Two-dimensional numerical simulation of the inertance tube

Zhang Y, Dai W, Luo E, Radebaugh R., Lewis M.

Technical Institute of Physics and Chemistry of Chinese Academy of Sciences, Beijing, 100080, China
Chinese Academy of Sciences, Beijing, 100029, China

Phase shifting is an important issue in the development of inertance tubes. Computational fluid dynamics (CFD) has the potential to assist in predicting operating conditions and designs that simulate the oscillating behavior inside inertance tubes. This paper reports the construction of a long inertance tube with a gas reservoir and a CFD model of this system using the commercial code, Fluent 6.1. These calculation results are compared with experimental results obtained by our group in cooperation with NIST of USA.

INTRODUCTION

In a pulse tube cooler, the acoustic power flow is proportional to the component of the mass flow in phase with the pressure. In most cases the optimum phase relationship is that phase between mass flow rate and pressure is zero at approximately the midpoint of the regenerator. Many methods were used to get the appropriate phase. Up to now, the introduction of double-inlet phase shifter has greatly improved the efficiency of pulse tube coolers. The multiple bypass inlet pulse tube coolers have also occurred. The Stirling coolers use two pistons to adjust the phase at the regenerator, but the pulse tube coolers only can use other passive methods that are discussed above to adjust the phase.

The inertance tube can serve as a phase shifter [1], which adjusts the phase relation between mass flow rate and pressure. It is important for oscillating flow systems such as high frequency pulse tube coolers and thermoacoustic machines to achieve high efficiency. Up to now, the experimental data on inertance tube is very limited and it is difficult to obtain a simple formula to analyze the experiments due to complex oscillating flow behavior. On the other hand, numerical research reports are also scarce, mostly based on linear model or one-dimensional model.

Most importantly, we want to find a good and relatively simple method to predict inertance tube performance and use it as a guideline for applications. And the first objective of the project is to check out if this design of structure will lead to optimal performance and efficient phase adjustment via an analysis of the power flow, volume flow rate and the pressure in the inertance tube. Computer modeling using commercial Computational Fluid Dynamics (CFD) packages, such as Fluent can generate pressure and mass flow rate data which are then tested experimentally in an inertance tube system. And this paper describes a test system which has been modeled using Fluent 6.1.

The research is financially supported by Chinese Academy of Sciences under Contract KJCX2-SW-W012-01.

CFD MODELS

Inertance tube system geometry and computational grid generation

The physical model is a long inertance tube connected with a reservoir (Figure 1). The geometrical model in this system consists of two subregions. The first is the inertance tube. Due to the long and thin shape of the long tube, a two-dimensional, axis-symmetry model was created. It consisted of a 1689 mm length, 1.016 mm diameter copper tube that allowed for the contained gas to vibrate in. This part was divided into almost 135000 quadrilateral cells, and they are all of the same size. The second subregion consists of the gas reservoir. With a dimension of 40 mm, the gas reservoir is 59mm length. This part was divided into almost 150000 cells. And the standard $k-\varepsilon$ model was used to simulate the system [2].

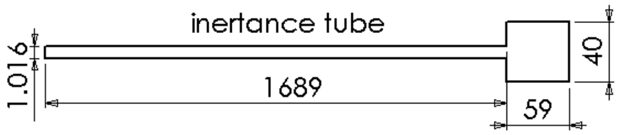


Figure 1 Diagram of the inertance tube system

Initial and boundary conditions

For solving the governing equations, the relevant initial and boundary conditions are as follows. Initially the system is at rest and at uniform temperature T=293K. The velocity is zero throughout the fluid field. The system is filled with He at a mean pressure of 2.5MPa, and the pressure ratio is set from 1.05 to 1.45. This oscillating pressure is set as the pressure boundary condition at the pressure inlet surface, which is at the leftmost of the system.

The thermal conductivity of all the walls was specified. Furthermore, a uniform convective heat transfer boundary condition was defined at the outer side of solid boundary. A uniform convective surface heat transfer coefficient $h = 40W/(m^2 \cdot K)$ was assumed at the boundary.

RESULTS AND DISCUSSION

The details of experiments for the verification of the CFD calculation can be found elsewhere[3]. The oscillation in the reservoir is not sinusoidal wave. In order to get the phase, we firstly analyze the data into sinusoidal wave via a fast-Fourier-transform algorithm. Figure 2 illustrates the phase angle difference of pressure between inlet and gas reservoir respectively obtained from experiment and CFD simulation. Figure 3 shows the measured and simulated phase angle of pressure leading mass flow rate at the pressure inlet surface.

Figure 4 shows the contrast of measured and simulated mass flow from the inertance tube to the reservoir. With the increase of the pressure ratio, the amplitude of the oscillating mass flow rate rises gradually. In the experiment, the gas reservoir was treated as an adiabatic gas capacitance. And the mass flow rate and volume flow rate passing in and out the gas reservoir can be calculated out. Just as what the figure illustrates, the experimental data and the simulation data agree well with each other.

Figure 5 shows the measured and simulated mass flow at the pressure inlet surface. The experimental data was measured by hot-wire anemometer. The measuring accuracy is not high due to the difficulty in measuring oscillating flow. So the discrepancy could be attributed to both experimental and calculation errors.

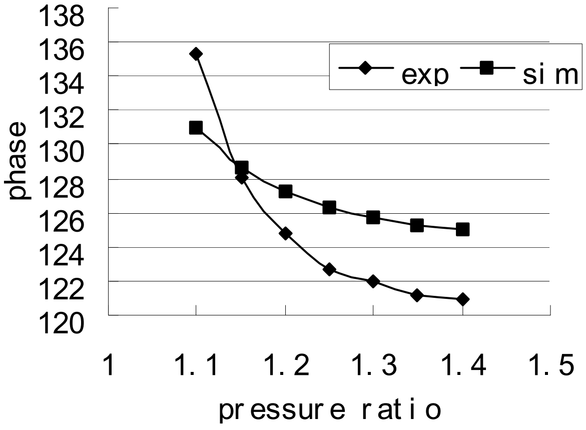


Figure 2 Measured and simulated phase difference of pressure between inlet and the reservoir

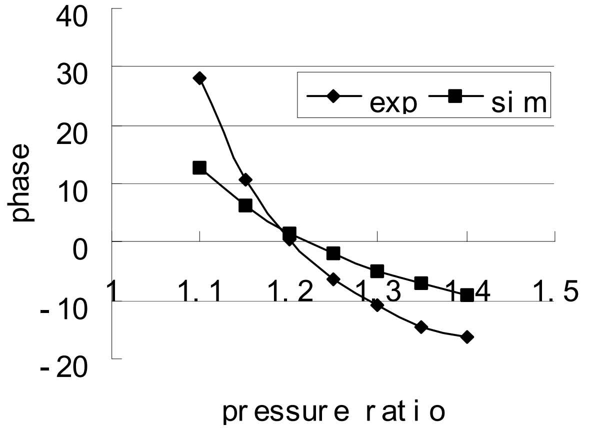


Figure 3 Measured and simulated phase difference between pressure and mass flow rate flow at the inlet

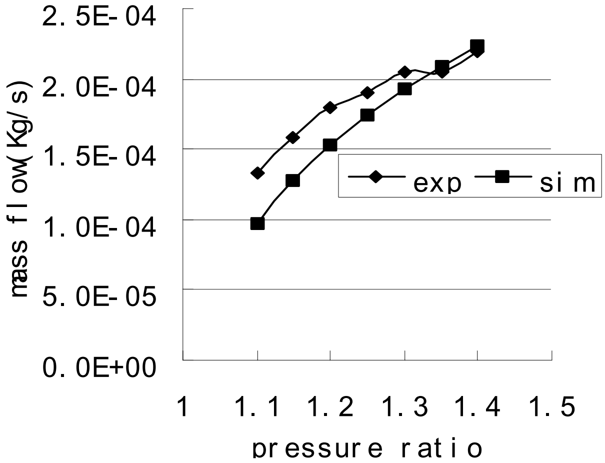


Figure 4 Measured and simulated mass flow rate from the inertance tube to the reservoir

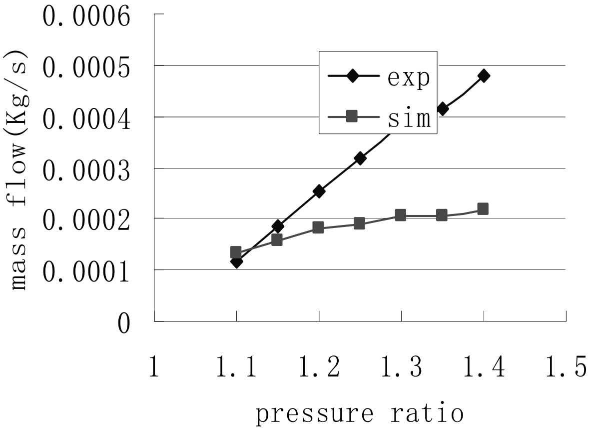
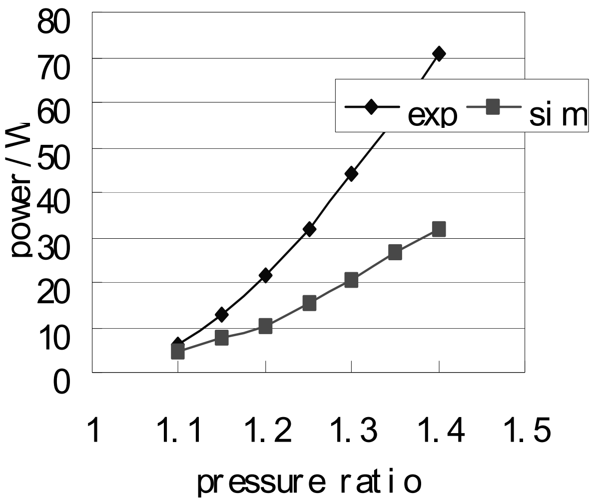


Figure 5 Measured and simulated mass flow rate at the pressure inlet

Figure 6 shows the acoustic power at the inlet. The acoustic intensity across the cross-sectional area of the channel:

$$W = \frac{1}{2}\left|\tilde{p}_1 \tilde{U}_1\right| \cos\theta_{Pu} = \frac{1}{2}\left|\frac{\tilde{p}_1}{p_0}\right| \dot{m} R_M T_0 \cos\theta_{pu} \tag{1}$$

Where θ_{pU} is the phase angle between $\tilde{p}_1$ and $\tilde{U}_1$ and the tilde denotes complex conjugation. This is the acoustic power flowing in the x direction. Both the experimental data and the simulation data are all calculated from equation (1). The power is basically in direct proportion to the product of corresponding mass flow rate.



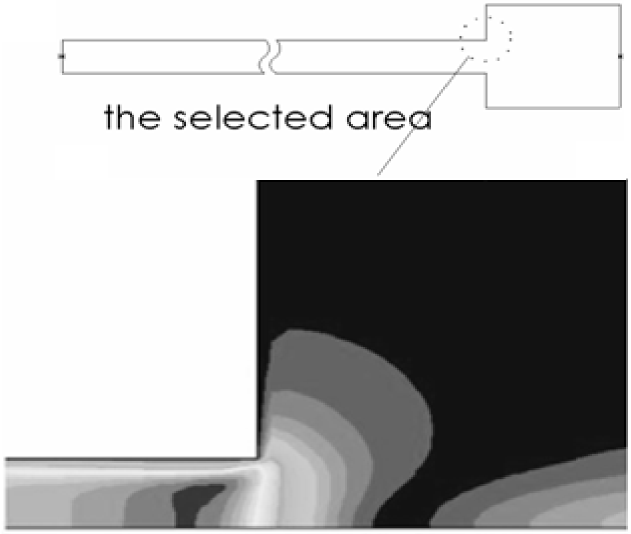


Figure 7 shows the contour of velocity magnitude at the joint of inertance tube and the gas reservoir. When the gas flows out of the inertance tube into the reservoir, the velocity quickly decreases. This will greatly influence the static pressure at this small inlet region in both phase and amplitude.

CONCLUSIONS

We use a two-dimensional, turbulence model to simulate the performance of an inertance tube. The sinusoidal pressure wave causes an unsteady oscillating flow in the tube. We contrast the simulation data, including phase between pressure at the inertance inlet and the inertance outlet, and that between pressure and mass flow rate at the inlet, also mass flow rate at inlet and outlet with the experimental data. Through these data, we can compute out the power at the inlet. The agreement between the values calculated by the previous model and experimental data is quite encouraging. Based on the present model, guidance for the design, operation, and control of inertance tube can be prepared for achieving optimal design, as well as safe and economical operation.

REFERENCES

1. D. L. Garden, G. W. Swift, Use of inertance in orifice pulse tube refrigerators, Cryogenics (1997) 37 117-121
2. Jacek Smolka, Andrzej J. Nowak , Luiz C. Wrobel, Numerical modeling of thermal processes in an electrical transformer dipped into polymerized resin by using commercial CFD package fluent, Computers & Fluids (2004) 33 859–868
3. E. Luo, Ray Radebaugh, M. Lewis, Inertance tube models and their experimental verification, Adv. Cryo. Eng., vol 49 (to be published), American Institute of Physics (2004)