Impeller Treatment for a Centrifugal Fan using Splitter Vanes – A CFD Approach

Madhwesh N.[1], K. Vasudeva Karanth[2] and N. Yagnesh Sharma[3], *Member*, *IAENG*

***Abstract*— In a centrifugal impeller the flow field is highly complex with flow reversal taking place on the suction side of impeller. Generally the performance of a centrifugal fan could be enhanced by introducing splitter vanes at judiciously selected locations. An extensive numerical whole field analysis on the effect of splitter vanes placed in discreet 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. It is found that splitter vanes located at the impeller leading edge improves the static pressure recovery of the fan. However, splitters provided near to the trailing edge of the impeller adversely affect the static pressure recovery of the fan.**

Index Terms: **splitter vanes, unsteady analysis, sliding mesh, recirculation zone, vortex, flow separation.**

I. INTRODUCTION

SEVERAL studies on the introduction of splitter vanes in the impeller passage have been conducted earlier. 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 [12], Bhargava and Gopalakrishnan [1], Fabri [3] performed computations on splittered centrifugal rotors based upon potential flow models. 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 and was circumferentially positioned half way between the main blades. Their study showed that the flow field at the rotor (impeller) exit became more homogenous when the splitters were used. Millour [10] examined the same configuration using a 3-D 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. Gui et al [2] 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 also present.

Teipel and Wiedermann [17] 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 was divided into two sub-passages by a splitter vane whose leading edge was located near 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 investigations were based on a time marching scheme for calculating inviscid transonic flow fields. Oana et al [11] 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 equal mass flow rate between the two channels. This increased the overall efficiency of the impeller at a given pressure ratio.

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

The first author wishes to gratefully acknowledge the financial support extended by the Manipal University, Manipal, India for sponsoring to this conference. The computational facilities were extended by Department of Mechanical and Manufacturing Engineering, Manipal Institute of Technology which is thankfully acknowledged.

[1] Assistant Professor, Dept of Mechanical and Manufacturing Engg, Manipal Institute of Technology, Manipal University, Manipal, Karnataka State, India. madhwesh_n@rediffmail.com

[2] Associate Professor, Dept of Mechanical and Manufacturing Engg, Manipal Institute of Technology, Manipal University, Manipal, Karnataka State, India. kvkaranth@gmail.com

[3]Corresponding Author, Professor, Dept of Mechanical and Manufacturing Engg, Manipal Institute of Technology, Manipal University, Manipal, Karnataka state, India. nysharma@hotmail.com, Ph: +91-9480485082.

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.

McGlumphy [9] 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 end wall 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 [4], Sorokes et al [16], Hillewaert and Van den Braembussche [7], 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.

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 the primary focus. Hence a numerical modelling of the flow domain which includes a portion of the inlet to the Impeller with volute casing has been carried out and moving mesh technique has been adopted for unsteady flow simulation of the centrifugal fan in the present work, as given in Fluent [5].

II. NOMENCLATURE

J : General Parameter
N : General parameter
t : Time in s
Δt : Time Step
p : Static Pressure (Pa)
p_t : Total Pressure (Pa)
γ : The angle of advance of a given impeller blade to its next adjacent blade position.(Deg.),
α :Static Pressure Recovery Coefficient
β :Total Pressure Loss Coefficient

Subscripts

1: Impeller inlet; 2: Impeller exit; 3: Volute exit.
Suffix: 1 – inlet, 2 – outlet.

III. NUMERICAL MODELING

A. *Geometric Model and Mesh Generation*

The centrifugal fan stage consists of an inlet region, an impeller, and a volute casing (Fig 1). The impeller consists of twelve 2-D backward swept blades. All the blades are of 3 mm thickness.

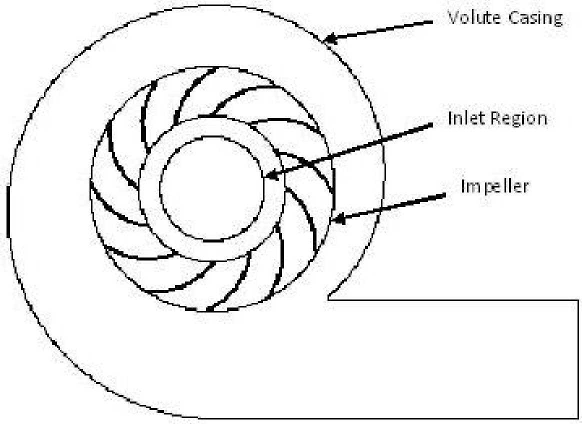

Fig 1. Model of the centrifugal fan used in the analysis

The specifications of the fan stage are illustrated in Table1.

Table 1 Specifications of the Centrifugal Fan

Impeller inlet radius, R_1	112.5 mm	Impeller inlet vane angle	30°
Impeller outlet radius, R_2	187.5 mm	Impeller outlet vane angle	40°
Volute Exit flange width (W)	100 mm	Number of impeller vanes	12
Channel height of impeller blade	80 mm	Channel height of volute casing	80 mm
Speed of the fan (RPM)	2860	Tongue Radius	3 mm

The geometric data for the modelling of the fan is taken from the experimental test rig. Unstructured meshing technique is adopted for establishing sliding mesh configuration. The CFD code used for the present study is dependent on unsteady condition. 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 75,018 nodes and 74,024 elements.

The impeller has 1,52,400 nodes and 1,44,000 elements. The inlet part of the domain has 6,018 and 5,664 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

ISBN: 978-988-19251-5-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

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 1.8%. Hence to save the computational time, elements with edge length of maximum 2 mm size are adopted.

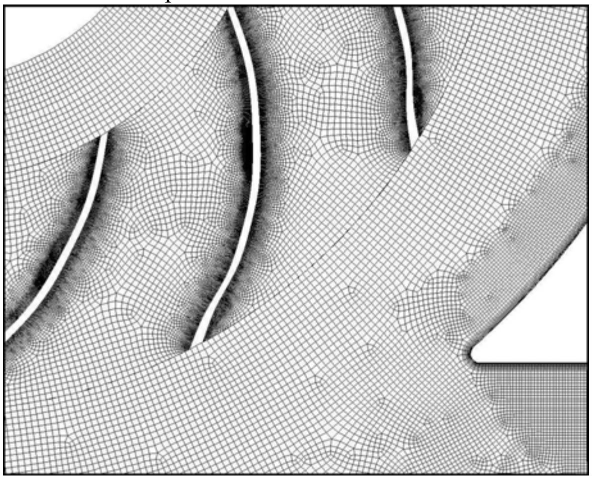

Fig 2. Localized view of the meshed impeller and volute casing of the centrifugal fan.

Figure 2 shows the meshed domain and it can be observed that a finer mesh is adopted near the blade surface of impeller and on the volute casing to capture the boundary layer effects using a suitable sizing algorithm as in CFD code Fluent[5].

B. *Unsteady Analysis Setup*

Two-dimensional, unsteady Reynolds-Averaged Navier-Stokes equations set to polar coordinate system are solved by the CFD code Fluent [5]. However for comparing the configurations with splitter vanes, an absolute velocity of 6.773 m/s which corresponds to the design point mass flow rate of the configuration without splitter vanes 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. The turbulence is simulated using a standard k-ε model available in Fluent [5]. Turbulence intensity of 5% and a turbulent length scale of 0.25 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 Fluent [5]. 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 [14] 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 – volute casing 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 5.8275 e -5 s, corresponding to the advance of the impeller by Δγ = 1° per time step for a rated speed of 2860 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^{-6}. 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 four complete rotations of the impeller. The salient locations chosen are the surfaces corresponding to, inlet to the impeller, impeller exit 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 rise coefficient α and the total pressure loss coefficient β for the diffusing domains of the fan are calculated using Eq. (1) and Eq. (2) respectively, based on the area and time weighted averages.

$$\alpha = \frac{1}{N}\sum_{j=1}^{j=N}\left(\frac{p_3-p_2}{p_{t2}-p_2}, t_{initial}+j\Delta t\right) \quad (1)$$

$$\beta = \frac{1}{N}\sum_{j=1}^{j=N}\left(\frac{p_{t2}-p_{t3}}{p_{t2}-p_2}, t_{initial}+j\Delta t\right) \quad (2)$$

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

C. *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 40% of the impeller in its radial height and is cambered towards the leading edge of the impeller for configurations A1, A2 and A3. The splitter vane is cambered towards the trailing edge of the impeller for configurations B1, B2 and B3.

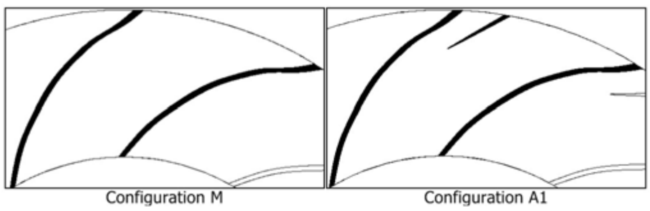

Fig 3. Configuration without splitters (M) and splitters at the trailing edge of the impeller suction side (A1)

Figure 3 shows the configuration M without splitter vanes. It also shows configuration A1 with splitter vanes provided about 25% of the circumferential distance between two impeller blades towards the suction side.

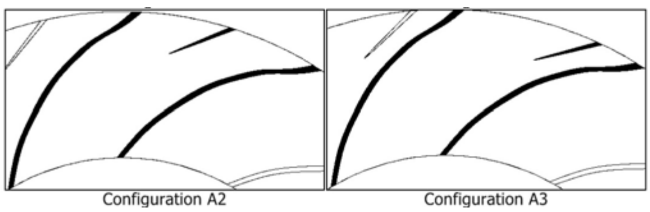

Fig 4. Configuration with splitters at the circumferential mid span (A2) and splitters at the trailing edge of the impeller pressure side (A3)

Figure 4 shows configuration A2 with splitter vane provided on the trailing edge of the impeller at the mid circumferential position between two impeller blades. It also shows configuration A3 with splitter vanes provided about 25% of the circumferential distance between two impeller blades towards the pressure side.

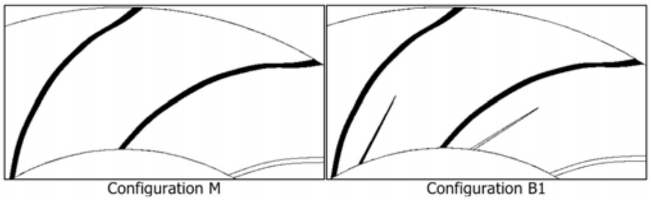

Fig 5. Configuration without splitters (M) and splitters at the leading edge of the impeller suction side (B1)

Figure 5 shows configuration B1 with splitter vanes provided between two impeller vanes towards the leading edge at about 25% of the flow channel at the suction side of the impeller vane.

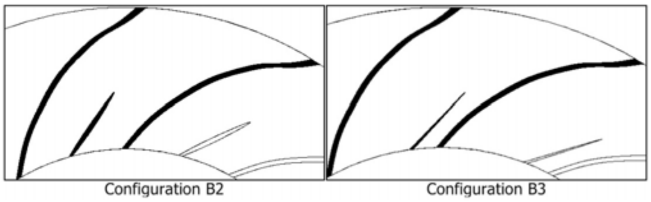

Fig 6. Configuration with splitters at the circumferential mid span (B2) and splitters at the trailing edge of the impeller pressure side (B3)

Figure 6 shows configuration B2 with splitter vanes provided between two impeller vanes towards the leading edge in the middle of the flow channel. It also shows configuration B3 with splitter vanes provided between two impeller vanes towards the leading edge at about 25% of the flow channel towards the pressure side of the impeller vane.

IV. RESULTS AND DISCUSSION

Figure 7 and 8 show static pressure rise coefficient with respect to different geometric types of splitter vanes used in the present analysis. It is clearly observed that the configuration types B1, B2 and B3 represent the best performance as regards to the fan static pressure rise coefficient. The configuration of type A3 shows marginal decrement of the performance for fan static pressure recovery when compared to configuration without splitter vanes. However configurations A1 and A2 completely annihilate the static pressure recovery. The above observed facts are corroborated by the respective fan total pressure loss coefficients as given in Figures 9 and 10.

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 5th rotation of the impeller.

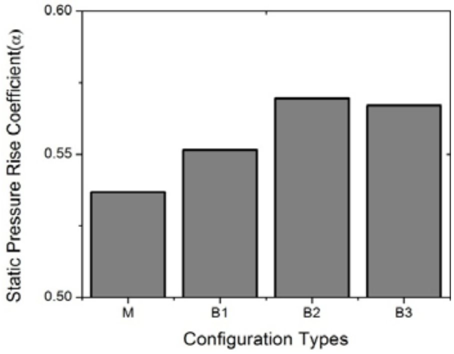

Fig 7. Static pressure rise coefficient across the fan for the configurations without splitters (M) and configurations B1, B2 and B3

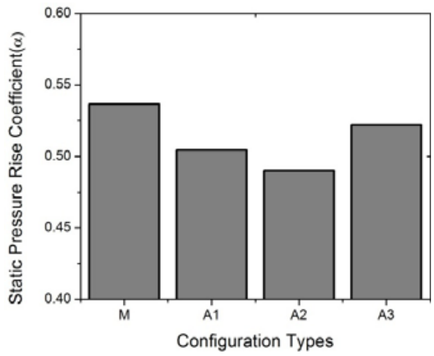

Fig 8. Static pressure rise coefficient across the fan for the configurations without splitters (M) and configurations A1, A2 and A3

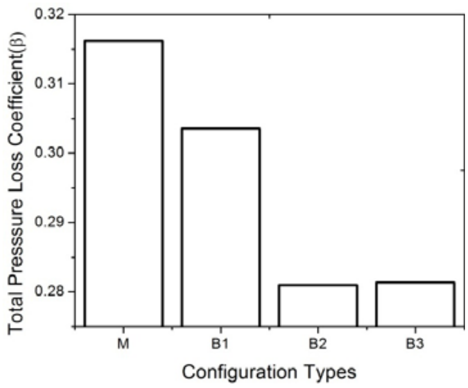

Fig 9. Total Pressure Loss Coefficient across the fan for the configurations without splitters (M) and configurations B1, B2 and B3

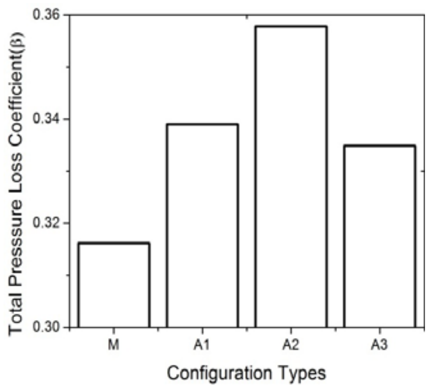

Fig 10. Total Pressure Loss Coefficient across the fan for the configurations without splitters (M) and configurations A1, A2 and A3

Referring to Figure 11, it is observed that a splitter vane provided at the middle of the flow path between two impeller blades at its trailing edge (configuration A2) and

ISBN: 978-988-19251-5-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

the splitter vane provided near the suction side of the impeller at its trailing edge (configuration A1), produce a large recirculation zone between the impeller blade suction side and the splitter vane 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 produce significantly lower static pressure recovery of the fan.

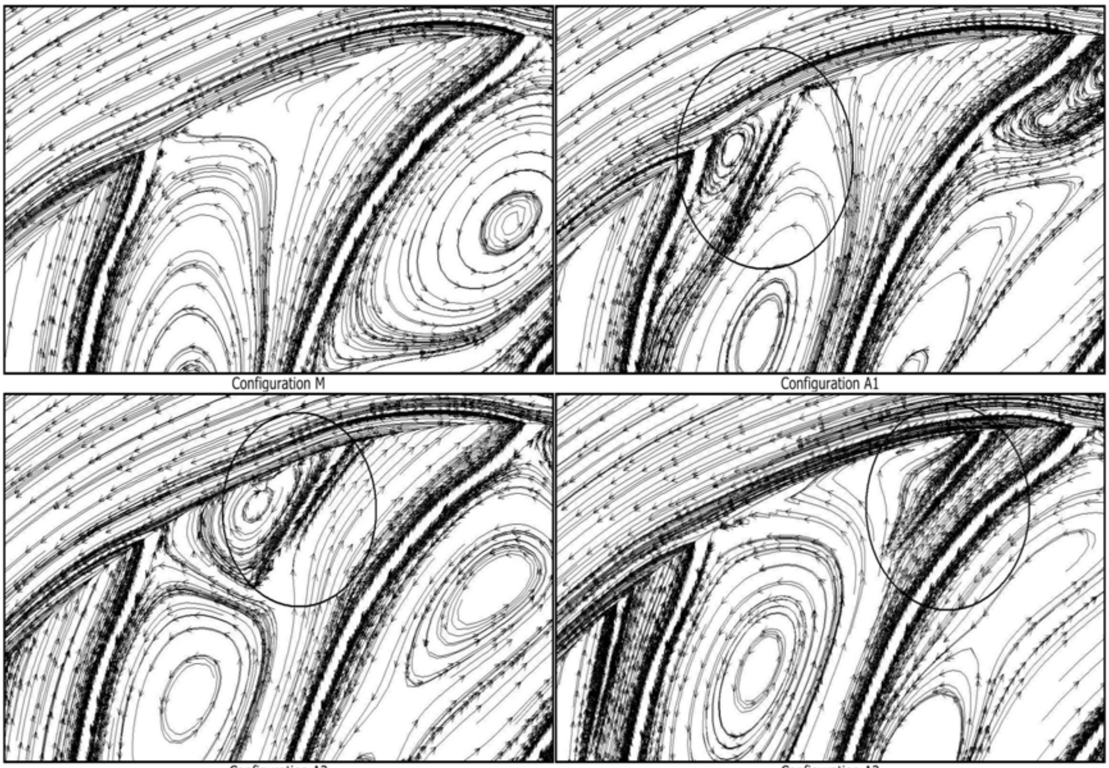

Fig 11. Instantaneous streamline plots for configuration without splitters(M) and configurations A1, A2 and A3

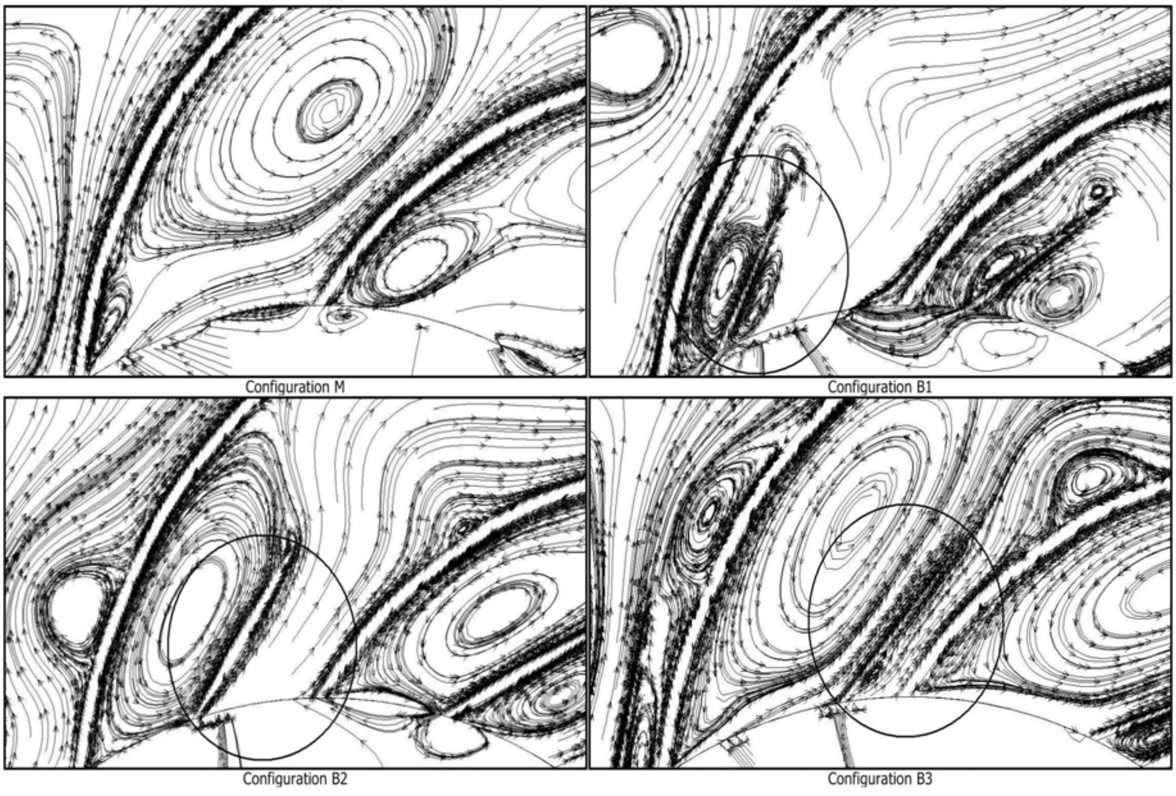

Fig 12. Instantaneous streamline plots for configuration without splitters(M) and configurations B1, B2 and B3

ISBN: 978-988-19251-5-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

It is discernable that splitters placed close to the pressure side of the impeller at its trailing edge (configuration A3) 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. This loss also tends to increase the total pressure loss coefficient significantly.

From figure 12, it is clearly seen that a splitter vane near to the suction side of the impeller inlet (configuration B1) facilitates in providing a good through flow near to the pressure side of the impeller. But contrastingly this produces a smaller recirculation zone or a wake region between the splitter vane and the suction side of the impeller unlike the very large recirculation zone in the configuration without splitters (M). The overall effect of the above observed phenomenon is to significantly improve the static pressure recovery of the fan.

Also it is observed that a splitter vane provided at the middle of the flow path between two impeller blades at its leading edge (configuration B2), neutralizes the large recirculation zone and provides larger through flow in the impeller passage, resulting in a significant improvement in the static pressure recovery of the fan. A similar trend can be observed for the configuration B3 also

V. CONCLUSION

In general, splitter vanes provided on impeller vanes 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.

- Splitter vanes provided at the impeller leading edge at the circumferential mid-span between two impeller vanes (configurations B2) provide relatively large static pressure recovery of the fan.
- Splitter vanes provided at the impeller leading edge near the pressure and suction side and the suction side (configurations B1 and B3) significantly improve the static pressure recovery of the fan.
- Splitter vanes provided at the impeller trailing edge near to the suction side and the circumferential mid-span (configurations A1 and A2) gives poor performance.
- Splitter vane provided at the impeller trailing edge near the pressure side (configuration A3) shows marginal decrement in the static pressure recovery of the fan.

REFERENCES

[1] 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*, I. I. T. Madras, India.

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

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

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

[5] Fluent 6.3, 2006, Fluent Inc., Cavendish Court Lebanon, NH, 03766, USA.

[6] Fradin, C., 1987, 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*, pp. 149 -155.

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

[8] Kergourlay, G., Younsi, M., Bakir, F. and Rey, R., 2007,Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison, *International Journal of Rotating Machinery*, Volume Article ID 85024, pp. 1 - 13.

[9] McGlumphy,J., 2008, *Numerical Investigation of Subsonic Axial-Flow Tandem Airfoils for a Core Compressor Rotor*, Thesis (Ph.D.), Mechanical EngineeringDepartment Virginia Tech. University.

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

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

[12] Ogawa, T. &Gopalakrishnan, G.,1977,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*,Baroda, India.

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

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

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

[16] 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.

[17] Teipel I.and Wiedermann, A. 1987, Computation of Flow fields in Centrifugal Compressor Diffusers with Splitter Vanes, *Proc. of the International Gas Turbine Congress,*vol. 2, pp.II-311- II-317.

ISBN: 978-988-19251-5-2
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)