Proceedings of the 2018 ASME International Conference and Exposition on Noise Control Engineering
NCAD2018
August 26-29, 2018, Chicago, IL, USA

NCAD2018-6138

CFD BASED LOCK-IN MODELING OF CAVITY-PIPE LINE SYSTEMS

Ted G. Bagwell
The Pennsylvania State University,
Applied Research Laboratory
State College, Pennsylvania, USA

Kristin L. Cody
Naval Nuclear Laboratory
Pittsburgh, Pennsylvania, USA

ABSTRACT

Flow over shallow cavities is a noise concern due to the possibility of flow tone lock-in with acoustic resonators. The principal aim of this work is to understand the factors that contribute to the onset of lock-in using Computational Fluid Dynamics (CFD) models.

CFD models of shallow cavity lock-in to longitudinal acoustic resonators are developed and validated against existing test data from Lehigh University. All simulations are performed using AcuSolve^{TM}. A key technical contribution is the development of admittance inflow and impedance outflow boundary conditions to model the effects of the pipe resonator. The general trends predicted by the CFD models agree with the test data. In particular, the resonator response at the strong interaction point is well represented.

INTRODUCTION

Turbulent flow over shallow and deep cavities can generate high levels of noise. This is especially true when the vortex shedding and the acoustic waves reinforce each other to generate a self-sustained interaction. This interaction is referred to as lock-in. Several researchers have empirically investigated lock-in to acoustic resonators in the past. In particular, Rockwell, *et al.* [1] examined the lock-in process for a shallow cavity-pipe configuration. Similarly, Yang [2] investigated the lock-in due to grazing flow over a deep cavity.

Numerical predictions of lock-in using eddy resolving Computational Fluid Dynamics (CFD) techniques have been successful in the past for deep cavities [3] as well as Helmholtz resonators [4]. To capture the shed vortices, the source region (cavity) always requires the high levels of mesh refinement typically associated with eddy resolving CFD methods, such as Large Eddy Simulation (LES). Although this is computationally expensive, it is achievable in many cases, especially if the noise source region is small. For the deep cavity and Helmholtz resonator cases, the flow in the acoustic domain is stagnant (apart from the acoustic wave). Hence, the large volume acoustic resonator does not require the high levels of mesh refinement typically associated with eddy resolving CFD methods. Lock-in cases where the base flow field and the acoustic resonance coincide, such as the cavity-pipe configuration investigated by Rockwell, *et al.* [1], have not been investigated with CFD, partly due to the cost and complexity required to resolve both the acoustic field and the flow field. In these cases, the large acoustic volume would require LES type mesh refinement, which is cost prohibitive. A practical alternative would be to replace the upstream and downstream sections of the resonator by admittance and impedance boundary conditions. Such boundary conditions must be able to provide an accurate representation of both the mean flow profile and the upstream acoustic environment. Especially critical items for lock-in are prescribing the correct turbulent boundary layer thickness and the acoustic resonator damping level.

Reymen, *et al*. [5], [6] have developed a time domain impedance formulation which could be used to describe an upstream acoustic environment within a CFD code. To date, this model has only been used as an impedance (wall) boundary condition. This model could however be extended to form an admittance formulation for inflow boundary conditions as well as an impedance formulation for the outflow boundary conditions required to model longitudinal resonators.

The principal aim of this work is to extend the rational function approach employed by Reymen, *et al*. to inflow and outflow boundary conditions. The basic approach is described in the next section. The model is demonstrated and compared to the experimental data from Rockwell, et al. [1]. All the CFD simulations utilize the commercial solver AcuSolve^{TM} [7] from Altair Engineering.

CFD CODE and MODELING

The Lehigh pipe-cavity system is illustrated in Figure 1. Modeling the entire acoustic resonator within a CFD calculation is extremely difficult due to several factors. First, the size of the acoustic resonator is much greater than the size of the noise source. Computing the entire resonator using LES would increase the cost of the simulation by at least an order of magnitude, which is prohibitively expensive. Second, it is difficult to adequately represent the acoustic damping in the resonator. For example, in the Lehigh test, the observed damping was found to be proportional to the inverse square root of the frequency. Although damping can be applied to the CFD calculation, it is difficult to reproduce this behavior in the time domain. It is difficult to satisfy the acoustic boundary conditions as well as the desired boundary conditions on the noise source simultaneously. In the Lehigh case, the upstream boundary condition is open. Although this could be handled with a total pressure boundary condition in the CFD, it would be difficult to specify the boundary layer characteristics incident to the cavity independently.

For CFD calculations of noise sources, it is desirable to replace the pipe line acoustic resonator with boundary conditions that effectively reproduce the desired response characteristics. For example, the upstream pipe could be removed as is illustrated in the second image Figure 1. In this case, the inflow boundary velocity would need to be a function of the inflow pressure, which is an admittance boundary condition. Similarly, the downstream pipe could be removed as is also illustrated in the second image in Figure 1. As pressure is typically set on the CFD outflow plane, the pressure on the outflow plane would be specified as a function of velocity, which is an impedance boundary condition.

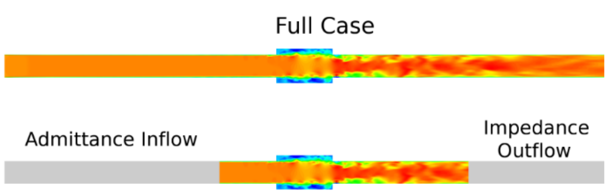

Figure 1. Illustration of pipe cavity system.

For the admittance or impedance boundary conditions to be physically correct, they must adequately represent the portion of the resonator that is not in the CFD simulation domain. Reymen, *et al*. [5], [6] introduced a time domain impedance formulation based on recursive convolution. This boundary condition was shown to be effective as an impedance boundary condition for acoustic liners and is the basis for the boundary conditions proposed in this paper.

The following discussion focuses on the admittance inflow boundary condition. The impedance boundary condition is identical if the roles of velocity and pressure are reversed. The first step in formulating the inflow boundary condition is to decompose the axial component of the inflow velocity into a base flow component and an acoustic component, as defined in Equation (1). For this work, it will be assumed that the acoustic resonator can be described by one dimensional modes. Hence, the frequency range of interest must be below the cut on for higher order acoustic modes.

$$
\begin{aligned}
&U(\underline{x},t) = U_{Flow}(\underline{x},t) + U_{Acoustic}(t) \\
&U_{Flow}(\underline{x},t) = \text{prescribed} \\
&U_{Acoustic}(t) = A * P'_{Inflow}(t) \\
&P'_{Inflow} = \left(P_{Inflow}(t) - \overline{P}_{Inflow}\right) \\
&P_{Inflow} = \text{Area Averaged Inflow Pressure} \\
&\overline{P}_{Inflow} = \text{Time and Area Averaged Pressure}
\end{aligned}
\tag{1}
$$

The base flow component, U_{Flow}, is prescribed. Here, the mean velocity profile is specified from the mean incident boundary layer in the Lehigh experiment. This could also be extended by including resolved turbulence characteristics from the experiment.

Following Reference [6], the acoustic component is specified in frequency space as:

$$
\begin{aligned}
&U_{Acoustic}(\omega) = A(\omega) \bullet P'_{Inflow}(\omega) \\
&A(\omega) = \sum_{k=1}^{S} \frac{B_k}{j\omega + \lambda_k}
\end{aligned}
\tag{2}
$$

The poles, λ_k, and coefficients, B_k, are constants which will be defined shortly.

Equation (2) is in the form of a rational function. This multiple pole form is especially well suited for representing the multiple resonances of a one dimensional resonator. Equation (2) is equivalent to a convolution in time space and can be calculated as a recursive convolution. The resulting discrete time domain form is given by:

$$
\begin{aligned}
&U_{Acoustic}(n\Delta t) = \operatorname{Re}\left\{\sum_{k-1}^{S} B_k \psi_k(n\Delta t)\right\} \\
&\text{where} \\
&\psi_k(n\Delta t) = p(n\Delta t)\frac{1-e^{-\lambda_k \Delta t}}{\lambda_k} \\
&\qquad + \left(p((n-1)\Delta t) - p(n\Delta t)\right)\frac{e^{-\lambda_k \Delta t}\left(-\lambda_k \Delta t - 1\right)+1}{\lambda_k^2 \Delta t} \\
&\qquad + \psi_k((n-1)\Delta t)e^{-\lambda_k \Delta t} \\
&p(n\Delta t) = P'_{Inflow}(n\Delta t) \\
&\Delta t = \text{time step increment}
\end{aligned}
\tag{3}
$$

The variables ψ_k are accumulators. Although Equation (3) appears to be fairly complicated, its implementation into a CFD code is very computationally efficient in terms of both memory and processing time. In particular, neither time derivatives nor long time histories are required to advance the solution. These equations are implemented in approximately 600 lines of user coding for AcuSolve™. The poles, λ_k, and coefficients, B_k, of the rational function can be fitted in the frequency domain using the *Vector Fitting* procedure of Gustavsen, *et al*. [8], [9], [10].

The actual form, or shape, of the admittance function in frequency space must be specified for the desired configuration. There are several methods to determine this including: empirical data, numerical modal solutions, other CFD calculations, or analytic solutions. Here, the analytic solutions are derived from the linearized Euler equations. The damping rate is set to match the experiment in the frequency range of interest. Outside of the frequency range of interest, the damping is adjusted in order to maintain stability in the CFD simulations. For the cases presented in this report, the damping is set to achieve a quality factor as follows. For $f < 265$ Hz: The quality factor is linearly proportional to frequency. This low frequency damping is used to minimize possible undesirable low frequency oscillations in the CFD calculation. For 265 Hz $< f <$ 1200 Hz: The quality factor is $Q=100\sqrt{f/530}$. This value is chosen to match the Lehigh experimental data. For $f >$1200 Hz: The quality factor is exponentially reduced to unity at the maximum frequency. This high frequency artificial damping is used to minimize oscillations outside of the region of interest. The maximum frequency is set at $1/(2\Delta t)$, where Δt is the physical time step size in the simulation.

The commercial CFD code, AcuSolve™ [7] is used for all calculations in this report. The fluid compressibility is modeled as isentropic air. The dynamic LES model is used to model turbulence [11], [12]. The time averages used in Equation (1) are defined by the running average procedure in AcuSolve™ using 500 time steps.

VERIFICATION of the BOUNDARY CONDITIONS

Verification is intended to show that the method is performing as expected. The configuration cited is the P37 cavity tested in air at Lehigh University [1]. The main pipe consisted of two 12 inch long, 1 inch inner diameter aluminum pipe section. An axisymmetric 2.5 inch long by 0.25 inch deep cavity is centered between the two pipe segments. The upstream end was terminated by a large settling chamber, while the downstream end was open to the atmosphere. Hence, both the pipe inlet and outlet are acoustically open or pressure release boundaries. A dynamic pressure transducer located 5 inches upstream of the cavity leading edge was used to measure the acoustic pressure.

In the experiments, the configuration is a circular pipe. For demonstrating the proposed CFD boundary conditions, a limited span, planar simulations are used in lieu of the circular experimental configuration. A 2.5 inch long and 0.25 inch deep cavity in a 1 inch channel is simulated, as is shown in Figure 1. Out of the total 12 inch inlet pipe length, 2.5 inches are in the CFD domain. The remaining 9.5 inch section is modeled with the admittance boundary condition. The mean flow profile is taken from the laser Doppler velocimetry (LDV) velocity data in the Lehigh test. Six inches of the downstream pipe are in the CFD domain. The remaining 6 inch section is modeled with the impedance boundary condition.

A view of the cavity mesh is shown in Figure 2. The block structured, hexagonal mesh is generated in Pointwise V17.3 [13]. There are 121 and 31 points across the cavity length and depth respectively. The maximum axial spacing is 0.029 inches. The near wall spacing is set to 0.005 inches. The span wise width of the cavity is 0.5 inches with 11 points across the span. Periodic boundary conditions are used in the span wise direction. It is noted that this reduced span planar model is insufficient to accurately model a fully three dimensional circular pipe configuration. However, it is capable of reproducing representative physics, while avoiding the spurious wake mode oscillations that can be encountered in two dimensional simulations [15]. In total there are approximately 348,000 vertices in the mesh. A 25 μ-second time step is used, which corresponds to approximately 76 time steps per period for a 530 Hz oscillation.

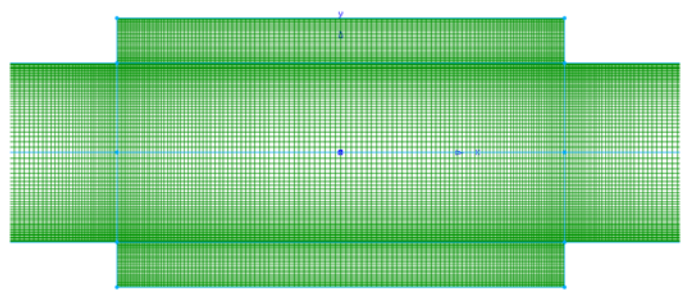

Figure 2. Cavity mesh for the planar configuration.

Prior to running flowing simulations, a zero flow, ring down (hammer or shaker) simulation is performed to ensure that the proposed boundary conditions can accurately represent the acoustic domain. The excitation (hammer) is an exponential time pulse of axial momentum located approximately two inches downstream of the cavity. The results are shown in Figure 3. The Quality factor for the 530 Hz acoustic resonance is 91, which is consistent with the experimental data.

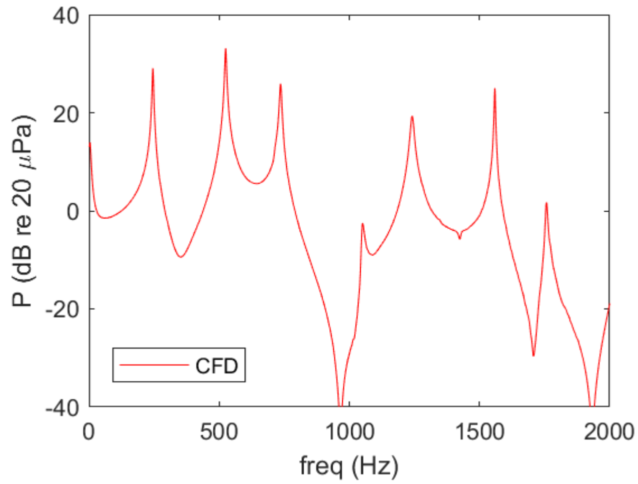

Figure 3. Ring down simulation resonator response.

As in experiments, flow sweeps are performed using the CFD model. The acoustic pressure response versus velocity and frequency is shown in Figure 4. In the experimental data, shown on the upper image, two strong Strouhal stages (or Rossiter modes) are evident, as are indicated by the lines on the plot. In particular, there is a strong acoustic mode-vortex shedding interaction that occurs for the 530 Hz acoustic resonance at approximately 35 m/s. This interaction (or lock-in) is used as the principal value for validating the CFD results, which are shown in the lower image. As in the experiment, the CFD simulations are predicting approximately the same Strouhal shedding source observed in the experiment.

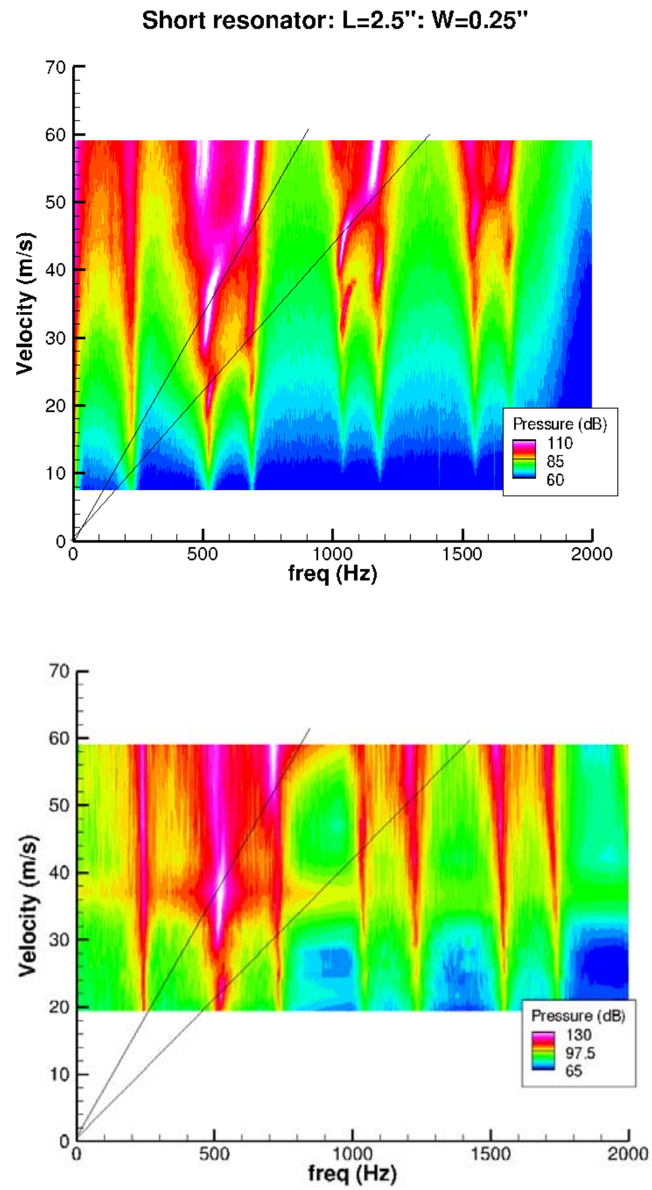

Figure 4. Pressure amplitude response versus velocity and frequency from the 2.5 inch long by 0.25 inch deep cavity and short pipe. Upper: Experiment. Lower: CFD.

The amplitude of the 530 Hz acoustic resonance amplitude versus flow is shown in Figure 5. The planar CFD simulations do show the same trends observed in the experiments. Quantitatively, the CFD results are between 10 and 20 dB higher than the experiments, which is expected for the planar CFD domain approximation. Using the Resonance Response Method (RRM) of Mendelson [14], the Strength of Lock-in (SoL), or dB amplitude of the peak beyond the linear background, is estimated to be 40 dB for the experiment and 30 dB for the CFD. Both values would be considered as a strong lock-in. The fact that the SoL is low for the CFD is primarily due to the fact that the CFD is predicting a stronger response off of lock-in response than the test data. This difference is partially due to fact that the CFD uses a coarser frequency resolution (or shorter time window) than the test.

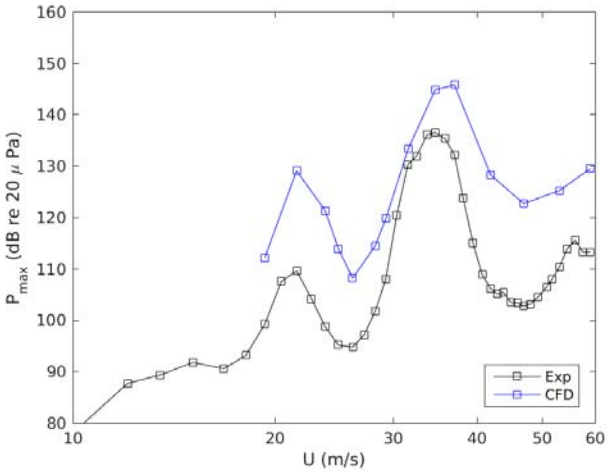

Figure 5. 523 Hz acoustic resonance response comparison between test and CFD.

In summary, this verification exercise has demonstrated that the inflow admittance and outflow impedance boundary conditions are capable of reproducing the effect of the inflow pipe and hence are acceptable boundary conditions for further study.

SUMMARY

Flow over shallow cavities is a noise concern due to the possibility of flow tone lock-in with acoustic resonators. CFD models of shallow cavity lock-in to longitudinal acoustic resonators are developed and validated against existing test data from Lehigh University. The key technical contribution is the development of an admittance inflow and impedance outflow boundary conditions to model the effects of the upstream and downstream pipe resonators. The general trends predicted by the CFD models agree very well with the test data.

Although substantial progress has been made towards validating CFD for lock-in applications, several future work items are required to fully demonstrate acceptability. Firstly, these simulations employ a planar approximation as an approximate, cost effective alternative to the fully circular pipe configuration. Further work should consider performing full pipe simulations. At present, the inflow admittance and outflow impedance boundary conditions are explicit to the linear solver. Hence, twice as many iterations are required to converge the solution at each time step. A fully implicit version would be beneficial for reducing the computing cost.

REFERENCES

1. Rockwell, D., Lin, J.-C., Oshkai, P., Reiss, M. and Pollack, M. "Shallow Cavity Flow Tone Experiments: Onset of Locked-on States". *Journal of Fluids and Structures*. Vol. 17, 2003. More detailed data from this report are given in the OSTI document LM-00K072.

2. Yang, Y. "*Generation of Tones Due to Grazing Shear Flow Past a Deep Cavity*". Ph.D. Thesis, Lehigh University, 2005.

3. Bagwell, T.G. "CFD Simulation of Flow Tones from Grazing Flow Past a Deep Cavity". **IMECE2006-15633**. *Proceedings of IMECE06, 2006 ASME International Mechanical Engineering Congress and Exposition*. November 5-10, 2006. Chicago, IL, USA.

4. Inagaki, M., Murata, O. and Kondoh, T. "Numerical Prediction of Fluid-Resonant Oscillation at Low Mach Number" *AIAA Journal*, Vol. 40, No. 9. 2002.

5. Reymen, Y., Baelmans, M. and Desmet, W. "Time-Domain Impedance Formulation Based on Recursive Convolution" **AIAA-2006-2685**. *Proceedings of the 12th AIAA/CEAS Aeroacoustics Conference*. May 8-10, 2006. Cambridge, MA, USA.

6. Reymen, Y., Baelmans, M. and Desmet, W. "Time-Domain Impedance Formulation Suited for Broadband Simulations" **AIAA-2007-3519**. *Proceedings of the 13th AIAA/CEAS Aeroacoustics Conference*. May 21-23, 2007. Rome, Italy.

7. AcuSolve Command Reference Manual. Version 13.0. Altair Engineering. 2014.

8. Gustavsen, B. and Semlyen, A., 1999 "Rational approximation of frequency domain responses by vector fitting", *IEEE Trans. Power Delivery*. Vol. 14, no. 3.

9. Semlyen, A and Gustavsen, B. 2000 "Vector fitting by pole relocation for the state equation approximation of nonrational transfer matrices", *Circuits Systems Signal Process*. Vol. 19, no. 6.

10. http://www.sintef.no/Projectweb/VECTFIT/.

11. Germano, M., Piomelli, U. Moin, P. and Cabot, W. H. 1991. "A Dynamic Subgrid-Scale Eddy Viscosity Model. *Physics of Fluids A*. Vol 7.

12. Jansen, K.E. 1999 "A Stabilized Finite Element Method for Computing Turbulence". *Computer Methods in Applied Mechanics and Engineering*. Vol. 174.

13. http://www.pointwise.com/.

14. Mendelson, R.S. 2003 "Methods of Measuring Lock-in Strength and Their Application to the Case of Flow over a Cavity Locking into a Single Side Branch" AIAA Paper 2003-3106.

15. Shieh, C. and Morris, P., 2001. "Comparison of two-and three-dimensional turbulent cavity flows" **AIAA-2001-0511**. *Proceedings of the 39th Aerospace Sciences Meeting and Exhibit*. January 8-11, 2001. Reno, Nevada.