كلية الهندسة بشبرا
FACULTY OF ENGINEERING AT SHOUBRA
ERJ
PRINT ISSN 3009-6049
ONLINE ISSN 3009-6022

ENGINEERING RESEARCH JOURNAL (ERJ)
Volume (52),Issue (4)
October 2023, pp:193-203
https://erjsh.journals.ekb.eg

Effect of Blade Slot Inclination Angle on Flow Separation in The Centrifugal Fan Impeller

S. A. Beskales[1,*], Ibrahim Shahin[1], Samir S. Ayad[1], Osama E. Abdullatif[1] and Tarek A. Mekhail[2]

[1] Department of Mechanical Engineering, Faculty of Engineering at Shoubra, Benha University, Shoubra, Cairo, Egypt1
[2] Department of Mechanical Engineering, Faculty of Energy Engineering- Aswan University, Aswan, Egypt.
**Corresponding author*

E-mail address:samer.aagaiby@feng.bu.edu.eg, ibrahim.shahin@feng.bu.edu.eg, samir_ayad@mail.com, osama.abdellatif@fen.bu.edu.eg, mina_tarek@hotmail.com

Abstract: The use of slotted blades for centrifugal fan impellers improves fluid flow steadiness and reduces fan noise. The slots create a jet of fluid from the pressure side to the suction side that increases the momentum of the main flow and postpones the flow separation. In addition, these slots make the fluid movement smoother and more efficient in impeller passages. In previous research, the optimum slot location was determined at 40% from the leading edge (at diameter $D_s=0.65\ D_{Impeller}$) with a 2.5 mm width. The present research concentrates on determining the optimum slot jet angle of a ten-blade backward curved centrifugal fan at the best efficiency point (BEP). Six values of slot inclination angles, namely θ_{S1}, θ_{S2}, θ_{S3}, θ_{S4}, θ_{S5} and θ_{S6}, ranging from an inward slot direction of (+60°) to an outward direction of (-80°), are simulated numerically using an unsteady Computational Fluid Dynamics (CFD) model. The CFD results have proven the benefits of the outward slots that ranged from (θ_S= -60° to -80°) on stall control of the fluid on the blade suction side compared with the inward slots. At the BEP, the computed performance of the tested fan with an outward slot inclination angle θ_{S6} =-80° showed a 2.6 percent efficiency improvement compared to fans with an inward slot of θ_{S1}=+60°. Generally, the outward slot direction has a positive effect on the boundary layer attachment on the suction side of the blade.

Keywords: Suction slots, slot inclination angle, Fan performance, CFD, Slotted blade.

1. Introduction

The fan impeller is one of the most important parts of a centrifugal fan. It spins at high speeds and creates airflow through centrifugal force. The vortex phenomenon may occur within the impeller of a centrifugal fan when the moving fluid in the boundary layer does not persist in the thin layer that adheres to the impeller blade surfaces [1]. This phenomenon is characterized by the formation of a complex and turbulent flow pattern and the thickening of the boundary layer, which can lead to reduced fan efficiency and increased noise levels. The slots technique resolves this issue by injecting the flow from the pressure side of the blade to the suction side; these slots are installed close to the point of separation [2], [3] to regulate the formation of the boundary layer on the blade surface. The design of slot jet angle in the blade can have a significant impact on its performance, pressure distribution on the blade surface and impeller outlet.

Many authors have employed computational fluid dynamics simulations to examine the flow field and performance of centrifugal fans with slotted blades with outward slot jet towards the blade tip. [3]studied numerically the effect of the slot on the performance of a centrifugal fan. The author used a divergence convergence slot from the pressure side to the suction side with dimensions of 2.5 to 1.5 mm, respectively. The slot was tested at five radial locations with slot jet directed to the impeller outlet. The numerical results showed that adding slots at 12.5% from the leading edge and trailing edge improved the static pressure, while slots at 25% from the trailing edge reduced the separation zones. The performance marginally improved when the slots were added at the midpoint. The slots cut in the diffuser was investigated by [4] with a constant width at variable locations and outward direction. It was shown that the cut slots at 30% from the trailing edge showed the best performance while slots near the leading edge partially reduced the stall.

[2]Investigated the effect of a 3 mm cut slot at 45% from the leading edge with an outward slot jet. The results showed an enhancement in the pressure distribution and fluid movement between the impeller passages. Also, reducing the slot width to 2mm with the same slot jet angle at a different location of 70 % from the leading edge, as reported by[5] reduced the pressure variations and energized the leakage flow. [6]performed unsteady numerical simulations of slot cuts near the impeller shroud and found that blade surface slots improved performance and flow field by 3.6 % for total pressure and 2.6 % for efficiency at the design mass flow rate. [7] investigated numerically the effect of the outward slot on backward curved fan impeller at six different locations (Ds = 0.5, 0.6, 0.65, 0.7, 0.8, and 0.9 of impeller diameter) with a slot width of 2.5 mm. Results revealed that the location of the slot at 0.65 of the impeller diameter increased the static pressure by 1.5 % and the efficiency by 2.7 % compared with the slot location near the leading edge at Ds=0.5of

impeller diameter. Also, the authors tested numerically the effect of slot widths (1.5, 2.5, 3.5, and 5 mm) on the performance of the centrifugal fan at the optimum slot location at the BEP. The results showed the vortex and flow separation is controlled at slot width of 2.5 mm and separation increased dramatically at 5mm slot width. [8]studied the effect of the slot from the leading edge of the pressure side to the midpoint of the blade at the suction side. The slot jet was directed to the blade tip with variable channel width. The results confirmed the slot benefits in reducing the boundary layer separation and stall control over different incident angles.

Although several studies have investigated the outward slot jet, little attention has been given to the perpendicular slot jet direction .[9]examined five slot configurations with variable diameters with slot jet perpendicular to the blade. This investigation proved the slot's benefits in reducing the noise and slightly increasing the fan pressure and efficiency. [10]investigated the perpendicular slot with single and twin slot configurations. The results indicated that the appropriate single slot size is 5 mm with a 5.5% increase in static pressure, whereas the twin slots located at 50% and 75% from the blade root with a slot size of 3 mm increase static pressure by 7% compared to the blade without slots.

Authors [11]–[14] investigated the effect of slot inclination angle on pump's performance. [11]improved the pump performance using 1 mm slot width with perpendicular direction on the blade. [12]conducted an orthogonal test using four sets of geoetrical parameters (slot position, width, depth, and slot inclination angle). The orthogonal test indicated that the slot angle range of 45° to 60° showed the best performance. [13]studied the effect of slot location, height, and inclination angle and showed that using a slot angle from -90 to -130 increased the impeller head from 5 to 10%. As reported by [14] it was shown that a slot inclination of 20° improved the pump efficiency and head compared to the base model

To the author's knowledge, few publications are available in the literature that discuss the effect of slot jet angle on the pump's performance. Most previous articles related to the centrifugal fan did not take into account the effect of slot jet inclination angle on the fan's performance. The objective of the present research is to improve the flow attachment in the fan impeller passages by reducing the size of the recirculation zones. This can be accomplished numerically using CFD model by cutting slots into the impeller's blades at a fixed location and width as designed by the authors [7] and controlling the slot's jet direction with respect to the blade to passively control the flow and push the flow separations towards the impeller tip.

2. NUMERICAL METHODOLOGY

2.1 Governing Equations and Turbulence Model

Since performing experiments is not available to visualize the vortex in the impeller passages, researchers use the CFD models to capture insights into fluid circulation and flow contours like pressure and velocity. Basic equations of conservation of mass and momentum for incompressible turbulent flow are given by equations (1) and (2)

$$\frac{\partial \overline{\rho}}{\partial t}+\frac{\partial}{\partial x_i}\left[\overline{\rho}\cdot\overline{U_i}\right]=0 \tag{1}$$

$$\rho\left(\frac{\partial \overline{U}_i}{\partial t}+\overline{U}_j\frac{\partial \overline{U}_i}{\partial x_j}\right)=-\frac{\partial \overline{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial \overline{U}_i}{\partial x_j}-\left(\rho\,\overline{u_j'u_i'}\right)\right)+\overline{F}_i \tag{2}$$

The Shear Stress Transport (SST) k- ω model is applied in the present research. The k- ω model consists of two equations. The Wilcox k- ω model [15] is more stable than the k- ε model in the viscous sublayer close to the wall and is one of the most well-known forms of k- ω. However, its results are particularly sensitive to the free stream value of ω in the free shear layer and adverse pressure gradient boundary layer flows. Consequently, the k- ε model is not the most appropriate model for wake area applications. In the wake areas, however, the k- ε model performs better, according to Menter[16], the combination of the two models is the optimum way to model the flow characteristics along the wall and in the boundary layer's wake. Accordingly, the SST k-ω turbulence is believed to be the best acceptable model for predicting the flow characteristics. The third equation in the model defines the turbulent kinetic energy (k), while the forth describes the turbulent dissipation rate (ω).

$$\frac{\partial \rho k}{\partial t}+\frac{\partial \rho k U_j}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_k\mu_t\right)\frac{\partial k}{\partial x_j}\right]+P_k-\beta^*\rho\omega \tag{3}$$

$$\frac{\partial \rho\omega}{\partial t}+\frac{\partial \rho U_j\omega}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_k\mu_t\right)\frac{\partial \omega}{\partial x_j}\right]+\gamma P\omega-\beta\rho\omega^2$$
$$+2\left(1-F_1\right)\rho\sigma_\omega 2\frac{\mu_t}{k}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{4}$$

2.2 Centrifugal Fan Geometry Model

The centrifugal fan model used in the current research is designed in the SPACE-CLAIM software. It consists of four main fluid domains (Inlet pipe, slotted impeller, volute, and outlet pipe). The impeller gap is considered in the present research. The length of the inlet and outflow ducts is more than five times the duct diameter to ensure a fully developed flow. The back distance between the hub and the volute wall is considered to be 33.5 mm. The impeller inlet and outlet angles are 31° and 41°, respectively. The modeled centrifugal fan in the present research ten backward impeller blades designed by [17]. The whole fan's dimensions are provided in Fig. 1.

In the following CFD simulation, the location of the slot is fixed at Ds of 0.65 of the impeller diameter and with a constant width of 2.5 mm as designed by the authors [7], as shown in Fig.2. With these constant parameters, the inclination angle of the slot was investigated from the inward direction to the outward direction, as shown in Fig.3.

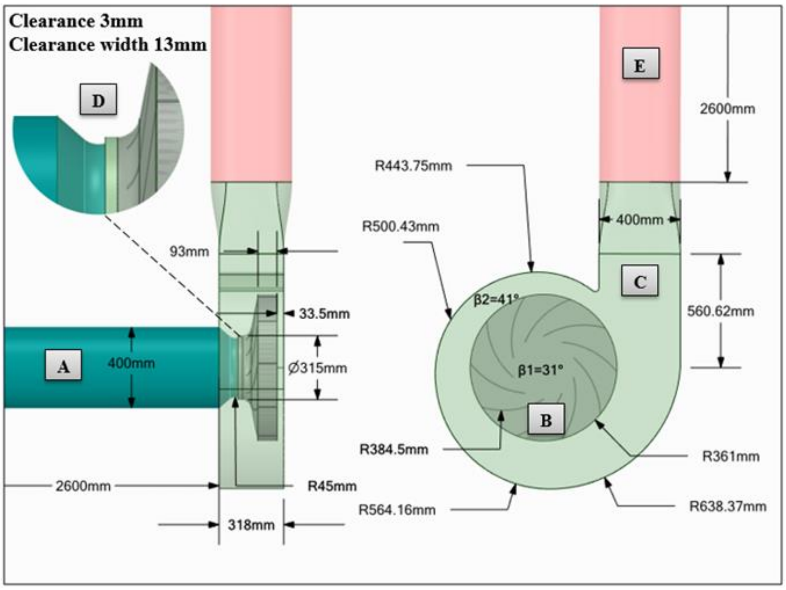


FIGURE 1. Centrifugal fan Dimensions:
[A] Inlet Pipe [B] Impeller [C] Volute [D] Gap clearance details [E] outlet pipe

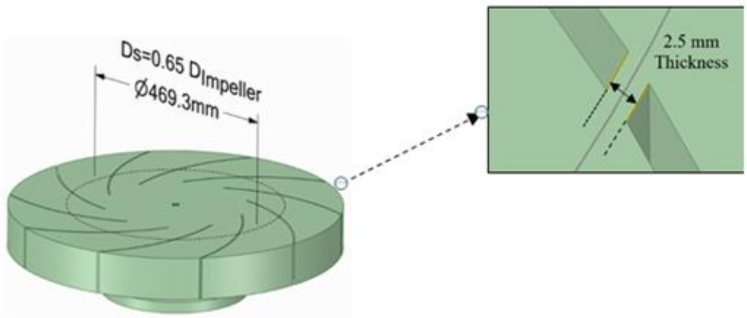


FIGURE 2. Slotted Impeller at Ds=0.65 Dimpeller with a fixed width of 2.5 mm

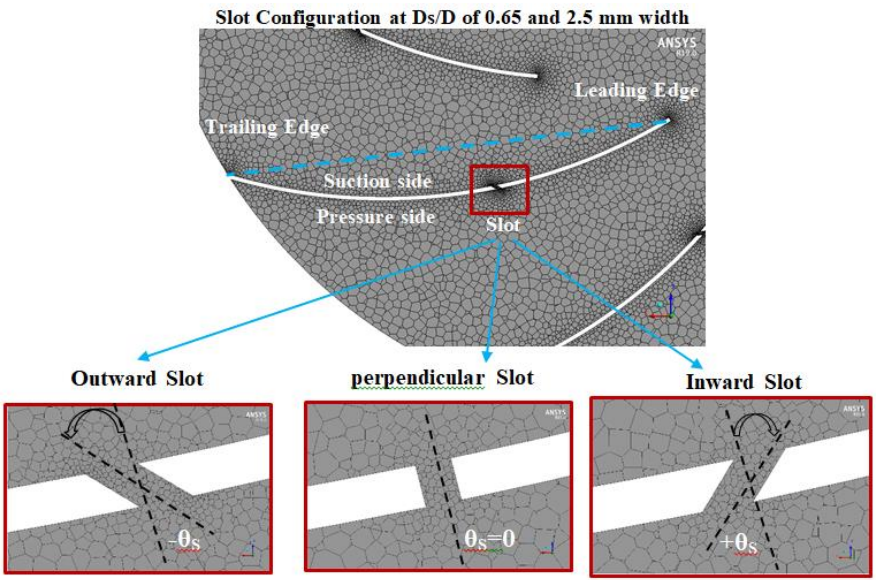


FIGURE 3. Slotted impeller with different inclination angles at Ds of 0.65 of impeller diameter and with 2.5 mm width.

2.3 Mesh Sensitivity Study

One of the most important factors affecting the results of numerical simulation is mesh size. Building the tiniest components surrounding the blades, such as the leading edge, trailing edge, and slot gap, is important for investigating the flow field over the most crucial locations of a centrifugal fan. A mesh convergence test is conducted To ensure that the experimental findings of[17] and the CFD validation results are in excellent agreement. This procedure has been done using six distinct points, ranging from coarse mesh (point 1) to fine mesh (point 6). Figure 4.illustrates the effect of the number ofelements on the static pressure rise between the inlet and outflow of the fan and the impeller torque for one of the validation study's cases with a flow coefficient of 0.027. The figures indicate that the results are nearly independent of the number of mesh elements for all values of the number of elements equal to or greater than 3,400,000.

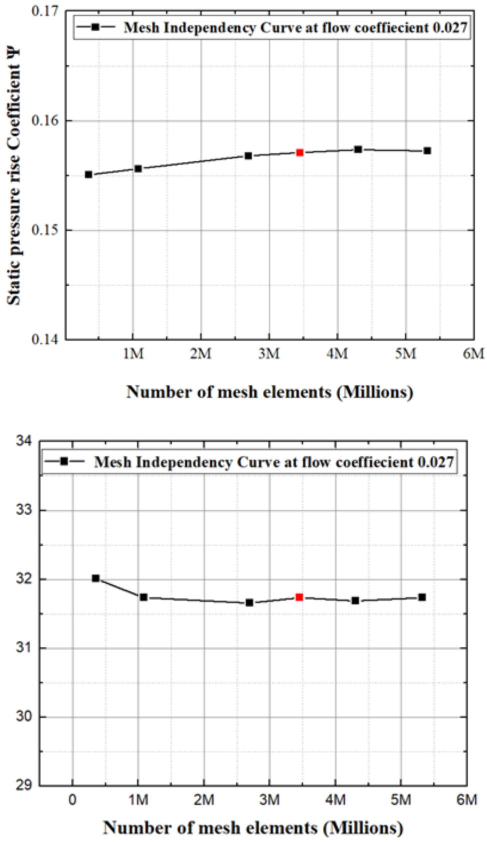


FIGURE 4. Number of mesh elements effect on the static pressure rise coefficient (left) and impeller torque (right) at flow coefficient 0.027

In the impeller domain, a fine mesh was created at the blade pressure and suction sides with a minimum and maximum element size of 0.5 mm to 3 mm and at the critical regions such as the leading edge, the slot gap, and the impeller gap with a minimum and maximum element size of 0.25 mm to 0.5 mm. The volumes of the impeller, volute, and inlet pipe were meshed with minimum and maximum element sizes of 0.25 mm and 7 mm, respectively. In all simulations, the volute casing and inlet pipe domain are the same, while the impeller domain is changed by changing the slot orientation. The meshing was created in the FLUENT MESHER program using a polyhedral mesh.The boundary condition is velocity inlet at the inlet pipe and outflow at the outlet of volute pipe. The blades, hub, shroud, and slot surfaces are considered walls with zero velocity relative to the adjacent fluid, while the volute casing and the inlet pipe are considered walls with zero absolute velocities. SIMPLE algorithm is used for velocity pressure coupling and second order scheme for turbulent kinetic energy and specific dissipation rate.

2.4 CFD Validations Results

The current simulation results are compared to the experimental results of [17]. Figure 5. (Left) compares the present steady and unsteady pressure rises to the experimental measurement by the authors [17]. The unsteady pressure curve shows good agreement with the experimental results throughout the entire flow coefficient range, with a 6% increase at the Best Efficiency Point (**BEP**) and with a 7.7% increase at the point of maximum flow rate. The unsteady pressure curve provides more accurate values than the steady calculations with a 10% increase in pressure at the BEP. However, when computational time is limited, a steady pressure curve could be used as an initial guess for estimating the fan pressure curve.

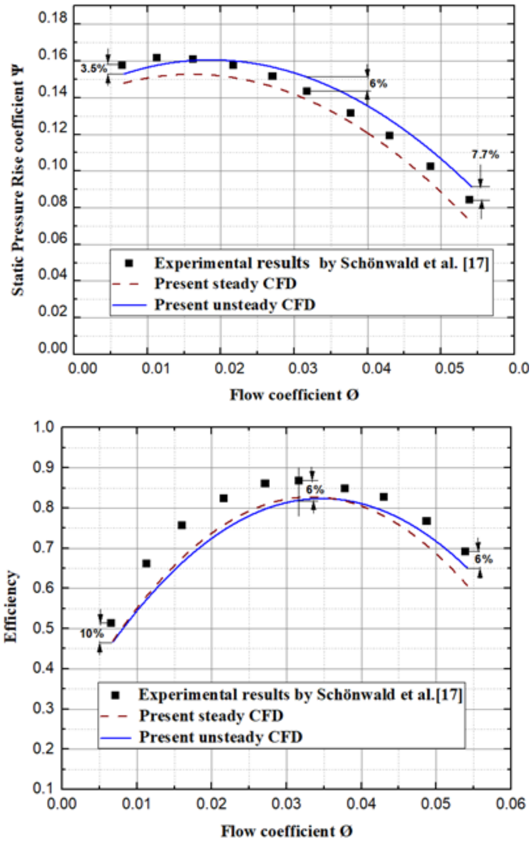


FIGURE 5. Comparisons of static pressure rise curves (Left) and efficiency curves (Right) for the fan (N=1500 RPM)

3. RESULTS AND DISCUSSIONS

3.1 Fan Performance Comparisons

The fan performance was calculated using the static pressure difference at the inlet and outlet pipes at the best efficiency point (BEP) at a flow rate of 1.9 m^3/s. The shaft power is calculated by multiplying the estimated torque times the angular velocity of the fan at N = 1500 RPM.

The static efficiency of the centrifugal fan is calculated and defined as:

$$\eta = (\Delta P * Q)/(T * \omega) \qquad (5)$$

Figure 6.shows the effect of changing the slot inclination angle from the inward direction to the outward direction on the performance of the centrifugal fan at the maximum efficiency point at a flow rate of 1.9 m^3/s. The outward slotted fan blade with θ_{S5} and θ_{S6} experienced an increase in efficiency of 2.6 percent compared with the inward slotted fan blade at θ_{S1}. On the other hand, the pressure decreased at the fan outlet as the slot changed from the inward to the outward direction. The main cause of increasing efficiency with the outward slot directions is decreasing the fluid blockage and enhanceing the stadiness of the fluid in the impeller passages. It will be revealed after visualizing the flow in the fan impeller in the next section.

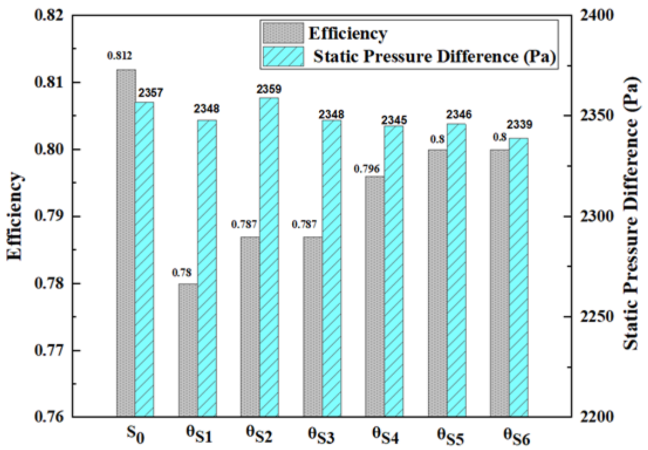


FIGURE 6. Effect of the slot inclination angles on the performance of centrifugal fan at BEP

3.2 Streamlines Visualization

Since the fluid is air and transparent, CFD flow visualization is needed to gather qualitative information about the fluid's behavior inside the impeller. Streamlines colored by the velocity magnitude were investigated at three different locations inside the impeller (near the hub, shroud, and mid-section), as shown in Fig. 7.

Streamlines between the impeller blades were investigated at three different locations: near the hub, shroud, and mid-section, as shown in Fig. 8. By investigating the figure, it is clear that the flow in the datum blades impeller S_0 near the shroud is unstable, and separation began at the midpoint of the blade towards the blade tip. Adding slots with inward directions increased the separation near the shroud, midsection and near the hub. Conversely, the slots with an outward directions energized the fluid throughout the whole impeller and reduced the circulation zones.

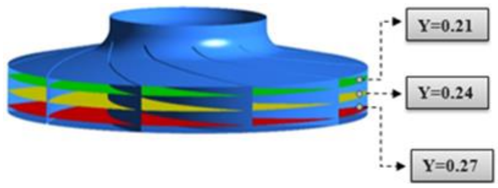
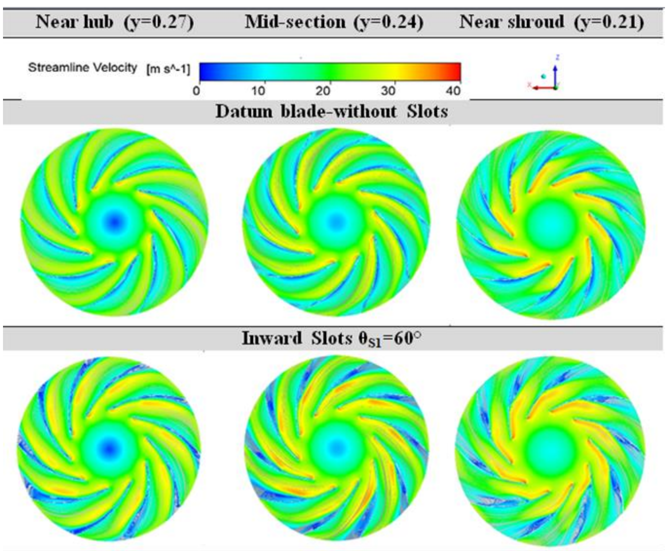


FIGURE 7. Impeller with three plans: near hub, mid-section and near shroud

To understand the effect of the slot inclination angle on the flow behavior inside the impeller passages, Fig. 9 shows the streamlines over one blade at different slot jet directions. For the slot jet direction θ_S equal to (-30, -60, 0), the fluid separated immediately after the slot to the blade tip. These separated fluids block the flow between the blade passages, reducing the performance of the centrifugal fan. Oppositely, using the outward slot direction from θ_S equal to (-30 to -80), the slot jet from the pressure to the suction side increased the momentum of the fluid on the suction side and pushed the separation towards the blade tip.

The jet of the fluid through the slot has a great impact on the steadiness of the fluid at the suction side. Figure 10.shows the effect of the slot angle direction on the stability of the fluid over the fan blade. For slot jet with inward directions from θS = -60 to perpendicular slot jet θS =0 the flow from the slot encounter the flow from the leading edge over the blade surface and a secondary flow appeared toward the blade tip. Conversely, the fan blade with outward direction from θS equal to -30 to -80 the fluid through the jet increase the main fluid momentum over the suction side and the fluid became more streamed and decreased the boundary layer thickness.



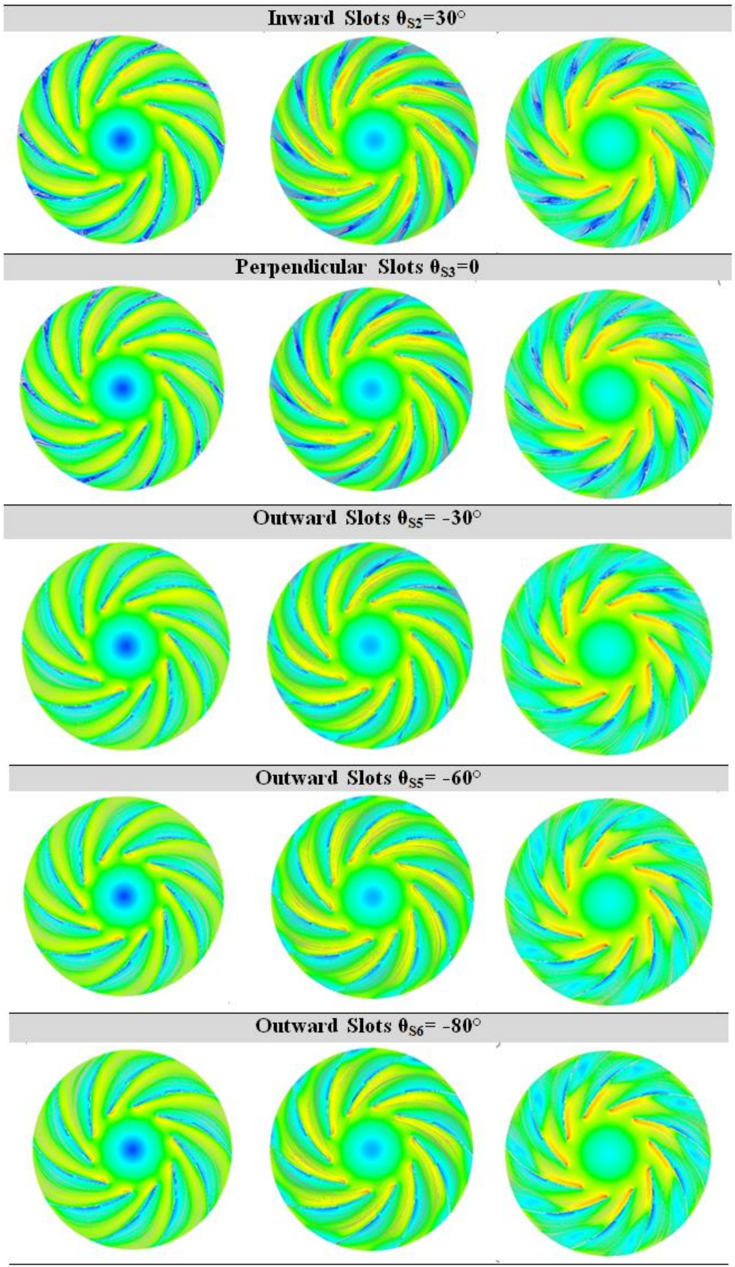


FIGURE 8. Streamlines comparisons at three different locations inside the impeller: Near hub, mid-section and near shroud

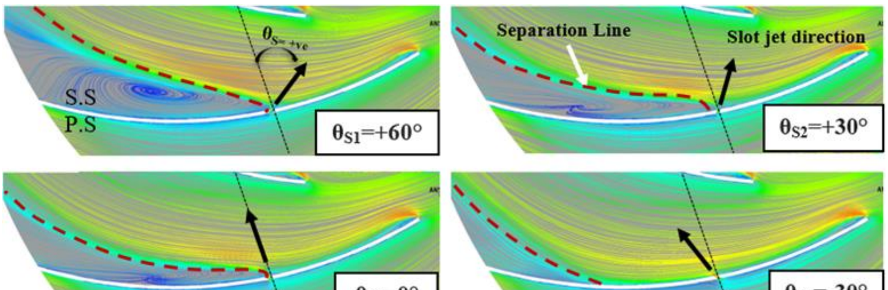


FIGURE 9. A close view to the streamlines comparisons colored by velocity magnitude over a blade from LE to TE at different inclination angle at y=0.24.

3.3 Turbulent Kinetic Energy

Turbulent kinetic energy is used to describe the turbulence flow and eddies inside the fan impeller. It is defined as the energy per unit mass (J/kg). It can be measured by the root mean square (RMS) of the velocity fluctuation components.

$$k = \frac{1}{2}\left(\overline{(u')^2} + \overline{(v')^2} + \overline{(w')^2}\right) \tag{6}$$

In Fig.11, For the datum blade, the turbulent kinetic energy (TKE) almost below the value of 1.3 (J/kg) in all section through the impeller domain. Adding a cut slot with inward direction and perpendicular increased the TKE especially near the shroud. Meanwhile, the outward slot showed a streamed flow over the blade surface with low vortices even near the shroud region.

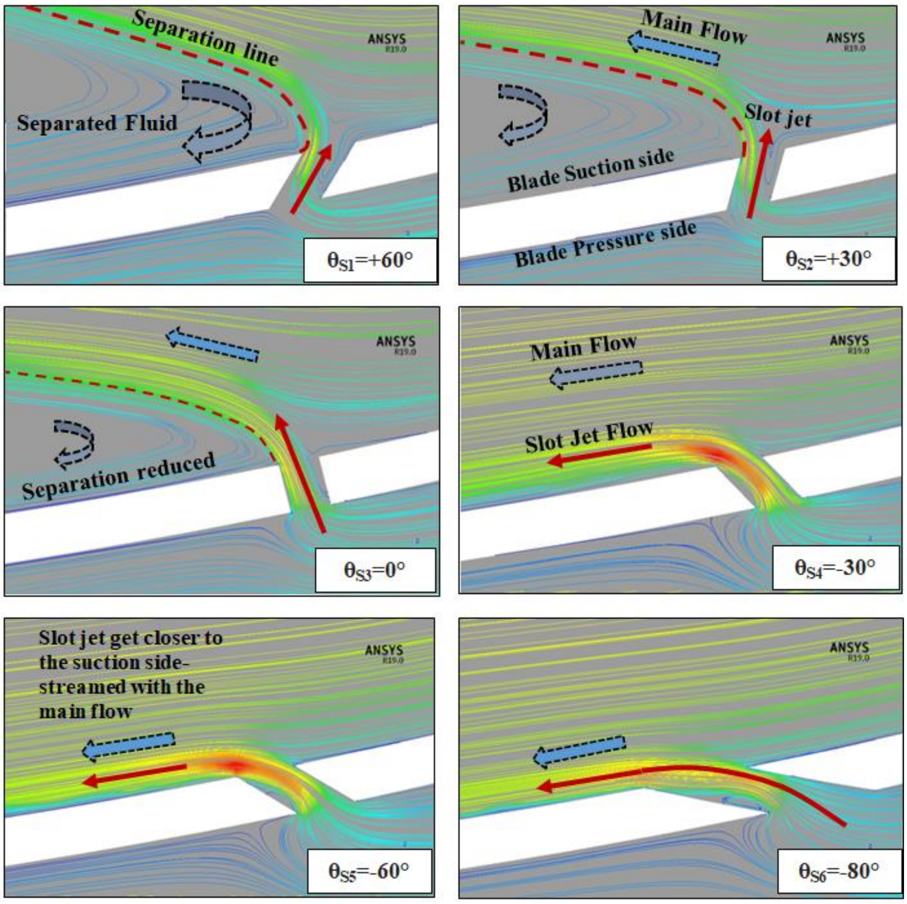
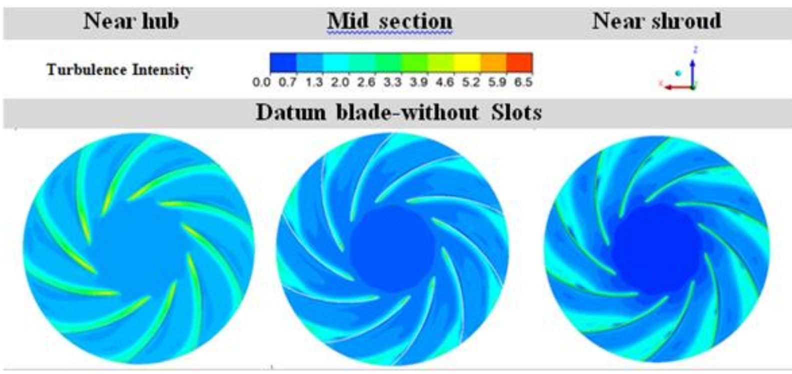


FIGURE 10. A close view to the streamlines colored by velocity magnitude through the slot at different inclination angles at mid-section plan y=0.24



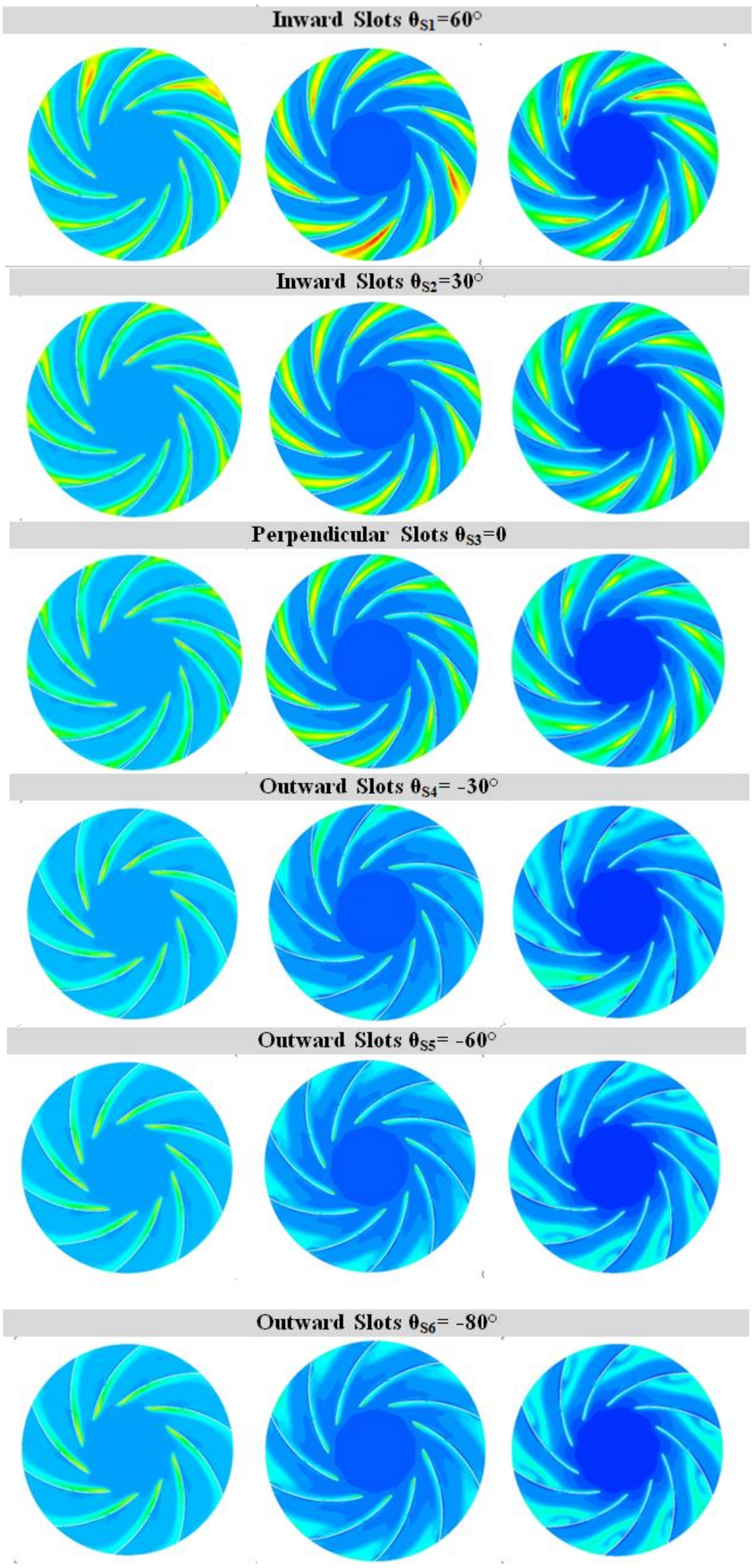


FIGURE 11. Turbulence kinetic energy comparisons at the three sections inside the impeller domain for different slot inclination angles at y=0.24

3.4 Blade Pressure Distribution Curve

To understand the effect of changing the slotjet angle on the internal pressure of the fan blades at the maximum efficiency point (at flow rate 1.9 m3/s), the pressure distribution on the surface of the fan blade is examined and plotted in x axis direction from the leading edge (x/c=0) to the trailing edge (x/c=1) as shown in Fig.12, at a polyline created in CFD-Post at the mid-section of the blade at y = 0.24.

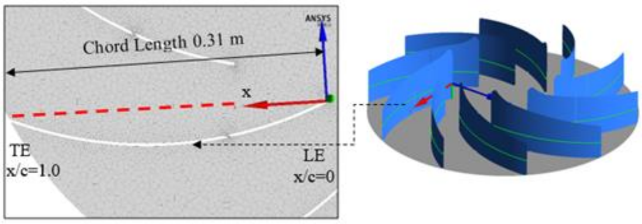
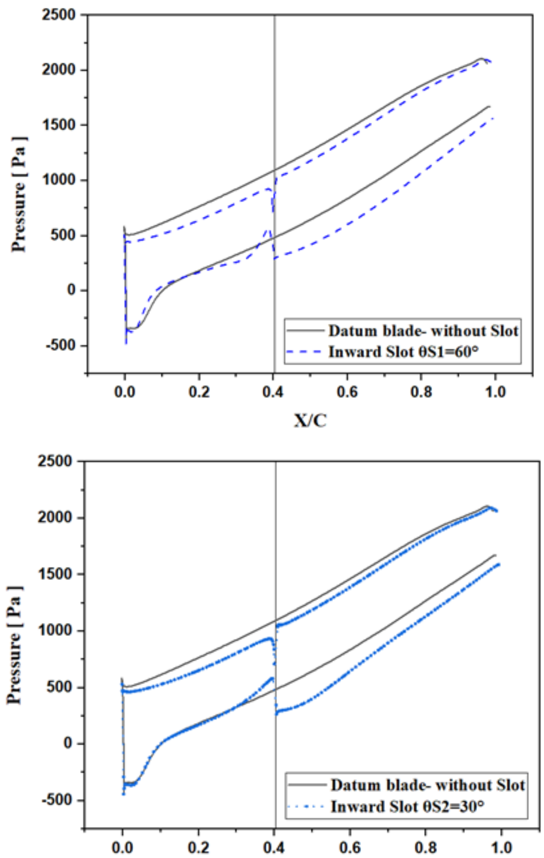


FIGURE 12 Fan blades with polylines at a x-z plane y=0.24m

Figure 13.shows a comparisons of the blade pressure distribution curves between a datum blade (without slot) and a fan blade with a slot at six different slot angles at a location at 0.65 of impeller diameter at BEP. Slots in the present research were added at the optimum location at (x/c=0.4) of the blade span from the leading edge [7]. By investigating the graphs, from the leading edge to (x/c=0.4), before the cut slot, the inward slots showed that the pressure difference across the blade surfaces decreased compared with the datum blade. Contrariwise, the fan blade with outward slots, the pressure difference between the suction and pressure sides are considered the same. After the cut slot immediately a considerable pressure drop in the suction side was observed about 39% compared with the datum blade, Although the pressure difference between the blade surfaces increased, the streamline visualization showed a strong separation at the suction side which decrease the fan efficiency of inward slot directions compared to the datum blade. The optimum slot angle was θ_S=-60 and -80° with a pressure drop and streamed flow over the suction side.



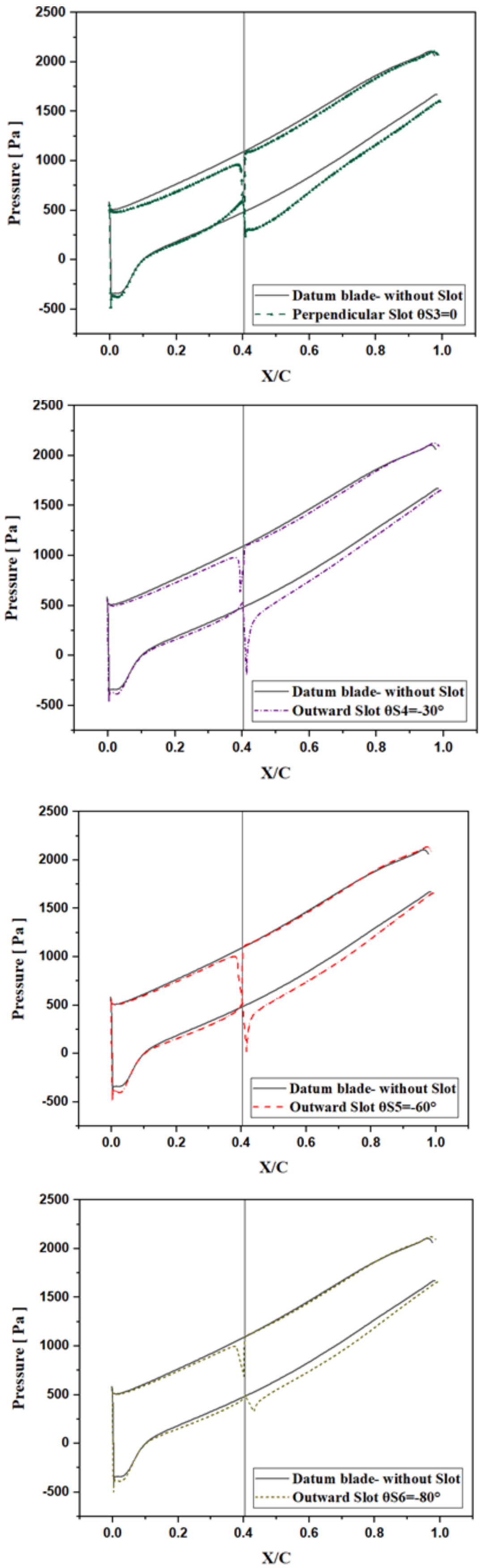


FIGURE 13. Blade pressure distribution comparison curves between datum blade and slotted blades with different slot inclination angles at mid-section (y=0.24m) at BEP

3.5 Vorticity Visualization

The Q-criterion is a method based on the velocity gradient tensor that is used for detecting and visualizing the vortex structures that are present in the complicated flow fields [18]. The velocity gradient tensor D_{ij} can be written as

$$D_{ij} = \frac{\partial U_i}{\partial x_j} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i}\right) \quad (6)$$

The first term in Eq. (6) represent the angular deformation tensor

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \quad (7)$$

And the second term represents the rotational tensor (vorticity)

$$\Omega_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i}\right) \quad (8)$$

This Q-criterion represented the relation of fluid rotation tensor (Ω_{ij}) and deformation tensor (S_{ij}) in the flow field.

$$Q = ½ (||\Omega_{ij}||^2 – ||S_{ij}||^2) \quad (9)$$

In Eq. 9, when the term Q > 0, it means that the antisymmetric component of the velocity gradient tensor, that is, the vorticity tensor, dominates over the symmetric one and the flow field's fluid is mostly rotating, whereas Q<0 indicates that the fluid is extensively deformed.

Figure 14.shows the vorticity at the slot outlet from the suction side for inward, perpendicular, and outward slots using the Q-criterion method, where Q = 0.03 > 0. By investigating the figure, it is clear that for the inward slot at θ_{S1} and θ_{S2} the backward vorticity that appears near the shroud acted like a barrier to the main impeller flow and decreased as the slot angle changed from the inward direction to the outward direction. The slot angles θ_{S5} and θ_{S6} exhibit negligible vorticity and are evenly distributed from the hub to the shroud.

4. Conclusions

This study aimed to numerically investigate the impact of the inclination angle of a single cut slot located at a fixed position (40% from the leading edge) and a width of 2.5 mm on the flow steadiness within the impeller blade passages and the fan performance at the BEP.The flow visualizations and blade load curves are lead to the following:

- A fan blade with an outward slot direction amended the flow attachment and energized the flow at the suction side to a certain point on the blade span. The slot jet pushes the separation point further towards the blade tip as the inclination increased toward the impeller outlet from θS= -30° to -80°.
- The blade load curves related to the outward slots displayed an identical pressure distribution on the fan blade as comparedd to the datum fan blade. The advantageous effect of the outward slot is subsequently observed in the reduction of pressure on the suction side, leading to increase of the diefference in pressure and an increase in fan efficiency up to 2.6% compared with inward slots.

The jet of fluid efflux from the slot in the inward direction thickened the boundary layer over the suction side, reduced the fluid momentum, increased the fluid blockage in the impeller passages, and reduced the fan efficiency.

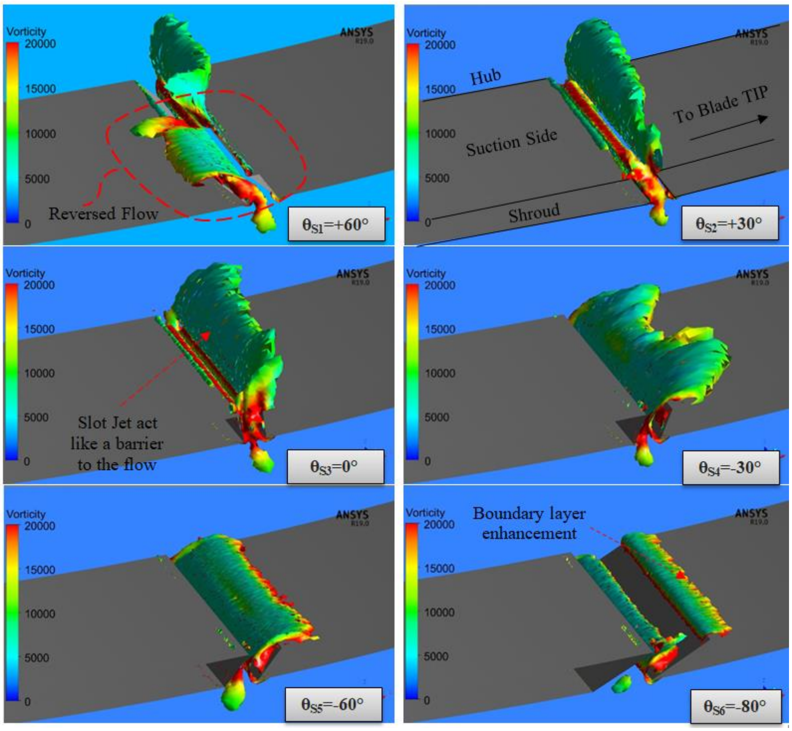


FIGURE 14. 3D vorticity at slot outlet at different slot angles at Q-criterion = 0.03

NOMENCLATURE

D_{ij}	velocity gradient tensor	S^{-1}	**Greek**		
Ds	Slot pitch Diameter	mm	ε	Turbulent energy dissipation rate	m^2/s^3
k	Turbulent kinetic energy	m^2/s^2	η	Static Efficiency	-
S_{ij}	Angular deformation tensor	S^{-1}	ω	Rotational Speed	S^{-1}
u'	Fluctuated velocity	m/s	ɷ	Flow coefficient = $Q/\omega D^3$	-
$\overline{U_i}$	Time averaged velocity	m/s	Ω_{ij}	Rotaional Tensor	S^{-1}

Abbreviations

BEP	Best Efficiency Point
CFD	Computational Fluid Dynamics
MRF	Moving Reference Frame
RMS	root mean square
SST	Shear Stress Transport
SIMPLE	Semi-Implicit Method for Pressure Linked Equations

5. References

[1] Schlichting, "Boundary Layer Theory," Seventh Ed. McCraw-Hill, 1968.

[2] M. S. Kassim, F. A. Saleh, and M. A. Kadhum, "Experimental and Numerical Investigation of Blades Slots on Rotating Stall Phenomenon in Centrifugal Blower," Univers. J. Eng. Sci., vol. 3, no. 2, pp. 24–37, May 2015, doi: 10.13189/UJES.2015.030203.

[3] K. V. Karanth and N. Y. Sharma, "CFD Analysis of a Centrifugal Fan for Performance Enhancement using Converging Boundary Layer Suction Slots," Int. J. Mech. Mechatronics Eng., vol. 3, no. 12, pp. 1493–1499, Dec. 2009, doi: 10.5281/ZENODO.1071632.

[4] K. Vasudeva Karanth and N. Yagnesh Sharma, "Numerical Analysis of a Centrifugal Fan for Performance Enhancement Using Boundary Layer Suction Slots," Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci., vol. 224, no. 8, pp. 1665–1678, 2009, doi: 10.1243/09544062JMES1990.

[5] M. S. Kassim, F. A. Saleh, and A. T. Aliwi, "Investigation The Effect of Blade's Slot on Rotating Stall of High Speed Blower," J. Eng. Sustain. Dev., vol. 23, no. 04, pp. 170–186, Sep. 2019, doi: 10.31272/JEASD.23.4.12.

[6] H. Zhang, F. Dong, X. Zhang, and W. Chu, "Numerical Investigation of Effect of Blade Surface Slot on the Performance and Flow Field of a Low Speed Centrifugal Fan." Sep. 21, 2020. doi: 10.1115/GT2020-14472.

[7] S. A. Beskales, I. Shahin, S. S. Ayad, O. E. Abdullatif, and T. A.Mekhail, "Effect of Slot Location and Width on the Performance of Centrifugal Fan," Eng. Res. J., 2023, [Online]. Available: In Press

[8] R. Rong, K. Cui, Z. Li, and Z. Wu, "Numerical Study of Centrifugal Fan with Slots in Blade Surface," Procedia Eng., vol. 126, pp. 588–591, Jan. 2015, doi: 10.1016/J.PROENG.2015.11.309.

[9] A. Pieruszka, T. Siwek, W. Kalawa, Ł. Lis, S. Stefański, and K. Sztekler, "The Effect of Blade Slots on Flow Behaviour in a Fan Impeller," EPJ Web Conf., vol. 02067, pp. 1–4, 2019, doi: 10.1051/epjconf/201921302067.

[10] N. Madhwesh and K. V. Karanth, "Effects of Innovative Suction Slots on the Performance of a Radial Bladed Impeller of a Centrifugal Blower - A Numerical Transient Analysis," espublisher, vol. 17, pp. 91–100, Dec. 2021, doi: 10.30919/ES8D587.

[11] D. X. Ye, H. Li, and Y. Wang, "Hydraulic Performance of a Low Specific Speed Centrifugal Pump with Spanwise-Slotted Blades," IOP Conf. Ser. Mater. Sci. Eng., vol. 52, no. 2, Dec. 2013, doi: 10.1088/1757-899X/52/2/022009.

[12] H. Wang, B. Long, C. Wang, C. Han, and L. Li, "Effects of the Impeller Blade with a Slot Structure on the Centrifugal Pump Performance," Energies 2020, Vol. 13, Page 1628, vol. 13, no. 7, Apr. 2020, doi: 10.3390/EN13071628.

[13] A. F. Ayad, H. M. Abdalla, and A. A. El-Azm, "Study of the effect of impeller side clearance on the centrifugal pump performance using CFD," ASME Int. Mech. Eng. Congr. Expo. Proc., vol. 7A-2015, no. May, 2015, doi: 10.1115/IMECE201550756.

[14] Q. Ke and L. Tang, "Performance Optimization of Slotted Blades for Low-Specific Speed Centrifugal Pumps," Adv. Civ. Eng., vol. 2023, p. 9612947, 2023, doi: 10.1155/2023/9612947.

[15] H. K. Versteeg and W. Malalasekera, An Introdaction to Computational Fluid Dynamics: The Finite Volume Method, vol. Second Edi. 1996.

[16] F. R. Menter, "Zonal Two Equation K- ω Turbulence Models for Aerodynamic Flows," AIAA 23rd Fluid Dyn. Plasmadynamics, Lasers Conf., 1993, doi: 10.2514/6.1993-2906.

[17] S. Schönwald, F. Kameier, and M. Böhle, "Influence of the Casing Width and the Impeller Position on Centrifugal Fan Performance - A CFD-Based Study on Cause and Effect," Proc. ASME Turbo Expo, vol. 1A, 2014, doi: 10.1115/GT2014-25319.

[18] I. Sadrehaghighi, Vortex & Vorticity. 2023. Accessed: Apr. 08, 2023. [Online]. Available: https://www.academia.edu/42191998/Vortex_and_Vorticity