Journal of Aeronautical and Automotive Engineering (JAAE)
Print ISSN: 2393-8579; Online ISSN: 2393-8587; Volume 1, Number 1; September, 2014 pp. 9-12
© Krishi Sanskriti Publications
http://www.krishisanskriti.org/jaae.html

Numerical Simulation of Effect of Lobed Nozzle on Temperature Mixing Using Large Eddy Simulation

Devi Archana Das

*Amity Institute of Space Science and Technology, Amity University,
Sector-125, Noida, Uttar Pradesh-201303, India*

***Abstract:** A numerical study has been conducted to analyze the effect of nozzle exit geometry (lobed nozzle) on temperature mixing at the wake of jet. A lobed nozzle is a plate with corrugated edges and facilitates efficient mixing of co-flow streams. It accelerates the flow field and involves faster spreading of the jet. A three-dimensional flow field is produced by a subsonic jet. The computational study is conducted using the turbulence model Large Eddy Simulation (LES). The numerical model consists of a jet from the exit of a circular nozzle and a lobed nozzle (6 lobes). The exit area of both the nozzle exit geometry is kept equal for accurate analysis. A comparative study is taken into account to clearly depict the difference in temperature mixing, variation in turbulence and formation of vortical structures for the two different exit geometries. The velocity magnitude, static temperature and vorticity magnitude contours for the circular and lobed jet nozzle are produced and discussed. The results reveal the difference in the diffusion rates of the two nozzles. Lobed nozzle diffuses at a higher rate than that of circular nozzle. Design optimization of the lobed nozzle would lead to higher rates of efficient temperature mixing. The Large Eddy Simulation (LES) model is found appropriate to simulate the flow field for analysis of turbulent structure formation. The complete modeling and meshing has been carried out in ANSYS Workbench. For accurate predictions of the flow field, the flow has been simulated with the help of FLUENT.*

***Keywords:** lobed nozzle, Large Eddy Simulation (LES), vortices formation, turbulence, jet*

1. INTRODUCTION

A lobed nozzle is designed with a splitter plate which consists of a convoluted trailing edge. It is a device which ensures efficient mixing of co-flow streams. It has found implementation as a vortex generator in the recent times, amongst industry and academic researchers. In engineering flows, it is an established fact that the mixing process is governed by the rate of mixing during combustion in the combustion chamber. The efficiency of a jet flow to mix with the ambient flow has a significant impact on combustion efficiency. Lobed nozzles have found application in reduction of jet noise, improvement of propulsive efficiency and reduction of Specific Fuel Consumption (SFC). A number of investigations have been performed experimentally and numerically to validate the above data. Researchers have carried out a number of experiments to demonstrate efficient mixing in a lobed nozzle with the help of geometry optimization. The rate of spreading of jet was found to be high in all the cases. The techniques used for validation were Particle Image Velocimetry (PIV) and Laser Induced Fluorescence (LIF) flow visualization techniques. There have been few experiments that have been performed numerically using Computational Fluid Dynamics (CFD) due to the issues related to setting up of the problem statement.

Lobed geometry at the nozzle exit introduces an efficient mixing of the fluid flow at various velocity, pressure and temperature conditions. It has been observed that there is an increase in the interfacial area in a lobed geometry. Paterson (1982) revealed on the basis of various measurements, the existence of large scale stream-wise vortices in mixing of co-flows in lobed nozzle. These have been introduced due to the lobed geometry. The formation of these large-scale vortices is due to the presence of the splitter plate in a lobed nozzle.

To analyze the physical characteristics of a turbulent flow, the numerical simulation must be based on the Navier –Stokes equations. These equations take into account the convective and viscous mechanisms effects as well as thermal effects. There are basically three approaches for turbulent flow analysis, namely Direct Numerical Simulation (DNS), Solution of Reynolds Averaged Navier-Stokes equations (RANS), and Large Eddy Simulation (LES). The LES approach is a good compromise between the DNS and the RANS approaches. In LES, the large-scale eddies are computed numerically whereas the small scales are modeled in the code.

For reduction of thermal signature of the military aircraft, to improve their survivability, lobed nozzles are being used for enhancement of the mixing process. The high velocity gas plume from the exhaust nozzle would diffuse at a higher rate in a lobed nozzle than in a circular nozzle. Therefore, this project was prompted to analyze the diffusion rate of the different nozzle configuration with the help of velocity, static temperature and vorticity contours.

The project is motivated due to the fact that few investigations have been conducted in this field. So, a CFD analysis would provide a better picture of the flow field. Thus numerical analysis of 3D flow field is carried out without mixing of streams. The analysis is focused only on the diffusion rate of various nozzle exit geometries i.e. circular nozzle and lobed nozzle. The obtained results were used to provide a comparison between the different exit nozzles.. These results are found to be in agreement with the experimental investigations through various flow visualization techniques.

2. MATHEMATICAL FORMULATION

The lobed nozzle used for analysis is a six (6) lobed nozzle. It is designed using **ANSYS Workbench 14.0**. A constant area is maintained for the lobed nozzle and the circular nozzle at a value of approximately 14 m^2.The domain is kept large enough for the flow field to develop completely. The problem setup uses a pressure based solver and transient time. The viscous flow turbulent model used is Large Eddy Simulation (LES). Velocity inlet conditions are defined at the exit of the jet for a subsonic flow. The pressure outlet is maintained at a value little higher than the initial static pressure. The temperature at the inlet of the nozzle is kept at 600K to analyze the temperature diffusion for different exit geometry.

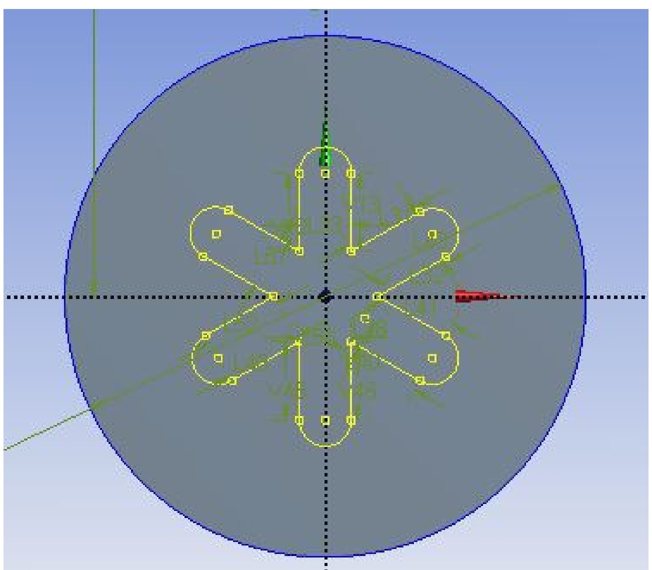

Figure 1: Lobed nozzle design

The grid generated is a structured mesh. For obtain an accurate analysis of the flow field, a fine mesh is generated in the jet flow region as shown in Figure 4. The number of mesh cells generated for lobed jet nozzle is 3754290 cells while that for a circular nozzle is 6148382 cells. The mesh is also generated using **ANSYS Workbench 14.0**. The length of the domain is limited to few hundred meters due to the limitation of computational power for grid generation. The flow time would be high to analyze a complex grid; hence the domain analysis was kept long enough to show comparison between the two flow fields of circular and lobed nozzle.

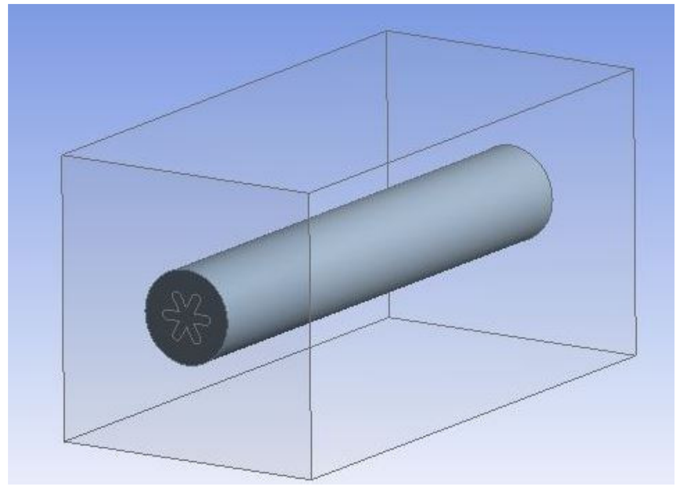

Figure 2: Analysis Domain

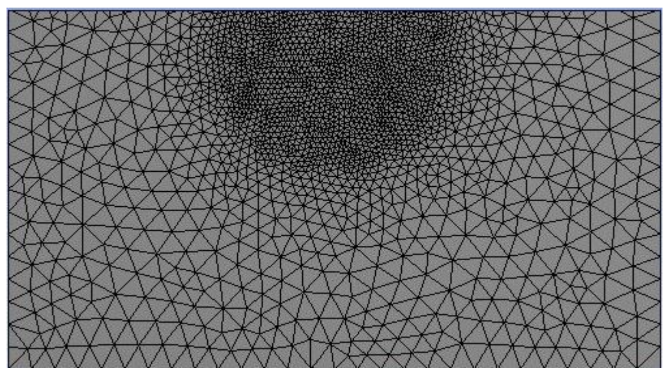

Figure 3: Triangular mesh generated

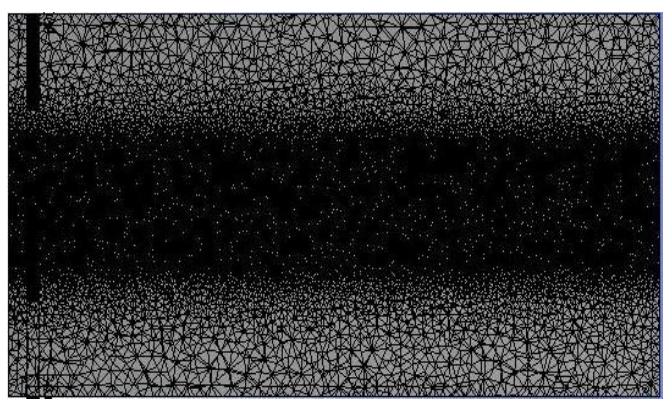

Figure 4: Fine mesh grid at the Jet flow region

The solution method used for this numerical simulation is Simple. The spatial derivatives can be approximated dimension by dimension. The spatial discretization scheme for gradient is Green gauss Node Based, the pressure and momentum being second order upwind and the transient formulation is also considered as second order implicit. The time step size is 1 * 10 ^-1 and the flow time is 3.5 seconds. The flow time allowed is sufficient enough for the flow to develop and provide data similar to real time data.

3. RESULTS AND DISCUSSIONS

Velocity field

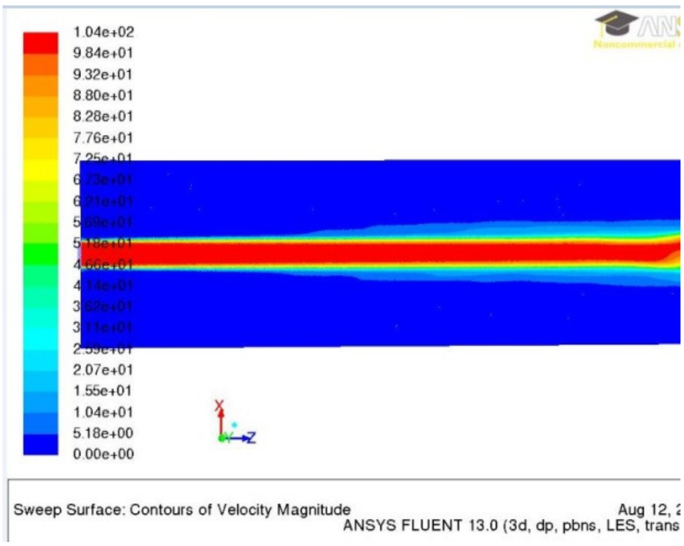

Figure 5: Contours of velocity for circular nozzle

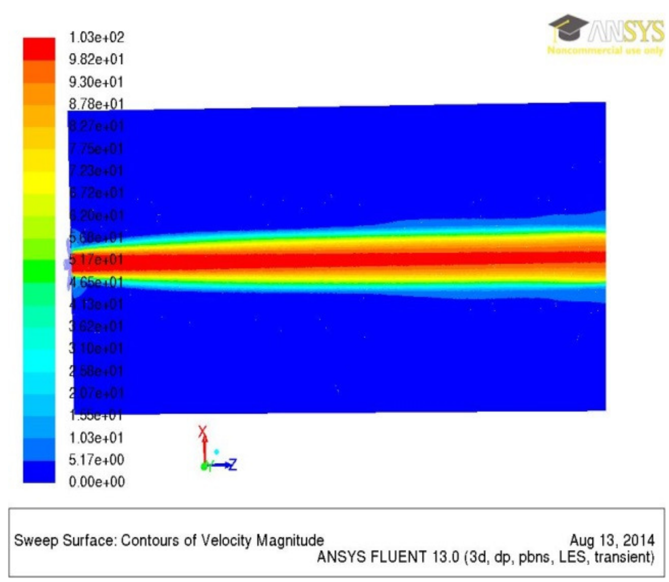

Figure 6: Contours of velocity for lobed nozzle

As per Figure 5 through 6, it can be observed that the circular nozzle has a steady flow till the end of the analysis domain, thus denoting that the flow has not accelerated for the give time of flow simulation. However, in the case of the lobed nozzle, the core is seen to change from red to orange, and the lobe jet troughs are seen to be blue, denoting acceleration of flow and increase in turbulence.

This also indicates that after a certain period of time of flow simulation, the flow would form vortical structures and thus enhancing the process of mixing, while in a circular nozzle the flow would have just started accelerating. The vortex interaction of primary and secondary vortices leads to efficient mixing in the lobed nozzle. The flow time was analyzed for a value of 3.5 seconds due to limitation of computational power. However, a clear difference between both the nozzle geometries can be observed in the above figure.

Vorticity Field

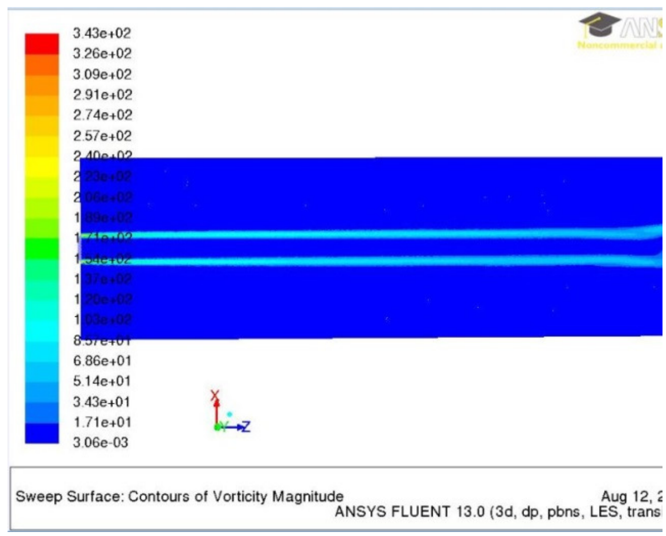

Figure 7: Contours of vorticity for circular nozzle

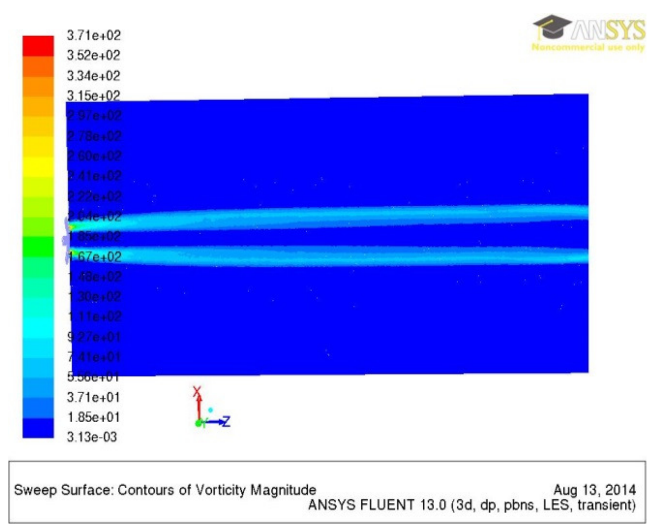

Figure 8: Contours of vorticity for lobed nozzle

The vorticity contour plots clearly depict the turbulence in a lobed nozzle. The turbulence intensity in a lobed nozzle is much higher than in a circular nozzle. The flow field accelerates during the initial stages of the flow thus enhancing the three stages of flow field development: laminar, transition and turbulence in a much lesser time period than a circular nozzle. The lobed nozzle creates perturbations in the flow field, thus leading to enhanced and efficient mixing.

This also throws light upon the fact that lobed nozzles have a shorter laminar flow field than circular nozzle. The prolonged effect of the stream wise vortices generated by the lobed nozzle on the secondary vortices leads to the enhanced energy mixing process of turbulence. They have smaller turbulent structures but large turbulent intensity. . The interaction of this flow field with the ambient air accelerates the mixing process.

Temperature field

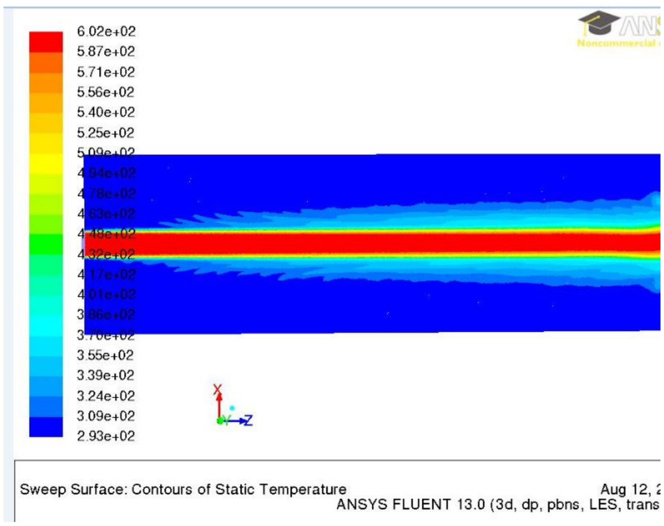

Figure 9: Contours of temperature for circular nozzle

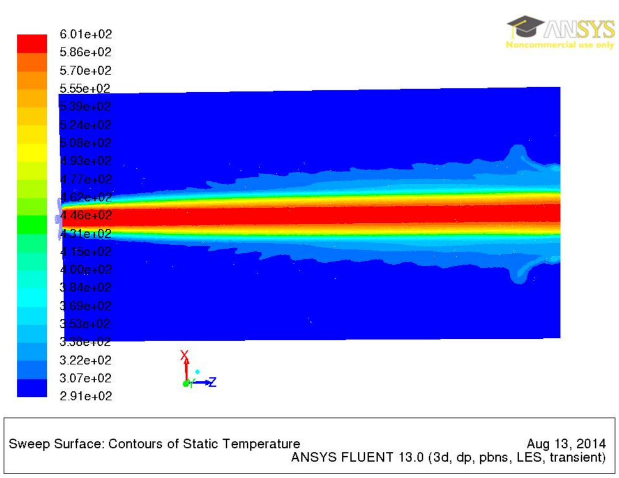

Figure 10: Contours of temperature for lobed nozzle

The temperature core of the Figure 9 through 10 shows a marked difference. For the lobed nozzle, the temperature has started to diffuse and the turbulence can be seen through the blue patches in Figure 10, which is a zoomed version to denote the turbulent characteristics clearly. As seen in Figure 9, the core is red in color and the flow field is not as turbulent as that of the lobed nozzle. The quick diffusion of the temperature flow field would suppress the thermal signature of the aircraft. Moreover, it would lead to a reduction in the intensity of heated parts of the aircraft surrounding the exhaust nozzle. The flow investigation could be modified by introducing ambient air through the sides, to observe the process of enhanced mixing.

4. CONCLUSION

The numerical simulation is found to agree with the experimental investigations conducted by few researchers in the view of temperature mixing. The lobed nozzle has a higher rate of diffusion than a circular nozzle. A circular nozzle is observed to take a larger flow time to accelerate and provide the same amount of turbulence as that of a lobed nozzle. The acceleration of the flow field in a lobed nozzle without any co-stream denotes the high level of turbulence due to formation of the vortical structures. Increased number of lobes optimized to reduce friction, would lead to better results, which could be applied to military aircraft. One of the most important attributes of a lobed nozzle is the jet noise reduction which has been numerically and experimentally verified by few research investigators. The future work aims at analyzing the efficient mixing with the admission of ambient air stream from the sides of the nozzle. The lobed nozzles are being used in the industry to make use of the various advantages, it provides to the aerospace industry.

REFERENCES

[1] Devi Archana Das, Nishant Madanagopal, " Numerical Simulation of Acoustic Performance of a Lobed Nozzle", 4D International Journal of management and Science, Volume-4, Issue -2, July 2014

[2] Hu hui, Toshio Kobayashi, Tatsuo Saga, Nobuyuki Taniguchi, Sigeka Segawa and Akiro Ono, "The Mixing Enhancement Performance of Lobed Nozzles by using PIV and LIF", Proceedings of FEDSM'98, June 21-25, 1988

[3] Vaclav Dvorak, "Study of Optimization of Lobed Nozzle for Mixing", Colloquim FLUID DYNAMICS 2007, October 24-26, 2007