Numerical Analysis of a Centrifugal Fan for Improved Performance using Splitter Vanes

N. Yagnesh Sharma[1] and K. Vasudeva Karanth[2]

Abstract—The flow field in a centrifugal fan is highly complex with flow reversal taking place on the suction side of impeller and diffuser vanes. Generally performance of the centrifugal fan could be enhanced by judiciously introducing splitter vanes so as to improve the diffusion process. An extensive numerical whole field analysis on the effect of splitter vanes placed in discrete regions of suspected separation points is possible using CFD. This paper examines the effect of splitter vanes corresponding to various geometrical locations on the impeller and diffuser. The analysis shows that the splitter vanes located near the diffuser exit improves the static pressure recovery across the diffusing domain to a larger extent. Also it is found that splitter vanes located at the impeller trailing edge and diffuser leading edge at the mid-span of the circumferential distance between the blades show a marginal improvement in the static pressure recovery across the fan. However, splitters provided near to the suction side of the impeller trailing edge (25% of the circumferential gap between the impeller blades towards the suction side), adversely affect the static pressure recovery of the fan.

Keywords—Splitter vanes, Flow separation, Sliding mesh, Unsteady analysis, Recirculation zone, Jets and wakes.

I. INTRODUCTION

THE idea of using splitter vanes in the blade passage of both impeller and diffuser is not new. Several works mostly experimental have been carried out to assess the suitability of the method. It is found that a numerical approach using a design analysis tool like CFD is of recent origin and the whole field flow analysis of the complex flow in a centrifugal fan has been the state of the art in the domain.

Ogawa and Gopalakrishnan [1, 2], Bhargava and Gopalakrishnan [3], Fabri [4] performed computations on splittered centrifugal rotors based upon potential flow models. Millour [5] examined the same configuration using a 3-D Euler analysis with simplified viscous forces. They observed that the primary effect of the splitters was to decrease the loading on the main blades, as well as to reduce the jet/wake effect at the rotor exit. Fradin [6] performed an extensive set of experiments on the flow fields of two centrifugal rotors: one with splitters, and one without. In both cases the flow

field was transonic. The geometry of the splitters was the same as the main blades. They were circumferentially positioned half way between the main blades. Their study showed that the flow field at the rotor (impeller) exit was more homogenous when the splitters were used. Gui et al. [7] performed a series of incompressible flow regime experiments on two centrifugal fans: one with no splitter and one with variable geometry splitters. They examined the effects of splitter length, circumferential position, and stagger angle. Results indicated that while splitters do reduce the load and velocity gradients on the main blades, they also introduce additional losses that are greatly dependent upon their geometry. It was shown that the pressure coefficient increases when the splitter is placed closer to the suction side of the main blade. Increasing the length of the splitter can raise the pressure coefficient with little or no effect on efficiency. However, they indicated no rule of thumb as to the limit on splitter length, which would certainly have to be taken into account in a transonic flow field, where shocks are present.

Kergourlay et al. [8] studied the influence of adding splitter blades on the performance of a hydraulic centrifugal pump. Velocity and pressure fields were computed using Unsteady Reynolds Averaged Navier- Stokes (URANS) approach at different flow rates. The sliding mesh method was used to model the rotor zone motion in order to simulate the impeller-volute casing interaction. The flow morphology analysis showed that, when adding splitter blades to the impeller, the impeller peripheral velocities and pressures became more homogeneous. Global and local experimental validations were carried out at the rotating speed of 900 rpm, for both the original and the splitter blade impellers. The head was evaluated at various flow rates corresponding to the best efficiency point (BEP). The experimental results matched the numerical predictions, so that the effect of adding splitter blades on the pump was acknowledged. Adding splitters had a positive effect on the pressure fluctuations which decreased at the casing duct. Teipel and Wiedermann [9] dealt with theoretical investigations of the flow field in radial diffusers for a high pressure ratio centrifugal compressor as it is used in compact gas turbine units. The diffuser was equipped with 19 blades. In addition each of those diffuser channels is divided into two sub-passages by a splitter vane whose leading edge was located nearby the throat of the main diffuser channel. For the splitter vanes it was assumed that they were infinitely thin and were mounted along the center line of each diffuser channel. It was the authors' purpose to show the influence of the splitter vanes on essential details of the flow pattern as well as on the global characteristics of the diffuser. These

[1] Corresponding Author: N. Yagnesh Sharma is with Manipal Institute of Technology, Manipal University, Manipal, India, as Professor and Head of the Department of Mechanical & Mfg Engineering. Ph No.:+91 0820 2571061 Extn: 25461, Fax: .:+91 0820 2571071, email I.D: nysharma@hotmail.com

[2] K. Vasudeva Karanth is with Manipal Institute of Technology, Manipal University, Manipal, India, as Additional Professor in the Department of Mechanical & Mfg Engineering.

World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:3, No:12, 2009

investigations are based on a time marching scheme for calculating inviscid transonic flowfields.

Oana et al. [10] in their work focused on the fraction of mass flow in the two splitter channels. Splitters were typically located at mid-pitch between the main blades. Maintaining this circumferential position, the splitter incidence angle was adjusted such that there was even mass flow rate between the two channels. This increased the overall efficiency of the impeller at a given pressure ratio.

McGlumphy [11] carried out 2D and 3D simulation to analyze the aerodynamic feasibility of using a tandem rotor in the rear stages of a core compressor. The results were constrained to shock-free, fully turbulent flow. The 3-D results were subjected to an additional constraint: thick endwall boundary layers at the inlet. A high hub-to-tip ratio 3-D blade geometry was developed based upon the best-case tandem airfoil configuration from the 2-D study. The 3-D tandem rotor was simulated in isolation in order to scrutinize the fluid mechanisms of the rotor, which had not previously been well documented. A geometrically similar single blade rotor was also simulated under the same conditions for a baseline comparison. The tandem rotor was found to outperform its single blade counterpart by attaining a higher work coefficient, polytrophic efficiency and numerical stall margin. An examination of the tandem rotor fluid mechanics revealed that the forward blade acts in a similar manner to a conventional rotor. The aft blade is strongly dependent upon the flow as it receives from the forward blade, and tends to be more three-dimensional and non-uniform than the forward blade.

According to Fatsis et al. [12], Sorokes et al [13], Hillewaert and Van den Braembussche [14], a jet-wake (or primary and secondary) flow pattern exists at the exit of the impeller. The wake (secondary) flow position is at the suction surface or at the shroud depending on the flow rate and the impeller geometry. The flow field entering the diffuser is unsteady and distorted, and it has a significant amount of kinetic energy to transfer to the static pressure. The pressure non-uniformity caused by the volute at the off-design condition further influences the flow fields in the diffuser.

Shi and Tsukamoto[15] in their study have shown that the Navier-Stokes code with the k-ε model is found to be capable of predicting pressure fluctuations in the diffuser. A part of the research work carried out in the current paper is validated with a paper by Meakhail and Park [16] which explores the study of impeller-diffuser-volute interaction in a centrifugal fan. These authors report measurement data in the region between the impeller and vaned diffuser and have obtained results of numerical flow simulation of the whole machine (impeller, vaned diffuser and volute) of a single stage centrifugal fan.

It can be noted from the above literature survey that a CFD analysis on the effect of splitter blades on the system performance of a centrifugal fan as well as its effect on Impeller-Diffuser interaction has not been properly explored so far. Hence a numerical modeling of the flow domain which includes a portion of the inlet to the Impeller as well as the diffuser with volute casing has been carried out and moving mesh technique [17] has been adopted for unsteady flow simulation of the centrifugal fan in the present work.

II. NUMERICAL MODELING

A. Geometry and Grid Generation

The centrifugal fan stage consists of an inlet region, an impeller, a vaned diffuser, and a volute casing (Fig. 1). The impeller consists of thirteen 2-D backward swept blades with an exit angle of 76^0 relative to the tangential direction. The radial gap between the impeller outlet and diffuser inlet is 15% of the impeller outlet radius. The diffuser ring has also the same number of vanes as that of the impeller. All the blades are of 5 mm thickness.

The specifications of the fan stage are illustrated in Table 1. The technical paper by Meakhail and Park [16] forms the basis for geometric modeling in the present work. Unstructured meshing technique is adopted for establishing sliding mesh configuration as the analysis is unsteady as is required in CFD code used in present analysis. A two dimensional flow computation is carried out about the cross-sectional view taken corresponding to the mid height of the blade. Grid for the volute part of the domain has 163,590 nodes and 162,113 elements. The diffuser has 163,213 nodes and 155,106 elements.

TABLE 1
SPECIFICATIONS OF THE CENTRIFUGAL FAN

Impeller inlet radius, R_1	120 mm	Impeller inlet vane angle	30^0
Impeller outlet radius, R_2	200 mm	Impeller outlet vane angle	76^0
Diffuser inlet radius, R_3	230 mm	Diffuser inlet vane angle	23^0
Diffuser outlet radius, R_4	300 mm	Diffuser outlet vane angle	38^0
Volute Exit flange width	450 mm	Number of impeller vanes	13
Channel height of diffuser	35 mm	Number diffuser vanes	13
Channel height of volute casing	90 mm	Speed of the fan (RPM)	1000

The impeller has 80,971 nodes and 74,143 elements. The inlet part of the domain has 5,536 and 5,190 nodes and elements respectively. The maximum size of the element is limited to elements having an edge length of 2 mm. However to establish grid independency, analysis were carried out with finer meshed models having element edge lengths of 1.5 mm and 1 mm. It was found from comparing the results that the variation in basic variable i.e. the static pressure was less than 1.5%. Hence to save the computational time, elements with edge length of maximum 2 mm size is adopted.

World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
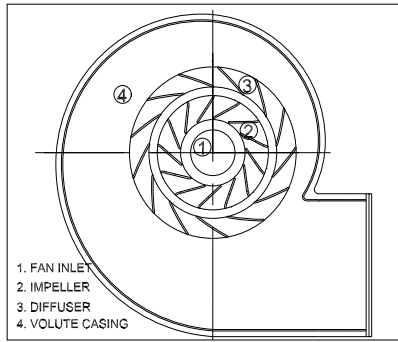Vol:3, No:12, 2009

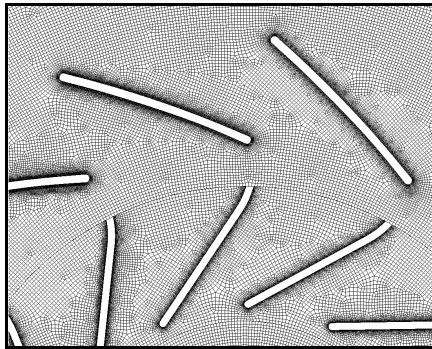Fig. 1 Model of the centrifugal fan used in the analysis.

Fig. 2. A view of the meshed portion between the impeller and diffuser of the centrifugal fan

Fig. 2 shows the meshed domain and it can be observed that a finer mesh is adopted near the blade surface of both impeller and diffuser as well as on the volute casing to capture the boundary layer effects using a suitable sizing algorithm as in CFD code [17].

B. Unsteady Calculations Setup

Two-dimensional, unsteady Reynolds-Averaged Navier-Stokes equations set to polar coordinate system are solved by the CFD code [17]. Due to two dimensional computational domain, the deceleration caused due to the difference in channel height between the diffuser and the volute casing is ignored in the present study. To obtain the flow characteristic curves of the fan, total pressure (gage) is applied at the inlet and static pressure (gage) is applied at the flange exit as the boundary condition. However for comparing the configurations with boundary layer slots, an absolute velocity of 5 m/s which corresponds to the design point mass flow rate of the configuration without slots is imposed at the inlet and a zero gradient outflow condition of all flow properties is applied at the flange exit of the fan, assuming fully developed flow conditions. No slip wall condition is specified for the flow at the wall boundaries of the blades, the vanes, and also the volute casing. The turbulence is simulated using a standard k-ε model [17]. Turbulence intensity of 5% and a turbulent length scale of 0.5 m which is the cube root of the domain volume are adopted in the study. The unsteady formulation used is a second order implicit velocity formulation and the solver is pressure based [17]. The pressure-velocity coupling

is done using SIMPLE algorithm and discretization is carried out using the power law scheme. The power law scheme developed by Patankar [18] is used in the analysis as it is computationally not so intensive and particularly gives good representation of the exponential behavior when peclet number exceeds 2.0. The interface between the inlet region - impeller and impeller - diffuser is set to sliding mesh in which the relative position between the rotor and the stator is updated with each time step. The time step Δt is set to 0.0001 s, corresponding to the advance of the impeller by $\Delta \gamma = 0.610$ per time step for a rated speed of 1000 rpm to establish stability criterion. The maximum number of iterations for each time step is set to 30 in order to reduce all maximum residuals to a value below 10^{-5}. Since the nature of flow is unsteady, it is required to carry out the numerical analysis until the transient fluctuations of the flow field become time periodic as judged by the pressure fluctuations at salient locations in the domain of the flow. In the present analysis this has been achieved after two complete rotations of the impeller. The salient locations chosen are the surfaces corresponding to, inlet to the impeller, impeller exit, diffuser exit, impeller vanes, diffuser vanes and the exit flange of the volute casing. The time and area weighted averages for the pressure and velocity fluctuations at each salient location in the computational domain are recorded corresponding to each rotation of the impeller by time step advancement. The static pressure recovery coefficient ζ_F and the total pressure loss coefficient λ_F for the diffusing domains of the fan are calculated using Eq. (1) and Eq. (2) respectively, based on the area and time weighted averages.

$$\zeta_D = \frac{1}{N} \sum_{j=1}^{j=N} \left(\frac{p_4 - p_2}{p_{t2} - p_2}, t_{initial} + j\Delta t \right) \tag{1}$$

$$\lambda_D = \frac{1}{N} \sum_{j=1}^{j=N} \left(\frac{p_{t2} - p_{t4}}{p_{t2} - p_2}, t_{initial} + j\Delta t \right) \tag{2}$$

Where generally $p = \dfrac{1}{N} \sum_{j=1}^{j=N} p(area\ node, j)$

C. Validation of the Model

The numerical model for the whole field flow calculations is validated by calibrating the results of the current numerical work with the experimental work carried out by Meakhail and Park [16]. The graph shown in Fig. 3 captures the validation results for the current work with the work cited above. The validation curve is a head coefficient (ψ) versus flow coefficient (ϕ) curve which shows a decrease in the head coefficient as the flow coefficient increases as is required for a backward swept impeller blade. The terms (ϕ) and (ψ) are calculated using Eq. (3) and Eq. (4) respectively.

$$\phi = \left(\frac{Q}{\pi R_2^2 U_2} \right) \tag{3}$$

World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:3, No:12, 2009

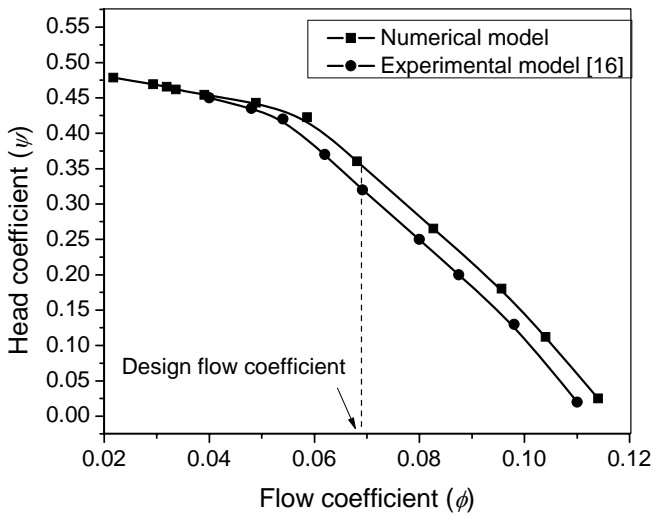$$\psi = \left(\frac{p_{exit} - p_1}{\rho U_2^2} \right) \qquad (4)$$

Fig. 3 Validation characteristic curve of Head coefficient vs. Flow coefficient.

The validation shows a reasonable agreement between the present numerical model and the experimental model of Meakhail and Park [16]. The reason for the computed higher head coefficient with respect to the experimental one as obtained by Meakhail and Park [16], is attributable to the 2-D numerical simulation not fully conforming with the 3-D (real flow) experimental data.

D. Geometric Modeling for Configuration with Splitter Vanes

A splitter vane is a flow re-aligning device and in the present work, is chosen to be made of an aerofoil which is 25% of the impeller and diffuser vane in its radial height and is cambered towards the trailing edge of the impeller for configurations S1 and S2, towards the leading edge of the diffuser for configuration S3 and towards the trailing edge of the diffuser for the configuration S4. Figure 4(a) shows the configuration without splitter vanes. Figure 4 (b) shows configuration S1 with splitters provided on the trailing edge of the impeller at the mid circumferential position between two impeller blades. Figure 4 (c) shows configuration S2 with splitter vanes provided about 25% of the circumferential distance between two impeller blades towards the suction side. Figure 4 (d) shows configuration S3 with splitter vanes provided between two diffuser vanes towards the leading edge in the middle of the flow channel. Figure 4 (e) shows configuration S4 with splitter vanes provided between two diffuser vanes towards the trailing edge in the middle of the flow path.
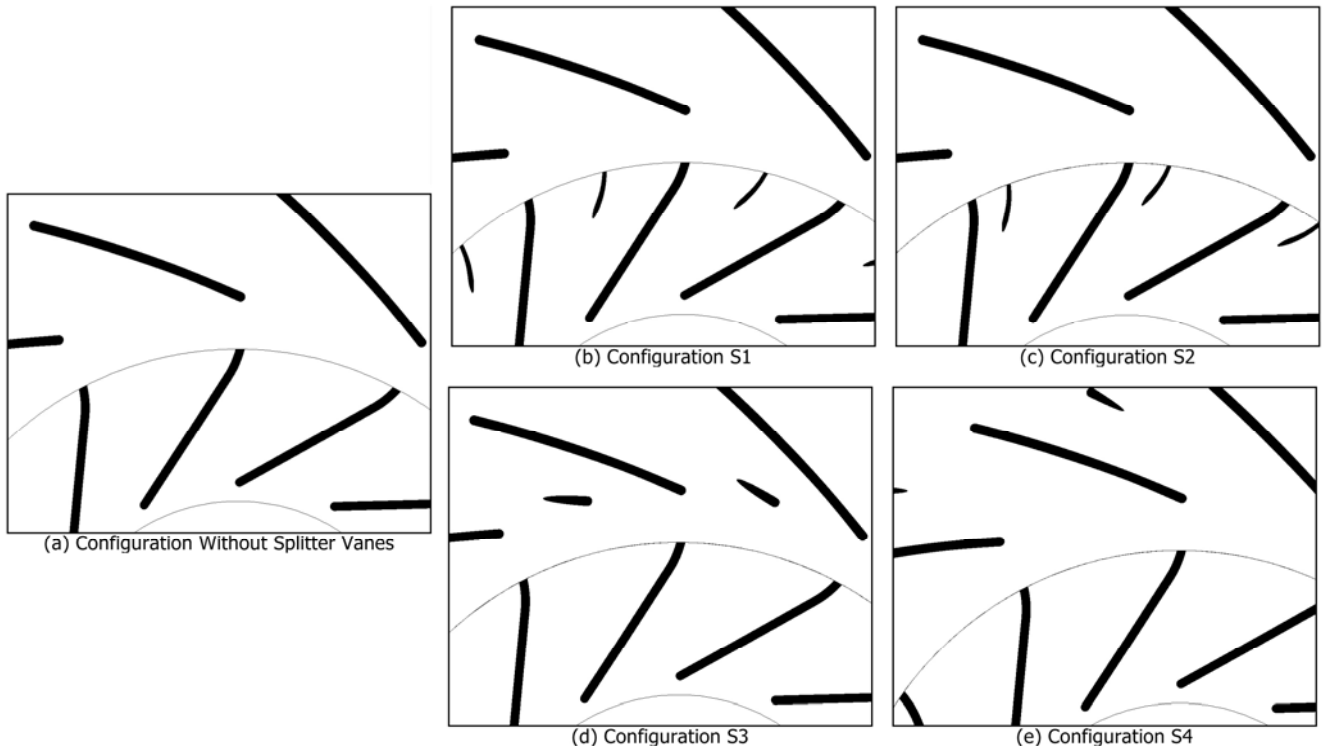

Fig. 4 Geometric Configuration for Splitter vanes provided on the impeller and the diffuser

World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:3, No:12, 2009

III. RESULTS AND DISCUSSION

Fig. 5 shows static pressure recovery coefficient with respect to different geometric types of splitter vanes used in the present analysis.
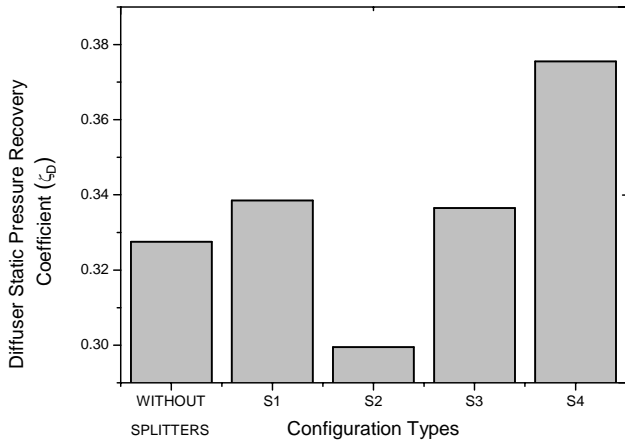
Fig. 5 Static pressure recovery coefficient at the diffuser exit for the various configurations
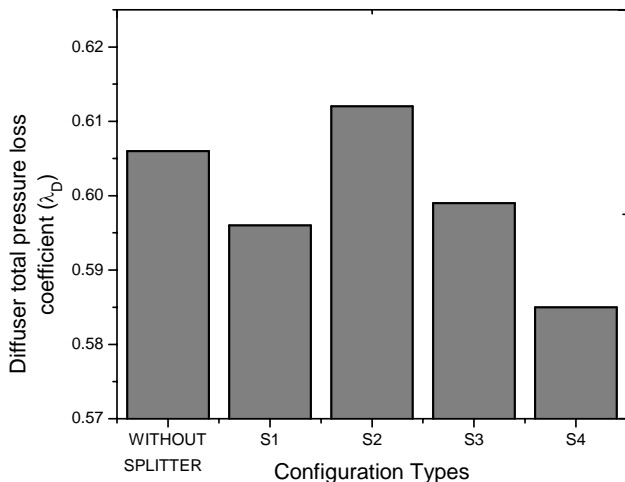
Fig. 6 Total pressure loss coefficient at the diffuser exit for the various configurations

It is clearly observed that the configuration corresponding to type S4 represents the best performance as regards to the diffuser static pressure recovery coefficient. The configuration of type S1 and S3 show reasonable improvement of the fan performance for diffuser static pressure recovery when compared to configuration without slots. However configuration S2 completely annihilates the static pressure recovery. The above observed facts are corroborated by the respective diffuser total pressure loss coefficients as given in Fig. 6.

The physical reasoning for the above observed phenomena could be deduced by carefully analyzing the instantaneous streamline plots obtained for each of the configurations mentioned above. This instantaneous streamline plots are frozen corresponding to a steady time periodic fluctuations at the end of the 4th rotation of the impeller.
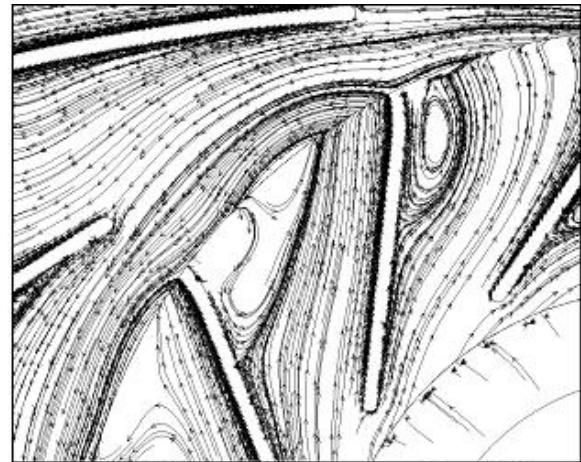
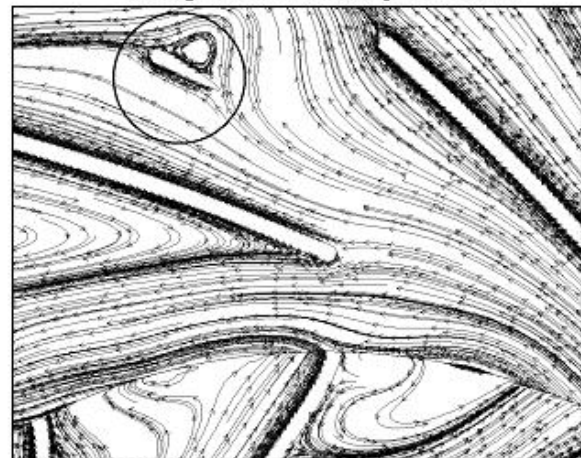Fig. 7 (a) Instantaneous streamline plot for configuration without splitter vanes

Fig. 7 (b) Instantaneous streamline plot for configuration S1
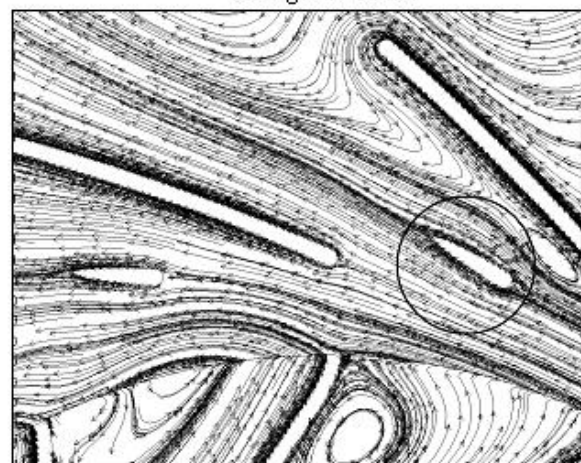
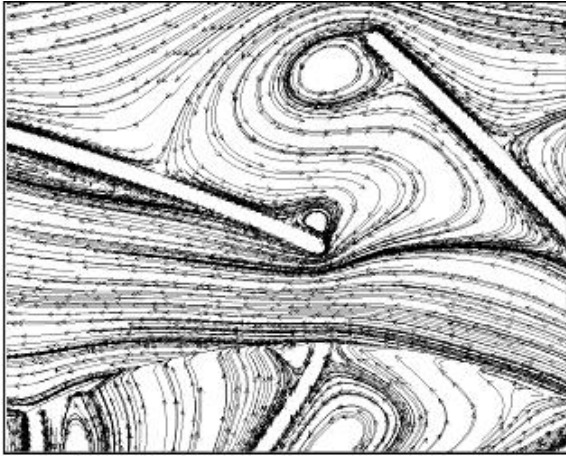Fig 7 (c) Instantaneous streamline plot for configuration S2

World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:3, No:12, 2009

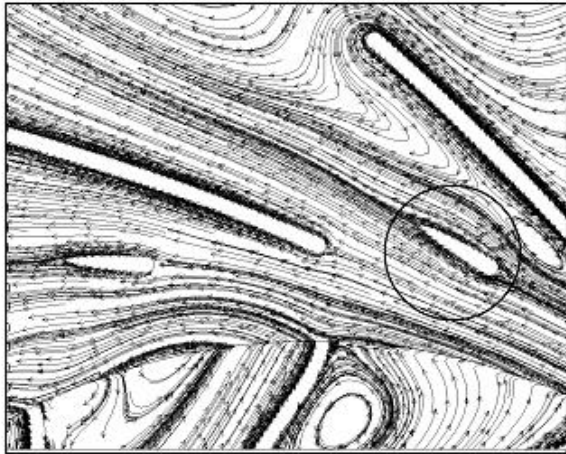Fig. 8 (a) Instantaneous streamline plot for configuration without splitter vanes

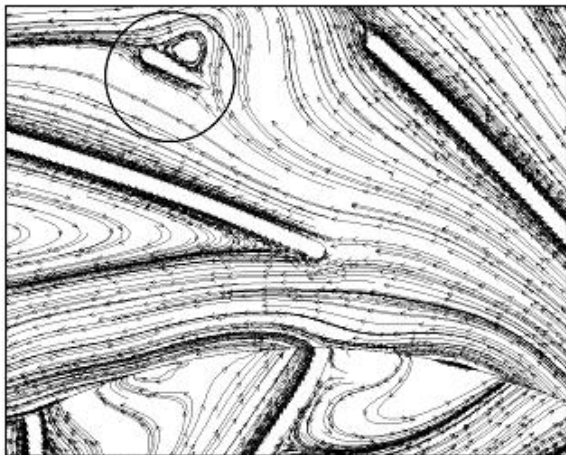Fig. 8 (b) Instantaneous streamline plot for configuration S3

Fig 8. (c) Instantaneous streamline plot for configuration S4

Referring to Fig. 7 (a) and Fig. 7 (b) it is clearly seen that a splitter vane at the mid section of the impeller exit facilitates in providing a good through flow near to the pressure side of the impeller, but contrastingly produces a larger recirculation

zone as well as a flow instability leading to the detachment of the boundary layer on the suction side of the flow path. The overall effect of the above observed phenomenon is to marginally improve the static pressure recovery. Referring to Fig. 7 (c), it is discernable that splitter placed close to the suction side of the impeller at its trailing edge, produces a narrow jet flow, whereas on the other side, a large stalling of the flow occurs and contributes to the annihilation of static pressure recovery.

Figures 8(a) 8(b) show the streamline plots for configuration without splitters and the configuration corresponding to S3. It is quite evident from the instantaneous streamline plot for S3 that the splitter has a positive effect of streamlining the flow in the diffuser channel and thereby decimating the formation of the rotating stall as seen in figure 8(a). This could be the reason for the splitter to show a better static pressure recovery (Fig. 5).

It is clear, by comparing configurations as seen in Fig. 8 (a) and 8 (c) that a splitter vane kept at the mid position of the diffuser exit greatly alleviates the high intensity trailing edge vortex found at the diffuser exit which is leading to rotating stall of the flow in the diffuser as seen in Fig. 8 (a), i.e. without splitter vanes. This has a mollifying effect on the overall through flow from the inlet of the impeller up to the exit of the diffuser. This could be the main reason for the maximum static pressure recovery coefficient observed for configuration with splitter vanes located at the exit of the diffuser.

IV. CONCLUSION

In general, splitter vanes provided on impeller and diffuser at judiciously chosen locations tend to improve the performance of the centrifugal fan, in terms of higher static pressure recovery coefficients and reduced total pressure loss coefficients. The above numerical analysis has also established this aspect and more specifically is able to reveal the following inferences.

1. A splitter vane provided at the diffuser trailing edge at the circumferential mid-span between two diffuser vanes (configuration S4) provides relatively large static pressure recovery of the fan.

2. A splitter vane provided at the diffuser leading edge in between two diffusers at the circumferential mid-span (configuration S3) marginally improves the static pressure recovery of the fan.

3. A splitter vane provided at the impeller trailing edge in between two impellers at the circumferential mid-span (configuration S1) helps to reasonably improve the static pressure recovery of the fan.

4. A splitter vane provided near to the suction side of the impeller at a circumferential distance of 25% between the impeller trailing edge (configuration S2) adversely affects the fan performance in terms of static pressure recovery.

V. ACKNOWLEDGMENT

VI. NOMENCLATURE

j : general parameter

N : general parameter

t : time step size in s

p : static pressure (Pa)

p_t : total pressure (Pa)

U_2 : tangential velocity at impeller exit (m/s)

Q : volume flow rate considering unit channel height (m^3/s)

ρ : air density (kg/m^3)

Φ : flow coefficient

Ψ : head coefficient

γ : the angle of advance of a given impeller blade to its next adjacent blade position.(Deg.),

ζ_D : static pressure recovery coefficient of the diffuser

λ_D : total pressure loss coefficient across the diffuser

VII. SUBSCRIPTS

1 : impeller inlet *2*: impeller exit *3*: diffuser vane inlet *4*: diffuser vane exit

exit: flange exit

initial : initial value

REFERENCES

[1] Ogawa, T. & Gopalakrishnan, G., "Flow Calculation in a Centrifugal Impeller by the Method of Singularities and its Experimental Verification," Proc. Of the Seventh National Conference on Fluid Power and Fluid Mechanics, 1977, Baroda, India.

[2] Ogawa, T. & Gopalakrishnan, G., "Use of Splitter Vanes in Centrifugal Compressor Impellers," Proc. Of the Eighth Canadian Congress of Applied Mechanics, 1981, Moncton, Canada

[3] Bhargava, R.K. & Gopalakrishnan, G., 1978, "Optimising Splitter Vane Locations Using the Method of Singularities," Proc. Of the First International Conference on Centrifugal Compressor Technology, 1978, I. I. T. Madras, India

[4] Fabri, J., "Flow Distribution in a Radial Impeller with Splitter Vanes," Proc. Of the First International Conference on Centrifugal Compressor Technology, 1978, I.I.T. Madras, India

[5] Millour, V., "3D Flow Computations in a Centrifugal Compressor with Splitter Blade Including Viscous Effect Simulation," 16th Congress, International Council of Aeronautical Societies, Volume 1, 1988, pp. 842 – 847

[6] Fradin, C., "Investigation of the Three Dimensional Flow near the Exit of Two Backswept Transonic Centrifugal Impellers," Proc. of the Eighth International Symposium in Air Breathing Engines, 1987, pp. 149 -155

[7] Gui, L., Gu, C., Chang, H., "Influences of Splitter Blades on the Centrifugal Fan Performances," 1989, ASME paper 89-GT-33

[8] Kergourlay, G., Younsi, M., Bakir, F. and Rey, R., Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison, International Journal of Rotating Machinery, Volume 2007, Article ID 85024, 13 pages.

[9] Teipel I. and Wiedermann, A. "Computation of Flowfields in Centrifugal Compressor Diffusers with Splitter Vanes", The International Gas Turbine Congress, 1987, (2) pp. II-311- II-317.

[10] Oana, M., Kawamoto, O., Ohtani, H., Yamamoto, Y., "Approach to High Performance Transonic Centrifugal Compressor," AIAA paper 2002-3536

[11] J. McGlumphy "Numerical Investigation of Subsonic Axial-Flow Tandem Airfoils for a Core Compressor Rotor", PhD Dissertation, 2008, Mechanical Engineering Department Virginia Tech.

[12] Fatsis, A., Pierret, S. and Van den Braembussche, R., "Three-dimensional unsteady flow and forces in centrifugal impellers with circumferential distortion of the outlet static pressure", Journal of Turbomachinery, 1997, 119, pp 94–102.

[13] Sorokes, J. M., Borer, J.C. and Koch, J.M., 1998, "Investigation of the circumferential static pressure non-uniformity caused by centrifugal compressor discharge volute", 98 GT 326, International Gas Turbine & Aeroengine Congress & Exhibition, Stockholm, 1998, Sweden.

[14] Hillewaert, K. and Van den Braembussche, R.A., "Numerical simulation of impeller-volute interaction in centrifugal compressors", Journal of Turbomachinery, 121, 1999, pp 603–608.

[15] Shi, F. and Tsukamoto, H., "Numerical Study of Pressure Fluctuations Caused by Impeller-Diffuser Interaction in a Diffuser Pump Stage", Transactions of the ASME, 9, Vol. 123, 2001, pp. 466 - 474.

[16] Meakhail, T. and Park, S.O., "A Study of Impeller-Diffuser-Volute Interaction in a Centrifugal Fan", ASME Journal of Turbomachinery, 127, 2005, pp. 84 – 90.

[17] Fluent 6.3, Fluent Inc., 2006.

[18] Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Tailor and Francis, (1980), pp. 90-92.