Numerical Simulation of Oscillating Fluid Flow in Inertance Tubes

C. Dodson[1, 2], A. Razani[1, 2] and T. Roberts[1]

[1]Air Force Research Laboratory
Kirtland AFB, NM, 87117-5776

[2]The University of New Mexico
Albuquerque, NM 87131

ABSTRACT

Inertance tubes are used in Pulse Tube Refrigerators (PTRs) to control the phase shift between the mass flow and pressure to increase performance of the refrigerator. Computational Fluid Dynamics (CFD) simulation of the oscillating fluid flow in the inertance tube for different tube geometries is presented in this study. Commercial CFD code FLUENT is used to carry out the simulations. The results of CFD simulations are compared to the available experimental results. In addition, the results of grid convergence studies, important in any numerical simulation, are presented and discussed. A numerical model is developed by dividing the inertance tube into sections where the conventional first order model is used in each section. The resulting simultaneous non-linear Ordinary Differential Equations (ODEs) are solved numerically to determine the input acoustic power and the phase shift between mass flow and pressure at the inlet of the inertance tube. The results of the first order model and CFD simulation are compared and discussed.

INTRODUCTION

Pulse Tube Refrigerators (PTRs) play an important role in satisfying the need for cryogenic cooling of space-based infrared detectors as well as many other applications requiring coolers with high reliability, low vibration, and high efficiency. Usually three types of phase-shifting processes exist on PTRs that control the phase shift between the mass flow rate and pressure[1]. The more conventional are Orifice Pulse Tube Refrigerators (OPTRs) where the mass flow rate and pressure are in phase at the orifice. In Double Inlet Pulse Tube Refrigerators (DIPTRs) a bypass valve between the warm end of pulse tube and warm end of the regenerator is used to provide a proper phase-shifting mechanism. In Inertance Tube Pulse Tube Refrigerators (ITPTRs), which are the focus of this study, the phase shifting is provided by an inertance tube replacing the orifice.[1-3] Figure 1 shows the important components of ITPTRs. A review of these phase-shifting mechanisms is given in previous publications.[4-5]
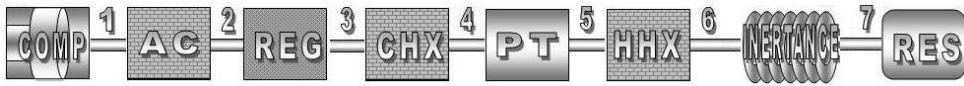
Figure 1. Inertance Tube Pulse Tube Refrigerator (ITPTR)

In first-order models, usually used in design analysis and parametric studies of ITPTRs, a lumped parameter approximation is used to take into account the inertance, compliance, and the fluid flow resistance associated with oscillating flow in the inertance tube.   In the last study[6], a previously developed and convenient correlation for entire ranges of laminar and turbulent flow is used with a modified distributed component model[2-3] and was integrated with REGEN 3.2 and a parametric model for pulse tube inefficiencies.   For the current study, the distributed model is used to model the inertance tube and is compared with CFD simulations[6] and previously published experimental data.[7] The distributed component method used in this study divides the inertance tube into several sections and applies the lumped parameter model to each section.

Grid convergence for numerical simulations is presented and discussed with respect to the issues of verification and validation for inertance tube studies.

MODELING THE INERTANCE TUBE

In this study, two inertance tube modeling techniques will be compared to experiment. An ODE model of the inertance tube will be compared to CFD simulations of the inertance tube. The inertance tube in this study is modeled by $2n+1$ ODEs using the distributed model of the inertance tube and is given in our last study.[8] The inertance tube was simulated in the CFD software Fluent and is meshed up with a uniform grid in the axial direction and a successive ratio method in the radial direction. The successive ratio method has more grid points near the wall/boundary layer than in the center of the tube. The finer successive ratio mesh had better mass conservation and no-slip boundary velocity values than the uniform mesh method in the radial direction. The successive ratio method is preferred over the uniform mesh because a uniform mesh with the smallest mesh equal to the smallest mesh of the successive ratio would result in a fine mesh throughout the whole radial direction and substantially increases simulation runtime. Turbulence techniques have shown better performance than laminar models.[2]   The $k-\omega$ turbulence technique was employed for the simulations as these simulations had better mass conservation values than the $k-\varepsilon$ turbulence technique. The walls of the inertance tube are assumed to be isothermal. Oscillating pressure at the inlet of the inertance tube is given by $P=P_a+P_d\sin\omega t$, where P_a is the average pressure, P_d is the dynamic pressure, $\omega=2\pi f$ is the angular frequency and f is the frequency. The dynamic pressure is typically given as a pressure ratio as such: $\mathrm{Pr}=(P_a+P_d)/(P_a-P_d)$.   The pressure at the reservoir side of the inertance tube is calculated as a function of the mass flow rate into and out of the reservoir by the following equations[9]:

$$m_{res}\left(t\right)=\frac{P_aV}{\gamma RT}+\sum_{n=0}^{N}2\pi m_F dt \qquad (1)$$

$$P_{res}\left(t\right)=\frac{\gamma RTm_{res}\left(t\right)}{V}-P_a \qquad (2)$$

where γ is the specific heat ratio of helium, R is the gas constant for helium, T is the temperature, m_{res} is the mass in the reservoir and m_F is the mass through the face going into the reservoir from the inertance tube, N is the number of time steps and dt is the time step size.

The input data is based on the experiments presented by Lewis, Bradley and Radebaugh (LBR)[7]. The inertance tube was 2.357 m long with a diameter of 5.7 mm. The charge pressure

P_a was 2.5 MPa with a pressure ratio of 1.3 and a frequency of 60 Hz. Reservoir volumes were varied from sizes of 30, 83, 134 and 334 cc in the LBR study. In the CFD study, reservoirs of size 0, 5, 10, 30, 83 and 104 cc were simulated. In the distributed model of the inertance tube, reservoirs of size 0, 1, 3, 5, 7, 9, 10, 30, 50, 70, 90, 200, 300, 400, 500, 600, 700 ,800, 900, 1000 and 10000 cc were simulated.

RESULTS AND DISCUSSION

Acoustic power and phase shift between mass flow rate and pressure are calculated as such:

$$\dot{W} = \frac{RT}{\tau} \int_0^\tau \dot{m} \ln \frac{P}{P_a} dt \tag{3}$$

$$\varphi = \tan^{-1} \left(\frac{\int_0^\tau \dot{m}(t) \cos \omega t dt}{\int_0^\tau \dot{m}(t) \sin \omega t dt} \right) \tag{4}$$

The comparison of the distributed ODE model, the CFD simulation, experimental data, and the model of the LBR experiment are shown in Figure 2 and 3 for the acoustic power and phase shift, respectively. It should be noted that the other half of this paper is about numerical issues, which have not been applied to the results of this part of the study, due to time constraints. Error bars would have been useful to see the extent of the simulation error compared to distributed model, and its corresponding error, and the experimental results.

NUMERICAL VERIFICATION AND VALIDATION

In computer simulations of physical models, one must deal with issues of verification and validation (V&V).[10,14,15] According to the AIAA CFD[10] standards paper "The fundamental strategy of verification is to identify, quantify and reduce errors in the computational model and
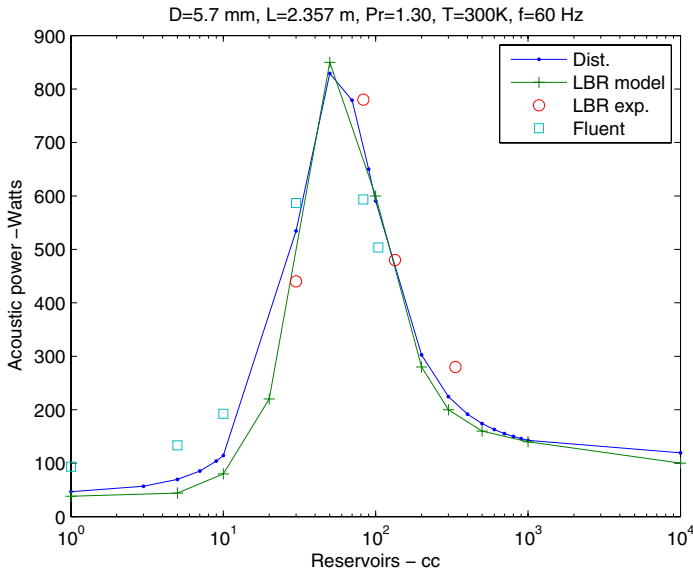

Figure 2. Comparison of acoustic power between different models and experimental results

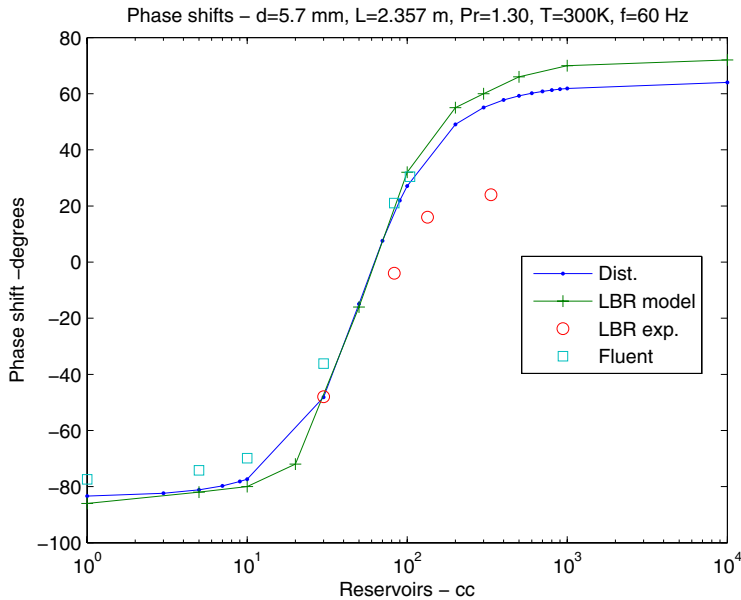Phase shifts – d=5.7 mm, L=2.357 m, Pr=1.30, T=300K, f=60 Hz

Figure 3. Comparison of phase shift between different models and experimental results

its numerical solution…The fundamental strategy of validation involves identifying and quantifying the error and uncertainty in the conceptual and computational models, quantifying the numerical error in the computational solution, estimating the experimental uncertainty, and then comparing the computational results with the experimental data." The primary reason for presenting these ideas to the cryogenics community is to give a base from which to talk about the issues of presenting simulation papers as solutions to cryogenic research. It should be noted that the AIAA's paper addresses the issue that there is no one stop shopping for all your CFD V&V needs. In this study of oscillating flow in inertance tubes, the issues of verification and validation will be discussed. This discussion will be tailored to the two simulation techniques of CFD solutions and ODE solutions used in our study.

VERIFICATION

Verification as restated from the AIAA paper is to identify and quantify the computational error in CFD solutions which is identified as coming from five primary sources. These five error sources are insufficient spatial discretization, insufficient temporal (time) discretization, insufficient iterative convergence (as most CFD solvers are iterative), computer round-off and computer programming. In addition to identifying and quantifying computational error sources, "Verification should demonstrate the stability, consistency and robustness of the numerical scheme."[10] Below are statements about how the ODE and CFD solution techniques relate to these five sources of error.

For the CFD solutions of this study, a grid convergence index (GCI) technique (presented later) was used to address the issues of identifying and quantifying spatial and temporal discretization. For the ODE solutions, space is a lumped parameter and thus GCI can be used to address temporal discretization errors. For the discretization techniques, the idea of CFL number should help yield statements about how the grid size and time step relates to numerical error and stability of the solver.

The CFD solver (Fluent) uses an iterative convergence technique, which could be used with GCI to better illuminate what iterative convergence criteria yields 'better' GCI results. It should be noted that picking poor iterative convergence criteria in certain Fluent simulations yields results that never converge or that take so long to converge that the simulation takes too long to be practical to run. The ODE solution is not an iterative technique, as it is a fourth order in time Runge-Kutta solver and thus does not have errors associated with iterative convergence.

Computer round-off errors are due to the use of computers to represent real numbers on finite-state machines and can be a function of direct or iterative methods. An example of this is the effect of calculating enthalpy flow with poor values of mass conservation, say 1e-6 for a frequency of 60 Hz with an average temperature about 300K and a specific heat of 5000 J/kg/K. For example, consider enthalpy flow calculations at any section in the tube over one cycle,

$$\dot{H} = \frac{1}{\tau}\int_0^\tau \dot{m}(t)c_p T(t)dt$$

if $T(t)$ is a slowly varying function of time

$$T(t) \approx \overline{T}(t) = 300K \Rightarrow$$

$$\dot{H} \approx \frac{1}{\tau}c_p \overline{T}(t)\int_0^\tau \dot{m}(t)dt$$

If the integral of mass flow rate is not zero, then

$$\dot{H} \approx c_p \overline{T}(t)\overline{\dot{m}}(t) \neq 0, \text{ if } \overline{\dot{m}}(t) \neq 0$$

$$\dot{H} \approx (5000)(1e-6)(300) = 1.5e6(1e-6) = 1.5 \text{ W}$$

If the mass flow average was three orders lower (1e-9) the enthalpy flow calculation would yield a value of .0015 W. So understanding the accuracy of mass flow average is essential for enthalpy flow and entropy flow calculations (see the grid convergence technique below). To make statements about whether mass streaming is occurring requires some error analysis of the solution technique. Without the error analysis, physical phenomena could be confused with the numerical error of the solution technique. Additionally, the poor mass flow average could be due to errors of the solution technique as a result of truncation errors or numerical instability of the solution technique.

The errors associated with computer programming include bad data input, bad data output analysis and bad code that does not do what was intended. For the CFD software, it is hard to know if the code is wrong or buggy, as well as for the ODE solution code. The test of bad code or bad input is in running various simulations that can be verified as similar to what might be expected or similar to one another when there are parameter changes. Sensitivity analysis and unexpected instability in simulations could be an indicator that the coding or input is incorrect. In our two cases, we do not believe that we have any coding issues as numerous test and checks have been done without any apparent anomalies, while it should be noted that numerous user errors have been corrected along the way.

VALIDATION

Once the errors of verification have been identified and quantified, validation can be used to compare with the results of experiment and its corresponding uncertainties. Additionally, validation concerns the usage of simulations to complex systems. It is typically suggested that a system be broken into subsystems and these subsystems simulated on their own, with the idea of carefully understanding the interactions of the various subsystems with respect to one another.

	Roache	JFE	Equation number
p	$\dfrac{\left\| \ln\dfrac{f_3 - f_2}{f_2 - f_1} \right\|}{\ln r}$	$\left\| \ln\left\| \dfrac{f_3 - f_2}{f_2 - f_1} \right\| \right\|$	(5) and (6)
ε	$\dfrac{f_1 - f_2}{f_1}$	$\dfrac{f_1 - f_2}{f_1}$	(7)
GCI	$\dfrac{\|\varepsilon\|}{r^p - 1} F_s$	$\dfrac{\|\varepsilon\|}{r^p - 1} F_s$	(8)
$f_{h=0}$	$\approx f_1 + \dfrac{f_1 - f_2}{r^p - 1}$	$\approx \dfrac{r^p f_1 - f_2}{r^p - 1}$	(9) and (10)

Table 1. Equations of interest for GCI studies.

For the current and past CFD studies[2,3,6,9,11], subsystem modeling has been done for the pulse tube and inertance tube. The issues with doing component level simulations are to understand what the boundary conditions for the components should be. For this study, pressure is input on the hot heat exchanger and reservoir sides, as though no boundary layer or junction transition effects are occurring. The turbulent model uses a value for kinetic energy as an input, whose effect of changing is not understood. The effect of the environment on the inertance tube wall temperature is not understood as in this study the tube wall is set to be isothermal, but in practice the heat transfer to the outside of the tube must be modeled as well.

For the case of the inertance tube study, validation is an ongoing process but the comparison to the experiment and ODE solution are useful to see that similarity exists. For simulations of multiple components, validation of each of the components needs to be done to ascertain if the combination of components is correct or not.

GRID CONVERGENCE INDEX RESULTS

The simulation concept of convergence describes the verification issues inherent to discretizing continuous equations in a computer simulation with the hope that the computer solution converges to the analytical solution of the continuous equations. Grid convergence techniques become important to address this issue and in this study two grid convergence index (GCI) techniques are presented and employed. The GCI technique usually requires simulations on several grids. For this study, three grids are meshed. Starting from a coarse grid of the inertance tube, a simulation is run on this grid. Then, the mesh is doubled from the coarse mesh to a finer mesh and a simulation run on this grid. This finer mesh is then doubled to the finest mesh and the simulation is run on this finest grid. The coarse grid shares meshing positions with parts of the finer grids. From our fluid simulations, various variables like pressure, mass flow rate and temperature can be compared on the grids, as described below, to find out their accuracy. The order of the method used in the simulations is usually given and indicates the order of the simulation method (this could show that our fourth order Runge-Kutta technique is fourth order). With the GCI methods used in this study, extrapolation can be done from the three discrete grids to yield the solution of the continuous case.
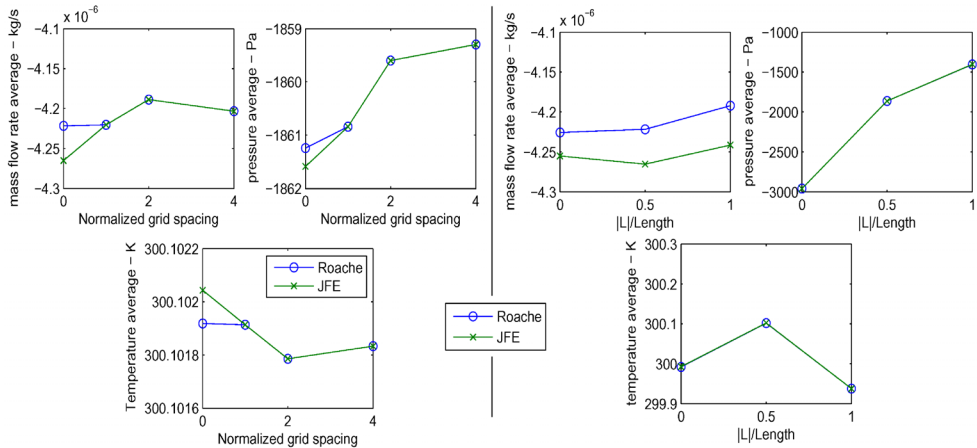
Figure 4. (Left) The Richardson extrapolation in the middle of the length of the tube for mass flow rate average, dynamic pressure average and temperature average. (Right)– The extrapolation values as a function of length for mass flow rate average, dynamic pressure average and temperature average.

There are several ways to define GCI criteria. In this study, two methods have been employed. The first is laid out in Roache's book[12] and the other as recommended by the Journal of Fluids Engineering (JFE)[13]. The two GCI techniques' equations of interest are shown in Table 1, where r is the mesh multiplication constant (i.e. 2 for doubling the grid), f_n is the value of the function (pressure, temperature, etc.) on the specified grid and F_s is the factor of safety and is 1.25 for the three grid case used in this study. The value for f_3 is the coarsest grid and the value f_1 is the finest and the value $f_{h=0}$ represents what is the Richardson extrapolation to the continuous grid. The *GCI* value is a representation of the percent of error for the value on the finest grid. The only differences between the two techniques are that the order and extrapolation are calculated in different manners. In the JFE method, determination of oscillating convergence can be done fairly easily. Oscillating convergence occurs when the middle mesh size had better or worse values than both the coarse and finest meshes.

For this part of the study an inertance tube of 1.28 m and 2 mm diameter is simulated with 16 uniform mesh points in the radial direction, 128 uniform mesh points in the axial direction, a pressure ratio of about 1.2, charge pressure of 2.5 MPa, 60 Hz inlet frequency, 300K isothermal wall and an adiabatic 300 cc reservoir. To test whether the grids used to mesh this representative inertance tube were good enough, three different grid convergence index (GCI) tests were done. The first was to test laminar flow in a tube by doubling grids in the radial direction. The next was to test a turbulent flow in an inertance tube by doubling grids in the radial direction. The last was to test GCI by doubling the number of time steps and keeping the grid the same for a turbulent flow. Shown in the Figures 4 and 5 are the turbulent time step GCI results. Cycle averages are given for mass flow rate, dynamic pressure and temperature over the last cycle of 11 cycles, with each cycle having 400 time steps per cycle, at a given circular cross-section along the length of the tube and the discrete time values were averaged using Boole's rule for numerical integration (7th order errors as a function of time step size). Shown in Figures 4 and 5 are the results of the two techniques' GCI Richardson extrapolation values of the averaged values in the middle of the tube, extrapolation values of the averages as a function of length of the tube, the GCI value as a function of tube length and the order as a function of tube length. In Figure 4, on the left is shown the Richardson extrapolation for the averages collected halfway between the inlet and outlet of the tube. The Richardson extrapolation value is the vertical value read at the zero value on the normalized grid spacing axis. The JFE method characterizes both
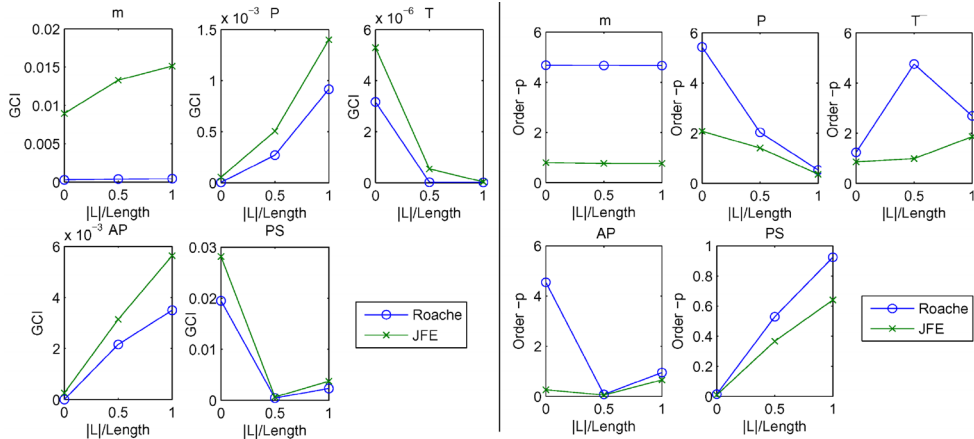
Figure 5. (Left) The value of GCI as a function of length for mass flow rate average, dynamic pressure average, temperature average, acoustic power (AP) and phase shift (PS). (Right) The order p as a function of length for mass flow rate average, dynamic pressure average, temperature average, acoustic power (AP) and phase shift (PS).

the mass flow rate and temperature averages as being oscillating. On the right side of Figure 4 are the values of the Richardson extrapolation value representing the averages at the inlet, middle and outlet of the inertance tube. As an example, the plots on the right show that dynamic pressure increases from the inlet side to the reservoir side, that there is a slight temperature peak in the middle of the inertance tube and that mass conservation is poor and either streaming might be occurring or that better mass conservation needs to occur. The plots on the right show that Roache's and the JFE techniques yield different extrapolation values for mass flow rate average but are very close for both dynamic pressure and temperature. In Figure 5, the value for GCI and order are plotted as functions of the tube length. On the left plots of the figure the GCI is given for mass flow rate (m), dynamic pressure (p), and temperature (T) averages, as well as the calculated values of acoustic power (AP) and phase shift (PS) as functions of tube length. According to the JFE technique, the GCI value is higher than Roache's for all the values plotted and especially for the mass flow rate average and phase shift. The GCI value is representative of the error bars needed for CFD plots and show that the mass flow rate average and phase shift have relatively high error bars. On the right of Figure 5 are shown the orders or the various variables as functions of tube length. The JFE technique yields lower orders than Roache's technique. Most of the solvers in Fluent are either first or second order methods or come with statements that say they are better than the second order methods for various cases, but don't implicitly state their order. In general, one would hope that these values yield something on the order of two, since we use Fluent solvers that state they are at least second order. This could mean that adjustments to the grid, time step size or other parameters need to be made before method order statements become more correct.

CONCLUSIONS

The results of the CFD simulation of the oscillating fluid flow in the inertance tube and numerical solutions of a first order distributed model are compared to the experimental results reported in the literature. The issue of numerical validation and verification of the CFD calculations are discussed and some results of Grid Convergence Studies (GCI) for the oscillating flow in the inertance tube are presented. It is not wise to draw conclusions on the physical phenomena occurring in the inertance tube without verification of the numerical results

of the CFD calculations and careful error analysis. More extensive verification of CFD calculations in application to the oscillating fluid flow in the inertance tube is under investigation.

REFERENCES

1. Shunk, L. *Experimental Investigation and Modeling of Inertance Tubes*, Master's thesis, University of Wisconsin-Madison, 2004.

2. Gustafson, S., Flake, B., Razani, A., "CFD Simulation of oscillating flow in an inertance tube and its comparison to other models," *Adv. in Cryogenic Engineering*, Vol. 51, AIP, Melville, NY (2006), pp. 1497-1504.

3. Gustafson, S. *CFD Simulation of Oscillating Flow in an Inertance Tube and Its Comparison to Other Models*, Master's Thesis, University of New Mexico, 2005.

4. Radebaugh, R., "Pulse tube cryocoolers for cooling infrared sensors," *Proceedings of SPIE*, Vol. 4130 (2000), pp. 363-379.

5. Richardson, R.N., Evans, B.E. "A review of pulse-tube refrigeration," *International Journal of Refrigeration*, Vol. 20, No. 6 (1997), pp. 367-373.

6. Dodson, C. *The comparison of CFD simulations and a first order numerical model of inertance tubes,* Master's Thesis, University of New Mexico, 2007.

7. Lewis, M., Bradley, P. and Radebaugh, R., "Impedance measurements of inertance tubes," *Adv. in Cryogenic Engineering*, Vol. 51, AIP, Melville, NY (2006), pp. 1557-1563.

8. Dodson, C., Lopez, A., Roberts, T. and Razani, A., "A model for the parametric analysis and optimization of inertance tube pulse tube refrigerators," *Adv. in Cryogenic Engineering*, Vol. 53A, Amer. Institute of Physics, Melville, NY (2008), pp. 685-692.

9. Flake, B. and Razani, A., "Phase shift and compressible fluid dynamics in inertance tubes," *Cryocoolers 13*, Kluwer Academic/Plenum Publishers, New York (2005), pp. 275-284.

10. Rahaim, C., Oberkampf, W., Cosner, R. and Dominik, D., *AIAA-2003-0844, AIAA Committee on standards for computational fluid dynamics – Status and Plans*, 41st AIAA Aerospace Sciences Meeting and Exhibit, 2003.

11. Stephens, S. *CFD modeling of pulse tube geometry and its effect on performance*, Master's Thesis, University of New Mexico, 2006.

12. Roache, P., *Verification and validation in computational sciences and engineering*, Hermosa Publishers, Socorro, August 1998

13. Journal of Fluids Engineering, *Statement on the control of numerical control of numerical accuracy*, http://journaltool.asme.org/Templates/JFENumAccuracy.pdf

14. Roy, C.J., "Review of code and solution verification procedures for computational simulations," *Journal of Computational Physics*, 205 (2005), pp. 131-156.

15. Gokaltun, S., Skudarnov, P.V. and Lin, C., "Verification and validation of CFD simulation of pulsating laminar flow in a straight pipe," *American Institute of Aeronautics and Astronautics*, AIAA, (2005), pp. 2005-4863.