2022

Numerical Study of the Aerodynamic Noise and Vibration Due to Pulsive Discharge Gas Jet in Hermetic Compressors

Dazhuang He

Yidan Cui

Davide Ziviani

Yangfan Liu

Numerical Study of the Aerodynamic Noise and Vibration Due to Pulsive Discharge Gas Jet in Hermetic Compressors

Dazhuang HE[1]*, Yidan CUI[1], Davide ZIVIANI[1], Yangfan LIU[1],

[1]Purdue University, School of Mechanical Engineering,
West Lafayette, Indiana, USA
he385@purdue.edu; cui182@purdue.edu; dziviani@purdue.edu; liu278@purdue.edu;

* Corresponding Author

ABSTRACT

The turbulent nature of refrigerant discharge flow and complicated flow path in hermetic compressors potentially render flow induced noise a source of noise, vibration and harshness (NVH) effects during the operation of hermetic compressors. In the presented study on flow induced of hermetic compressors, a fluid-structure interaction simulation model was developed, where the interior refrigerant flow field and the structural vibration are coupled together. The thermomechanical aspects of compressor operation are resolved in the compressor mechanistic model, based on a compressor modelling platform developed in previous studies. The fluid-structure interaction simulation describes the two-way coupled interaction between the compressor discharge flow and hermetic shell vibration. The interior hermetic shell surface is subjected to turbulent fluid load due to compressor discharge, and the fluid domain is bounded by vibrating shell. Simulations were conducted to validate the models developed in the current work, and the exterior noise radiation is evaluated based on shell vibration to demonstrate how these modeling tools can help compressor manufacturers to gain better understanding of the physical reasons behind NVH effects of compressors.

1. INTRODUCTION

The operation of HVAC compressors inevitably yields to undesired NVH effects. Hermetic compressors are widely used in refrigeration and air conditioning applications and mitigating NVH effects is an important factor not only to ensure customer satisfaction, but also to meet noise standards depending on the application.

One of the major sources of noise and vibration is the discharge refrigerant flow and its interactions with the structure of the compressor. Specifically, there are two aspects of the discharge refrigerant flow and each result in different noise and vibration characteristics: (1) the periodic pulsive fluid volume change at the discharge valve location; and (2) the turbulent perturbations caused by the discharge gas jet flow. The volume change of the compression chambers during a working cycle acts as a periodic acoustic monopole source which usually contributes to the gas pulsation noise, i.e., the narrowband peaks in the noise and vibration spectra (He *et al.*, 2021). Moreover, the turbulence in the gas jet, when interacting with the inside shell surface of the discharge cavity, contributes to the aerodynamic noise, i.e., broadband features in the noise and vibration spectra.

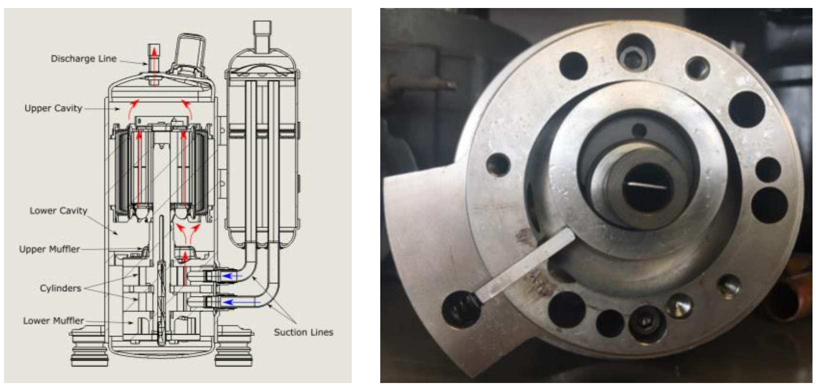
The gas-borne noise and vibration of hermetic compressor has been extensively studied based on the four-pole method (Soedel, 2007), in which discharge gas pulsation is modelled as acoustics perturbation that propagates in a system consisting of acoustic filter components with lumped parameters. The four-pole method is capable of resolving narrow band features associated with compressor operation speed. The generation of broadband noise and vibration features, however, involves fluid-structure interactions that cannot be described by lumped compressor characteristics. In order to predict broadband mechanism numerically, higher order models are necessary. Specifically, interior fluid field must be solved in order to evaluate broadband gas-borne noise and vibration. Thus, computational fluid dynamics (CFD) is an appropriate technique to tackle the broadband noise mechanisms. CFD has not been widely used for noise and vibration analysis of compressors, but it has been a useful tool for the analysis of refrigeration compressor performance. Attempts were made to CFD to predict pressure variation in compressor cavity and its effect on thermal performance of compressors (Nakano and Kinjo, 2008). The internal flow field within hermetic shell was solved to analyze how the choice of refrigerant affects the pressure drop (Birari *et al.*, 2006). Researchers have also used CFD to solve the pressure field in compression chamber with respect to compressor operation and the resulting fluid load applied on compressor structures (Wu *et al.*, 2019).

The studies mentioned above are limited for a specific aspect of compressor. However, a compression process consists of different phenomena which interact with each other in a short period of time. The gas pulsation noise and vibration are generated through acoustical response of the compressor cavity as well as vibration response of the hermetic shell, both of which require modeling of three-dimensional continuous systems. In addition, the thermodynamic processes in the working chamber(s), the valve motion, gas pressure pulsations, turbulent gas flows, shell vibration and sound radiation are strongly coupled.

In a study of narrow band gas pulsation noise, multiple simulation models are developed to resolve such a multi-physical phenomenon (He, 2021). A similar approach can be taken to resolve broadband aerodynamic noise and vibration generation of hermetic compressor.

2. COMPRESSOR MECHANISTIC MODEL

In this work, a rotary rolling-piston compressor for air conditioning applications has been used as a case study. The typical structure of the compressor is illustrated in Figure 1(a). Rolling piston compressors usually feature either a single or dual-cylinder configurations depending on the capacity range as well as if the compressor is a single or two-stage. A close-up view of the vane and cylinder assembly is reported Figure 1(b).



(a) (b)

Figure 1: (a) typical structure of a rolling piston compressor; (b) compression chamber of a rolling piston compressor

Compressor mechanistic model is the simulation model that provides the discharge velocity/pressure profile through the compressor discharge port. Theoretically, the compressor mechanistic model is based on solving mass and energy conservation equations. The core structure includes working chamber volume calculations, evaluation of leakage flows, heat transfer within the working chamber, mechanical and frictional losses, suction and discharge valves (where applicable) and an overall energy balance to account for additional heat losses through the compressor shell. A complete geometric model of rolling piston compressor has been developed and integrated into the existing PDSim platform (Bell *et al.*, 2020) (Ziviani *et al.*, 2020). The resulting discharge velocity profile is shown in Figure 2, which serves as the input boundary condition in the fluid simulations in this study.

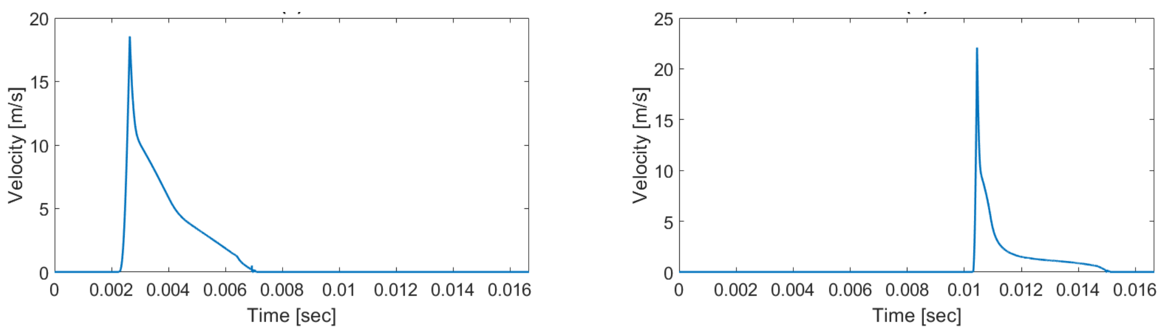


Figure 2. Discharge velocity of the compressor

3. INTERIOR FLOW AND ACOUSTICS

Aerodynamically induced noise is typically generated by the turbulent flow field inside the hermetic shell, the relation between interior flow field and acoustics needs to be clarified. In principle, sound is a class of fluid medium perturbation that is governed by wave equation and propagates at the speed of sound. In other words, sound generation and propagation, in principle, can be resolved if fluid field is solved. In many applications where aerodynamic noise is of interest, the non-linear flow region and linear sound propagation region can be separated, such as free jets or fans. In these cases, where non-linear fluid domain and linear acoustic domain can be distinctively separated, the acoustics can be solved separately for the sake of computation efficiency. There are two approaches for solving acoustics separately: (1) acoustic analogy and (2) linearized equations. Acoustic analogy method is based on the analogy between non-linear noise inducing mechanism and simple acoustic sources, which was coined by Lighthill (1952). Linearized equation method, specifically linearized Euler/Navier-Stokes equation (LEE/LNS) method, is based on the decomposition of fluid variables into a mean variable and a perturbation variable, which allows the linearization of fluid equations. It was discovered that acoustic analogy method is more suitable for predicting sound radiated from turbulent flow regions to free space and are not capable to predict aerodynamic noise within a turbulent flow field which, in the application of this study, is the turbulent refrigerant flow inside the compressor cavity. The linearized equation method is capable to resolve flow induced sound with the presence of a background flow field. However, it is not usually implemented in interior problems. In typical application of LEE/LNS method, the computation domain of LEE/LNS and RANS/LES fluid simulations are separated. At the boundary between the two computational domains, LEE/LNS requires an input boundary condition based on the fluid simulation results. For hermetic compressors, the interior acoustic domain and fluid domain occupy the same region, so the LEE/LNS method is not directly applicable in the interior of hermetic shell. For all these reasons, a direct computation of the unsteady interior fluid field is needed to resolve acoustic perturbations. The fluid perturbation is solved, and the techniques of fluid-structure interaction (FSI) were used to compute the response of the hermetic shell. It should be noted that the resolved perturbations using this method is not necessarily acoustic perturbations. In a sense, interior acoustics in this study implies general fluid perturbations, instead of exclusive acoustic perturbations.

4. FLUID-STRUCTURE INTERACTIONS

As mentioned in previous sections, noise and vibration are caused by the turbulent unsteady fluid loads applied on the interior structure of a hermetic compressor. Commonly, co-simulation of fluid and acoustics is the way to resolve the aerodynamic noise, in which either acoustic analogy method or linearized Euler/Navier-Stokes equations can be used to solve for the aerodynamic noise based on the fluid field solutions. In these methods, the one-way coupling between fluid dynamics and acoustics has been assumed – that is, acoustic perturbation does not affect fluid field. This assumption is true in most of applications involving aerodynamic noise, such as fans, vehicles, etc., where the far-field acoustic perturbation is much smaller in magnitude than fluid field variations. However, in the application of hermetic compressors, there is no acoustic far-field within the shell, and the disparity in magnitude between acoustic and fluid dynamics perturbation is not as significant, because the cavity is pressurized. In addition, the hermetic shell and interior acoustic field affects each other due to the abrupt discharge process. All these characteristics of our application support the use of fluid-structure interaction to resolve the aerodynamically induced noise and vibration. Fluid-structure interaction can be used to model phenomena where fluid and deformable shell structure affect each other. Both the unsteady fluid load on the structure and the structural velocity transmission to the fluid are taken into account. In our application, the displacements of the shell are assumed to be small enough for the geometry of the fluid domain to be considered as fixed during the interaction. Figure 3 shows the coupling between different computation domains

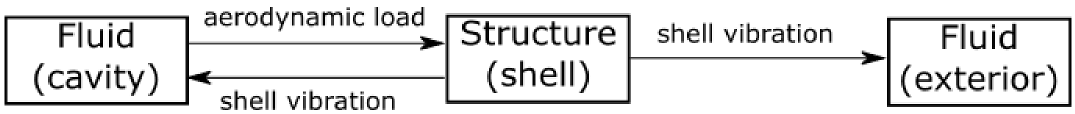


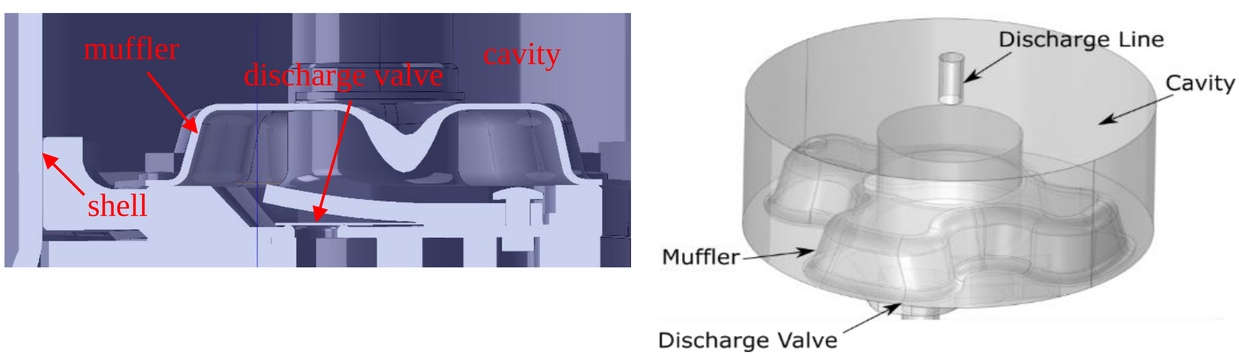
Figure 3: Couplings of FSI simulation

The acoustics excitation is total stress (pressure + viscous stress) on the interior wall. The value of total stress is extracted from CFD simulation. The effect of turbulence is included in the viscous stress, specifically the additional stress resulted from turbulence eddy viscosity, which is an intermediate result of turbulence model. The reason to use total stress as the excitation is because that is the dipole (loading) type of acoustic excitation. In this study, the boundary is impermeable, so that surface monopole does not exist on the domain boundary except for domain inlet. The effect of inlet monopole at the domain inlet has been studied by He *et al*. (2021). After excluding surface monopoles, the surface dipole is the most relevant sound source, and according to acoustic analogy theory (Ffowcs-Williams and Hawkings, 1969), the surface dipoles are induced by total wall stress.

5. RESULTS AND DISCUSSION

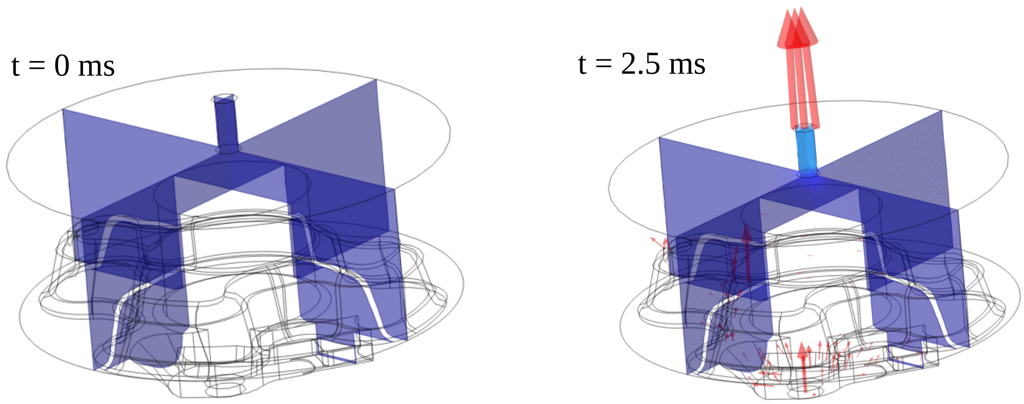
5.1 Configurations of CFD-acoustics coupled simulation

A compact compressor shell and cavity computation domain was used to test simulation procedure. The topology of the compressor cavity interior structure was shown in Figure 4. The discharge pulsation profile obtained from compressor mechanistic model (Figure. 2) is the variation of refrigerant discharge velocity in one cycle of compressor operation (1/60 s) and is applied on the discharge port as inlet boundary condition. Snapshots of resulting flow field is shown in Figure 5.



(a) (b)

Figure 4: (a): Vicinity of compressor discharge port; (b): Compact CFD simulation domain



(a) (b)

Figure 5. (a) Initial condition of flow field; (b) flow field during discharge process

The effect of fluid-structure coupling is investigated based on the simulation using the above-mentioned compact computation domain (Figure 4) for the sake of efficiency. There are three different ways to deal with fluid-structure coupling: structure velocity on fluid, fluid load on structure and two-way coupling. Specifically, the fluid field exerts a fluid loading on structure and thus induces motion of structures. The moving structure, on the other hand, induces fluid motions because structure velocity introduces fluid perturbations. If both effects are considered, the coupling between fluid and structure is referred as a two-way coupling. For the purpose of computation efficiency, one-way coupling is a compromise of the complete two-way coupling, in which either the fluid-to-structure coupling (fluid load on structure) or structure-to-fluid coupling (structure velocity on fluid) is considered. Since the excitation is on the fluid domain, the one-way coupling simulation conducted in this project assumes that structure velocity on fluid would yield zero shell vibration, thus it only considers fluid load on structure when computing shell vibration. Results from one-way and two-way couplings are compared based on the fluid velocity magnitude at muffler outlet and the side shell vibration displacement magnitude.

Figure 6 shows the comparison of the simulated fluid velocity magnitude and shell vibration displacement magnitude. It is found that the fluid-structure coupling mechanism has little influence on fluid side but can significantly affect the shell vibration response. When two-way coupling is used, the shell vibration displacement is much smaller than the result with only one-way coupling. This result can be explained by the lack of fluid reactance, because the shell vibration velocity and fluid velocity are in-phase with each other in most of frequencies. Therefore, with two-way coupling, the energy of shell vibration can be absorbed by fluid without delay. The fact that difference between one-way and two-way coupling is large in terms of shell response means that the exterior sound radiation is heavily affected by coupling too. In order to capture exterior sound radiation accurately, the inclusion of two-way coupling in simulation is necessary.

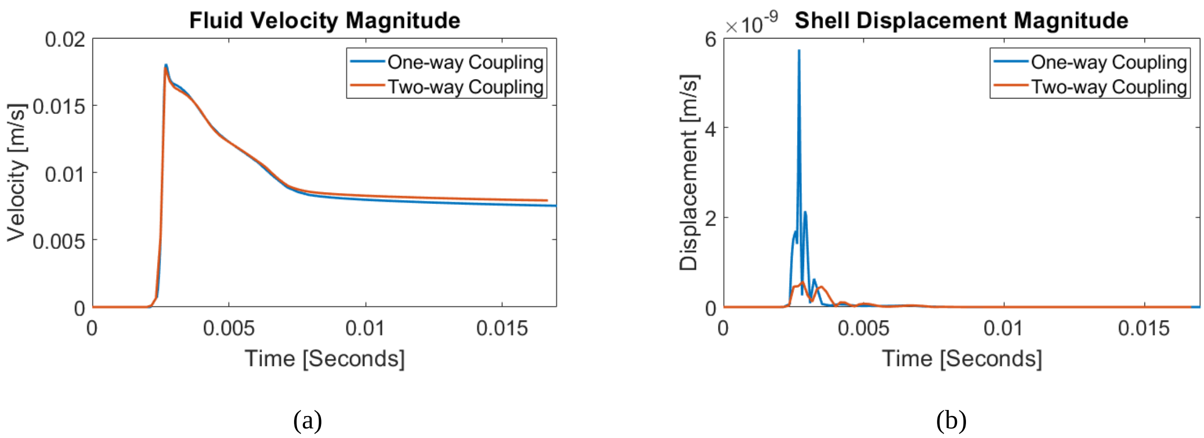


Figure 6. Comparison between one-way and two-way fluid-structure coupling

Another simulation configuration parameter that needs to be determined in this application is the choice of turbulence model. As shown in Figure 18, the flow field is pulsive. The velocity grows from zero to its maximum value in 1 millisecond, which requires an abrupt decrease in time step size. The abrupt decrease may induce convergence issues in CDF simulations, and the convergence of simulation depends on the choice of turbulence model. After trying 7 different turbulence models (k-epsilon, k-omega, SST, Spalart-Allmaras, v2-f, L-VEL, algebraic), only k-epsilon, k-omega and Spalart-Allmaras turbulence models yielded converging solution.

Fluid velocities simulated by using different turbulence models at three locations (two muffler outlets and a cavity outlet) are compared and the effect of turbulence models on CFD solution is shown in Figure 7. In general, the difference simulation results with or without a turbulence model is significant, but the exact choice of turbulence does not have noticeable impact on the solution. As a result, the choice of turbulence model for our application should be made mainly based on their robustness and convergence rate. Among the tested turbulence models, k-epsilon model renders the fastest convergence. Therefore, k-epsilon model is used in the simulation for this application.

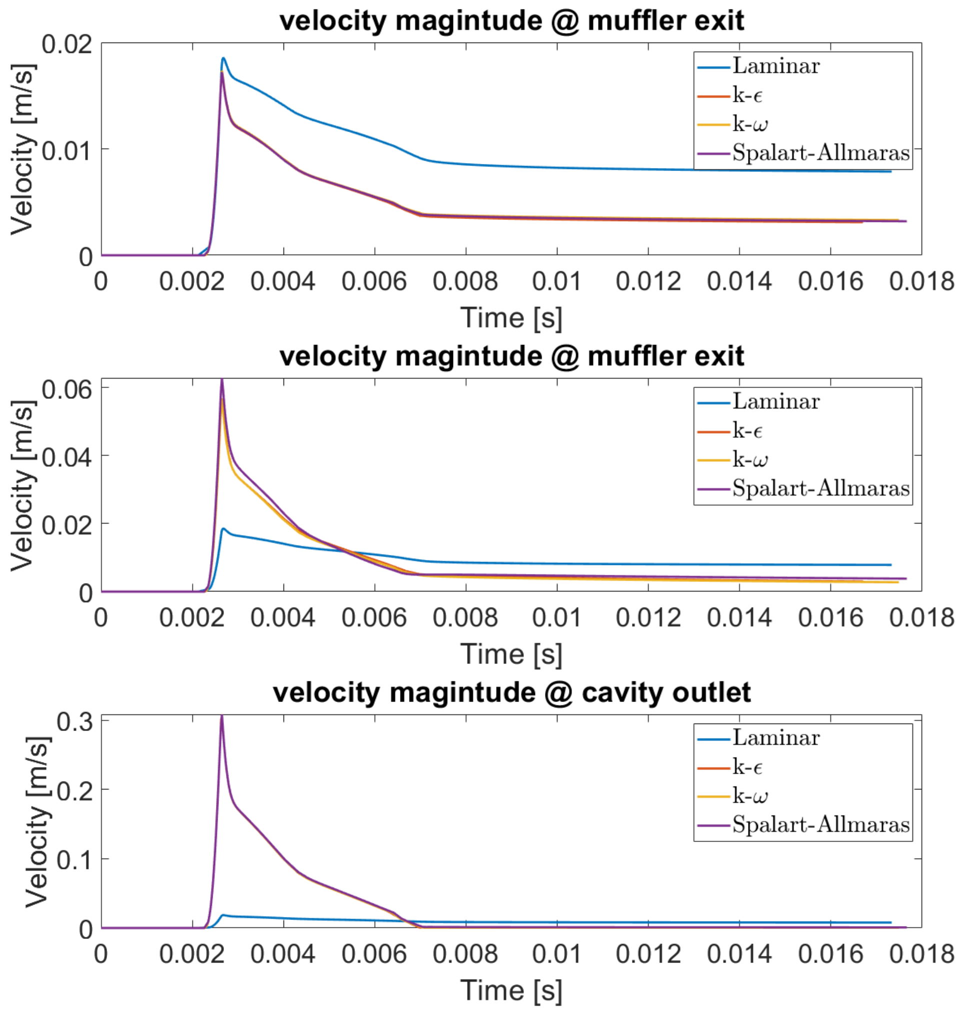
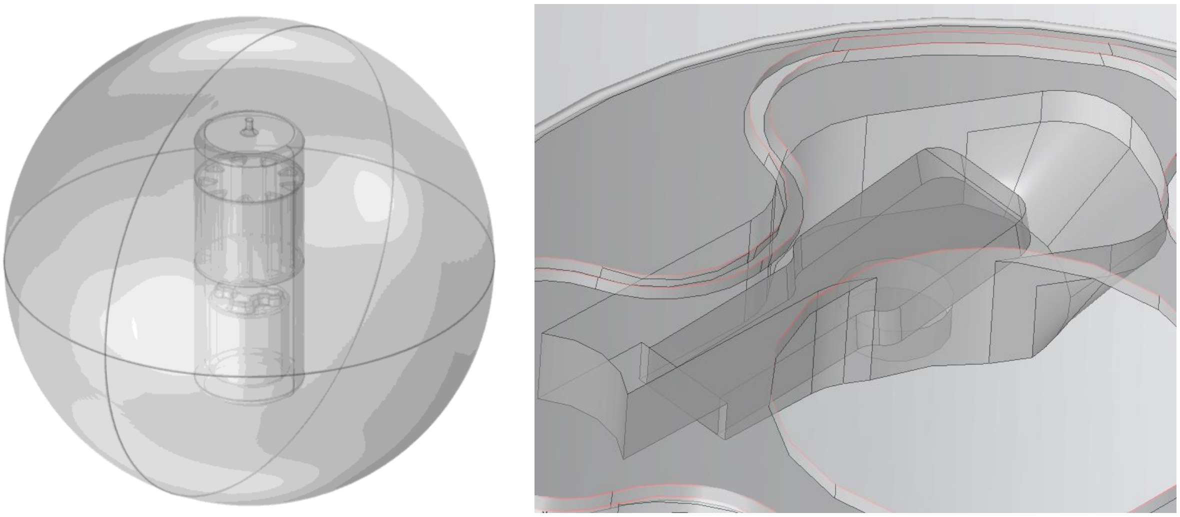


Figure 7. Effect of turbulence model on fluid field solution

5.2 CFD-acoustics coupled simulation for compressor geometry

Based on the configurations explained in previous sections, CFD-acoustics coupled numerical simulation with fluid-structure interaction is conducted to study the aerodynamically induced noise and vibration of hermetic compressors. Figure 3 shows the general setup of the simulation. The interior geometry of the compressor is simplified, and mesh is generated based on a simplified geometry (Figure 8). The turbulent refrigerant gas flow is coupled with shell vibration and exterior sound radiation, with input boundary condition generated by compressor model developed in PDSim (Figure 2).



(a) (b)

Figure 8. (a) Computation domain; (b) close-up view of discharge port

A snapshot of time domain solution is shown in Figure 9, in which the solution on all three domains (interior fluid, shell, exterior fluid) are shown. The result of simulation indicates that the aerodynamically induced noise and vibration mainly occur on the upper and lower caps of the shell. In the study of gas pulsation noise, numerical solution also revealed that the upper and lower caps are where major round radiation occurs (He *et al.*, 2021).

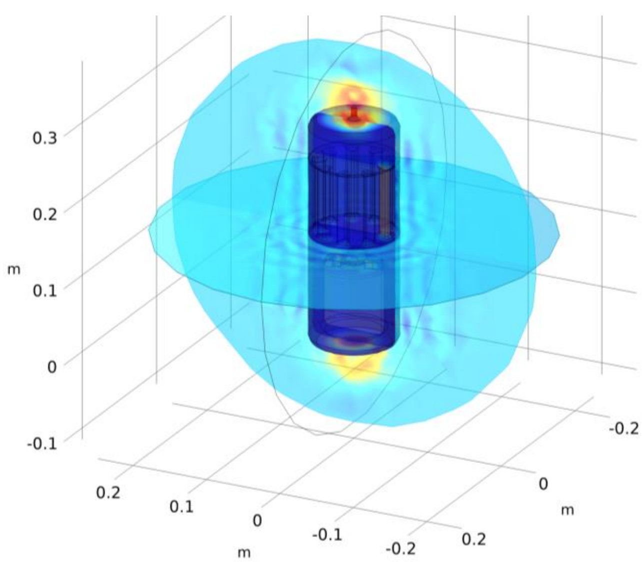


Figure 9. A snapshot of the solution in cavities, on shell and exterior computation domains

Twelve domain/boundary probes in total are setup, and the frequency domain results are shown in Figure 10. Probes are located in interior flow field, on the shell and in the exterior acoustic field to extract time domain responses --- interior flow field velocity, shell vibration displacement and exterior sound pressure. The power spectral densities of time domain results are computed and shown in Figure 10. The power spectral densities of exterior sound also imply that the top/bottom caps are where aerodynamically induced noise radiation occurs. The power spectral density of cavity velocity and shell vibration shows strong linear relation (coherence = 1) across all the frequency up to 50 kHz, which indicates that the acoustic perturbations do not cause shell resonances, and that the shell vibration linearly induce fluid perturbations. Due to the fact that fluid flow is a non-linear phenomenon while shell vibration and sound emission are linear phenomenon, strong linear relation between cavity flow field and shell vibration means the shell induces little non-linear perturbation in the cavity.

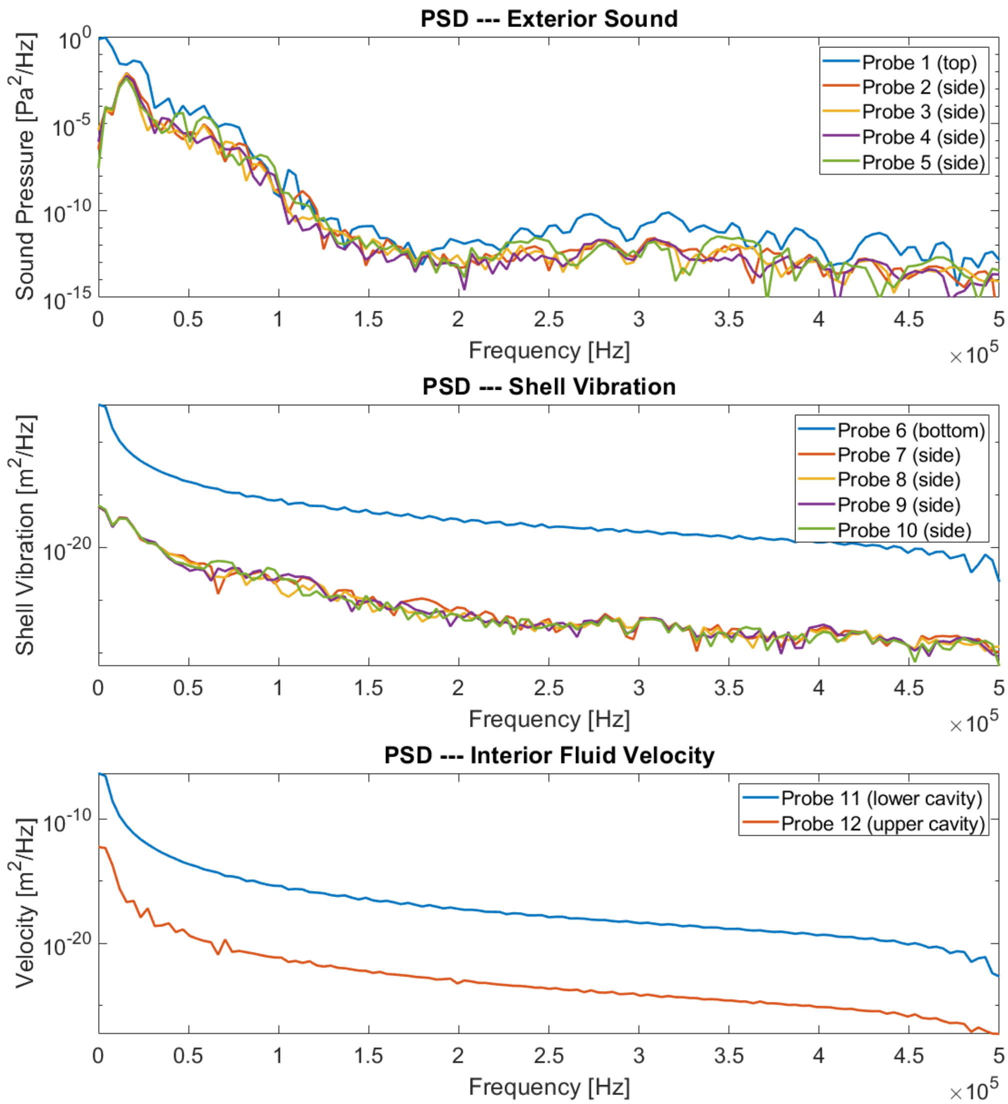


Figure 10. Probe results

Probe 1-5 --- exterior sound field; Probe 6-10 --- shell; Probe 11-12 --- interior flow field

6. CONCLUSIONS

A simulation model that couples the thermodynamic and mechanical aspects of the compression process with the fluid-structure interaction between compressor shell and cavity, with a focus on noise and vibration induced aerodynamically by refrigerant discharge is developed. Coupled numerical analysis is done by feeding the mass flow variation obtained from compressor mechanistic model into the two-way coupled FSI simulation, and noise and vibration response induced by turbulent discharge gas jet in the compressor cavity is computed. Effects of fluid-structure interaction is investigated, and modeling techniques with different levels of fluid-structure interaction was evaluated to find the most suitable model coupling method for the compressor aerodynamic noise and vibration application. The simulation results of flow field and vibrational response of a hermetic indicate that the aerodynamically induced noise and vibration mainly occur on the upper and lower caps of the shell, which provide possible guidelines on NVH oriented design optimization of compressors.

NOMENCLATURE

CFD	computational fluid dynamics
FSI	fluid-structure interaction
HVAC	heating, ventilation, and air conditioning
LEE/LNS	linearized Euler/Navier-Stokes equation
PSD	power spectral density
RANS/LES	Reynolds averaged Navier-Stokes/large eddy simulation

REFERENCES

Bell, I. H., Ziviani, D., Lemort, V., Bradshaw, C.R., Mathison, M., Horton, W.T., Braun, J.E., Groll, E.A. (2020). PDSim: A general quasi-steady modeling approach for positive displacement compressors and expanders. *International Journal of Refrigeration*, *110*, 310–322.

Birari, Y. V., Gosavi, S. S. and Jorwekar, P. P. (2006). Use of CFD in Design and Development of R404A Reciprocating Compressor. *18th International Compressor Engineering Conference at Purdue,* Paper 1727. West Lafayette, IN, Purdue University.

Ffowcs Williams, J. E. and Hawkings, D. L. (1969). Sound generation by turbulence and surfaces in arbitrary motion. *Phil. Trans. R. Soc. Lond. A Math. Phys. Sci.*, vol. 264, pp. 321–342.

He, D., Ziviani, D. and Liu, Y. (2021). Theoretical Analysis of Noise and Vibration Generated by Gas Pressure Pulsation in Hermetic Compressors. *25th International Compressor Engineering Conference at Purdue*, Paper 1527. West Lafayette, IN, Purdue University.

Lighthill, M. J. (1952). On sound generated aerodynamically. i. general theory. *Proc. R. Soc.Lond. A Math. Phys. Sci.*, vol. 211, pp. 564–587.

Nakano, A. and Kinjo, K. (2008). CFD Applications for Development of Reciprocating Compressor. *19th International Compressor Engineering Conference at Purdue*, Paper 1842. West Lafayette, IN, Purdue University.

Soedel, W. (2007). *Sound and vibrations of positive displacement compressors*. Boca Raton, FL: CRC Press.

Wu, H., Huang, H., Zhang, B., Xiong, B. and Lin, K. (2019). CFD Simulation and Experimental Study of Working Process of Screw Refrigeration Compressor with R134a. *Energies* 12, 2054.

Ziviani, D., Bell, I. H., Zhang, X., Lemort, V., De Paepe, M., Braun, J. E., Groll, E. A. (2020). PDSim: Demonstrating the capabilities of an open-source simulation framework for positive displacement compressors and expanders. *International Journal of Refrigeration, 110*, 323–339.

ACKNOWLEDGEMENT

The author would like to thank the Center for High Performance Buildings group for the support and encouragement of this project, and especially from those members on the Project Management Subcommittee.