Numerical Investigation on the Flowseparation Control of S809 Airfoil with Slot

K Sarath Kumar
Department of Mechanical Engineering
PG student, SKIT, Sri Kalahathi,india

Dr. C. Bhaskar Reddy
Associate Professor in Mechanical Engineering,
SKIT, Sri Kalahathi.

M.Sivaramakrishnaiah

Assistant Professor in Mechanical Engineering,
SVCET, RVS Nagar, Chittoor.

Abstract: This study focuses primarily on designing the blade for regions of low wind power density. The aerodynamic airfoils of wind turbine blades have crucial influence on aerodynamic efficiency of wind turbine. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. In this paper S809 airfoil profile is considered for analysis of wind turbine blade. Geometry of the airfoil is created using ANSYS. And CFD analysis is carried out using FLUENT at various angles of attack from 0° to 15˚. With slot and without slot varies with different angle of attack. The coefficient of lift and drag values are calculated for 5×10^5 Reynolds number.

Key points: Fluent, Ansys, Angle of attack, Wind turbine blade.

I. INTRODUCTION

CFD study of airfoils to predict its lift and drag characteristics, visualisation and surveillance of flow field pattern around the body, before the endeavour of the experimental study is almost patent. In the present study aerodynamic characteristics of a well -documented airfoil, NACA 0012, equipped with plain flap is investigated. Wing with flap is usually known as high lift device. This ancillary device is fundamentally a movable element that supports the pilot to change the geometry and aerodynamic characteristics of the wing sections to control the motion of the airplane or to improve the performance in some anticipated way.CFD facilitates to envisage the behaviour of geometry subjected to any sort of fluid flow field. This fast progression of computational fluid dynamics (CFD) has been driven by the necessity for more rapid and more exact methods for the calculations of flow fields around very complicated structural configurations of practical attention. CFD has been demonstrated as an economically viable method of preference in the field of numerous aerospace, automotive and industrial components and processes in which a major role is played by fluid or gas flows. In the fluid dynamics, for modelling flow in or around objects many commercial and open source CFD packages are available. The computer simulations can model features and details that are tough, expensive or impossible to measure or visualize experimentally. This study does not provide any experimental data for the flow over the flapped airfoil. Therefore, to reduce the scepticism associated the results obtained; the simulation process for the study is validated instead. In the validation course the results for flow over no flapped NACA 0012 is compared with published standard data by NASA [1], as nearly same computational method is used to study flapped NACA 0012 airfoil. Eleni [2] studied variation of lift and drag coefficients for different viscous turbulent model. His study shows that for flow around NACA 0012 airfoil k-ω Shear Stress Transport (SST) model is the most accurate. Many researchers have studied aerodynamic characteristics of NACA 0012 using different methods and operating conditions. The Abbott and von Doenhoff data [3] were not tripped. The Gregory and O'Reilly data [4] were tripped, but were at a lower Re of 3 million. Lift data are not affected too significantly between 3 million and 6 million, but drag data are [5]The shear-stress transport (SST) k- ω model was developed by Menter to effectively blend the robust and accurate formulation of the k- ω model in the near-wall region with the free stream independence of the k- ω model in the far field. To achieve this, the k- ω model is converted into a k- ω formulation. The SST k- ω model is similar to the standard k- ω model, but includes the following refinements:

The standard k-ω model and the transformed k-ω model are both multiplied by a blending function and both models are added together. The blending function is designed to be one in the near-wall region, which activates the standard k- ω model, and zero away from the surface, which activates the transformed k- ω model. The SST model incorporates a damped cross-diffusion derivative term in the ω equation. The definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress. The modeling constants are different. These features make the SST k- ω model more accurate and reliable for a wider class of flows (for example, adverse pressure gradient flows, airfoils, and transonic shock waves) than the standard k- ω model.

II. METHODOLOGY

Modeling and CFD analysis of airfoil S809 profile Problem Solving Approach in CFD

The basic steps involved in solving any CFD problem are as follows:

- Identification of flow domain.
- Geometry construction or Component Modeling.
- Grid generation.
- Specification of boundary conditions and initial conditions.
- Selection of solver parameters and convergence criteria.
- Results and post processing.

The airfoil S809 profile is modeled and analysis is carried out by following above steps.

2.1 2D Geometrical Model of Seals and blades: - Uses of 2D Design:

The flow can be understood and predicated on a lift and drag coefficient but some aspects of the flow must be considered as fully 2D and quasi wind turbines are very useful in the preliminary design process. When it comes high pressure blading, the blade heights are high the flow in complex and 2D in nature. The flow domain of 2D models of Seals and blades generated by using fluent software. Later these models exported into ANSYS WORKBENCH to generate the mesh.

It is a Semi-automated meshing module which present the rapid generation of multi-block structured quadrilateral cell meshes. Surfaces can be interactively adjusted by splitting it to the underlying CAD geometry or generate H-grids can be generated by the system automatically.

2.2. Modeling:

The modeling of below 4mm slot to 3mm nozzle effect airfoil profile Modified S-809 created and design in Reynolds number 5 x 10^5.In this point data collected in script.js format. ANSYS software has been used for modeling. In this profile are drawn and created 2D model as shown below fig.2

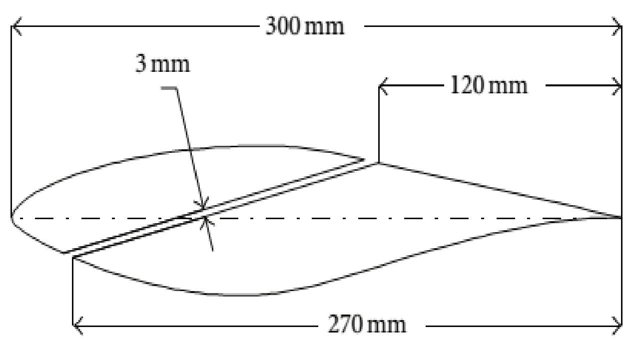

Fig1 Geometry of split

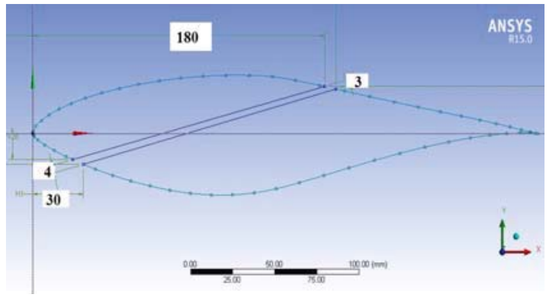

Fig.2Modified slot 4-3mm

2.3. Identification of Flow Domain:-

Before constructing grid, it is required to understand the exact flow domain properly. The flow domain in the case of airfoil wind turbine blade. It is therefore required that before going ahead with 2D modeling and grid generation, the common interfaces should be clearly defined. The nature and complexity of the geometry of blade profile, leading and tailing edge are varied to achieve the better performance of flow domain.But to get the flow established, so that realistic boundary conditions can be given at the inlet and outlets.

The boundary wall is the region where no slip condition exists and the velocity gradually increases and reaches to mainstream velocities. That means, velocity gradient exists there and that region close to the boundary wall should have fine grids.

2.4. Grid generation

Values of the flow properties such as the velocity, pressure etc. CFD require the subdivision of the domain in a number of smaller, non-overlapping sub domain then order to solve the flow physics within the domain geometry that has been created this results in the generation of a grid of elements over laying the whole domain geometry .The essential fluid flows that are described in each of this elements are usually solved numerically so that the discrete.

2.4.1. Grid Generation of Modified S-809 Profile:

Airfoil S809 grid is designed for the creation of high quality computational meshes for periodic blade geometries. Pre defined grid topology templates are used to minimize grid set-up time and mesh for the given application.

ANSYS workbench enables the user to generate computational grids quickly through the automatic management of grid topology, periodic Boundaries and grid attachment.

Steps Involved In Creating Grid:

1. Select a ansys work bench
2. Import the airfoils809 profile file from geometry
3. Create edge sizing for the one edge to parallel edges and bias created with the smooth transition and bias factor
4. Select the all surfaces and right click than select the mapped face meshing
5. Creates the mesh.
6. Review the mesh.

Before importing the data files select suitable sketches, create the surface by using the surface from sketches option and create the both airfoils809 profile and the rectangular boundary. Combine airfoil S809 profile and boundary by use the option Boolean. Using the project option to project the points respected lines and spilt the faces by using the face spilt option and create the named selections on the surface

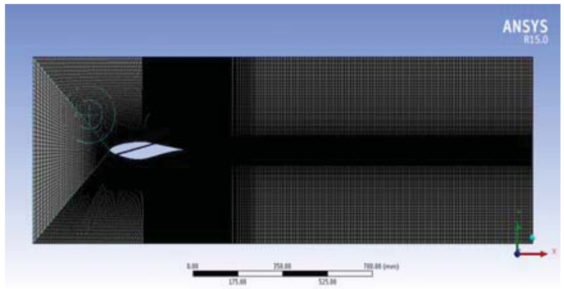

Fig.3 mesh refinement

Quadrilateral cells were used for this simple geometry because they can be stretched easily to account for different flow gradients in different directions. In the present case, the gradients normal to the airfoil wall are much greater than those tangent to the airfoil. Consequently, the cells near the surface have high aspect ratios. For geometries that are more difficult to mesh, it may be easier to create a mesh refinement cells as shown above fig.3

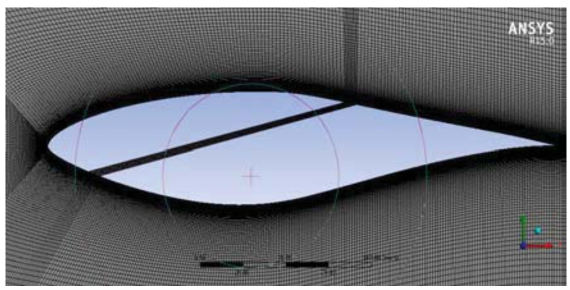

Fig.4 smooth mesh in the interior of the domain

A wind tunnel boundary was chosen to represent the inlet and outlet boundary, enabling the construction of a smooth mesh in the interior of the domain as shown above fig.4

III. PRE AND POST PROCESSING

A significant percentage of the total time spent on a CFD analysis is involved in pre- and post processing activities. Both the setup for an analysis and the evaluation of results

Require considerable effort on the part of the component designer. Therefore, the use of Software tools to automate or facilitate these activities has the potential to substantially reduce the time required for the analysis and improve the overall efficiency of the Process. Preprocessing involves the definition of the boundaries of mesh elements. The basic boundary conditions used for an element as given in the table 1and fig 5.

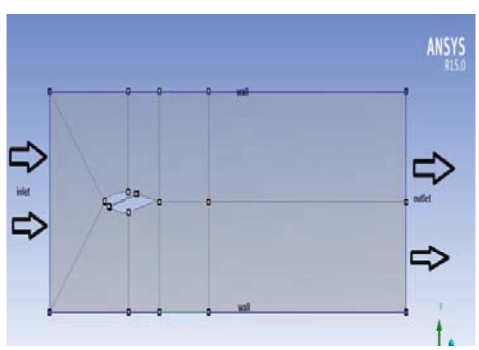

Fig.5 Boundary conditions definition

Domain	fluid type
Turbulence model	SST, k- ω
Type	fluid-fluid
Boundary conditions	
Inlet	velocity inlet
Outlet	pressure outlet
Wall	specified shear
Wall-nozzle	specified shear
Velocity inlet	30 m/s

Table 1. Boundary conditions are specified.

3.1. Physics definition

Physics definition involves defining the physical parameters such as pressure, velocity, etc. and other boundary conditions relevant for the problem. Preprocessing involves the following steps. The software used was ANSYS 15.0

3.2 Defining the domain and boundaries:

The wind tunnel selection like inlet, outlet, and blade are defined. The upper and lower boundary were treated as walls. The boundary conditions were applied at inlet and outlet.

3.3 Initial conditions:

CFD-Solver will terminate the run when the equation residuals calculated using the method specified is below the Residual Target value.

For the present simulation Solver Parameters are specified as follows:

a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.

b. Retain the default selection of Green-gauss node based from the Gradient drop-down list in the Spatial Discretization group box.

c. Retain the default selection of PRESTO! From the Pressure drop-down list.

d. Select power law from the Modified Turbulent Viscosity drop-down list, and maintain density and momentum power law

Maximum Iterations : 2000
Residual Convergence criteria: RMS
Run the **solver monitor**.
The solver is allowed to run till the required convergence is obtained.

3.4. Post processing Results of airfoil S809 profile

After completion of the pre-processing the post processing is carried subsequently the following results are obtained as given below fig,6

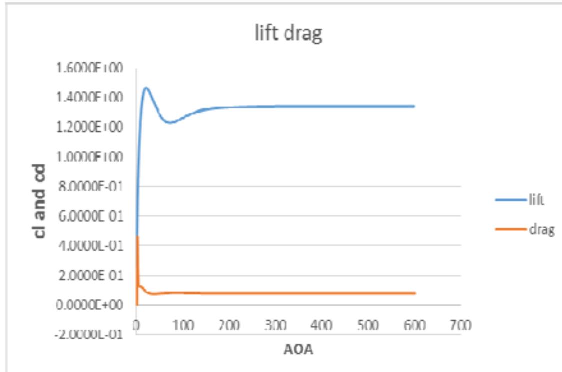

Fig.6 Convergence graph

3.5 Airfoil lift and drag

From the contours, we see that there is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper surface of airfoil. From Bernoulli equation, we know that whenever there is high velocity, we have low pressure and vice versa. Figure 7 to 10.shows the simulation outcomes of absolute pressure at angles of attack 0° to 15° with SST model. The pressure on the lower surface of the airfoil was greater than that of the incoming flow stream and as a result it effectively "pushed" the airfoil upward, normal to the incoming flow stream.

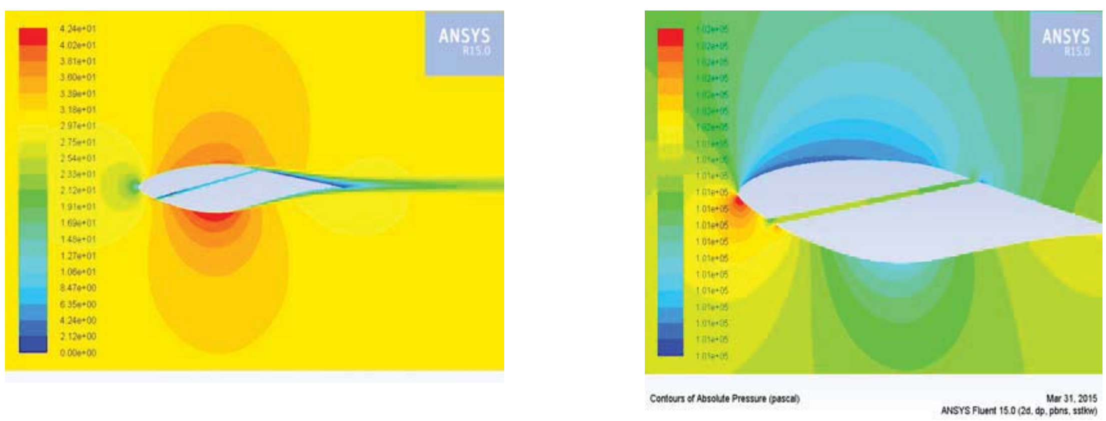

Fig 7 Absolute pressure and velocity plot at 0° Angle of attack.

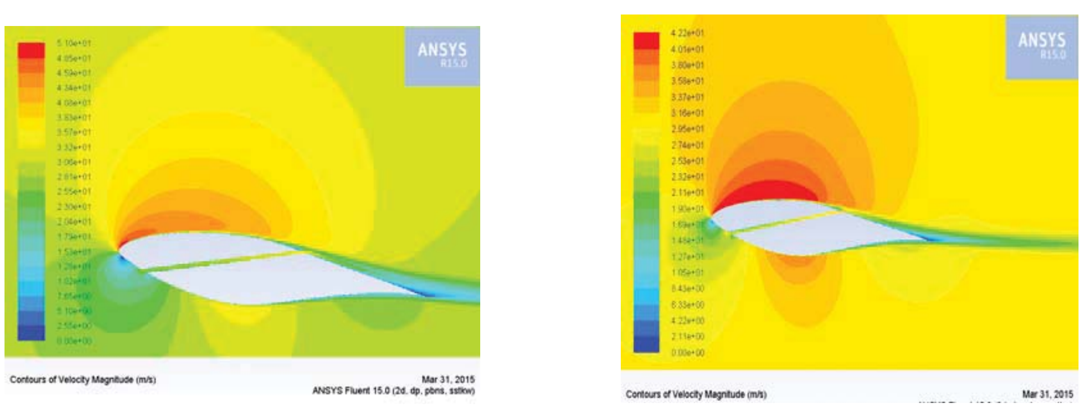

Fig.8 Absolute Velocity and pressure plot at 5° Angle of attack.

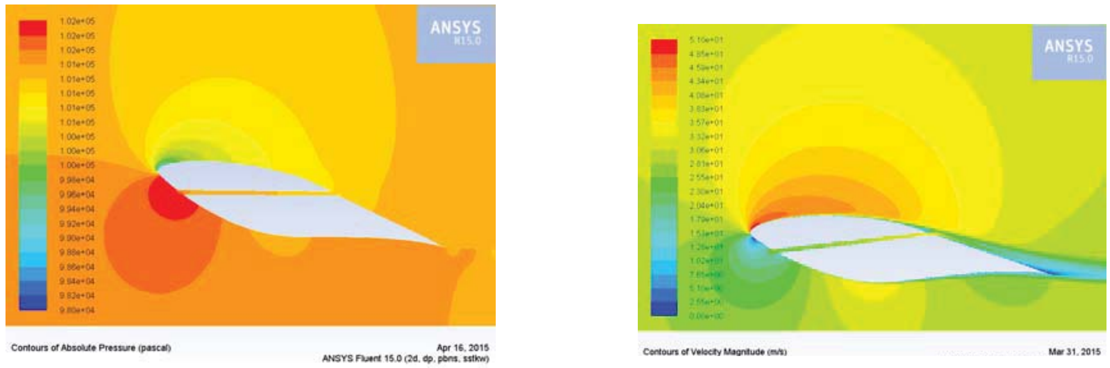

Fig.9 Absolute Velocity and pressure plot at 8° Angle of attack.

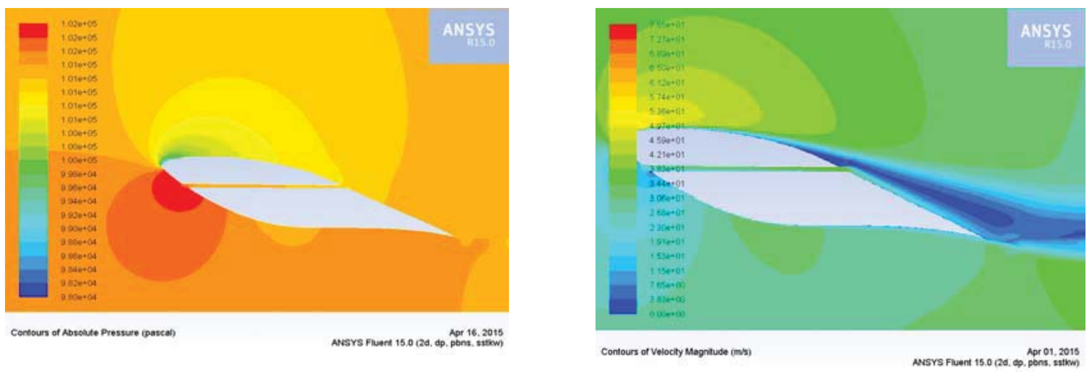

Fig.10 Absolute Velocity and pressure plot at 15° Angle of attack.

The curves of the lift and the drag coefficient are shown Fig 15 for various angles of attack, computed with SST model and compared with data 0f NACA S809 for Re = 5×10^5

IV. RESULTS

The result shows that different angle of attack different coefficient of drag and lift. Mean while with and without slot simulated as shown in table 2,3

Table 2 Coefficient of Lift (CL) for different angle attack

Alpha	airfoil without slot	Slot 3mm	Slot 2mm	Slot 3-2mm	Slot 4-3mm	Slot 4mm
α	cl	cl	cl	cl	cl	cl
0	0.131526	-0.0341927	0.0165654	0.0167011	0.00416255	-0.0411081
5	0.698463	0.563639	0.605701	0.611573	0.603797	0.562178
8	0.82786	0.900526	0.950168	0.968076	0.959731	0.893721
10	0.917101	1.09592	1.14718	1.15439	1.16909	1.08616
12	0.749814	1.28094	1.28555	1.30897	1.34055	1.26523
15	0.655195	1.12448	0.933926	1.01158	1.21533	1.26715

alpha	airfoil without slot	Slot 3mm	Slot 2mm	Slot 3-2mm	Slot 4-3mm	Slot 4mm
α	cd	cd	cd	cd	cd	cd
0	0.020264	0.028408	0.02587	0.026529	0.027496	0.02943
5	0.028114	0.034831	0.033706	0.033172	0.03487	0.036644
8	0.046977	0.046656	0.046869	0.046002	0.047752	0.049245
10	0.069841	0.060654	0.060264	0.059145	0.060734	0.063506
12	0.116154	0.07673	0.078371	0.076893	0.078006	0.081654
15	0.200691	0.154593	0.179544	0.1664	0.146197	0.14396

Table 3 Coefficient of drag (Cd) for different angle attack

Figure 15 shows that at low angles of attack, the dimensionless lift coefficient increased linearly with angle of attack. Flow was attached to the airfoil throughout this regime. At an Angle of attack of roughly 9° to12°, the flow on the upper surface of the airfoil began to separate and a condition known as stall began to develop. All three models had a good Agreement with the experimental data at angles of attack from 0˚ to 15° and the same Behavior at all angles of attack until stall. It was obvious that the SST k- ω model Turbulence model had the same behavior with the experimental data as well as after stall Angle. As angle increases above stall angle, lift coefficient decreased and the drag coefficient increased drastically. Lift coefficient and stall angle was improved effectively by using slots when compared with airfoil without slot. Also Figure 11 shows that the slotted airfoil causes significant reduction in drag coefficient at higher angle of attacks.

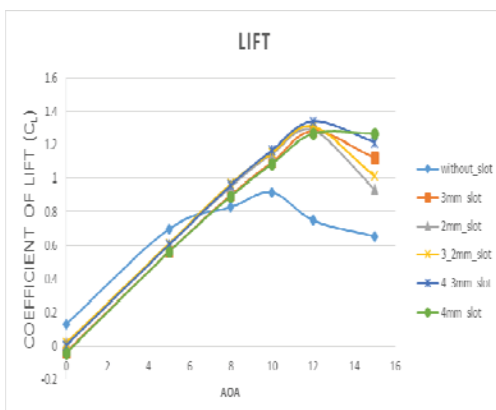

Fig 11 Difference without slot and different angle of attacks for lift

Since the actual airfoil has laminar flow over the forward half. The turbulence models cannot calculate the transition point from laminar to turbulent and consider that the boundary layer is turbulent throughout its length. From theory, the turbulent boundary layer carries more energy and C_D is much greater than at the viscous boundary layer, which carries less energy. The computational results must be compared with experimental data of a fully turbulent boundary layer. This was done only for C_D as C_L is less sensitive to the transition point.

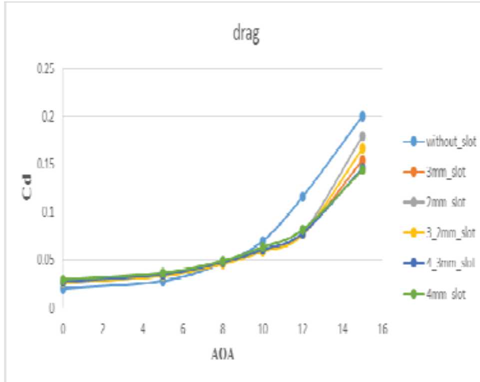

Fig.12 Difference without slot and different angle of attacks for drag

V. CONCLUSIONS

A new passive flow control method which is opening a slot in the airfoil was studied at different angle of attack. Two types of slots straight and nozzle with different ranges of width's are studied to choose the better one than the existing 3mm. The different ranges of slot width for straight are 2mm 3mm 4mm and for nozzle 3 to 2mm and 4to3mm are considered.

At high angle of attack flow separation will be predominant that causes reduction in lift and increment in drag. This passive technique work better at higher angle of attack than lower angle of attack by energizing the flow. The results from the numerical simulations have good agreement with the above physical.

(1). The results shows that the airfoil with slot performs much better than plain airfoil at higher angle of attack.
(2). Adverse effect was find at the lower angle of attack.
(3).Among all nozzle slot of 4 to 3mm width works better at the higher angle of attack.
(4). But at very high angle of attack say highly stalled angle of straight slot at 4mm is performing better than all.

REFERENCES

[1] 2DNACA0012Airfoil Validation Cases available.http://turbmodels.larc.nasa.gov/naca0012_val.html.
[2] Eleni, Douvi C., Tsavalos I. Athanasios, and Margaris P. Dionissios. "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil." Journal of Mechanical Engineering Research 4.3 (2012): 100 -111
[3] Abbott, I. H. and von Doenhoff, A. E., "Theory of Wing Sections," Dover Publications, New York, 1959
[4] Gregory, N. and O'Reilly, C. L., "Low -Speed Aerodynamic Characteristics of NACA 0012 Aerofoil Sections, including the Effects of Upper-Surface Roughness Simulation Hoar Frost," NASA R&M 3726, Jan 1970.
[5] W. J. McCroskey, "A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil." NASA Technical Memorandum 1987