AERODYNAMIC ANALYSIS OF RETURN CHANNEL VANES IN CENTRIFUGAL COMPRESSORS

P. RAVINDER REDDY[1] & M. SAIKIRAN [2]

[1]Professor & Head, Department of Mechanical Engineering, CBIT, Hyderabad, Telangana, India

[2]M.E Student, Department of Mechanical Engineering, CBIT, Hyderabad, Telangana, India

ABSTRACT

A compressor is a device capable of efficiently transferring energy to the fluid medium so that it can be delivered in large quantities at elevated pressure condition. Compressors have numerous applications ranging from aircraft and process industries to household appliances such as refrigerators and air conditioners. There are numerous types of compressors, each suitable for a particular application. The return channel provides the connection and carries the flow between two stages of a multistage centrifugal compressor. These return channel vanes are located between the two stages through which the fluid passes from one stage to another stage in a centrifugal compressor. Its main function is to rectify the outlet swirling flow from the former stage with possible little flow loss before it enters the impeller of next stage. There are many complexities involve designing the return channel passages, namely; the total pressure loss coefficient, static pressure recovery coefficient and uniform blade loading. The design parameter of a return channel on the performance of a multistage centrifugal compressor is studied through numerical investigations. To find an optimum configuration of compressor with minimization of losses the CFD analysis is performed by changing the design parameter. In the present work a new return channel vane profile is designed and its performance is evaluated using CFD code FLUENT. The aerodynamic studies include the geometry generation of the return channel vanes, meshing, CFD-calculations and performance evaluation. The parameters such as total pressure loss coefficient, static pressure recovery coefficient, vane load distribution on the return channel passages are studied.

KEYWORDS: Multi Stage Compressor, Return Channel Vane, Flow Separation, Divergence Angle

INTRODUCTION

Multistage centrifugal compressors are widely used in applications in the oil and gas fields where compressors are operated for long periods, and hence their reliability is very important. Impellers of multistage compressors are linked by a diffuser, cross-over and return channel the main task of the vane less or vaned diffuser is to transform as much as possible kinetic energy, available at the impeller exit, into static pressure rise. The function of the return channel vanes is to guide the swirling flow leaving the diffuser to the eye of the subsequent impeller stage with near-zero inlet swirls. B. Balakrishna, S. Venkateswarlu, P. Ravinder Reddy [1], have studied the flow analysis of an atmosphere reentry vehicle using CFD techniques. K. Srinivasa Reddy, G.V. Ramana Murthy, Dasgupta and Sharma [2] (2010) reported experimental and numerical studies in the crossover system of a typical centrifugal compressor stage with two different configurations of return channel vanes: RCV1, and RCV2. In their study, the ensemble of the static components of the centrifugal compressor stage, i.e., the crossover bend, cascade of return channel vanes, and the exit L-turn ducting, was referred to as the "crossover system".

The RCV1 configuration was observed to give better performance than RCV2. Simon & Rothstein [3] (1983) conducted experimental investigations on a static test rig with three different geometries of return channel vanes. They reported on the nature of flow through the return channel vanes and emphasized the need to describe the flow with the aid of simplified calculation models. In a similar fashion, Inoue and Koizumi [4] (1983) conducted experimental investigations on a static test rig and reported the presence of secondary flow in the U-turn and the exit L-turn sections. They concluded that two-dimensional blades are adequate for the de-swirl vanes and that losses in the de-swirl vanes are due mainly to separation on the vane surfaces. Lenke and Simon [5] (2000) conducted computational fluid dynamics (CFD) studies and showed that the deceleration of the flow introduces large separations and recirculation, which decreases the efficiency. An inverse design and optimization of a multistage radial compressor stage comprising a vane less diffuser, crossover bend, and return channel, was presented by Veress and Braembussche [6] (2004).

They also studied the impact of vane lean on secondary flows and showed performance improvements with negative lean. Oh, Engeda and Chung [7] (2005) solved the U-bend problem by using the FLUENT solver. An inverse method to design a circular cascade for the return channel of a centrifugal turbo machine, whose vane height varies in the radial direction, was developed by Toyokura, Kanemoto and Hatta [7] (1986) using a singularity method. The main purpose of the return channel design is to rectify the outlet swirling flow from the former stage with possible little flow loss before it entering the impeller of the next stage.

A numerical study of the U-turn bend in return channel systems for multi stage centrifugal compressors was conducted by Oh ET. Al. They have discussed in detail the loss mechanisms in the U-turn bend along with the effect of turbulence models on the flow behavior. Aalburg ET a1 [7] sought to reduce the diffuser radius ratio (outer radius/inner radius) from 1.45 to 1.19 without losses in performance. Despite the increase in flow turning and higher vane loading, optimization of the return channel vane and end walls gave an increase of up to one point in stage efficiency across the operating range. Thygesen [10] has shown that higher efficiencies are possible by extending the swirl vanes upstream of the return bend. Similar experience is reported in. It provides a way to increase, the overall compressor performance without increasing the dimensions. The dependence of the solution on the turbulence model has not been evaluated in this study, but a comprehensive assessment on the computation of turbo machinery flows using the k-ε model has been made by Lakhsminarayana [11]. K. Bala Showry, A.V.S. Raju, Dr. P. Ravinder Reddy [12] (2012), have analyzed the multi-dimensional modeling of direct injection diesel engine and their effects of split injection, and B. Balakrishna, P. Sri Seshavani, P. Ravinder Reddy [12] (2013) used the CFD techniques for the studies on the effect of angle of flap on lift and drag in a flapping wing flight. Further the authors B. Balakrishna, Sushma Indan, P. Ravinder Reddy [14] (2013), carried out the investigation of supersonic flow through conical nozzle with various sngles of divergence. The detailed study of Lenke and Simon [4] indicates that calculations with a standard k-ε model underestimate the separation length at lower Reynolds and that a modification of the turbulence model may be needed when comparing with laboratory testing at low-pressure or compressors with extremely low flow coefficients.

MATERIALS AND METHODS

According to the reference 2 the different configurations of return channel are given in Figure 1.

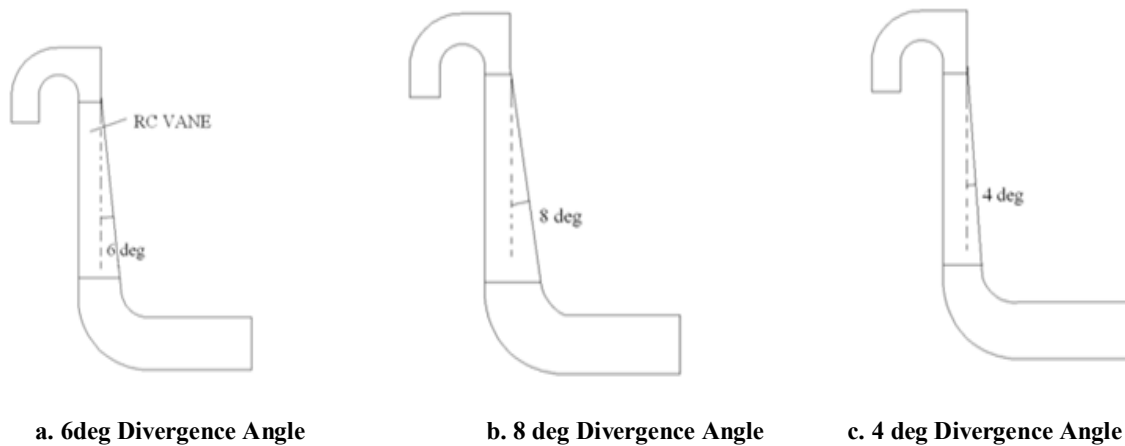| a. 6deg Divergence Angle | b. 8 deg Divergence Angle | c. 4 deg Divergence Angle |

Figure 1: 2d Model of RCV with the Different Divergence Angle

The geometry is divided in to the 3 parts such as U-bend, Return channel vane, and L-bend. The geometric model is created by using the GAMBIT preprocessor with three different blocks: crossover bend, return channel vane, and exit L-turn ducting. Three-dimensional sector models are used because the flow passages are axi-symmetric. The crossover bend and the L-turn ducting are meshed with structured hexahedral volumes. The return channel vane sector is meshed with unstructured hex/wedge volumes. Boundary layers are created on the vane surface to resolve the flow conditions at critical regions where flow separation is likely occur. The three blocks are coupled with the "interface" feature available in the program. The Figure 2 shows the governing equation with the boundary conditions as well as the inlet, outlet conditions. The three dimensional sector models are appropriate as the flow passages are axi-symmetric.
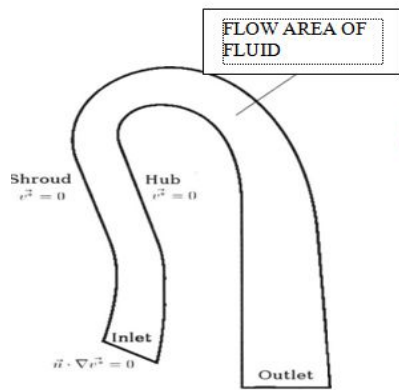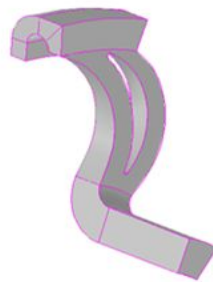

| **Figure 2: Boundary Conditions** | **Figure 3: Geometric Model of Flow Path** | **Figure 4: RCV Grid Set Up** |

The sector model of flow path geometry used for simulation is shown in the Figure 3 and the computational grid for RCV is shown in the Figure 4. Structured hexahedral 3-D elements are used for U-bend and exit 90° bend sections while unstructured hex/wedge elements are used for the grid setup in return channel vanes. The exit section of 180^0 U-bend and inlet section of return channel vanes are coupled with "interface" feature available in the program.
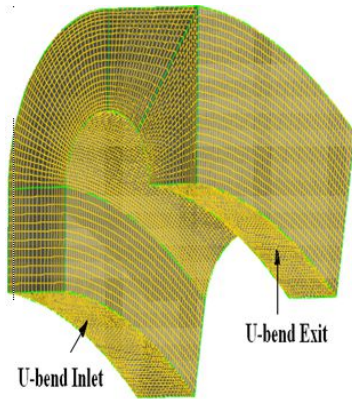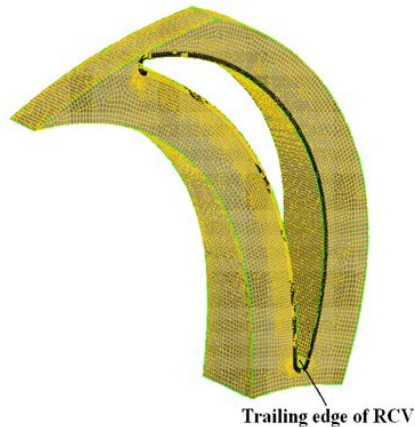
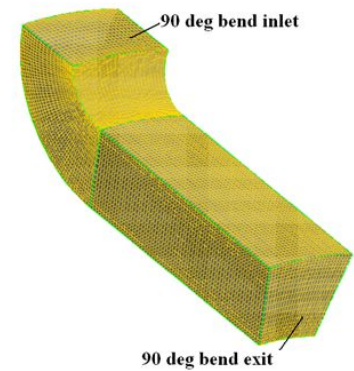**Figure 5: U-Bend Profile with Mesh**   **Figure 6: RCV with Mesh**   **Figure 7: L-Bend Profile with Mesh**

Similarly the exit section of return channel vanes and inlet section of 90^0 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 90° 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. 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 29^0 references are computed and 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. The u-bend in3d with grid size of 0.002 and inlet as well as the exit of the flow through the u-bend is shown in the below Fig. 5 and the return channel vane with trailing edge of a centrifugal compressor with the mesh is shown in the Figure 6. The L-bend with mesh is shown in the Figure 7 as the fluid takes 90^0 turn so the L-bend is designed according to the flow and leaves the fluid exactly at the eye impeller of next stage

BOUNDARY CONDITIONS

The boundary conditions for the U-bend, L-bend, and Return channel vanes are: the inlet condition is taken as pressure inlet and the outlet is taken as interface and the sides of U-bend are taken as wall and the other side's such as thickness are taken as a periodic. The inlet condition is taken as a interface and the outlet condition is taken as pressure outlet and the other side's is taken as walls and the other side's is taken as periodic. The sides of the vane and surface of the blade is taken as a wall and the inlet and outlet is taken as interface's because at the inlet it will be interacting with the U-bend interface and the outlet interface will be interacting with the L-bend inlet interface. The fluid enters through the U-bend inlet and takes 180^0 turn and enters into the return channel vane and flows over the vane during this period the swirling motion which is occurred at the U-bend outlet due to180^0 turn will be removed and at the exit fluid takes 90^0 turn enters into the L-bend Inlet an through the L-bend exit enters into the next stage. The geometry after reading the 3 different parts (U-bend, return channel passage, L-bend) of entire geometry is shown in the Figure 8.
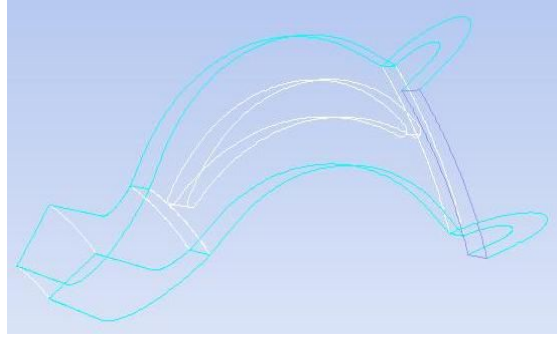
Figure 8: Complete Geometry after Joining the 3 Parts Using T-Grid

SOLVER SETTINGS & FLOW CONDITIONS

A velocity inlet with parabolic velocity profile is selected to reduce the loss from viscous dissipation near the inlet. In contrast, a uniform inlet velocity profile with the same mean yields nearly three times the loss in the flow field. A pressure outlet and non-slip walls are also used as the boundary conditions for the flow field. The laminar viscosity is set to the cell volume average effective viscosity based on a compressible turbulent flow return channel calculation in ANSYS Fluent to provide some connection to the actual situation. The grid generated for the entire stage starting from U-bend inlet to L-turn exit consists of three separate blocks. They are a sector of U-bend, a sector of de swirl vane and a sector of L-turn exit. There are 18 de-swirl vanes in the return channel geometry, of which only one de-swirl vane covering a rotational angle of 200 is considered for simulation. Similarly the circumferential U-bend and L-turn ducting were also modeled as sector elements covering a rotational angle of exactly 200. The grid is generated using GAMBIT preprocessor, which is an in-built modeling tool that comes along with FLUENT package. The steps involved in the grid generation process are explained in the following sections.

The inlet of U-bend was defined as "Pressure Inlet" boundary, while L-turn exit was specified as "Pressure Outlet", with target mass flow rate and "Radial Equilibrium Pressure Distribution" option. The exit of U-bend and inlet of L-turn section were specified as "INTERFACE" boundaries. The inlet and exit of the vane sector were defined as "INTERFACE" boundaries. Since sector slices were used in the simulation, the periodic faces were defined with appropriate "Periodic" boundary condition. The data obtained from experimental investigations were specified as inlet boundary conditions in the present study for ease of comparison. At the exit section of 90^0 bend, the static pressure with radial equilibrium pressure distribution option with target mass flow rate was used as outlet boundary condition. The pressure based solver with implicit formulation under 3-dimensional steady flow conditions with absolute velocity formulation is chosen. In the present study the standard k–ε model is used to predict the turbulence. The near wall treatment is handled with standard wall functions. The second order upwind scheme is used for discretization of convection terms. The solution was obtained with the maximum residual values equal to 1e-06. 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. Typical performance parameters: (i) Static pressure: it is the pressure of the fluid which enters at diffuser and leaves at the L-duck(at 90° turn), (ii) dynamic pressure is the kinetic energy per unit volume of a fluid particle, (iii)total pressure loss: it is the sum of the both static and dynamic pressure. To increase the efficiency of the compressor total pressure loss should be minimum. (iv) Swirl angle, swirl is the tangential flow component of the velocity vector. It should not exceed more than ±2or3°.

FLUID PROPERTIES

Density=1.0836999 kg/m^3, Cp (Specific Heat) = 1006.43 j/kg-k, Thermal Conductivity=0.0242 w/m-k, Viscosity=1.7894001e-05 kg/m-s, Molecular Weight=28.966 kg/kg mol, L-Characteristic Length=3.711 angstrom, L-J Energy Parameter=78.6 k.

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 4^0 configuration studied at U-bend inlet flow angle of 29^0 is shown in Figure 9. The vector plot of total velocity as indicated in the three configurations shows the flow separation taking place towards the downstream of suction side. The flow separation for 8^0 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.
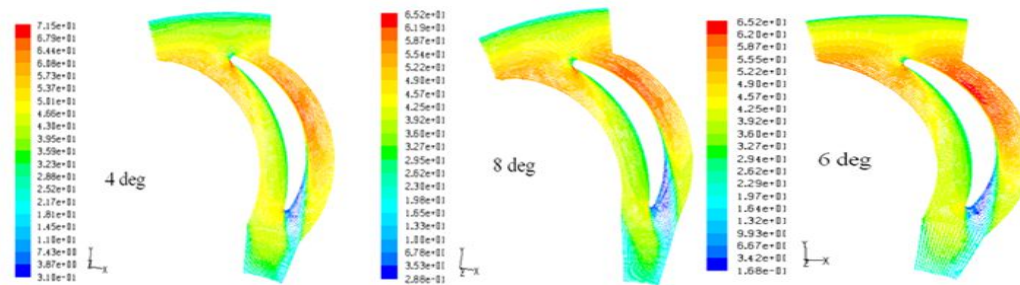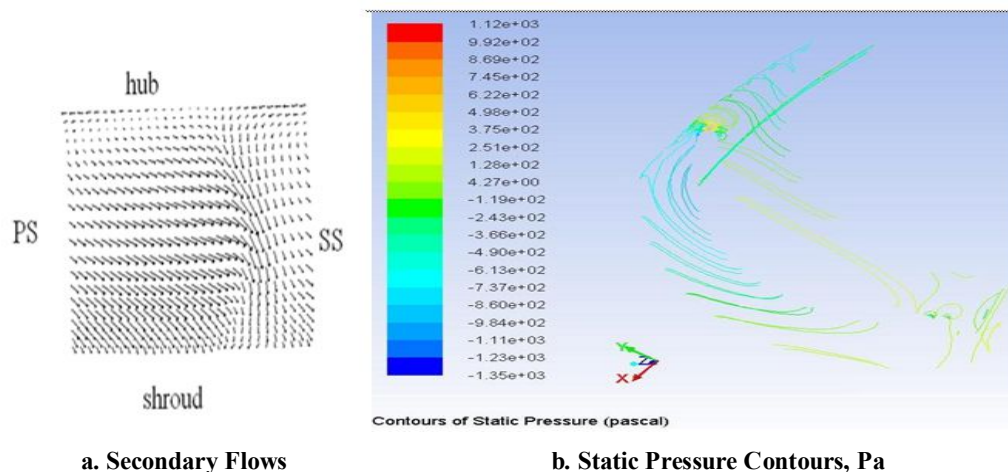

Figure 9: Velocity Vector Plot on a Plane Passing Through Mid Span (α_1= 29 deg)

In all the three configuration studied, the flow is observed to migrate from the Pressure Side (PS) to Suction Side (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 Figure 10. 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.

a. Secondary Flows   **b. Static Pressure Contours, Pa**

Figure 10: Secondary Flows Calculated at the RCV Exit (6 deg) (α_1= 29 deg) and Static Pressure

The static pressure is maximum at the U-bend inlet and decreases at the return channel vane due to the separation of fluid takes place at the RCV inlet the fluid loses its pressure and again increases at the RCV exit that is at the trailing edge of the RCV and L-bend inlet due free movement of fluid flow at these areas.
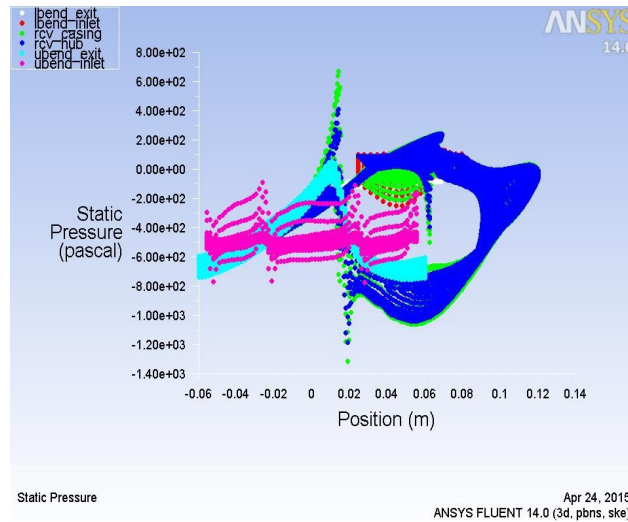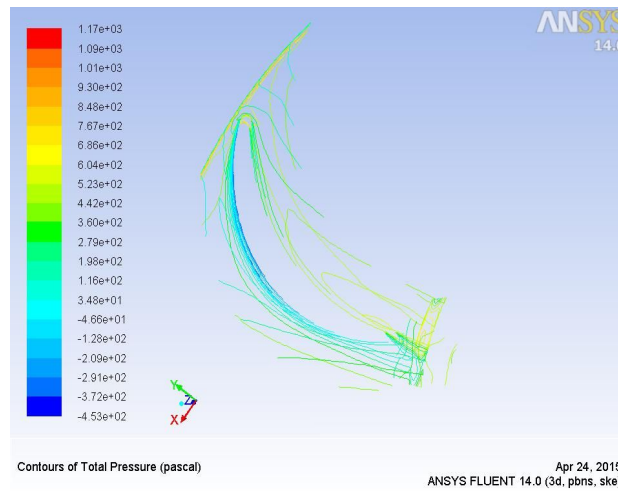

Figure 11: Static Pressure with Position

Figure 12: Total Pressure Contours

Figure 11 shows the XY plot of static pressure shows the effect of static pressure at all the boundaries U-bend inlet & exit, RCV casing & hub, L-bend inlet &exit. Total pressure(Figure 12) is maximum at the RCV inlet and moderate throughout the RCV and increase at the L-bend inlet this due to as the fluid enters the RCV fluid passes over the RCV leading edge and the all over the RCV blade will act as a wall and when the fluid passes over the RCV the swirling motion will be removed as a result there will be reduce in total pressure compared to the U-bend inlet and at the L-bend inlet the fluid will flow freely without any obstacle. The total pressure of fluid is linear at the L-bend inlet, U-bend inlet, U-bend exit and the changes in the total pressure takes place in the return channel passages.
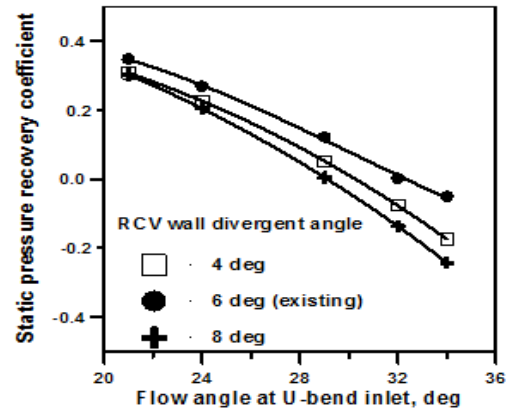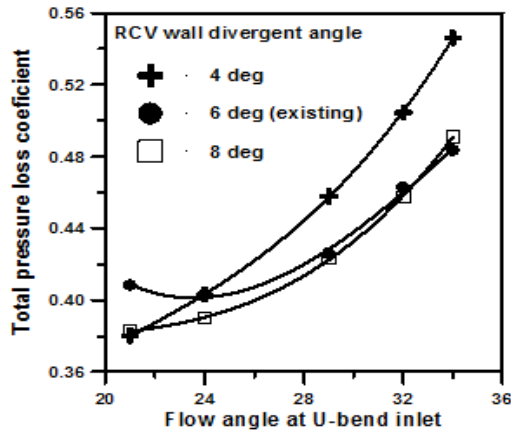
Figure 13 (a): Total Pressure loss Coefficient   **Figure13 (b): Static Pressure Recovery Coefficient**

The variation of total pressure loss coefficient and static pressure recovery coefficient are shown in Figure 13 (a) and (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. The particle path line profile shown in Figure 14, the nature of particle after enters in to the flow area the behavior of fluid seems to be maximum at the U-bend inlet and moderate at the RCV and again increases in the L-bend inlet and then passes to the next stage.
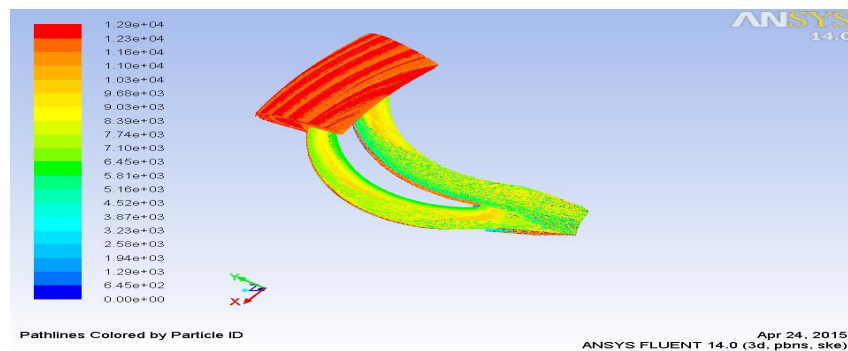

Figure 14: Particle Path Lines

CONCLUSIONS

According to the velocity and pressure profiles of the entire three divergence angle's for 4, 6, 8 deg it is concluded that 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. Dr. B. Balakrishna, S. Venkateswarlu, Dr P. Ravinder Reddy, Flow Analysis of an Atmosphere Reentry Vehicle, International Journal of Engineering Research and Development, Volume 3, Issue 4 (August 2012), PP. 50-55, e-ISSN: 2278-067X, p-ISSN: 2278-800X

2. K. Srinivasa Reddy, G.V. Ramana Murty and K.V. Sharma, Aerodynamic studies in the static components of a centrifugal compressor stage, Journal of Mechanical Engineering and Sciences (JMES) Volume 1, pp. 75-86, December 2011.

3. 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.

4. 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.

5. Lenke, L. J., & Simon, H. (2000). *Aerodynamic analysis of return channels of multistage centrifugal compressors*. ASME PID-Vol. 5, Challenges and Goals in Industrial and Pipeline Compressors.

6. 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.

7. 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.

8. Toyokura, T., Kanemoto, T., & Hatta, M. (1986). Studies on circular cascades for return channels of centrifugal turbomachinery. *Bulletin of JSME*, 29(255), 2885-2892. Veress, A., & Braembussche, R. V. (2004). Inverse design and optimization of a return channel for a multistage centrifugal compressor. *ASME, Journal of Fluids Engineering*, 126(5), 799-806.

9. Aalburg C., Simpson A., Carretero J., Nguyen T., and Michelassi V. "Extension of the Stator Vane Upstream Across the 180 Degree Bend for a Multistage Radial Compressor Stage". Proceedings of ASME Turbo Empo 52009: Power for Land,Sea and Air, 2009

10. Thygesen, R., 2000, ''Optimization of Return Channel Blades for Radial Compressors,'' von Karman Institute PR 2000–21, Sint-Genesius-Rode, Belgium.

11. Akaike S and Toyakura T., 1979. Flow in Inter stage Return Bend of Centrifugal Turbo mach nary. In proceedings of the 6th Conference on Fluid Machinery, Budapest, 1979, Vol. 1,, pp. 11-20.

12. K. Bala Showry, Dr. A.V.S. Raju, Dr. P. Ravinder Reddy, Multi-Dimensional Modeling of Direct Injection Diesel Engine and Effects of Split Injection, International Journal of Scientific and Research Publications, Volume 2, Issue 1, ISSN 2250-3153, pp. 1-8,January 2012

13. B. Balakrishna, P. Sri Seshavani, Dr. P. Ravinder Reddy, Effect of Angle of Flap on Lift and Drag in a Flapping Wing Flight, International Journal of Mechanical Engineering (IJME), ISSN: 2319-2240, Vol.2, Issue 1 Feb. 2013,pp. 29-36, IASET

14. B. Balakrishna, Sushma Indan, P. Ravinder Reddy, Investigation of Supersonic Flow Through Conical Nozzle With Various Angles of Divergence, International Journal of Mechanical Engineering (IJME), ISSN: 2319-2240, Vol.2, Issue 1., Feb. 2013, pp. 09-16, IASET