NUMERICAL MODELLING ON SUPersonic fluid flow past a cavity with spoiler for investigations on pressure field and OASPL

Dr. Nirmal Kumar Kund
Associate Professor, Department of Production Engineering
Veer Surendra Sai University of Technology, Burla 768018, India

Abstract—The current study implicates the development of the proper numerical model be relevant to the supersonic flow past a 3D open cavity with a length-to-depth ratio of 2. The Mach number of the supersonic free-stream is 2 in addition the Reynolds number of the flow is $10^5$. The numerical simulations have been executed by deploying Large Eddy Simulation (LES) technique. The Smagorinsky model is included for this purpose. The simulation results have been portrayed in the shape of pressure field and overall sound pressure level at the centreline of the front wall of the open cavity. As pragmatic, the feedback loop mechanism illuminates the trend of flow field of the open cavity very nicely. Very large pressure fluctuations are also detected inside the open cavity without spoiler. Yet, the reduction of the pressure fluctuations inside the open cavity is attained by integrating a spoiler in the form of one-fourth of a cylinder at the leading edge of the open cavity. Thus, the pressure less than the free-stream pressure is attained inside the open cavity. With the integration of the spoiler, the overall sound pressure level at the centreline of the front wall of the open cavity is also suppressed by some amount. Congruently, the sound pressure level is suppressed by almost 16 dB at the front wall and around 12 dB at the aft wall of the stated open cavity. The vicissitudes in the flow characteristics of the open cavity integrating the spoiler is also explored. In general, the comparisons between the numerical predictions of the cavity flows with and without the integration of the spoiler is also accomplished.

Keywords—Numerical Simulation, Open Cavity, Spoiler, LES, Pressure Field, OASPL.

I. INTRODUCTION

Very huge noise is witnessed in our day-to-day life originating from exhaust pipes, vacuum cleaners, ventilation systems, fans etc. The flow induced noise generation is the greatest ever challenge in ample engineering usages relating to all kind of automotive sectors like surface transport, aviation and marine applications. Airframe noise is resulting from very large pressure fluctuations. One such prime airframe noise is the cavity noise as experienced in open wheel wells along with the weapon bays of the aircrafts in addition to door gaps, side mirrors together with the open sun roof of automobiles. The wheel wells, weapon bays and door gaps may be modelled as rectangular cavities and the cool, calm and collected flow outside the cavity may be taken to be smooth and uniform. Even though the rectangular cavity is simple in shape, it is very rich in diverse dynamic and acoustic occurrences, indeed surrounded by an aeroacoustic feedback loop subject to the shape/size of the cavity along with the flow circumstances. Very severe tone noises may be engendered down to the vortex shredding at the upstream edge of the cavity during the flow over a cavity.

The flow-induced pressure oscillations in shallow cavities are described by Heller et al. [1]. The investigations on the tones and pressure oscillations induced by flow over rectangular cavities are carried out by Tam and Block [2]. Mach 0.6 to 3.0 flows over rectangular cavities are performed by
Kaufman et al. [3]. The high resolution schemes are used by Sweby [4] in applying flux limiters on hyperbolic conservation laws. The numerical simulation on supersonic flow over a 3D cavity are reported by Rizzetta [5]. The very fundamentals of computational fluid dynamics is demonstrated by Anderson and Wendt [6]. The achievements and challenges of large-eddy simulation are described by Piomelli [7]. The numerical simulations of fluidic control for transonic cavity flows are carried out by Hamed et al. [8]. The LES studies on feedback-loop mechanism of supersonic open cavity flows are conducted by Li et al. [9]. A validation study on unsteady RANS computations of supersonic flow over 2D cavity is done by Vijayakrishnan [10]. The lid-driven cavity flows of viscoelastic liquids are very well illustrated by Sousa et al. [11]. The experimental investigations on double-cavity flows are studied by Tuerke et al. [12]. It is pragmatic that an extensive study on cavity flow has been accomplished both experimentally and numerically for increasing the aerodynamic adeptness. Still, aside from its standing, the intricate flow physics of flow over a cavity has spellbound the investigators around the globe for further researches and yet occupies a very frontier area of research.

II. OBJECTIVES

Even if, very extensive experiments have been executed to overcome the pressure fluctuations within the cavity, still, more than a few are not just as effective for all flow circumstances. The ratings of the control devices are considerably influenced by the Mach number. Control devices intended to be fabricated so that they perform over a wide range of Mach numbers. The incoming boundary layer is also another vital factor which controls the operation of the control devices. Keeping this stance in attention, the motivation of this investigation is to study the flow occurrence in a 3D open cavity supersonic flow. Furthermore, the reduction of cavity fluctuations by passive technique has been examined by keeping spoiler at the leading edge of the open cavity. The comparison between the open cavity flows with and without the integration of spoiler has also been accomplished. In general, the present researches pertain to the establishment of a three-dimensional numerical model for obtaining comparative numerical results of the open cavity flows in terms of pressure field and overall sound pressure level (OASPL) at the centreline of the front wall of the open cavity with and without the integration of the spoilers.

III. DESCRIPTION OF PHYSICAL PROBLEM

Supersonic flow past a three-dimensional cavity is studied numerically. The streamwise length, depth and spanwise length of the cavity are 20 mm, 10 mm, and 10 mm, respectively. The length-to-depth ratio (L/D) for the cavity is 2. The width-to-depth ratio (W/D) is 1. The cavity is three-dimensional with streamwise length-to-spanwise length ratio (L/W) > 1. In addition, the Mach number of the free-stream along with the Reynolds number based on the cavity depth are taken as 2 and 10^5, respectively, for setting the inflow conditions.

3.1. Geometric model

The geometric domain of the cavity with the spoiler is shown in figure 1. The size of this domain, as stated previously, is 2D×D×D (length × depth × width). The spoiler at the cavity leading edge has a dimension of 0.6D. One-fourth part of a cylinder has been used for the shape of the spoiler. The domain is three-dimensional with L/W > 1. The upper boundary is at a distance of 4D above the cavity to ensure that no reflected shock affects the flow features inside the cavity.

3.2. Initial and boundary conditions

The initial conditions for the cavity involving supersonic flow are Mach number = 2 with \( P_\infty = 101.325 \text{ kPa} \) and \( T_\infty = 300 \text{ K} \) at the inlet, Reynolds number of the flow being \( 10^5 \), based on the cavity depth.
No-slip adiabatic wall boundary conditions is applied at the wall boundaries. Zero-gradient condition is applied at all the outflow boundaries. Periodical boundary condition is applied in the spanwise direction of the cavity. No-slip adiabatic wall boundary condition is employed for the spoiler.

IV. MATHEMATICAL FORMULATION

4.1. Generalized governing transport equations

The 3D compressible Navier-Stokes equations are the governing equations which comprise the continuity equation (1), the momentum equation (2), and the energy equation (3) which are as mentioned below:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \]  
\[ \frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) - \nabla \cdot (\mu \mathbf{U}) = -\nabla p \]  
\[ \frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{U}) - \nabla \cdot (\rho \mu e) = -\rho \nabla \cdot \mathbf{U} + \rho \left[ \frac{1}{2} (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) \right]^2 \]

Where,  
\( \mathbf{U} = \) velocity vector = \( u \mathbf{i} + v \mathbf{j} + w \mathbf{k} \)  
\( \frac{1}{2} (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) = \) strain rate tensor.

The equations (1), (2) and (3) symbolise the conservation form of the Navier-Stokes equations. The conservation form of these governing equations are reached from a flow model fixed in space [6]. The stated equations are relevant to viscous flow, except that the mass diffusion is not involved. It is supposed, in aerodynamics, that the gas is a perfect gas. The equation of state for a perfect gas is,  
\[ p = \rho R T \]  
Where, \( R = \) specific gas constant = \( C_p - C_v \)

For a calorically perfect gas (constant specific heats), the caloric equation of state is,  
\[ e = \text{internal energy per unit mass} = C_v T \]
4.2. LES Turbulence Modelling

The turbulent flows may be simulated applying three different methods: Reynolds-Averaged Navier-Stokes equations (RANS), direct numerical simulation (DNS), and large eddy simulation (LES). Direct numerical simulation has high computational necessities. DNS resolves all the scales of motion and for this it desires a number of grid points proportional to \((Re)^{9/4}\) and computational scales’ cost is proportional to \((Re)^3\) [7].

In the current investigation, behaviours of the turbulent flow field have been simulated applying LES as it is suitable for unsteady complex flows and noise induced flows. LES computes the large resolved scales and also models the smallest scales. The turbulence model is incorporated by dividing the time and space varying flow parameters into two components, the resolved one \(f\) and \(f’\), the unresolved portion:

\[
(x, t) = f(x, t) + f'(x, t)
\]

(7)

LES uses a filtering operation to separate these resolved scales from the unresolved scales. The filtered parameter is represented by an over bar [7]. The top-hat filter smooth both the fluctuations of the large-scale as well as those of small scales. The filtering operation whenever introduced to the Navier-Stokes equation, it results in:

\[
\frac{\partial \overline{p}}{\partial t} + \nabla \cdot (\overline{\rho \overline{U}}) = 0
\]

(8)

\[
\frac{\partial (\overline{\rho \overline{U}})}{\partial t} + \nabla \cdot (\overline{\rho \overline{U} \overline{U}}) - \nabla \cdot (\overline{\mu \overline{U}}) = -\nabla \overline{\rho}
\]

(9)

\[
\frac{\partial (\overline{\rho \overline{U}})}{\partial t} + \nabla \cdot (\overline{\rho \overline{U} \overline{U}}) - \nabla \cdot (\overline{\mu \overline{U}}) = -\rho \overline{\nabla \cdot \overline{U}} + \mu \left[ \frac{1}{2} (\nabla \overline{U} + \nabla \overline{U}^T) \right]^2
\]

(10)

Nevertheless, the dissipative scales of motion are rectified poorly by LES. In a turbulent flow, the energy from the large resolved structures are passed on to the smaller unresolved structures by an inertial and an effective inviscid mechanism. This is called as energy cascade. Therefore, LES employs a sub-grid scale model to mimic the drain pertaining to this energy cascade. Most of these models are eddy viscosity models connecting the subgrid-scale stresses \((\tau_{ij})\) and the resolved-scale rate of strain-tensor \((\overline{S}_{ij})\),

\[
\tau_{ij} - (\delta_{ij}/3) = -2\nu\overline{S}_{ij}
\]

(11)

Where, \(\overline{S}_{ij}\) is the resolved-scale rate of strain tensor \(=(\partial \overline{u}_i/\partial x_j + \partial \overline{u}_j/\partial x_i)/2\). In most of the circumstances it is supposed that all the energy received by the unresolved-scales are dissipated instantly. This is the equilibrium assumption, i.e., the small-scales are in equilibrium [7]. This simplifies the problem to a great extent and an algebraic model is found for the eddy viscosity:

\[
\mu_{sgs} = \rho C \Delta^2 |\overline{S}| \overline{S}_{ij}, |\overline{S}| = (2\overline{S}_{ij}\overline{S}_{ij})^{1/2}
\]

(12)

Here, \(\Delta\) is the grid size and is generally considered to be the cube root of the cell volume [7]. This model is termed as the Smagorinsky model and \(C\) is the Smagorinsky coefficient. In the current investigation, its value has been considered to be 0.2.

V. NUMERICAL PROCEDURES

5.1. Numerical scheme and solution algorithm

The 3D compressible Navier-Stokes governing transport equations are discretized over an outline concerning finite volume method (FVM) through the SIMPLER algorithm. Here, the turbulent model utilized for large eddy simulation is Smagorinsky model, on account of its minimalism. The spatial
derivatives like Laplacian and convective terms are computed by second order scheme based on Gauss theorem. Furthermore, the viscous terms are calculated by second order scheme. Additionally, the implicit second order scheme is utilized for time integration. The numerical fluxes are calculated by using Sweby limiter in central differencing (CD) scheme, which is a total variation diminishing (TVD) scheme. The central differencing (CD) is an unbounded second order scheme, while, the total variation diminishing (TVD) is a limited linear scheme. The developed solver is utilized to predict flow behaviours of the related flow variables pertaining to supersonic flow over an open cavity.

5.2. Choice of grid size, time step and convergence criteria
The computational domain is again decomposed into upper cavity and inside cavity region as illustrated in figure 2. The grid is refined at the regions near to the wall (where very high gradient is expected) to determine the behaviour of shear layer satisfactorily. A comprehensive grid-independence test is performed to establish a suitable spatial discretization, and the levels of iteration convergence criteria to be used. As an outcome of this test, the optimum number of grid points used for the final simulation, in the upper cavity region as 360 × 150 × 1 and those of in the inside cavity region as 200 × 150 × 1. The grid spacing at the leading edge of the cavity denoted as $\Delta X^+$, $\Delta Y^+$ and $\Delta Z^+$ being 5.0, 12.5, and 1.0, respectively. However, the total number of grid points is 81000 for this cavity. Corresponding time step chosen in the numerical simulation is $10^{-6}$ seconds. Even though, it is tested with smaller grids of 128000 in numbers, it is witnessed that a finer grid structure does not change the results considerably.

The convergence in inner iterations is confirmed only when the condition $\left| \frac{\phi - \phi_{old}}{\phi_{max}} \right| \leq 10^{-4}$ is fulfilled concurrently for all variables, where $\phi$ represents the field variable at a grid point at the current iteration level, $\phi_{old}$ stands for the corresponding value at the previous iteration level, and $\phi_{max}$ is the maximum value of the variable at the current iteration level in the whole domain.

![Figure 2. Computational grid of cavity with spoiler in X-Y Plane](image-url)
VI. RESULTS AND DISCUSSIONS

6.1. Comparisons of pressure distributions with and without spoiler

Figure 3 shows the pressure fields, at an instant of time $t = 0.1$ sec, for supersonic flow past an open cavity with and without the usage of spoiler. The cavity flow characteristics with spoiler is very different from that of the cavity flow without spoiler. The disparity in the flow fields may certainly be revealed from the pressure fields. One shedding vortex is noticed in the cavity with spoiler against two shedding vortices witnessed in the cavity without spoiler. Mostly, the pressure inside the stated open cavity is less than that of the free stream pressure. The recirculation regimes for both the above-mentioned cavities are reasonably different from one another. A shock is observed to be reflected from the upper boundary, nevertheless, it does not have any influence on the flow field inside the open cavity. In addition, the pressure fields, at a different instant of time $t = 0.3$ sec, are also illustrated in the figure 4. The comparisons of the flow characteristics may also be done from both the said figures obtained at two different instants of times.

![Figure 3. Pressure field at instant of time, $t = 0.1$ sec, with and without the use of spoiler](image)

6.2. Comparisons of overall sound pressure level (OASPL) with and without use of spoiler

The comparison of the OASPL (Overall Sound Pressure Level) distributions at the centreline of the front wall of the open cavity with and without the use of spoiler has also been made. The OASPL is given as:
\[ OASPL = 10 \log_{10} \left( \frac{\overline{p_a^2}}{q^2} \right) \]  
(13)

Where, \( \overline{p_a^2} = \frac{1}{t_f-t_i} \int_{t_i}^{t_f} (p - \overline{p})^2 \, dt \)  
(14)

\( q \) is the acoustic sound reference level with a value equal to \( 2 \times 10^{-5} \) Pa  
\( \overline{p} \) is the time-averaged static pressure  
\( t_f \) and \( t_i \) are the initial and final times, respectively.

The OASPL distribution at the centreline of the front wall of the open cavity with spoiler has been compared with that of without the use of spoiler. The comparison of both the said circumstances are depicted in the figure 5. It is witnessed that at the centreline of the front wall of the open cavity on an average the OASPL gets suppressed by about 14 dB. It is further witnessed that with the use of spoiler the OASPL is suppressed by approximately 12 dB at the aft wall of the open cavity and also suppressed by approximately 16 dB at the front wall of the stated open cavity. However, the OASPL distribution at the centreline of the front wall of the open cavity with spoiler remains almost uniform throughout.

**Figure 5. OASPL distribution at the centreline of the front wall of the cavity with and without the use of spoiler**

**VII. CONCLUSIONS**

In the present investigation, the numerical simulation has been accomplished for supersonic flow past an open cavity with and without the integration of spoiler. The cavity has length-to-depth ratio of 2 besides the Mach number of the free-stream is 2.0. The simulation is executed through the LES based on Smagorinsky model. The results are depicted in the form of both cavity flow-field analysis along with the aeroacoustic analysis epitomized by the overall sound pressure level at the centreline of the front wall of the cavity. The LES model benefits to envisage all the principal flow structures of the open cavity. The overall sound pressure level at the centreline of the front wall of the cavity with spoiler is compared with that of without the spoiler. There ensues both qualitative and quantitative agreement between those two. It is observed that with the use of spoiler on an average the overall sound pressure level at the centreline of the front wall of the cavity gets decreased by nearly 14 dB. Nevertheless, with the combination of spoiler, the stated sound pressure level get reduced by nearly 12 dB and 16 dB at both aft and front walls of the open cavity, respectively. But, the overall sound pressure level at the centreline of the front wall of the open cavity with spoiler is almost constant.
The nature of results of both the cavities are alike and therefore are comparable. The pressure inside the open cavity is also decreased and the values less than the free-stream pressure is accomplished. In general, in this study, a 3D model is established for an open cavity and a spoiler is used at its leading edge to examine the flow structures and to reduce both pressure fluctuations and overall sound pressure level at the centreline of the front wall of the open cavity.

ACKNOWLEDGMENTS
The author would like to thank the editor and the reviewers for extending their constructive comments, valuable time and contributions for giving perceptive reviews to the research article.

REFERENCES