PASSIVE AND ACTIVE CONTROL OF ACOUSTIC RESONANCE IN CAVITY FLOWS USING FFOWCS-WILLIAMS-HAWKINGS EQUATIONS

Ece AYLI [1, +]
[1]Mechanical Engineering, Cankaya University, Ankara, Turkey
eceayli@cankaya.edu.tr

Abstract

In the aerospace industry, interior storage carriages, that carry items such as weapons and bombs form cavities. The turbulence-cavity interaction causes significant vibration, sound pressure levels, resonance, and structural problems. Therefore, control methods are can be useful to reduce drag, minimize pressure fluctuations and SPL levels. This work studies the passive flow control methods to reduce the noise induced by the flow over the cavity. For this purpose, cavity leading, and trailing edge wall modifications were made such as inclination, placing a block upstream of the cavity, blowing from the cavity walls. Broadband nature of the noise sources is captured generally with DNS or LES approach. Large Eddy Simulations (LES) is used to compute the flow field to reduce computational cost. ANSYS Fluent software is utilized to solve compressible, two-dimensional, transient subsonic cavity flow. For the determination of sound pressure levels, Ffowcs-Williams–Hawkings (FW-H) integral method is used.

Keywords: Passive and Active control, compressible flow, Ffowcs-Williams–Hawkings equations, CFD, Aeroacoustics

KAVİTE BÖLGELERİNDEKİ AKUSTİK REZONANS KONTROLÜ İÇİN AKTİF VE PASİF KONTROL METODLARININ FFOWCS-WILLIAMS-HAWKINGS DENKLEMLERİ YARDIMI İLE İNCELENMESİ

[+] This paper has been presented at the ICAT'20 (9th International Conference on Advanced Technologies) held in Istanbul (Turkey), August 10-12, 2020.

Özet

Havacılık endüstrisinde uçakların mühimmat depolayan bölümleri kavite olarak adlandırılmaktadır. Kavite bölgesinde, türbülans-kavite etkileşimi önemli ölçekte titreşime, yüksek ses basınç seviyelerine, rezonansa ve yapısal problemlere sebep olmaktadır. Bu sebeplerden ötürü, kavite bölgelerinde meydana gelen basınç salınımını minimize etmek, sürtünme kat sayısını ve ses basınç seviyelerini (SPL) düşürmek amacı ile kontrol metodları geliştirilmekttedir. Bu çalışmada, kavite giriş ve çıkış duvarlarında geometrik modifikasyonlar yaratarak pasif, kavite duvarlarından hava üflemesi yaparak ise aktif akış kontrol yöntemleri geliştirilmiştir. Akış alanını çözümlemek için LES yaklaşımı kullanılmıştır. Sıkıştırılabilir, iki boyutlu,zamana bağlı değişen ve türbülanslı kaviye akışı ANSYS Fluent programı kullanılarak modellenmiştir. Ses seviyelerinin belirlenmesi için ise Ffowcs-Williams–Hawkings (FW-H) integral metodu kullanılmıştır.

Anahtar Kelimeler: Pasif ve aktif kontrol, sıkıştırılabilir akış, Ffowcs-Williams–Hawkings (FW-H) denklemleri, HAD, aeroakustik

1. Introduction

Cavities are mostly appearing in the aerospace applications, such as weapon bays, on aircrafts. Transporting the ammunition in the aircraft reduces the risk of catching up on the radar and reduces aerodynamic load, drag force and aerodynamic heating. On the other hand, the high velocity values around the cavity region create many complex structures in the flow area, and this poses a major problem in aviation applications. This complex flow field comprises small-scale pressure fluctuations, that is typical of turbulent flow, resonance formations whose frequency and magnitude vary depending on some parameters. Acoustic modes and pressure fluctuations that cause resonance can damage the structure of the aircraft and negatively affect the possibility of ammunition reaching the target. Although, controlling the flow around the cavity is difficult, storing weapons in internal bays of fighter aircrafts offers several benefits including an expanded flight envelope, greater maneuverability and higher penetration speed resulting in less time over target, which increases the survivability of the aircraft. Therefore, cavities that have complex flow field have an indispensable application area in aerospace [1].

Due to the complexity of the cavity flow, it has been classified in the literature in different ways. Some of these classification types are classifications based on cavity geometry, flow properties and Mach number [2]. Cavities classified as deep or shallow cavities depending on their geometry. Cavities in which the L/ D ratio is less than 1 represent deep cavities, whereas cavities in which they are larger than 1 represent shallow cavities. In deep cavities, two circulation regions are observed generally, while stronger formations occur in shallow cavities [2].

Tracy et al. [3] kept the depth of the cavity constant while changing the width at subsonic and transonic regime. According to their results, the pressure distribution varies depending on the depth of the cavity and depth has a dominant effect on the the flow behavior. According to the Block [4], when the L / W ratio is less than 1, two-dimensional acoustic field is obtained, while the L / W value is greater than 1, a three-dimensional acoustic field is observed. Acccording to the flow behavior cavity flow is classified as open and closed cavities. Closed cavity flow occurs in shallow cavities that L / D ratio is greater than or equal to 13. Open cavity flow occurs in deep cavities with L / D <10 [5]. Tray et al [3] claimed that cavity flow is not only depends on the cavity geometrical parameters but also it is strongly related with the Mach number. In the open and transition regime, L/D range is independent from the Mach number and W/D. On the other hand, in the closed cavity regime, L/D range varies between 9 to 14 according to the Mach number. In the literature several studies are performed in order to examine the cavity flow experimentally and numerically. Casper et al [6], performed a parametric study to examine the geometrical parameters effects on Mach=0.8 cavity flow experimentally. They used different cavity configurations like, cavity with scoop and tooth, cavity with side ramp, cavity with doors and without doors. According to their results, configuration with tooth reduces the amplitude of the modes and frequencies. Shih et al. [7] used the k-ε turbulence model to analyze an open cavity flow for L/D=5.07 with 1.5 Mach number. They verify their numerical results with the experimental study of Kaufman et al. [8]. The flow on a two-dimensional flat plate was numerically resolved and used as an inlet boundary condition of, cavity. According to the results of the study, the flow becomes complex because of the shear layer. Mass entrance and exit occur due to the shear layer formation. Mass entrance causes pressure fluctuations inside the cavity zone which causes aeroacoustics and aerodynamic problems.

Rizetta and Visbal [9] analyzed supersonic cavity flow with the LES method. Mach number is 1.19, L / D ratio is 5 in the study. The mesh structure comprises 20 million mesh elements and no wall functions are required. Large-scale eddies are formed at the entrance of the cavity and these eddies are transferred behind the cavity. The SPL levels and Rossiter frequencies compared with the experimental value are consistent with each other. Bres and Colonius [10] used the DNS method to analyze three-dimensional open cavity flow, to investigate the mechanism that triggered cavity oscillations. The obtained results confirm the Rossiter mechanism. Chung [11] and Heller et al. [12] claimed that the highest amplitude pressure fluctuations are obtained in the rear edge of the cavity. The shear layer and rear wall interaction is the main reason of this complex flow field. Therefore, in the literature the effect of the rear edge modifications on pressure oscillations are examined. Saddington et al. [13] compare the passive flow control methods in the cavity for transonic regime. Both leading and trailing edge modifications are tried experimentally. According to their results square tooth spoiler has a serious effect to suppress the tones with reducing SPL 8.8 dB. Gupta and Roy [14] observed that inserting a passive receptive channel to trailing edge shows an appreciable effect on the acoustic tones in the cavity.

In this paper, the effect of passive control techniques has been examined and their effectiveness in the suppression of cavity tones, SPL levels and pressure fluctuations are compared, numerically. Firstly, the nature of the cavity flow is investigated with numerical methods also verification study is performed to be sure that numerical methodology is accurate. For this purpose, analysis is performed with same conditions of experimental study from the literature [15].

2. Methodology

2.1. Aeroacoustics Methodology

Sound is formed according to the pressure fluctuation and the sound sources are generated by motion, either by the free fluid motion or by a solid body–fluid interaction. The sound production is very small when it is compared with the fluid flow energy therefore; it is much more difficult to predict. Acoustic analogy is described by Lighthill [16] in 1952, firstly. His equation connects the fluid mechanics and acoustics with taking the derivative of continuity equation and subtracting the divergence of momentum equation [17]. In 1969, Curle, Powell and Ffowcs Williams extended Lighthill's ideas and include the solid boundaries effects on aeroacoustics regime. Ffowcs Williams and Hawkings [18] add a source term to the Lighthills equation. Curle [19] developed this analogy by taking into account the effect of immobile solid objects in the flow field and expanded the Lighthill equation. Ffowcs William–Hawkings equations are given as follow:

$$\left(\frac{\partial^2}{\partial t^2} - c_0^2 \frac{\partial^2}{\partial x_i \partial x_i}\right)(H(f)\rho') = \frac{\partial^2}{\partial x_i \partial x_i}\left(T_{ij}H(f)\right) - \frac{\partial}{\partial x_i}(F_i\delta(f)) + \frac{\partial}{\partial t}(Q\delta(f)) \quad (1)$$

Where t, Tij, Q and f represents time, Lighthill's stress tensor, and source terms, respectively. Function F denotes the domain outside the source surface. To reduce the sound, the source efficiency of the noise, the kinetic energy that is converted to the noise should be minimized and active sound waves with out-of-phase waves should be eliminated.

2.2. Aerodynamic Methodology

To solve the two-dimensional, compressible and turbulent cavity flow ANSYS Fluent software is used. The LES model with the subgrid scale model is utilized in order to solve the cavity flow mechanism and cavity noise. To solve the compressible flow field; the conservation of mass, momentum, and energy equations given in equations (2)-(4) are solved.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (2)$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial t_{ji}}{\partial x_j} \quad (3)$$

$$\frac{\partial}{\partial t}\left[\rho(e + \frac{1}{2}u_i u_i)\right] + \frac{\partial}{\partial x_j}\left[\rho u_j(h + \frac{1}{2}u_i u_i)\right] = -\frac{\partial}{\partial x_i}(u_i t_{ij}) + \frac{\partial q_j}{\partial x_j} \quad (4)$$

To solve Navier-Stokes equations SGS stress model is used:

$$\tau_{ij} = \frac{1}{3}\tau_{kk}\delta_{ij} - 2\mu_t\bar{S}_{ij} \quad (5)$$

For compressible flows, τ_{ij} is the stress tensor that comprises the subgrid viscosity, μ and kronocer delta, δ_{ij}. τ_{kk} is the isotropic part of the subgrid scale. Subgrid viscosity is calculated as

$$\mu_t = (C_t\Delta)^2|\bar{S}| \quad (6)$$

Where, C_t is the Smagorinsky constant and the stretch tensor |S| is calculated as:

$$|\bar{S}| = \sqrt{2\bar{S}_{ij}\bar{S}_{ij}} \quad (7)$$

2.3. Computational Method and Flow Parameters

The baseline cavity is shown in Figure 1. The cavity is placed 28 cm the aft of the leading edge of the flat plate. The cavity length to depth ratio is 4.5. For the verification study, the cavity length and depth are taken as 27.9 cm long and 2.5 cm deep. The tests are conducted for 65.2 m/s, which corresponds to Reynolds number of 1200000 and Mach number of 0.4. All of the geometrical and flow parameters are kept same with the experiment of [15] in order to make verification.

The meshing domain is divided to 4 different zones to create a structured mesh that leads to more accurate results and a better convergence. Mesh independency study is performed with using three different mesh structures with an element number of 106,1.5x105,2x105. After one million elements the relative error reduces to 0.02% as it is shown in Figure 1. Therefore, one million number of mesh element is used for the study.

The simulations are performed in ANSYS Fluent Software. LES with dynamic Smagorinsky model is used in the study. For discretization least square cell based is used. For time resolution, second order implicit method is used. Convergence criteria is chosen as 10-6. Outer and upper wall is selected as pressure far field. For cavity walls, no slip wall is utilized. At the inlet, velocity inlet boundary condition is used with 65.2 m/s velocity value. The wall y+ value is less than 3.5 which is sufficient for the wall function to resolve the viscous sub-layer.

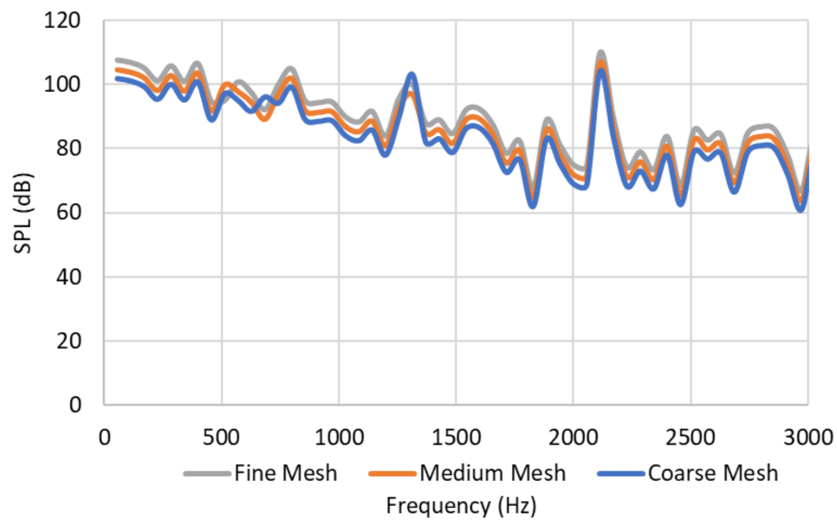

Figure 1. Mesh independency study

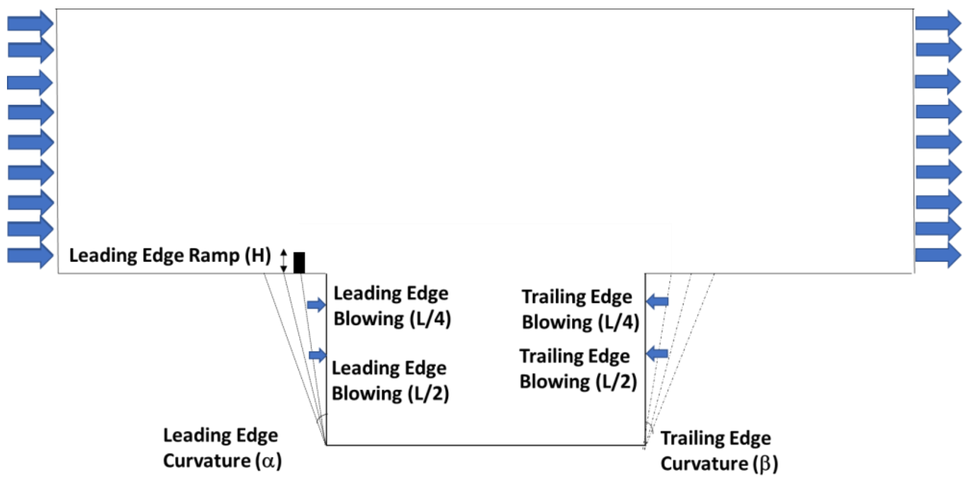

Figure 2. Schematic view of the modifications

Table 1. Test cases

	Test Case	Control Method	b	a	L
Passive Control	Case 1	TE Curvature	15	0	0
	Case 2	TE Curvature	30	0	0
	Case 3	TE Curvature	45	0	0
	Case 4	TE Curvature	60	0	0
	Case 5	LE Curvature	0	15	0
	Case 6	LE Curvature	0	30	0
	Case 7	LE Curvature	0	45	0
	Case 8	LE Curvature	0	60	0
	Case 9	LE Ramp	0	0	L/4
	Case 10	LE Ramp	0	0	L/6
	Case 11	LE Ramp	0	0	L/8
Active Control	Case 12	LE Üfleme	0	0	L/2
	Case 13	TE Üfleme	0	0	L/2
	Case 14	LE Üfleme	0	0	L/4
	Case 15	TE Üfleme	0	0	L/4

3. Results

3.1. Verification Study and Flow Field Analysis of Base Case

As mentioned in the previous section to be sure that numerical methodology that is used is accurate numerical results are compared with the experimental study of Meganathan [15]. The comparison graphic as a function of SPL is depicted in Figure 3. Also in Table 2, for several frequency values SPL values are compared with experimental study. The maximum error is calculated at 2000 Hz as 7.86%. Mode 1 and mode 2 are accurately obtained in numerical study as it is shown in Figure 3.

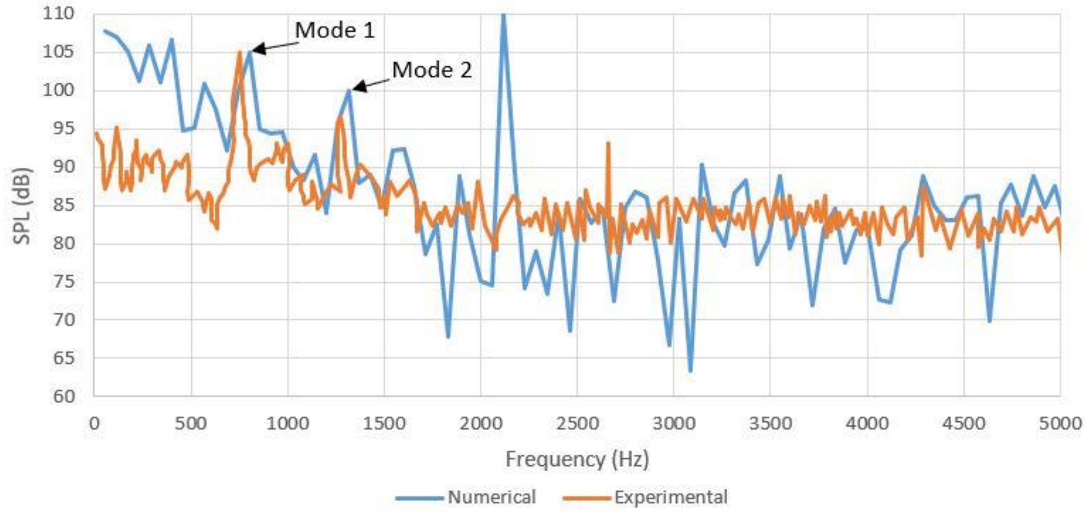

Figure 3. Verification study graphic

Table 2. Comparison of experimental and numerical SPL values

Frequency (Hz)	SPL (dB, experiment)	SPL (dB,computation)	% difference
850	105	103	1,904
1000	94.7	94.2	0,527
2000	88.6	79.7	7,861
3500	84.8	84.2	0,707

According to the observation of the velocity field of the cavity that is given in Figure 4, it is seen that the shear layer is formed above the cavity due to the pressure difference between the free stream and the cavity. Vortices are formed at the leading edge of the cavity and, travels through the downstream, grows with convection and impinges to the trailing edge. The second up-coming structure forces the first structure to ejection. This is the main mechanism in the closed cavity. The detailed flow physics of the cavity flow is given in our previous research [1]. The mass transfer in the cavity causes pressure fluctuations in the rear edge of the cavity. Those fluctuations create acoustic waves which propagate upstream.

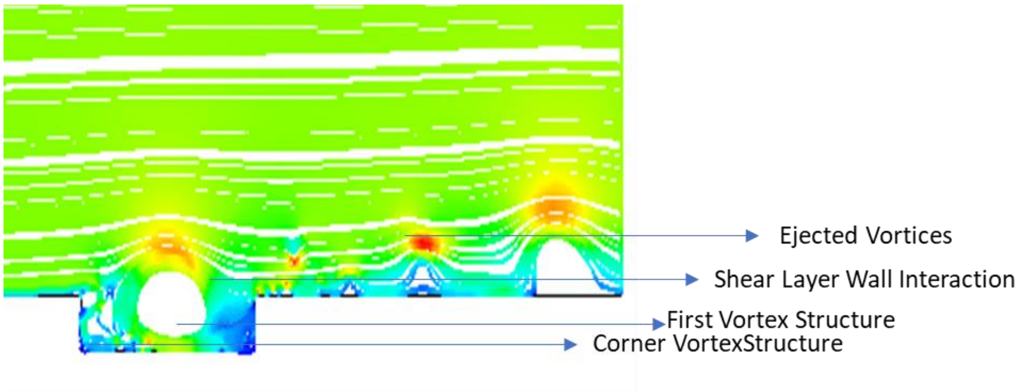

Figure 4. Flow characteristics for open cavity flow (L/D=4.5)

3.2. Curvature Effect

The most popular passive control method is creating modification to the walls of the cavity. In this part of the research, those modifications are test for both front and aft walls. The obtained results are compared with each other and the base cavity results.

For trailing edge and leading edge curvature is given between 0° to 60° in Figure 5 and 6. In Figure 6, for 30° trailing and leading edge ramp pressure contours are compared with the base flow physics at the same time. It is seen that maximum pressure value decreases for both of the curvature modification. The maximum pressure value is obtained at the rear edge of the cavity for all of the cases. The trailing edge curvature show the maximum attenuation of cavity tones and acoustic levels.

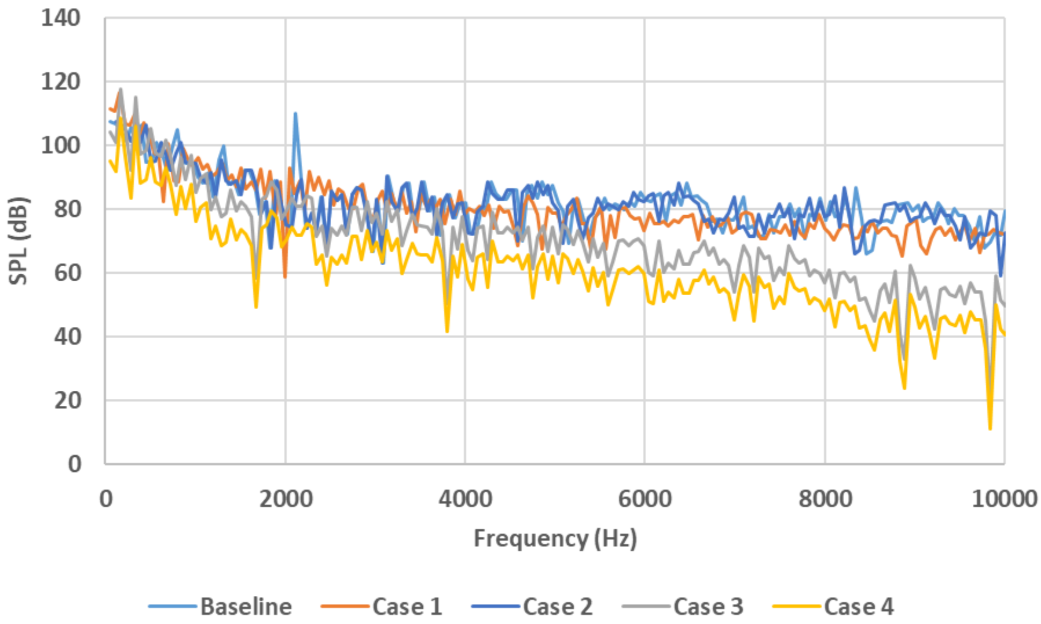

Figure 5. Trailing edge curvature effect as a function of SPL

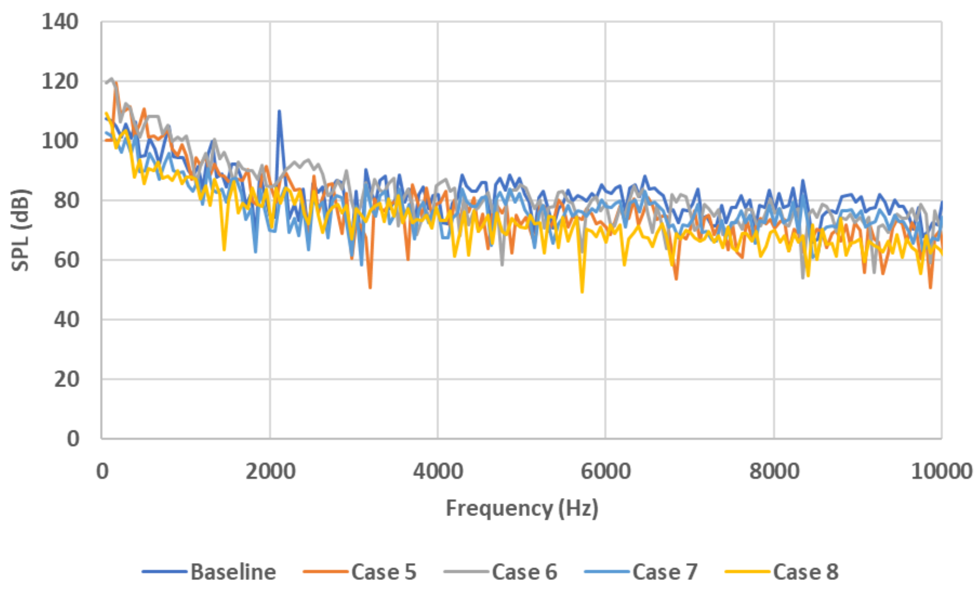

Figure 6. Leading edge curvature effect as a function of SPL

Increasing the curvature angle on both edges decreases the SPL values. Another observation from Figure 5 and 6 are decrease in the tonal frequencies with defining curvature to the edges. No additional frequency modes are observed in both modifications. With trailing edge curvature average SPL values reduces up to 20.3%. With trailing edge curvature average SPL values reduces up to 7.47%. The absolute pressure is reduced by an 24.4% with case 4, while it is reduced 12.7% with case 8. This pressure reduction causes a significant decrease the SPL levels and possibly also prevent the vibrations and reduces the structural deformations.

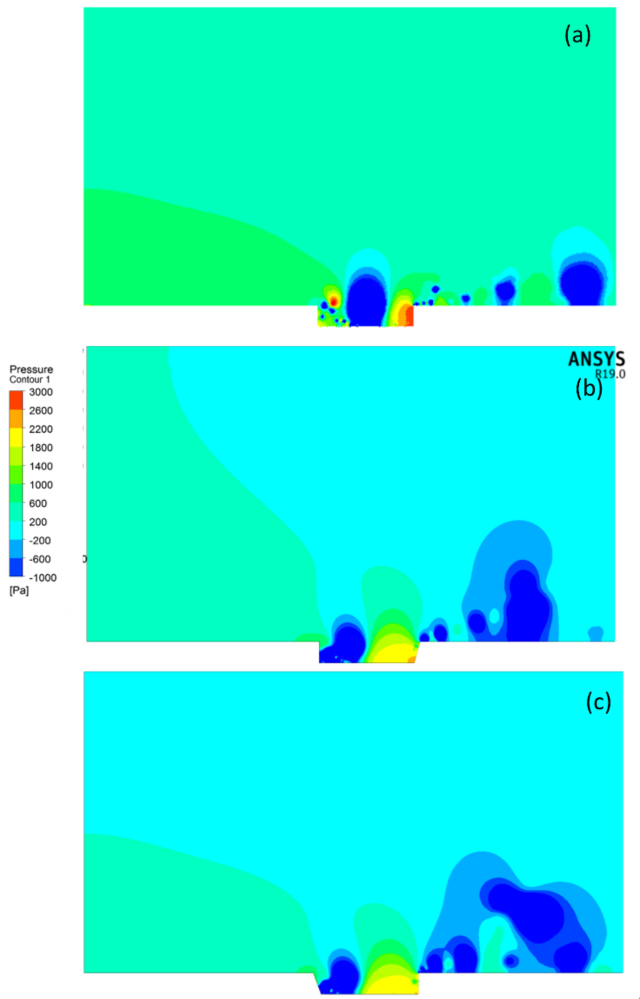

Figure 7. Pressure contour for (a) base cavity (b)30° trailing edge curvature cavity (c) 30° leading edge curvature cavity

3.3. Ramp Effect

As a different sound pressure control method ramp is added to the leading edge of the cavity. It is thought that the high of the ramp plays a critical role in the pressure suppression ability. However, when Figure 8 is examined it is seen that increasing the ramp height does not have an dominant effect on the SPL levels. Although ramp decreases the SPL levels up to 6.81%, SPL distribution trend is almost same for all of the cases (case 9-case 11)

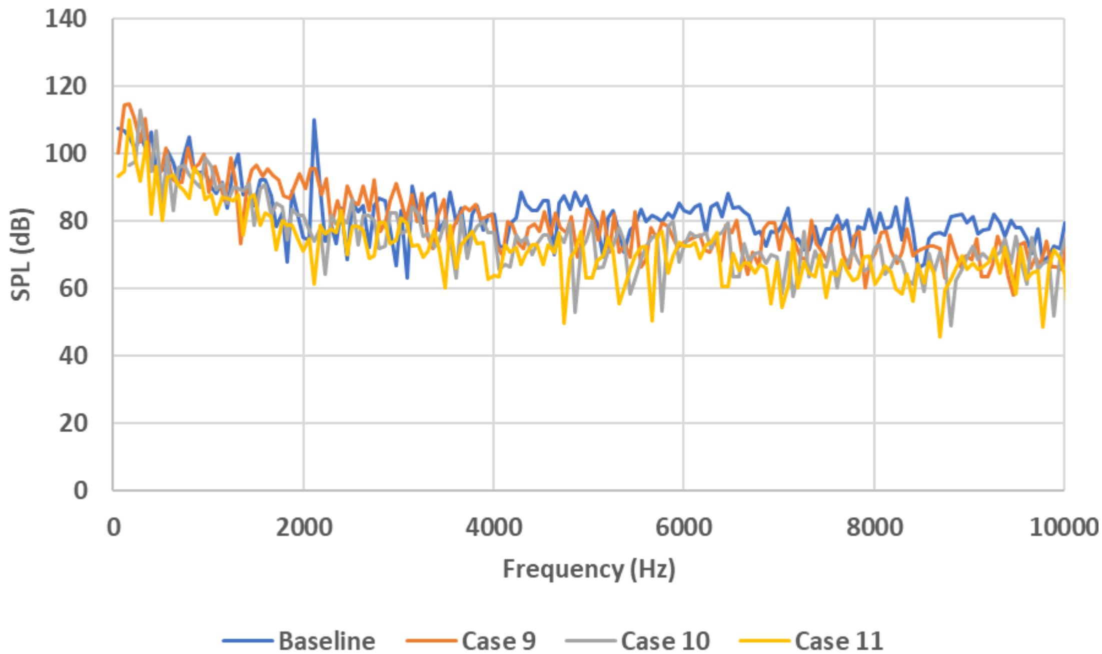

Figure 8. Leading Edge Ramp Effect as a function of SPL

Based upon the observations of Figure 9, it is seen the with adding ramp to the leading edge, mass entering length distance is increased. In the base cavity mass enters to the cavity when x/L=0.1, when ramp is added to the cavity (Case 9) mass entering length x/L is equal to 0.33. That means that even the length of the ramp is so small it has a direct effect on the mass entering length distance.

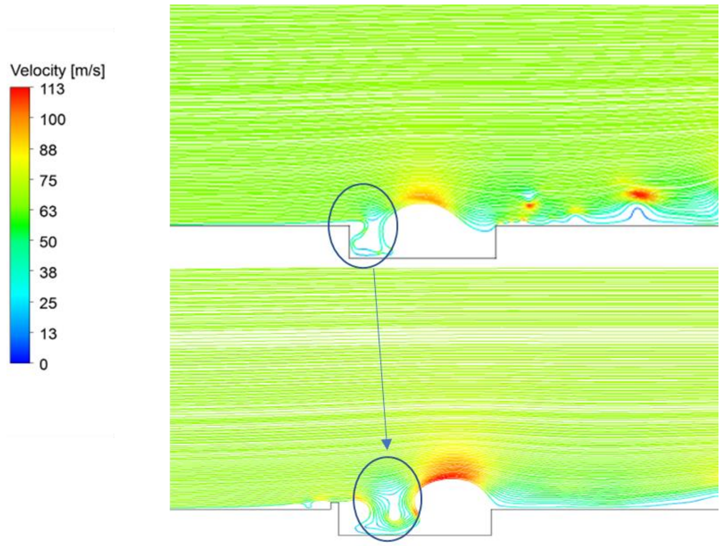

Figure 9. Velocity streamline for base cavity and case 9

3.4. Blowing Effect

From the observations of baseline cavity, it is seen that, the oscillation of the shear layer disturbs the fluid inside the cavity and vortices move towards the cavity region. To restrict the moving capability of the vortices air is blown through the holes from outside to the inside of the flow field. Slot air velocity is taken as 10% of the free stream velocity. From Figure 10, it is seen that in the most effective blowing position 15.3 dB reduction is provided that means that this active control technique is an effective SPL reduction method. In addition, another advantage of this method is according to the flight conditions blowing location and mass flow can be varied. On the other hand, passive methods cannot adapt itself to the flight conditions.

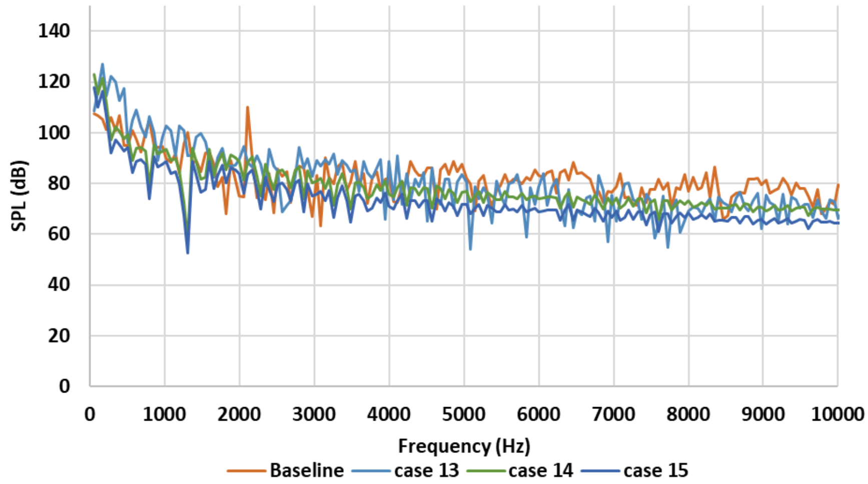

Figure 9. Blowing Effect as a function of SPL

3.5. Comparison of all methods

A comparative study is conducted for different active and passive control techniques for subsonic cavity. SPL analysis reveals that trailing edge curvature method is the most effective in suppressing the tones and minimizing the pressure fluctuations so SPL levels. Trailing edge curvature is followed by the jet blowing technique with reducing to SPL level from 85.7 dB to 70.4 dB. Also leading edge cavity modification seems as a promising technique with decreasing SPL levels 7.47%. Table 3, shows the average reductions in SPL levels with the control methods.

Table 3. Comparison of active and passive techniques with base cavity

Reduction Method	Average SPL(dB)	% reduction
Baseline	85.7	-
Case 4	68.3	20,30
Case 8	73.4	7,47
Case 11	78.4	6,81
Case 15	70.4	10,20

4. Conclusion

Flow over cavity is investigated for practical purposes such as reducing the drag, noise levels, energy consumption, maneuverability of the aircraft. Cavities not only represent the ammunition zones in the aircrafts but also the landing gears, space between wagons of trains, sunroof, windows of the cars. In the all of the application areas, in the cavity zones unavoidable noise is generated to the high amplitude pressure fluctuation. In order to minimize this noise levels, drag reduction and pressure fluctuation passive and active control methods are utilized. In this study, effects of ramped cavity, curvature cavity, and air blowing on sound generated aerodynamically is investigated and compared with each other. According to the results, it is discovered that creating curvature in trailing edge wall among all tested methods is the efficient method with decreasing the SPL level 20.3%. The periodic pressure fluctuation is eliminated and amplitude is lowered efficient with trailing edge curvature modification. Introducing a jet blowing is the second most effective method that reduces SPL levels up to 70.4 dB that means 10.2 % reduction. In this study, mass flow rate of the blowing is not investigated. As a further study, the effect of the blowing velocity should be investigated as at the optimized mass flow rate value, efficiency of the jet blowing method can be increased.

If the results obtained are summarized briefly, with this study it is seen that the tried passive and active control methods are able to reduce the pressure fluctuations so the SPL levels; however, the optimization of the active and passive techniques parameters (angle, location, height, velocity…) should be performed to maximize the efficiency.

References

[1] Koçak, E., Ayli,E. Turkoglu, H., "Numerical Analysis of Aerodynamic and Aeroacoustics Characteristics of Subsonic Rectangular Cavity with Different Aspect Ratios", 5th International Anatolian Energy Symposium,15-17 April 2020.

[2] Meganathan, A.J., An Experimental Study of Low Speed Open Cavity Flows, MSc Thesis, Unıversity of Tennessee, Knoxville, 2000.

[3] Tracy, M.B., Plentovich, B., Cavity Unsteady Pressure Measurements at Subsonic and Transonic Speeds, NASA Technical Paper 3669, Springfield, U.S.A., 1997

[4] Block, P.J.W., Noise Response of Cavities of Varying Dimensions at Subsonic Speeds, NASA Technical Report, TN D-8351, U.S.A., 1976.

[5] Ayli,E., Numerical Analysis of Supersonic cavity flow, MSc Thesis, TOBB ETU, 2012.

[6] Casper, K.M., Wagner, J.L., Beresh, S.J., Henfling, J.F., Spillers, R.W., Pruett, O.M., "Complex Geometry Effects on Subsonic cavity Flows, 53rd AIAA Aerospace Sciences Meeting, Florida, 5-9 January 2015.

[7] Shih, S.H., Hamed, A., Yeuan, J.J., Unsteady Supersonic Cavity Flow Simulations Using Coupled k-epsilon and Navier-Stokes Equations, AIAA Journal, 1994, 32(10), 2015-2021, 1994.

[8] Kaufman, L.G., Maciulaitis A., Clark, R.L., Mach 0.6 To 3.0 Flows Over Rectangular Cavities, Air Force Wright Aeronautical Labs., AFWAL-TR-82-3112, New York, U.S.A., Kasım 1993.

[9] Rizetta ,D.P., Visbal, M.R., Large Eddy Simulation of Supersonic Cavity Flowfields Including Flow Control, AIAA 41(8), 2003.

[10] Bres, G.A., Colonius, T., Direct Numerical Simulations of Three Dimensional Cavity Flows, Journal of Fluid Mechanics: 599, 309-339, Cambridge, UK, Mart 2008.

[11] Chung, K.M., Characteristics of Compressible Rectangular Cavity Flows, Journal of Aircraft, 37(3), 463-468, 2000.

[12] Heller, H., Holmes, H., Covert, E.E., Flow Induced Pressure Oscillations in Shallow Cavities, Journal of Sound and Vibration, 1970,18(4), 545-545.

[13] Saddingtoni, A., Thangamani, T., Knowles,K., Comparison of Passive Flow Control Methods For a Cavity in Transonic Flow, Journal of Aircraft, 2016.

[14] Gupta, A.D., Roy, S., Noise control of subsonic cavity flows using passive receptive channels, 53rd AIAA Science Meetings, 2015.

[15] Meganathan, A.J., An Experimental Study of Low Speed Open Cavity Flows, MSc Thesis, Unıversity of Tennessee, Knoxville, 2000.

[16] Ligthill, M. J "On Sound Generated Aerodynamically," Proc. Roy. Soc. London Ser., Vol. 211(1107), 1952, 564-587,

[17] Yenigelen, E., Aeroacoustic Analysis of open cavities with rounded edges, MSc Thesis, Istanbul Techical University, 2015.

[18] Ffowcs Williams, J. E. and Hawkings, D. L. "Sound Generation by Turbulence and Surfaces in Arbitrary Motion," Philos. Trans. Roy.Soc., A264, 1969., 231-342.

[19] Curle, N. "The Influence of Solid Boundaries Upon Aerodynamic Sound," Proc. Roy. Soc. London Ser., A 231, 1955, 505-514.