Numerical Modelling on Supersonic Fluid Flow Over A Cavity Concerning Velocity Field, Coefficient of Pressure And Overall Sound Pressure Level

Dr. Nirmal Kumar Kund
Associate Professor, Dept of Production Engineering
Veer SurendraSai University of Technology, Burla, Odisha, India

Abstract- *A suitable numerical model is developed to estimate supersonic flow past a 3D open cavity. The studies involve supersonic flow over the 3D open cavity having length-to-depth ratio of 2, including the supersonic free-stream Mach number of 2 in addition to the flow Reynolds number of* 10^5. *The numerical simulation has been conducted by utilizing the Large Eddy Simulation (LES) method. The Smagorinky model is used for this investigation. The results acquired have been compared with both experimental as well as numerical simulation predictions reported in the literature. The results have been shown in the form of both coefficient of pressure (Cp) and overall sound pressure level (OASPL) at the centreline of the aft wall of the open cavity. The coefficient of pressure appears to match both qualitatively and quantitatively with the existing experimental and numerical results available from the other investigators. Nevertheless, the overall sound pressure level at the centreline of the aft wall of the open cavity is over-predicted by nearly 30-40 dB. In addition, very large recirculation is also witnessed within the cavity as observed from the velocity vectors and therefore these require to be suppressed. But, the attachment of a spoiler is also planned for the future to alter the flow characteristics within the cavity which can bring about the reduction in both recirculation and overall sound pressure level at the centreline of the aft wall of the open cavity.*

Keywords- Numerical Simulation, Cavity, LES, Velocity Vector, Coefficient of Pressure, OASPL.

I. INTRODUCTION

The occurrence of noise is because of the aerodynamic forces acting on the surfaces caused by the air flow. Noise generated by a flow is a major problem in many engineering applications such as military vehicles, submarines, aircrafts, automobiles, etc. Airframe noise is a considerable component of overall noise. Noise from gear, flaps, slats etc. are regarded as airframe noise. One of the most significant airframe noises is the cavity noise. They arise from open wheel wells, weapon bays, door gaps, side mirrors, open sun roof, etc. These noises groups also influence the comfort within the vehicles. The door gaps, wheel wells and weapon bays can be modelled as rectangular cavities and the serene flow external to the cavity can be regarded to be smooth. Though the rectangular cavity is easy in shape, it is rich in various dynamic as well as acoustic portents, likely overlooked by an aeroacoustic feedback loop pertaining to the shape/size of the cavity in addition to the flow situations. Quite severe tone noises may be shaped owing to the vortex shredding at the upstream edge of the cavity, while the flow over a cavity.

II. LITERATURE REVIEW

Heller et al. [1] illustrated on flow-induced pressure oscillations in shallow cavities. Tam and Block [2] studied on the tones and pressure oscillations induced by flow over rectangular cavities. Kaufman et al. [3] reported on Mach 0.6 to 3.0 flows over rectangular cavities. Sweby [4] applied high resolution schemes using flux limiters on hyperbolic conservation laws. Rizzetta [5] performed numerical simulation on supersonic flow over a three-dimensional cavity. Anderson and Wendt [6] described about the fundamentals of computational fluid dynamics. Piomelli [7] demonstrated on achievements and challenges of large-eddy simulation. Hamed et al. [8] conducted numerical simulations of fluidic control for transonic cavity flows. Li et al. [9] carried out LES study of feedback-loop mechanism of supersonic open cavity flows. Vijayakrishnan [10] executed a validation study on unsteady RANS computations of supersonic flow over two dimensional cavity. Sousa et al. [11] discussed about thelid-driven cavity flow of viscoelastic liquids**.** Tuerke et al. [12] investigated the experimental study on double-cavity flow. It is apprehended that a general study on cavity flow has been done both experimentally as well as numerically for enhancing the aerodynamic effect. Conversely, above and beyond its significance, the challenging flow physics of flow over a cavity has spellbound the researchers around the world for further investigations and remains as a front-line area of investigation.

III. OBJECTIVES OF PRESENT RESEARCH WORK

Despite the fact, the flow over a cavity has been studied experimentally/numerically by many investigators, nevertheless, complete modelling of both large and small scales of motions at a time, not yet done which is one of the leading inadequacies. But, Large Eddy Simulation (LES) is the technique which resolves the large eddies as it is and models the small eddies that can provide sensibly more convincing results too. The importunity of this investigation work is to study the flow physics and modes of oscillations in a 3D open cavity supersonic flow. It includes details about the governing equations and the establishment together with the implementation of the LES method counting the sub-grid scale modelling. The discretization trials have also been discussed. The simulation predictions have been provided in the form of pressure flow field, coefficient of pressure (Cp) and overall sound pressure level (OASPL) at the centreline of the aft wall of the open cavity. The numerical results of supersonic flow past an open cavity have also been compared with the experimental as well as numerical results reported in the literature. In overall, very good agreement between the above said results is also witnessed from the current studies. Nonetheless, the investigations relating to the usage of passive control methods/devices for the reduction of recirculation within the open cavity is planned for the future. As these devices perform over a broad range of parameters, definitely it will influence the flow physics of incoming boundary layer for offering equally efficient flow circumstances.

IV. 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.

A. Geometric model

The computational domain of the cavity used in the present simulation is shown in figure 1. The size of the computational domain, as mentioned earlier, is 2D×D×D (length × breadth × width). The inlet boundary is located at a distance of D upstream from the leading edge of the cavity. The outlet boundary is located at a distance of 4D downstream from the trailing edge of the cavity. The upper boundary is also located at a distance of 4D above the cavity.

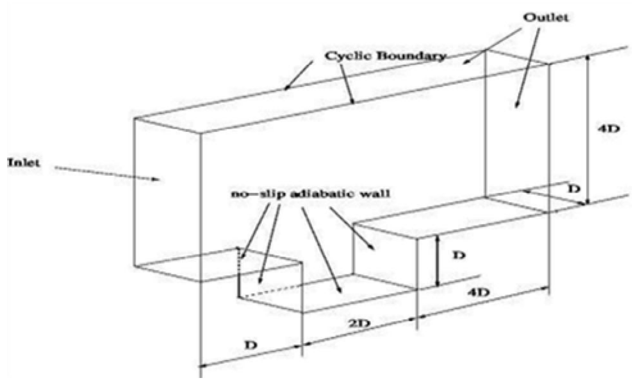

Figure1.Computational domainofcavity

B. Initial and boundary conditions

The inflow boundary conditions are initialized with free-stream conditions of $M_\infty = 2$, $P_\infty = 101.325$ kPa, and $T_\infty = 300$ K. The Reynolds number of the flow used in the simulation is 10^5, which is 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.

V. MATHEMATICAL FORMULATION

A. Generalized governing transport equations

The three-dimensional compressible Navier-Stokes equations are the governing equations which include the continuity equation (1), the momentum equation (2), and the energy equation (3) which are as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \boldsymbol{U}) = 0 \quad (1)$$

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla.(\rho \boldsymbol{U}.\boldsymbol{U}) - \nabla.\nabla(\mu \boldsymbol{U}) = -\nabla p \quad (2)$$

$$\frac{\partial (\rho e)}{\partial t} + \nabla.(\rho e \boldsymbol{U}) - \nabla.\nabla(\mu e) = -p(\nabla.\boldsymbol{U}) + \mu\left[\frac{1}{2}(\nabla \boldsymbol{U} + \nabla \boldsymbol{U}^{\mathrm{T}})\right]^2 \quad (3)$$

Where,

$\boldsymbol{U}$ = velocity vector = $u\hat{\imath} + v\hat{\jmath} + w\hat{k}$

$\frac{1}{2}(\nabla \boldsymbol{U} + \nabla \boldsymbol{U}^{\mathrm{T}})$ = strain rate tensor.

The equations (1), (2) and (3) represent the conservation form of the Navier-Stokes equations. The conservation form of these governing equations are achieved from a flow model fixed in space [6]. The above equations are applicable to viscous flow, except that the mass diffusion is no included.

It is assumed, in aerodynamics, that the gas is a perfect gas.

The equation of state for a perfect gas is, $p = \rho RT$ (4)

Where, R = specific gas constant = C_p - C_v (5)

For a calorically perfect gas (constant specific heats), the caloric equation of state is,

e = internal energy per unit mass = C_vT (6)

B. LES turbulence modelling

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

In the present study, features of the turbulent flow field have been simulated using LES as it is appropriate for unsteady complex flows as well as noise induced flows. LES computes the large resolved scales and also models the smallest scales. The turbulence model is introduced by splitting the time and space varying flow variables into two constituents, the resolved one $\bar{f}$ and f', the unresolved part:

$$(x,t) = \bar{f}(x,t) + f'(x,t) \quad (7)$$

LES uses a filtering operation to separate these resolved scales from the unresolved scales. The filtered variable is denoted by an over bar [7]. The top-hat filter smooth both the fluctuations of the large-scale and those of small scales as well. The filtering operation when applied to the Navier-Stokes equation gives:

$$\frac{\partial \bar{\rho}}{\partial t} + \nabla.(\overline{\rho \boldsymbol{U}}) = 0 \quad (8)$$

$$\frac{\partial (\overline{\rho \boldsymbol{U}})}{\partial t} + \nabla.(\overline{\rho \boldsymbol{U}.\boldsymbol{U}}) - \nabla.\nabla(\overline{\mu \boldsymbol{U}}) = -\nabla \bar{p} \quad (9)$$

$$\frac{\partial (\overline{\rho e})}{\partial t} + \nabla.(\overline{\rho \boldsymbol{U} e}) - \nabla.\nabla(\overline{\mu e}) = -\overline{p(\nabla.\boldsymbol{U})} + \mu\left[\frac{1}{2}(\nabla\bar{\boldsymbol{U}} + \nabla\bar{\boldsymbol{U}}^{T})\right]^2 \quad (10)$$

However, 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 known as energy cascade. Hence, LES employs a sub-grid scale model to mimic the drain related to this energy cascade. Most of these models are eddy viscosity models relating the subgrid-scale stresses (τ_{ij}) and the resolved-scale rate of strain-tensor ($\bar{S}_{ij}$),

$$\tau_{ij} - (\delta_{ij}/3) = -2\nu_T\bar{S}_{ij} \quad (11)$$

Where, $\bar{S}_{ij}$ is the resolved-scale rate of strain tensor = $(\partial \bar{u}_i/\partial x_j + \partial \bar{u}_j/\partial x_i)/2$.

In most of the cases it is assumed that all the energy received by the unresolved-scales are dissipated instantaneously. 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 obtained for the eddy viscosity:

$$\mu_{sgs} = \rho C\Delta^2|\bar{S}|\bar{S}_{ij},\ |\bar{S}| = (2\bar{S}_{ij}\bar{S}_{ij})^{1/2} \quad (12)$$

Here, Δ is the grid size and is usually taken to be the cube root of the cell volume [7]. This model is called as the Smagorinsky model and C is the Smagorinsky coefficient. In the present study, its value has been taken to be 0.2.

VI. NUMERICAL PROCEDURES

A. Numerical scheme and solution algorithm

The three-dimensional compressible Navier-Stokes governing transport equations are discretized through a framework pertaining to finite volume method (FVM) using the SIMPLER algorithm. Here, the turbulent model used for large eddy simulation is Smagorinsky model, because of its simplicity. The spatial derivatives such as Laplacian and convective terms are computed by second order scheme based on Gauss theorem. In addition, the viscous terms are evaluated by second order scheme. Furthermore, the implicit second order scheme is used for time integration. The numerical fluxes are evaluated by applying Sweby limiter to central differencing (CD) scheme, which is a total variation diminishing (TVD) scheme. The central differencing (CD) is an unbounded second order scheme, whereas, the total variation diminishing (TVD) is a limited linear scheme. The established solver is used to predict flow behaviours of the associated flow variables relating to supersonic flow over an open cavity.

B. Choice of grid size, time step and convergence criteria

Figure 2demonstrates that the computational domain comprises of two regions: upper cavity region and inside cavity region. 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. Thus, the total number of grid points is 84000. The values of ΔX^+, ΔY^+ and ΔZ^+ at the leading edge of the cavity are 5, 12.5 and 1.0, respectively. Corresponding time step taken in the simulation is 0.000001 seconds. Though, it is checked with smaller grids of 132000 in numbers, it is observed that a finer grid system does not alter the results significantly.

Convergence in inner iterations is declared only when the condition $\left|\frac{\varphi-\varphi_{old}}{\varphi_{max}}\right| \leq 10^{-4}$ is satisfied simultaneously for all variables, where φstands for the field variable at a grid point at the current iteration level, φ_{old}represents the corresponding value at the previous iteration level, and $\varphi_{max}$is the maximum value of the variable at the current iteration level in the entire domain.

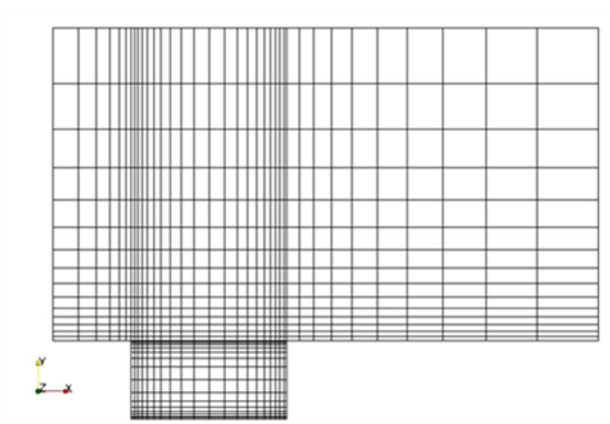

Figure 2.Computational gridofcavityin X-YPlane

VII. RESULTS AND DISCUSSION

A. Velocity distributions

Figures 3 and 4 illustrate the velocity vectors (as found from the numerical simulations for supersonic fluid flow past an open cavity) at two different instants of times like t = 0.1 sec as well as t = 0.3 sec, respectively. The tenacity of the velocity vector is to display the recirculation zone within the open cavity. Quite large recirculation zone occurs near the aft wall as a result of mass injection at high speed close to the trailing edge. Very small recirculation zone is observed in the left bottom corner. The shocks produced at both leading and trailing edges of the open cavity may be clearly noticed in the simulated predictions of the velocity vectors.

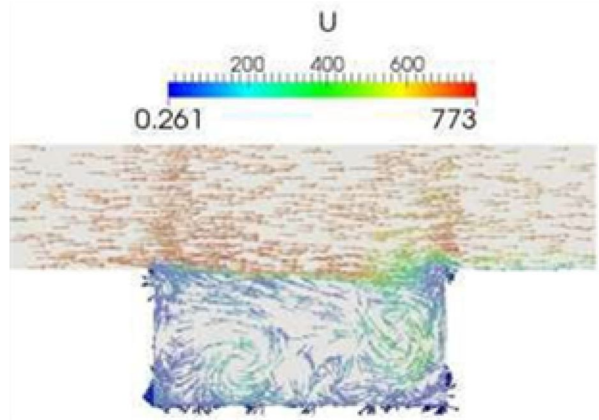

Figure 3.Velocity vector at time, t = 0.1 sec

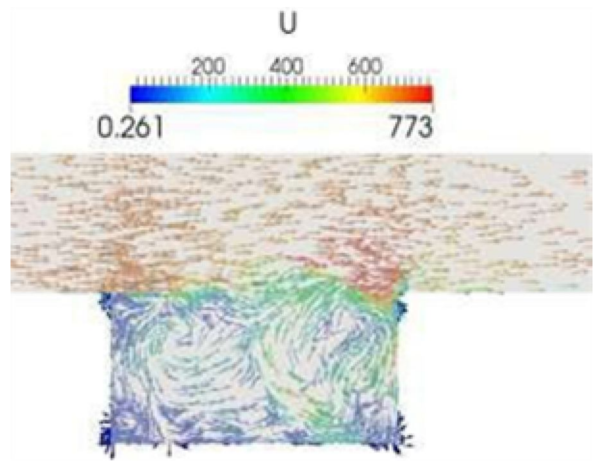

Figure 4.Velocity vector at time, t = 0.3 sec

B.Comparisons with other numerical and experimental results

(i).Comparison of coefficient of pressure (Cp)

The comparison of coefficient of pressure has been accomplished at the centreline of the aft wall of the open cavity which is exemplified in figure 5. Furthermore, the coefficient of pressure at the centreline of the stated open cavity wall is observed to be in both qualitative and quantitative agreement with the experimental along with the numerical results reported in the literature. The deviation in the results (at the centreline of the aft wall of the open cavity) from the experimental data of Kaufman et al. is caused by the numerical errors occurred throughout the simulation practices. In addition, the variation from the Vijaykrishnan research work is owing to the three-dimensionality effect and also the variation from the Rizzetta research work is because of the difference in the Mach number.

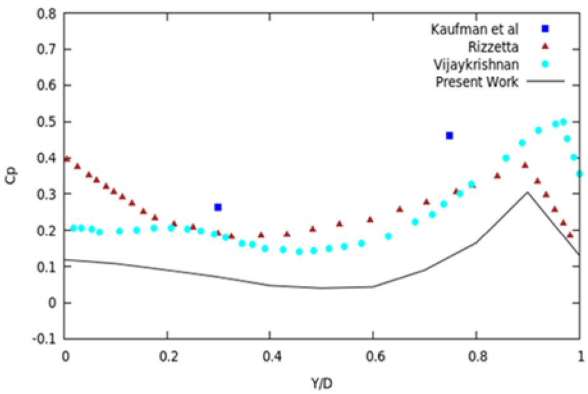

Figure 5.Coefficient of pressure at the centreline of the aft wall of the cavity

(ii).Comparison of overall sound pressure level (OASPL)

The comparison has also been accomplished with the OASPL (Overall Sound Pressure Level) distributions at the centreline of the aft wall of the open cavity. The OASPL is denoted as:

$$OASPL = 10\log_{10}(\overline{p_d^2}/q^2) \quad (13)$$

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

$q$is the acoustic sound reference level with a value of 2×10^{-5} Pa

$\bar{P}$is the time-averaged static pressure

$t_f$andt_i are the initial and final times, respectively

The OASPL distribution at the centreline of the aft wall of the open cavity is depicted in figure 6. It is estimated to be about 25 dB higher than the experimental results (of Kaufman et al.). Certainly, the trend of results are comparable to the research works done by the other investigators. Nonetheless, the trend of result is very similar to the Kaufman et al. and Rizzetta research works rather than Vijaykrishnan research work. The sound pressure level observed at the aft wall is almost 5 dB higher than the front wall. The OASPL distribution at the centreline of the aft wall of the open cavity is over-predicted (from both Kaufman et al. and Rizzetta research results) by nearly 30-40 dB between aft and front walls. Conversely, the over-prediction from Vijaykrishnan research work is about 50-90 dB between aft and front walls. Furthermore, from the current research, the observed sound pressure level at the centrelineof theaft wall of the open cavity seems to be nearly uniform throughout.

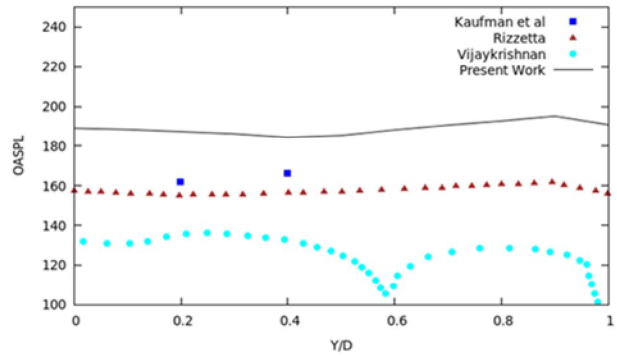

Figure 6. OASPL distribution at the centreline of the aft wall of the cavity

VIII.CONCLUSION

In the present investigational work, the numerical simulations have been carried out for supersonic flow past a 3D open cavity. The open cavity has length-to-depth ratio of 2 in addition the Mach number of the free-stream is 2.0. The simulations are performed by means of LES based on Smagorinsky model for the stated open cavity. The numerical simulation predictions are done in the form of both cavity flow-field as well as aeroacoustic analyses. In addition, the aeroacoustic analysis is also done in the form of both coefficient of pressure (Cp) and overall sound pressure level (OASPL) at the centreline of the aft wall of the open cavity. The present simulation results are compared with both experimental and numerical results exist in the literature. The LES model enables to foresee all the central flow behaviours of the open cavity. In addition, there also exists both qualitative and quantitative agreement of the coefficient of pressure with the experimental along with the simulation results available in the literature by the other researchers for supersonic flow past the said open cavity. Instead, the overall sound pressure level at the centreline of the aft wall of the open cavity is over-predicted by 30-40 dB. Besides, very large recirculation is observed within the open cavity as witnessed from velocity vectors and hence these must be reduced. But, the incorporation of a spoiler in the form of one-fourth of a cylinder at the leading edge of the open cavity is also planned for the future to modify the flow behaviours within the open cavity that can suppress both recirculation and overall sound pressure level at the centreline of the aft wall of the open cavity.

IX. ACKNOWLEDGMENT

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

[1] Heller, H. H., Holmes, D. G., & Covert, E. E. (1971). Flow-induced pressure oscillations in shallow cavities. Journal of sound and Vibration, 18(4), 545-553.

[2] Tam, C. K., & Block, P. J. (1978). On the tones and pressure oscillations induced by flow over rectangular cavities. Journal of Fluid Mechanics, 89(02), 373-399.

[3] Kaufman, I. I., Louis, G., Maciulaitis, A., & Clark, R. L. (1983). Mach 0.6 to 3.0 flows over rectangular cavities (No. AFWAL-TR-82-3112). Air force wright aeronautical labs wright-patterson AFB, OH.

[4] Sweby, P. K. (1984). High resolution schemes using flux limiters for hyperbolic conservation laws. SIAM journal on numerical analysis, 21(5), 995-1011.

[5] Rizzetta, D. P. (1988). Numerical simulation of supersonic flow over a three-dimensional cavity. AIAA journal, 26(7), 799-807.

[6] Anderson, J. D., & Wendt, J. F. (1995). Computational fluid dynamics (Vol. 206). New York: McGraw-Hill.

[7] Piomelli, U. (1999). Large-eddy simulation: achievements and challenges. Progress in Aerospace Sciences, 35(4), 335-362.

[8] Hamed, A., Das, K., &Basu, D. (2004). Numerical simulations of fluidic control for transonic cavity flows. AIAA Paper, 429, 2004.

[9] Li, W., Nonomura, T., Oyama, A., &Fujii, K. (2010). LES Study of Feedback-loop Mechanism of Supersonic Open Cavity Flows. AIAA paper, 5112, 2010.

[10] Vijayakrishnan, K. (2014) Unsteady RANS computations of supersonic flow over two dimensional cavity using OpenFOAM-A validation study. AIAA 2014.

[11] Sousa, R. G., et al. (2016).Lid-driven cavity flow of viscoelastic liquids**.** Journal of Non-Newtonian Fluid Mechanics, 234, 129-138, 2016.

[12] Tuerke, F., Pastur, L. R., Sciamarella, D., Lusseyran, F., &Artana, G. (2017). Experimental study of double-cavity flow. Experiments in Fluids, 76, 2017.