, PP 187-194.
- [3] Arpad Veress., and Vanden Braembussche, R., 2004, Inverse Design and Optimization of a Return Channel for a Multistage Centrifugal Compressor, ASME Journal of Fluids Engineering-Vol. 126, pp 799-806.
- [4] Inoue, Y., and Koizumi, T., 1983, Experimental Study on Flow Patterns and Losses in
- [5] Return Passages for Centrifugal Compressor,” ASME Fluids Engineering Division, Vol-3, pp 13-21.



- [6] James M Sorokes, Bradley R Hutchinson, 2000, “The Practical Applications of CFD In the Design of Industrial Centrifugal Compressors”, ASME PID-Vol. 5, Challenges and Goals in Industrial and Pipeline Compressors.
- [7] Lenke, L.J., and Simon, H., 2000, Aerodynamic Analysis of Return Channels of Multi- Stage Centrifugal Compressors, ASME PID-Vol. 5, Challenges and Goals in
- [8] Industrial and Pipeline Compressors.
- [9] Lütke, K. H., 2004, Process centrifugal compressors, Springer, Berlin, PP 86-88.
- [10] Oh, J M., Engeda, A., and Chung, M K., 2005, A numerical Study of the U-turn Bend in Return Channel Systems for Multistage Centrifugal Compressors, Proc. I Mech E Vol. 219, part C: J. Mechanical Engineering Science.
- [11] Simon, H., and Rothstein, E., 1983, On the Development of Return Passages of Multistage centrifugal compressors, ASME FED-Vol. 3, Return Passages of Multistage Turbo machinery.
- [12] Siva Reddy T. Ch, Ramana Murty G. V , MVSSSM Prasad, DN Reddy, 2003, Theoretical and Experimental Studies on Low Solidity Vaned Diffusers of a Centrifugal Compressor Stage, Proc. Of 30th National Conference on Fluid Mechanics and Fluid Power, December 11-13, NIT, Surathkal, India, pp. 621- 628.



INNO SPACE
SJIF Scientific Journal Impact Factor
Impact Factor: 8.423



ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

9940 572 462 **6381 907 438** **ijirset@gmail.com**



www.ijirset.com

Scan to save the contact details