Proceedings of the 48th National Conference on Fluid Mechanics and Fluid Power (FMFP)
December 27-29, 2021, BITS Pilani, Pilani Campus, RJ, India.

FMFP2021–04–022

Turbulent Characteristics of Compressible Jet from a Chevron Nozzle: A numerical study

Shreyas Kotian, Nishant Jain, Nachiket Methekar, and Shailesh Nikam

Department of Mechanical Engineering, K.J. Somaiya College of Engineering, Mumbai- 400 077, India.

ABSTRACT

The aim of the present study is to computationally investigate the turbulent characteristics of a high-speed jet at exit Mach 0.8 from a chevron nozzle. A Chevron nozzle with a chevron crest angle of 45° is used and flow characteristics are investigated and compared with base nozzles at two different axial locations from nozzle exit; x/D = 0.5 and x/D = 5. Contours of turbulent intensity suggest that the chevron helps in the reduction of turbulence downstream of the potential core, which results in lowering jet noise. Turbulence is reduced by the enhanced mixing between the two fluids which is mainly due to the formation of a pair of vortices rotating in opposite directions which is evident in the vorticity contours. The development of a counter-rotating pair of vortices results in the formation of lobed structures in the flow field close to the chevron nozzle exit. Similar results can be observed for turbulent intensity at the lip line as well as the centerline. The baseline nozzle causes more turbulence than the chevron nozzle thus increasing the sound level.

Keywords: Numerical simulation, Chevron nozzle, turbulent characteristics, Compressible jet.

1.

1. INTRODUCTION

In recent years, the deployment of chevrons at the end of nozzles has gained prominence because it acts as a passive control that helps in reducing jet engine noise. Through experiments and numerical simulations, researchers have established the effectiveness of using chevrons to reduce engine noise. They are basically serrated edge structures attached to the lip of the nozzle forming petal-like structures. The flow after passing through the chevrons forms a pair of vortices rotating in the opposite direction which enhances the mixing of the hot and cold stream which helps in the reduction of turbulence and thereby a source of potential noise. Detailed study of literature shows that researchers have used experiments and computer simulations as an effective tool in analyzing flow characteristics. Various researchers through computational analysis and experimental investigations have studied these parameters. Hussain and Clark [2] using numerical simulations and experimental observations predicted the effect of turbulent characteristics of the initial boundary layer on the development of flow in a jet. Four different conditions were analyzed. It was observed that when the flow is laminar when the mass flux and turbulent intensities are higher than when the flow is turbulent. It was also found that the nondimensional entrainment rates have a higher value downstream of the jet exit for the laminar initial boundary layer cases. Ko and Davies [3] studied the near pressure field within the potential core of a subsonic circular jet. It was found that up to a distance of 4.5D, the fluid stream was in phase with the velocity of sound. The difference in pressure field characteristics was caused due to the formation of vortices at low velocity and eddies at higher velocities. The dominant components of the flow are a function of the Strouhal number and also depend on the axial and radial positions. Subramanian et al. [4] studied the effect of the mixing process of an incompressible jet from a convergent nozzle using experiments. Results were compared of three different nozzles- a baseline nozzle, a convergent chevron nozzle, and a tabbed chevron nozzle. It was found that using a tabbed chevron nozzle over a convergent chevron nozzle promotes the formation of vortex structures which develops into enhanced entrainment. The consistency of streamwise structures was an important parameter in determining its mixing characteristics. It was also concluded that the presence of a confining tube causes a faster decay of the centerline velocity. Raman et al. [5] through experimental studies analyzed the effect of turbulence characteristics on the development of a jet and discrete tone excitation. Turbulence intensity was changed between 0.15-5%. It was reported that the natural jet decay remains unaffected by the varying levels of turbulence. For a fixed value of tonal excitement, increasing the turbulence levels damped out the instability wave. It was also observed that the initial boundary layer had a profound effect on the evolution of the jet as well as its excitability. Callender and Gutmark [6] evaluated the far-field acoustic characteristics of a chevron nozzle. Experiments were conducted for a baseline nozzle and three different chevrons with varying geometries over a range of flow conditions. It was observed that the presence of chevrons decreases the noise levels with a reduction of 3-6 dB being reported. The effectiveness of chevrons is largely dependent on the engine operating point. An increased number of lobes affects the high-frequency effects of the chevron. Brown and Bridges [7] investigated the link between azimuthal modes in jet noise and the acoustic efficiency with the help of chevron nozzles. A baseline nozzle and a nozzle with 3 and 4 primary chevrons were experimented with cold and hot jets to study the impact of chevrons on the sound field. Chevrons largely impact the mean axial velocity but remotely affect the fluctuating axial velocity. Comparison of sound levels of a chevron nozzle with a baseline nozzle reveals the fact that chevrons help in the

S. Bhattacharyya and H. Chattopadhyay (eds.), *Fluid Mechanics and Fluid Power (Vol. 1)*,
Lecture Notes in Mechanical Engineering, https://doi.org/10.1007/978-981-19-7055-9_14

reduction of sound levels. Engel et al. [8] using computational analysis tried to predict the acoustic characteristic of a chevron nozzle using the RANS-based method. Two chevrons with different penetration angles were numerically investigated for M= 0.9 and Re = 1,380,000. Comparison of numerical results with experimental data yielded the fact that the RANS method can be safely used to predict the acoustic characteristics of a chevron nozzle. Koch et al. [9] conducted experiments and numerical simulations on a chevron nozzle with alternating patterns. Experiments were performed on the imported model from the NASA Glenn Research Center. WIND/MGBK codes were written to predict the trends in the reduction of noise for the baseline nozzle and the nozzle with chevrons. The numerical results were in good agreement with the experimental data available and further work is proposed to use this set of codes for the development of quieter turbofan engines.

2. GEOMETRY OF CHEVRON

In the present study chevron nozzle with a crest angle of 45° has been used. The geometries have been adopted from Nikam et al. [1]. **Fig. 1**, **Fig. 2** and **Fig. 3** shows the geometry of the chevron used for the study. Flow condition at the nozzle exit is maintained as Mach 0.8.

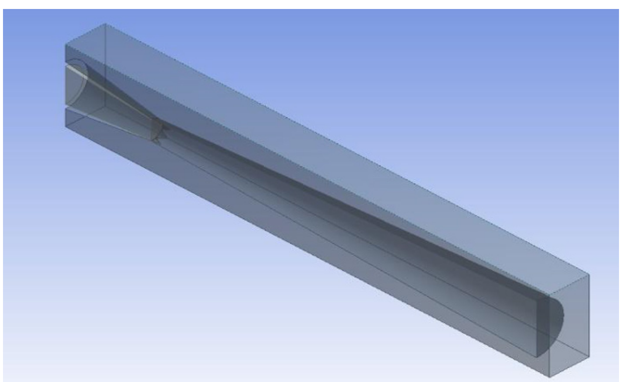

Figure 1: Computational domain of the chevron nozzle

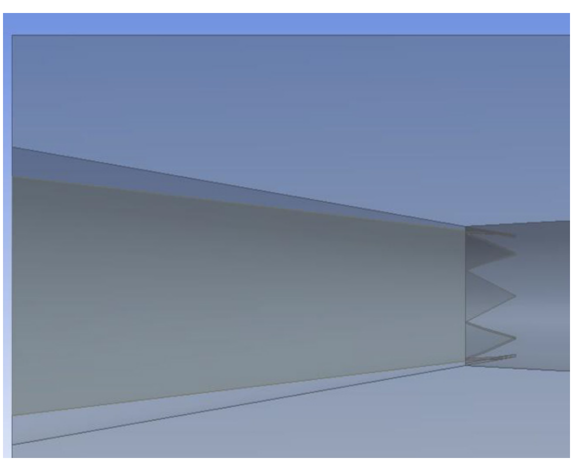

Figure 2: Magnified view of the chevron nozzle.

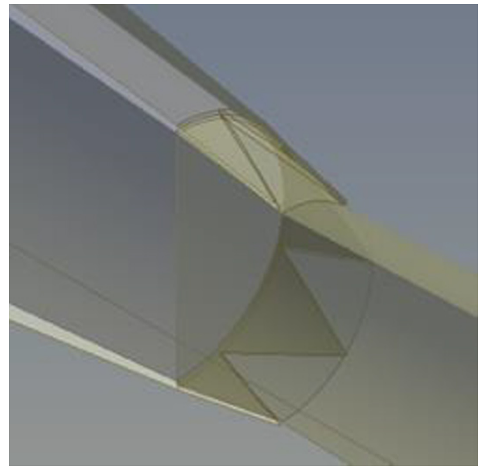

Figure 3: Close-up view of the chevrons.

3. CFD METHODOLOGY

In the current investigation, the geometry of the model was displayed utilizing SOLIDWORKS, and meshing was finished utilizing ANSYS 2020 R1. Three unique models, with varying geometry (different crest angles of chevron), were researched. The schematics of the models used are shown in Fig. The solver used is a pressure-based solver in steady-state. The fluid used is air where its density, specific heat, and thermal conductivity are assumed to be constant and viscosity is Sutherland. The molecular weight of air is assumed to be 28.966 kg/kmol.

Mesh Topology

The ANSYS Meshing tool was being used to mesh the data. For the simulations, linear element order was used with a global mesh size of 20 mm. The fluid domain inside the nozzle was meshed by the Patch conforming method. Body sizing was used for the domain starting from the inlet of the nozzle to the end of the domain for which analysis is carried out. Edge sizing is also used to refine the mesh at the boundaries and bifurcation, increasing the degrees of freedom in the areas of interest. The final mesh size consisted of 203683 nodes and 941796 elements (**Fig. 4**).

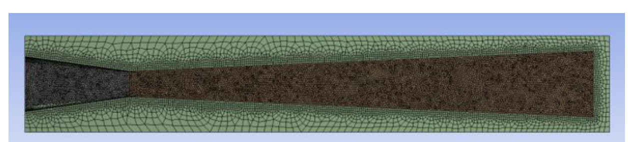

Figure 4: Final mesh of the chevron nozzle.

Governing Equations

A pressure-based solver with coupled scheme along with a Least Square cell-based gradient was used for the simulations. SST k-ω turbulence model was used in the study, considering its stability and precision. Reynolds Averaged Navier Stokes equations (RANS) are used in the present study.

Boundary Conditions

Limit conditions were set like that of Nikam et al. [1]. Pressure inlet was specified with a total gauge pressure of