

A Numerical study of effect of Return Channel Vanes Shroud Wall Divergence Angle on the Cross-over System Performance in Centrifugal Compressors

* K.Srinivasa Reddy *, M Sai Kiran **, G.V.Ramana Murty *** ,

* Department of Mechanical Engineering, CMR Engineering College, Hyderabad, India,
dean.cmrec@gmail.com (corresponding author)

** M.Tech Student, Chaitanya Bharathi Institute of Technology, Hyderabad, India

***Jawaharlal Nehru Technological University, Hyderabad, India

ABSTRACT

The cross-over system of a centrifugal compressor stage consists of 180° U-bend, a cascade of return channel vanes and exit L-turn (90° bend). In a multistage centrifugal compressor, the exit flow from the impeller of a stage enters a vaned / vane less diffuser followed by the cross-over system. The main function of the cross-over system is to smoothly guide the flow from the exit of the diffuser of a stage to the eye of the impeller of subsequent stage with minimum aerodynamic losses and swirl-free flow. A numerical study was carried out on three cross-over system configurations at design and off-design operating conditions. The three configurations of cross-over system have different RCV semi-divergence angles namely, 4°, 6°, and 8°. The turbulence is predicted with 2-equation k-epsilon model. The grid is refined in critical regions to capture the flow pattern accurately. Three dimensional sector models were used as the flow is nearly axi-symmetric. The results are presented both in qualitative and quantitative forms. The total pressure loss coefficient variation is observed to be favorable with 8° configuration. However the calculated static pressure recovery is found to be better with 6° configuration.

I. INTRODUCTION

In a multistage centrifugal compressor, the exit flow from the impeller of a stage enters a vaned / vane less diffuser followed by the cross-over system. The main function of the cross-over system is to smoothly guide the flow from the exit of the diffuser of a stage to the eye of the impeller of subsequent stage with minimum aerodynamic losses and swirl-free flow. The cross-over system of a centrifugal compressor stage consists of 180° U-bend, a cascade of return channel vanes and exit L-turn (90° bend) as shown in Fig.1. The aerodynamic performance of 180° U-bend, return channel vanes and exit ducting which form the cross-over system, influences the overall performance of the stage of a centrifugal compressor. The performance optimization of these individual components is crucial in reducing the power consumption requirements of the centrifugal compressors. The flow coming out of the centrifugal compressor impeller is having significant tangential velocity component as it enters the cross-over 180° circumferential bend. The swirling flow is subjected to sharp curvature in 180° U-bend resulting in intense energy exchanges before it enters the return channel vanes. In the return channel vanes the flow is further decelerated in the process of removing the swirl. This deceleration of flow which has significant tangential

component of velocity leads to the development of cross-flows. Simon and Rothstein [1] created a test bed for carrying out systematic measurements on return channel passages 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 [2] conducted experimental investigations on an entire flow model for return passages including a U-turn bend, a deswirl vane section and an L-turn section at the exit. They reported the presence of secondary flow in U-turn and exit L-turn sections. Because of the secondary flow in deswirl vanes; the flow has a swirl component at the return channel exit. They also concluded that most of the losses in deswirl vane section can be attributed to the flow separation for small inlet flow angle. Lenke and Simon [3] conducted CFD studies and showed that for small flow coefficient flows, the frictional losses are more dominant and increase the loss coefficients whereas for higher flow coefficients secondary flow increases and care has to be taken to avoid small streamline radius of curvature within the cross over bend. Veress and Braembussche [4] presented the inverse design and optimization of a multistage radial

compressor stage consisting of a vane less diffuser, cross over bend and return channel. They studied the impact of vane lean on secondary flows and showed performance improvements with negative lean. A numerical study of the U-turn bend in return channel systems for multi stage centrifugal compressors was conducted by Oh et. al. [5]. They have discussed in detail the loss mechanisms in the U-turn bend along with the effect of turbulence models on the flow behavior. Toshiaki Kanemoto and Tomitaro Toyokura [6] designed a circular cascade for a return channel of a centrifugal turbo machine, whose vane height varies in the radial direction using singularity method. They also developed a circular cascade model and tested its performance experimentally [7]. They concluded that the minimum flow loss is given at a small positive incidence angle and the mixing loss downstream of the cascade is considerable. In the present numerical study, the effect of semi-divergence angle of the return channel vane on the performance of the cross-over system is presented. The meridional view of the chosen three cross-over system configurations is shown in Fig.2.

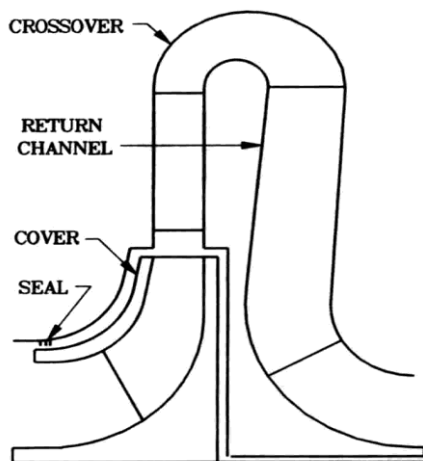


Fig.1 Meridional View of the Cross-Over System

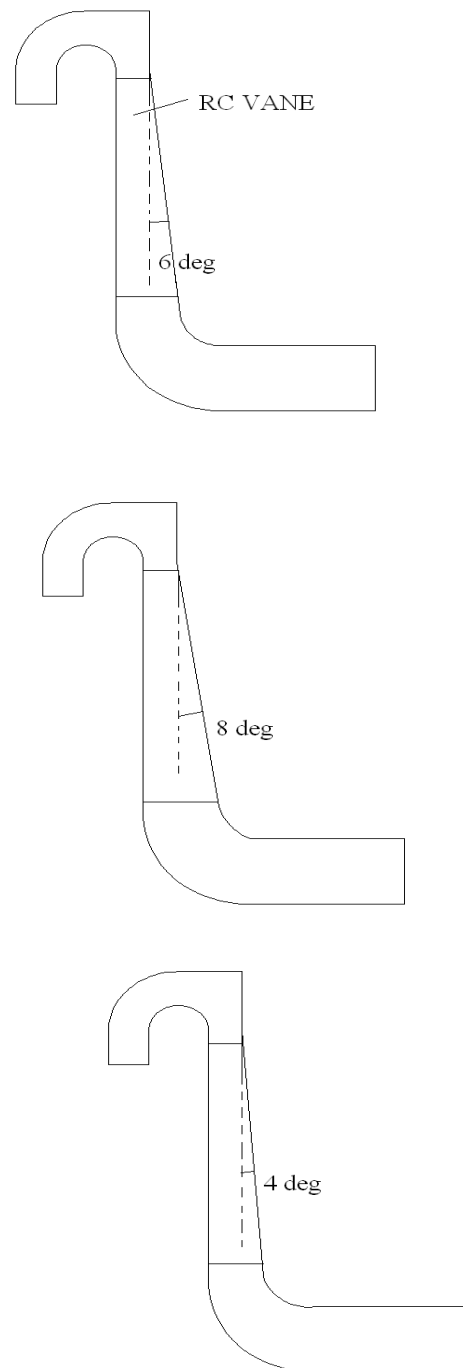


Fig.2. Configurations showing different RCV wall divergence angles

II. NUMERICAL SOLUTION

2.1 Grid Setup

The three dimensional sector models are appropriate as the flow passages are axi-symmetric. This procedure also minimised the computer memory requirement and allowed grid refinement in critical

regions [9]. The sector model of flow path geometry used for simulation is shown in Fig 3(a) and the computational grid for RCV in Fig 3(b). Structured hexahedral 3-D elements are used for U-bend and exit 90o bend sections while unstructured hex/wedge elements are used for the grid setup in return channel vanes. The exit section of 180⁰ U-bend and inlet section of return channel vanes are coupled with “interface” feature available in the program. Similarly the exit section of return channel vanes and inlet section of 90⁰ bend are coupled with interface feature. Near the walls of flow geometry grid is refined. To capture the flow separation, fine grid features were used on the return channel vane surface and wall surfaces. At the inlet section “Total Pressure” was specified along with flow component directions. Also the density of air is specified. The data used for experimental investigations only is specified as inlet boundary conditions in the present study for comparison. At the exit section of 90o bend the “static pressure” with “radial equilibrium pressure distribution” option with target mass flow rate is used as outlet boundary condition. Grid independence studies were conducted for each case.

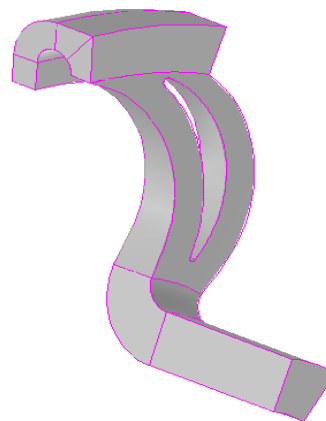


Fig 3(a) Flow path geometry

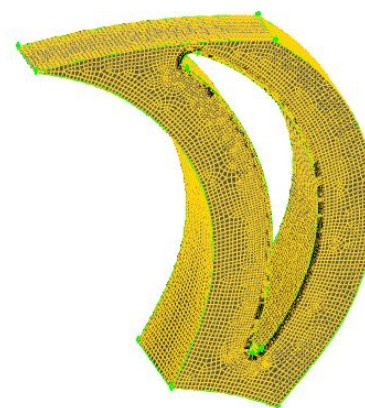


Fig 3(b) RCV grid setup

2.2 Grid independence studies

To ensure that the solution is obtained with sufficient grid spacing for accuracy, grid sensitivity studies were conducted with different interval spacing. The total pressure loss coefficient is chosen as the basic parameter to decide the optimal grid size. The details of studies carried out for an average U-bend inlet flow angle of 29⁰ are shown in Table 1.

Table 5.1 Grid independence study

S no	Grid interval size	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

Based on the study, the solution is found to be grid independent with an interval size of 0.002. Hence all the solution runs were conducted at this grid size for the chosen geometry.

2.3 Solver environment settings

The 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, [5] 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. The solution is assumed to converge when the maximum residual values are equal to 1e-06.

III. RESULTS AND DISCUSSIONS

The flow simulations were carried out for 70%, 80%, 100%, 110% and 120% of the design flow arte and their corresponding U-bend inlet flow angles [8]. The vector plots on the hub surface of the 4⁰ configurations studied at U-bend inlet flow angle Of 29⁰ is shown in Fig.4. The vector plot of total velocity as indicated in the three configurations show

the flow separation taking place towards the downstream of suction side. The flow separation for 8° wall divergence angle is observed to occur early than the other configurations. In all the configurations, acceleration of flow is observed to take place near the suction side of the vane at the upstream section, while deceleration is observed on the pressure side. In all the three configuration studied, the flow is observed to migrate from the PS to SS after the T.E due to the formation of low pressure zone which resulted from the flow separation phenomena. The reason for flow separation is quite obvious and is attributed to the vane curvature. The secondary flows calculated at the exit of the RCV for the 6 deg configuration is shown in Fig 5. The secondary flows are strongly oriented towards the shroud wall, migrating from PS to the SS just after the exit of the RCV. These secondary flows are responsible for the flow losses as well as increase in the exit swirl angle. The swirl angle should be reduced by the time the flow exits the stage.

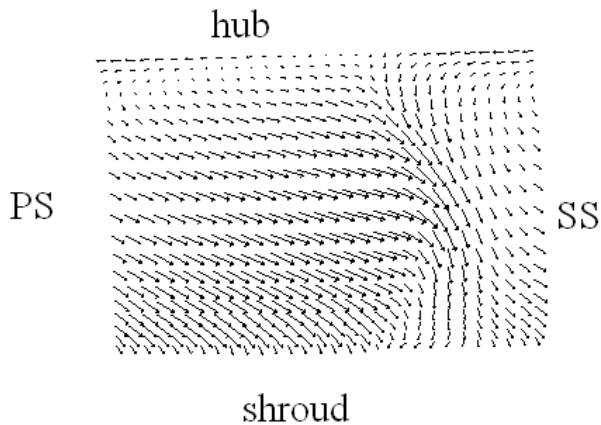


Fig.5. Secondary flows calculated at the RCV exit (6 deg) ($\alpha_1= 29$ deg)

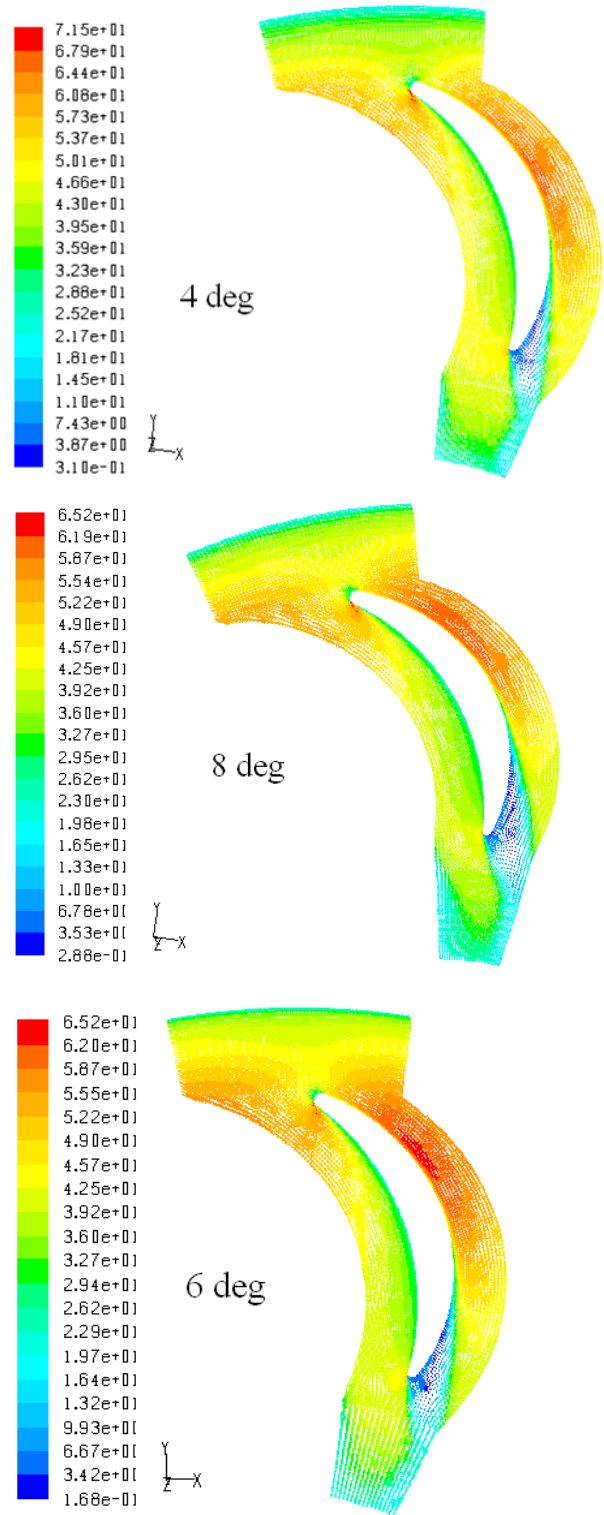


Fig.4. Velocity Vector Plot on a plane passing through mid span ($\alpha_1= 29$ deg)

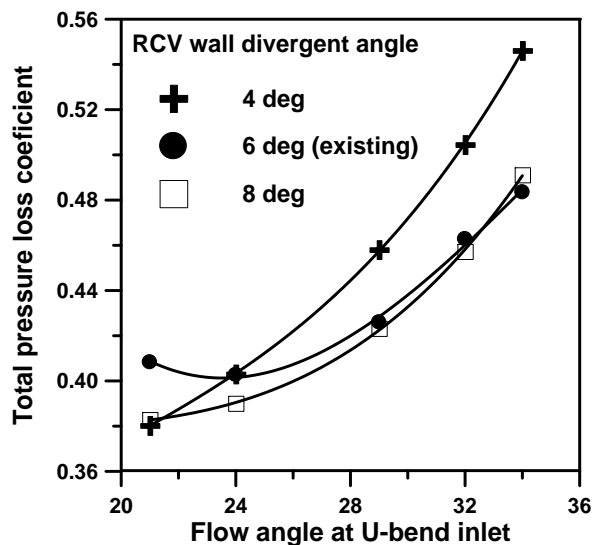


Fig.6 (a) Total pressure loss coefficient

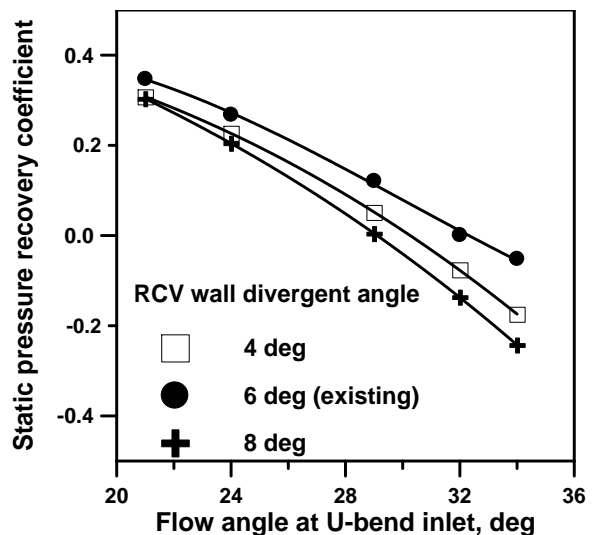


Fig.6 (b) Static pressure recovery coefficient

The variation of total pressure loss coefficient and static pressure recovery coefficient are shown in Fig 6(a) and 6(b) respectively. The total pressure loss coefficient is seen to decrease first for 6 deg configuration and is observed to increase thereafter with increase in the flow angle at U-bend inlet. In the other configurations the total pressure loss is seen to increase continuously with the increase in the flow angle at U-bend inlet. The variation of total pressure loss coefficient with increase in U-bend inlet flow angle is observed to be favorable with 8 degree wall divergence angle. The reason for decrease in the total pressure loss may be attributed to the increased flow area in the downstream direction with increased wall divergence angle. However the static pressure recovery is found to be lower with 8 degrees of RCV wall divergence, when compared to the existing case. Therefore a wall divergence angle of 7 degrees may be chosen for the present study to give better cross-over stage performance.

IV. CONCLUSIONS

- The flow through return channel passages is subjected flow separation on the suction side due the vane curvature.
- The secondary flows are dominantly seen at the RCV exit which is migrating from PS to SS.
- The wall divergence angle of 8 deg is seen to be favorable in reducing the total pressure losses in the stage.

- The static pressure recovery is superior in the case of 6 deg wall divergence angle compared to the other cases.

REFERENCES

- [1] 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.
- [2] Inoue, Y., and Koizumi, T., 1983, "Experimental Study on Flow Patterns and Losses in Return Passages for Centrifugal Compressor," ASME Fluids Engineering Division, Vol-3, pp 13-21.
- [3] 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 Industrial and Pipeline Compressors.
- [4] 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.
- [5] 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.
- [6] Tomitaro Toyokura and Toshiaki Kanemoto, "Studies on Circular cascades for Return

- Channels of Centrifugal Turbomachinery”
1st report-Inverse Method and Cascade Design, Bulletin of JSME, Vol 29, No. 255, September 1986, Paper No: 255-22.
- [7] Tomitaro Toyokura and Toshiaki Kanemoto, “Studies on Circular cascades for Return Channels of Centrifugal Turbomachinery” 2nd report-Flows and Performances of Cascades, Bulletin of JSME, Vol 29, No. 255, September 1986, Paper No: 255-23.
- [8] T Ch Siva Reddy, GV Ramana Murty, MVSSM Prasad, DN Reddy, “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, 2003, NIT, Surathkal, India, pp 621-628.
- [9] James M Sorokes, Bradley R Hutchinson, “The Practical Applications of CFD In the Design of Industrial Centrifugal Compressors”, ASME PID-Vol. 5, Challenges and Goals in Industrial and Pipeline Compressors

NOTATIONS

C_p	static pressure recovery coefficient $[(p_{s3} - p_{s1}) / (p_{t1} - p_{s1})]$
C_{pv}	vane surface pressure coefficient $[(p_s - p_{s1}) / (p_{t1} - p_{s1})]$
L.E	leading edge
PS	pressure side
p	pressure
RCV	return channel vanes
SS	suction side
T.E	trailing edge
α	absolute flow angle
ϕ	flow coefficient
ζ	total pressure loss coefficient $[(p_{t1} - p_{t3}) / (p_{t1} - p_{s1})]$

SUBSCRIPTS

s	static
t	total
1	U-bend inlet
2	U-bend exit
3	90 ⁰ bend exit