

# CFD Analysis of a Centrifugal Fan for Performance Enhancement using Converging Boundary Layer Suction Slots

K. Vasudeva Karanth<sup>1</sup> and N. Yagnesh Sharma<sup>2</sup>

**Abstract**—Generally flow behavior in centrifugal fan is observed to be in a state of instability with flow separation zones on suction surface as well as near the front shroud. Overall performance of the diffusion process in a centrifugal fan could be enhanced by judiciously introducing the boundary layer suction slots. With easy accessibility of CFD as an analytical tool, an extensive numerical whole field analysis of the effect of boundary layer suction slots in discrete regions of suspected separation points is possible. This paper attempts to explore the effect of boundary layer suction slots corresponding to various geometrical locations on the impeller with converging configurations for the slots. The analysis shows that the converging suction slots located on the impeller blade about 25% from the trailing edge, significantly improves the static pressure recovery across the fan. Also it is found that Slots provided at a radial distance of about 12% from the leading and trailing edges marginally improve the static pressure recovery across the fan.

**Keywords**—Boundary layer suction converging slot, Flow separation, Sliding mesh, Unsteady analysis, Recirculation zone, Jets and wakes.

## I. INTRODUCTION

SLOTS are made at the point of separation so as to attenuate the boundary layer by which boundary layer buildup can be controlled in the centrifugal impeller. These slots have been designed in the present work in such a way that a jet of fluid effluxes through the converging slot from the pressure to the suction side. This results in moving the separation region closer to the tip of the impeller, thus reducing the slip as well as flow losses. A host of studies have been undertaken, mostly experimentally, to assess the efficacy of these boundary layer suction slots in the impeller or diffuser of various types of Turbomachines. Experimental results indicate improvement in the pressure ratio and efficiency as reported by Boyce [1]. According to Reacock [2], corner separation existing at the junction between a cascade sidewall and a moderately loaded compressor blade in cascade was eliminated by means of a boundary-layer suction

technique. Khan et al. [3] studied the basic fluid phenomena associated with a Blasius boundary layer over a flat plate approaching a two-dimensional suction slot experimentally. Also numerically the complete two-dimensional Navier-Stokes equations using an alternating direction implicit scheme was solved. Results exhibited considerable upstream and downstream influence upon boundary layer integral parameters and wall shear due to the suction. Seal and Smith [4] carried out experimental evaluation of the effects of localized surface suction on a turbulent junction flow using Particle Image Velocimetry (PIV). The results indicated that surface suction can (i) weaken both the instantaneous turbulent vortex and its associated surface interactions in the symmetry plane, (ii) effectively eliminate the presence of the average turbulent necklace vortex in the symmetry plane, and (iii) weaken the average downstream extensions of the vortex. Raghunathan and Cooper [5] carried out experiments in short wide-angle diffusers and the control surface was made of slots. The experiments included slots normal to the surface and a combination of slots normal and inclined to the surface. Passive control with inclined slots produced modest improvement in pressure recovery but significant increases in the stall angle. Wursthorn and Schnerr [6] in their work have carried out the control of cavitation by providing a slot near the leading edge of the impeller blade connecting the suction side from the pressure side of the impeller. They carried out 2-D numerical calculations in a rotating frame of reference for an impeller of a radial pump of low specific speed. From these investigations they concluded that, this method was most effective to reduce leading edge suction peaks in the partial capacity operating range. Hubrich et al. [7] carried out numerical and experimental study aiming at the enhancement of the working range of a transonic compressor via boundary layer suction. Shojaefard et al. [8] have carried out a numerical study of flow control on a subsonic airfoil with suction and injection slots on the suction side of the airfoil. The results showed that the surface suction could significantly increase the lift coefficient and injection decreased the skin friction. Atik et al. [9] in their study have considered a high-speed incompressible flow past a thin airfoil in a uniform stream. It was found by them that substantial delays in separation could be achieved even when the suction is weak, provided that the suction is initiated at an early stage. Suction into small two- or three-dimensional surface slots inside an otherwise planar boundary-layer was examined theoretically through a combined analytical and computational approach by Gajjar and Smith [10]. It was shown that increasing suction

<sup>1</sup> K. Vasudeva Karanth is with Manipal Institute of Technology, Manipal University, Manipal, India, as Additional Professor in the Department of Mechanical & Mfg Engineering.

<sup>2</sup> 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

strength would lead to enhanced nonlinear structures with flow reversals or trailing vortices. Dovgal and Sorokin [11] carried out a wind-tunnel study of the influence of flow suction on laminar boundary-layer separation behind a two-dimensional step on the surface.

It was found that this method of flow control allowed suppressing the formation of large-scale vortices determined by global stability properties of the separation. Song et al. [12] investigated numerically the effect of boundary layer suction via a slot on the suction surfaces of a compound lean compressor cascade having large camber angles with a conventional straight compressor cascade for comparison. The result showed that the total loss for all the cascades reduced significantly by boundary layer suction, and the largest reduction occurred at the highest suction flow rate. Atik and van Dommelen [13] by means of computational technique explored the possibility of autogenous suction, in which the local pressure difference which leads to separation was itself used to create the suction as it leads to delay or avoid separation. One of the important ways of improving turbo machinery compressor performance is to control three-dimensional (3D) separations, which form over the suction surface and end wall corner of the blade passage. Based on the insights gained into the formation of these separations, the paper by Gbadebo et al. [14] illustrates how an appropriately applied boundary layer suction of up to 0.7% of inlet mass flow can control and eliminate typical compressor stator hub corner 3D separation over a range of operating incidence.

According to Fatsis et al. [15], Sorokes et al. [16], Hillewaert and Van den Braembussche [17], 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 [18] in their study have shown that the Navier-Stokes code with the  $k-\epsilon$  model is found to be capable of predicting pressure fluctuations in the diffuser. Sofiane et al. [19] have carried out the numerical unsteady flow analysis in a vaned centrifugal fan.

A part of the research work carried out in the current paper is validated with a paper by Meakhail and Park [20], 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 boundary layer suction slot especially with converging profile on the system performance of a centrifugal fan as well as its effect on Impeller-Diffuser interaction has not been 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 [21] has

been adopted for unsteady flow simulation of the centrifugal fan in this analysis.

## 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  $76^\circ$  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 [20] 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 I  
 SPECIFICATIONS OF THE CENTRIFUGAL FAN

|                                 |        |                            |            |
|---------------------------------|--------|----------------------------|------------|
| Impeller inlet radius, $R_1$    | 120 mm | Impeller inlet vane angle  | $30^\circ$ |
| Impeller outlet radius, $R_2$   | 200 mm | Impeller outlet vane angle | $76^\circ$ |
| Diffuser inlet radius, $R_3$    | 230 mm | Diffuser inlet vane angle  | $23^\circ$ |
| Diffuser outlet radius, $R_4$   | 300 mm | Diffuser outlet vane angle | $38^\circ$ |
| 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.

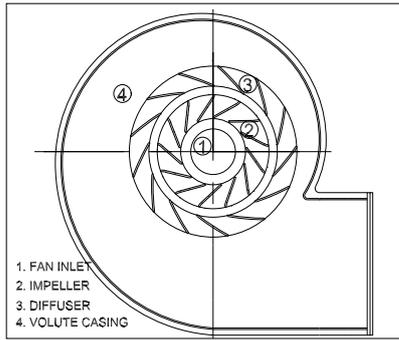


Fig. 1 Model of the centrifugal fan used in the analysis.

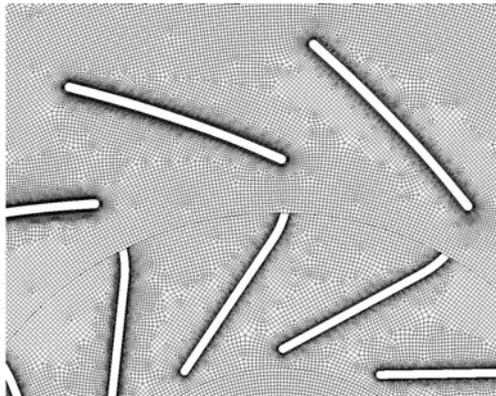


Fig. 2. A view of the meshed portion between the impeller and diffuser of the centrifugal fan

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 [21].

### B. Unsteady Calculations Setup

Two-dimensional, unsteady Reynolds-Averaged Navier-Stokes equations set to polar coordinate system are solved by the CFD code [21]. 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 [21]. 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 [21]. 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 [22] 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 Δγ = 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<sup>-5</sup>. 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<sub>F</sub> and the total pressure loss coefficient λ<sub>F</sub> for the diffusing domains of the fan are calculated using Eq. (1) and Eq. (2) respectively, based on the area and time weighted averages.

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

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

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

### 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<sup>(20)</sup>. 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.

$$\phi = \left( \frac{Q}{\pi R_2^2 U_2} \right) \quad \text{-- (3)} \quad \psi = \left( \frac{P_{exit} - P_1}{\rho U_2^2} \right) \quad \text{(4)}$$

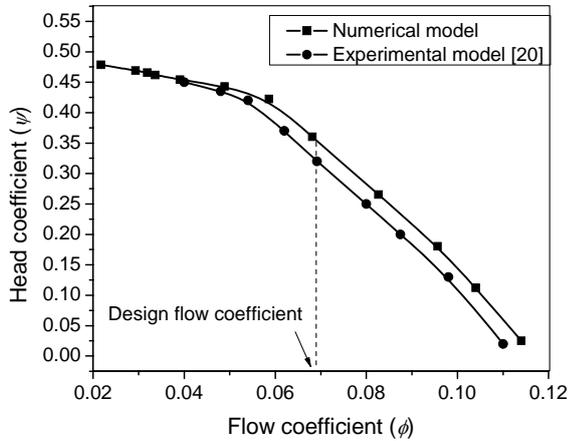


Fig. 3 Validation characteristic curve of Head coefficient vs. Flow coefficient.

The validation shows a reasonable agreement between the present numerical model and the experimental model of Meakhal and Park [20]. The reason for the computed higher head coefficient with respect to the experimental one as obtained by Meakhal and Park [20], 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 Boundary Layer Suction slots

A boundary layer suction slot is a flow re-aligning device and in the present work, is chosen to be made of a 2.5 mm to 1.5 mm converging slot from the pressure side to suction side cut through the impeller blade in the tangential direction so as to connect the pressure side with the suction side.

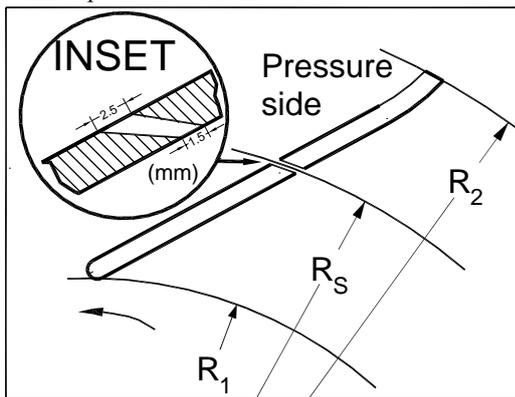


Fig. 4 Geometric Configuration for Boundary layer suction slot provided on the Impeller

Location of the slot is determined judiciously and is specified corresponding to the radial distance from the axis of the fan. Table 2 specifies the location of the slot in terms of

radial distance ratio ( $R_I$ ). Configurations  $M_1$  to  $M_5$  represent the five different geometric configurations used in the study. The Fig.4 shows the geometric details pertaining to the slot location. The radius at which the slot is located from the centre of the fan is obtained by Eq.5

TABLE II RADIAL DISTANCE RATIO FOR CONFIGURATION WITH SLOTS ON THE IMPELLER

| Configuration with slots on the impeller blade |                              |
|--|------------------------------|
| Configuration type                             | Radial distance ratio, $R_I$ |
| M1   | 0.12                         |
| M2   | 0.25                         |
| M3   | 0.50                         |
| M4   | 0.75                         |
| M5   | 0.88                         |

$$R_I = \frac{R_S - R_I}{R_2 - R_I} \quad \text{(5)}$$

### III. RESULTS AND DISCUSSION

Figure 5 shows static pressure recovery coefficient at the diffuser exit and Figure 6, the corresponding total pressure loss coefficient.

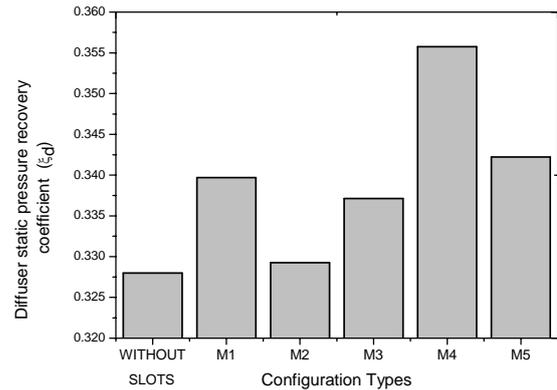


Fig. 5 Static pressure recovery coefficient at the diffuser exit for the various configurations

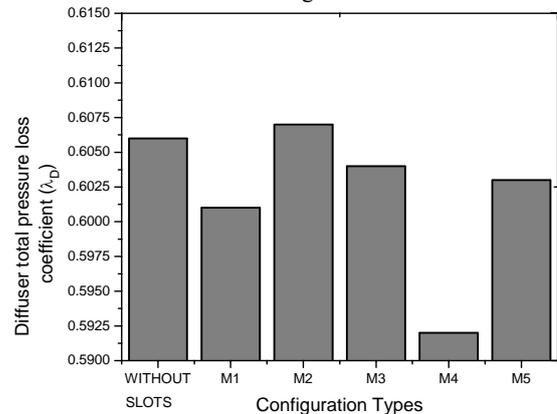
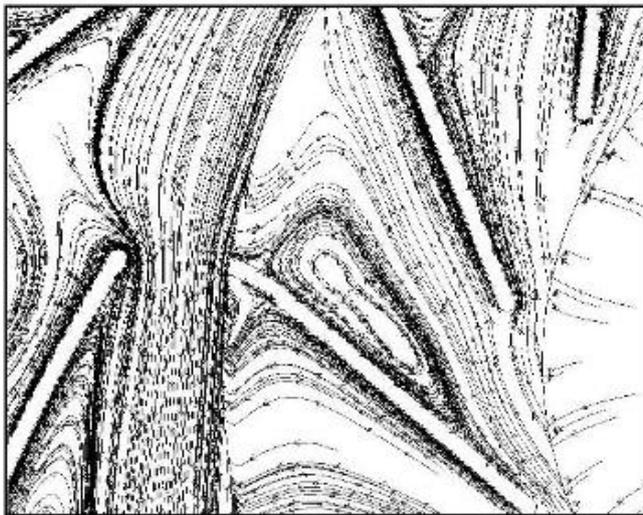
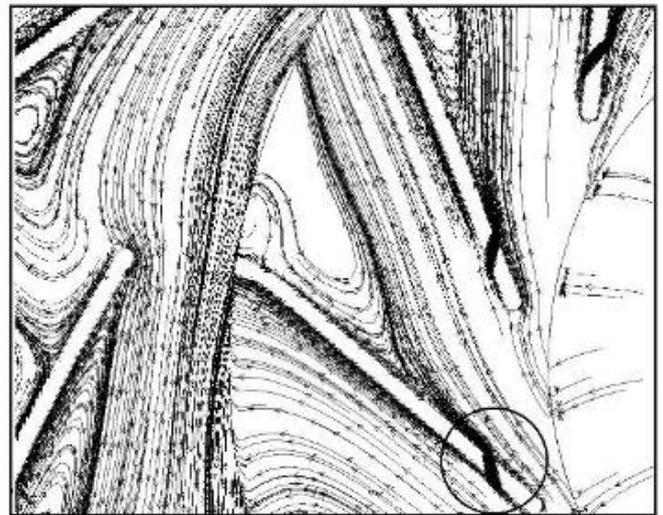


Fig. 6 Total pressure loss coefficient at the diffuser exit for the various configurations



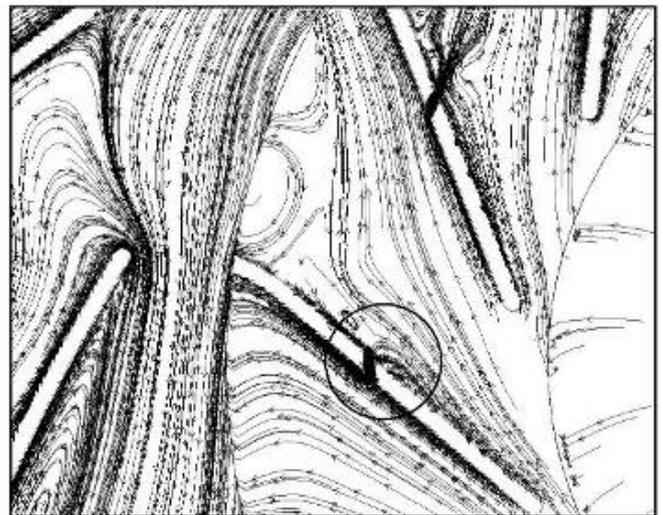
(a) Configuration without slots



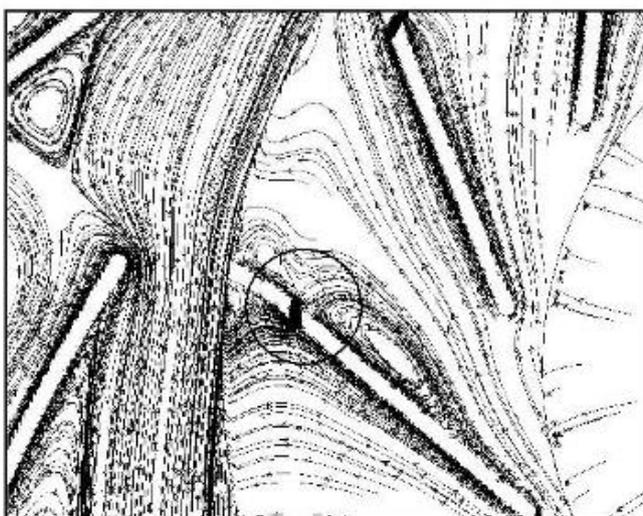
(b) Configuration M1



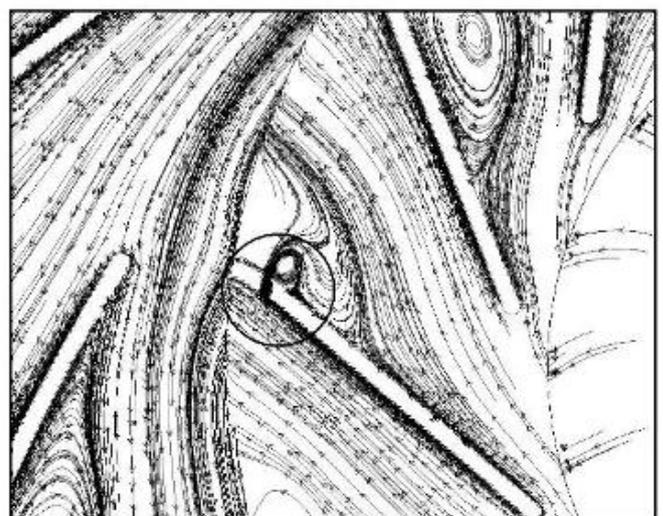
(c) Configuration M2



(d) Configuration M3



(e) Configuration M4



(f) Configuration M5

Fig. 7 Enlarged view of instantaneous streamline plots corresponding to one of the flow passages for various configurations.

It is found from the above figures that the configuration M4 which corresponds to slot position at 25% from the trailing edge yields the best possible static pressure recovery coefficient as well as matching lowest total pressure loss coefficient. Configuration M1 and M5 also shows significantly improved yields of static pressure recovery coefficient while compared to configuration without slot. But configuration M3 shows only marginal improvement while configuration M2 shows practically no improvement compared to configuration without slot. The above inferences are corroborated by Figure 6 which represents the total pressure loss coefficient for respective configurations explained above.

The physical reasons and implications for the above observed phenomenon can be deduced by carefully analyzing the instantaneous streamline plots obtained for the above configurations corresponding to stabilized time periodic fluctuations at the end of the fourth rotation of the impeller. Figure 7 (a) shows the streamline plots for the configuration without slots in which it is clearly discernable that a large high intensity recirculation zone lies on the suction surface almost stalling 50% of through flow and also producing a secondary low intensity vortex at the trailing edge. Hence corresponding to this configuration, the static pressure recovery coefficient is admittedly low as shown in fig 5.

It is clearly seen from Figure 7 (e) corresponding to configuration M4 that the suction slot which is placed about 25% from the trailing edge, not only annihilates the recirculation zone shown in Figure 7 (a) but also evens out the vortex at the trailing edge in sharp contrast to the phenomena observed with respect to Figure 7 (a). This has a highly beneficial effect on the static pressure recovery coefficient as is observed in Figure 5 and 6, which shows highest possible recovery coefficient as well as lowest pressure loss corresponding to this configuration. Though there appears a minor recirculation zone as a cause and effect of the suction slot corresponding to configuration M4, it is observed that this also brings about a positive effect in terms of improved through flow all along the impeller passage.

Figure 7 (b) and 7 (f) show comparatively improved results with respect to configuration without slots. The physical reason for this may be ascribed to the following. In Figure 7 (b), the role of the suction slot seems to be that of driving away the strong recirculation zone that was observed in configuration without slot [Figure 7 (a)] as well as decimation of the low intensity vortex. These two coupled phenomena facilitate better through flow and hence the significant improvement in the static pressure recovery coefficient as shown in Figure 5 for this configuration. In a similar vein, it could be explained for configuration 7 (f) where also the above explained phenomena occurs but with an appearance of recirculation zone behind the suction slot, near the emerging jet.

The configuration M3 which is located midway of the impeller produces in a contrasting way a doublet of vortex near the trailing edge of the impeller due to which static pressure recovery coefficient only marginally improves compared to configuration without slots. But the configuration M2 appears to accentuate the strong recirculation zone

towards the trailing edge of the impeller and thereby effectively stalling the flow. Hence this configuration produces no improvement as regards to static pressure recovery coefficient.

#### IV. CONCLUSION

In general, boundary layer suction slots provided on impeller blades 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 converging boundary layer suction slot provided at 25% radial distance from the trailing edge of the impeller blade (configuration M4) helps to attenuate the formation of recirculation zone resulting in a highly appreciable improvement in the performance for the fan.

2. A converging boundary layer suction slots located at a radial distance of 12.5% from the leading edge and trailing edge of the impeller (configurations M1 and M5) show significant improvement in static pressure recovery of the fan.

3. The converging boundary layer suction slot located at a radial mid span of the impeller blade (configuration M3) tends to marginally improve the static pressure recovery of the fan

4. A boundary layer suction slot at a radial distance of 25% from the leading edge of the impeller does not contribute to the performance of the fan.

#### V. ACKNOWLEDGMENT

The authors wish to acknowledge and thank Tarek Meakhail and Seung O Park [20], for readily giving the centrifugal fan drawing for their numerical modeling. They also wish to thank Manipal Institute of Technology, Manipal University, for providing computational resources for undertaking this study.

#### 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 <sup>3</sup> /s)                       |
| $\rho$ :    | air density (kg/m <sup>3</sup> )   |
| $\Phi$ :    | flow coefficient   |
| $\Psi$ :    | head coefficient   |
| $\gamma$ :  | the angle of advance of a given impeller blade to its next adjacent blade position.(Deg.), |
| $\zeta_D$ : | static pressure recovery coefficient across the diffuser                                   |

$\lambda_D$  : total pressure loss coefficient across the diffuser  
 $R_S$  : radius at which suction slot is located  
 $R_f$  : impeller suction slot radial distance ratio

## VII. SUBSCRIPTS

1 : impeller inlet 2: impeller exit 3: diffuser vane inlet 4:  
diffuser vane exit  
exit: flange exit  
initial : initial value

## REFERENCES

- [1] Boyce, M.P., Gas Turbine Engineering Hand Book, Gulf Professional Publishing, Third Edition, USA, 2006, pp. 238 – 239.
- [2] Reacock, R. E., “Boundary-Layer Suction to Eliminate Corner Separation in Cascades of Aerofoils”, Research report, University Engineering Department, Cambridge, 1965.
- [3] Khan, M.M.S., Cornelius, K.C. and Tassa, Y., “Effect of suction on boundary-layer flow over a 2-D slot”, American Institute of Aeronautics and Astronautics, Aerospace Sciences Meeting, 20th, Orlando, FL, Jan. 11-14, 1982, 11 p
- [4] Seal, C.V. and Smith, C.R., “The control of turbulent end-wall boundary layers using surface suction”, *Experiments in Fluids*, Springer Publications, Volume 27, Number 6, November, 1999, pp. 484-496
- [5] Raghunathan, S. and Cooper, R.K., “Passive Boundary Layer Control with Slots in Short Diffusers”, *ASME Journal of Fluids Engineering*, 3, Vol. 122, 2000, pp. 177-179.
- [6] Wursthorn, S. and Schnerr, G.H., “Numerical investigations of cavitation control in pumps”, *Journal of Applied Mathematics and Mechanics*, vol. 80, 2000, pp. S663-S664
- [7] Hubrich, K., Bolcs, A. and Ott, P., “Boundary Layer Suction via a Slot in a Transonic Compressor: Numerical Parameter Study and First Experiments”, *Proceedings of ASME Turbo Expo 2004 Power for Land, Sea, and Air GT2004-53758*, 2004, Austria
- [8] Shojaefard, M.H., Noorpoor, A.R, Avanesians, A. and Ghaffarpour, M., “Numerical Investigation of Flow Control by Suction and Injection on a Subsonic Airfoil”, *American Journal of Applied Sciences*, Science Publications, 2 (10), 2005, pp. 1474-1480.
- [9] Atik, H., Kim, C.Y., Van-Dommelen, L. and Walker, J.D.A., “Boundary-layer Separation Control on a Thin Airfoil using Local Suction”, *Journal of Fluid Mechanics Cambridge University Press*, 535, 2005, pp. 415-443.
- [10] Gajjar, J.S.B. and Smith, F.T., “Vortices and flow reversal due to suction slots”, *Philosophical Transactions of the Royal Society*, Vol. 363, No. 1830, 2005, pp. 1199 – 1208.
- [11] Dovgal, A. and Sorokin, A., “Application of flow suction for controlling the shedding of large-scale vortices at boundary-layer separation”, *Journal of Applied Mechanics and Technical Physics*, Volume 47, Number 4, 2006, pp. 510-514
- [12] Song, Y., Chen, F., Yang J. and Wang, Z., “A Numerical Investigation of Boundary Layer Suction in Compound Lean Compressor Cascades”, *ASME Journal of Turbomachinery*, 5, Volume 128, Issue 2, 2006, pp. 357 – 366.
- [13] Atik, H. and van Dommelen, L., “Autogenous Suction to Prevent Laminar Boundary-Layer Separation”. *Journal of Fluids Engineering* 1, Vol. 130, Issue 1, 2008, 011201 (8 pages)
- [14] Gbadebo, S.A., Cumpsty, N.A. and Hynes T.P., “Control of Three-Dimensional Separations in Axial Compressors by Tailored Boundary Layer Suction”, *ASME Journal of Turbomachinery*, 1, Volume 130, Issue 1, 2008, 011004 (8 pages)
- [15] 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”, *ASME Journal of Turbomachinery*, 119, 1997, pp. 94–102.
- [16] Sorokes, J.M., Borer, J.C. and Koch, J.M., “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] Hillewaert, K. and Van den Braembussche, R.A., “Numerical simulation of impeller-volute interaction in centrifugal compressors”, *ASME Journal of Turbomachinery*, 121, 1999, pp. 603-608.
- [18] 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.
- [19] Sofiane, K., Smaïne, K., Farid, B. and Robert, R., “Flow Study in the Impeller-Diffuser Interface of a Vaned Centrifugal Fan”, *ASME Journal of Fluid Engineering*, 5, vol. 127, 2005, pp. 495 – 502.
- [20] Meakhail, T. and Park S.O., “A Study of Impeller-Diffuser-Volute Interaction in a Centrifugal Fan”, *ASME Journal of Turbo machinery*, 127, 2005, pp. 84 – 90.
- [21] Fluent 6.3, Fluent Inc., 2006.
- [22] Patankar, S.V., Numerical Heat Transfer and Fluid Flow, *Taylor and Francis*, 1980, pp. 90-92.