A computational approach for coupled 1D and 2D/3D CFD modelling of pulse Tube cryocoolers

T Fang¹, P S Spoor² and S M Ghiaasiaan¹

¹Georgia Tech Cryo Lab, G.W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology, Atlanta, GA 30332 USA

²Chart, Biomedical Division, Chart Inc., Troy, New York 12188 USA

Email: mghiaasiaan@gatech.edu

Abstract. The physics behind Stirling-type cryocoolers are complicated. One dimensional (1D) simulation tools offer limited details and accuracy, in particular for cryocoolers that have non-linear configurations. Multi-dimensional Computational Fluid Dynamic (CFD) methods are useful but are computationally expensive in simulating cryocooler systems in their entirety. In view of the fact that some components of a cryocooler, e.g., inertance tubes and compliance tanks, can be modelled as 1D components with little loss of critical information, a 1D-2D/3D coupled model was developed. Accordingly, one-dimensional-like components are represented by specifically developed routines. These routines can be coupled to CFD codes and provide boundary conditions for 2D/3D CFD simulations. The developed coupled model, while preserving sufficient flow field details, is two orders of magnitude faster than equivalent 2D/3D CFD models. The predictions show good agreement with experimental data and 2D/3D CFD model.

1. Introduction

The high reliability and mechanical simplicity of stirling-type pulse tube cryocoolers (PTCs) make them especially attractive for some applications. PTCs with inertance tubes were first introduced in 1990s [1, 2]. Although PTC contains multiple components, most of the primary loss processes are found inside their cold-tip assemblies. Some of the mechanisms that lead to a loss have been studied by previous researchers, including Rayleigh Streaming [3], Gedeon Streaming [4], shuttle heat transport [5] and convective losses. Among these loss mechanisms, convective losses are probably the most complicated and significant.

The convective losses caused by misalignment with respect to gravity have been studied experimentally by several researchers. Thummes et al [6] found that many PTCs can experience significant loss in cooling power when their cold-ends are not pointing downward. Swift and Backhaus [7, 8] experimentally analyzed the buoyancy effect and developed an analogous theory to explain the convective loss caused by misalignment of a PTC with respect to gravity. In their theory, a non-dimensional pulse tube convection number, $N_{ptc}$, was derived as an indicator of gravitational...
sensitivity. Experiment and simulation have shown that the convective condition in a PTC can be very complicated and a simple model may not be sufficient to fully represent the physical processes. Detailed experimental measurements of flow field without affecting the performance of PTC is difficult. As a result, CFD is the primary tool used by many researchers and can be very helpful in resolving the flow details. Mulcahey [9] developed system-level and component-level CFD approaches to investigate the convection phenomena that underlie the performance deterioration in a single-stage PTC caused by its misalignment with gravity. The system-level model addressed the entire PTC system including wave generator, cold-tip, inertance tube and compliance tank. Typically, the component-level model simulates three components that include the component that is the focus of the analysis and its immediate neighboring components. An example is the pulse tube and the warm and cold heat exchangers attached to the two ends of the pulse tube. Fang et al. [10] recently compiled more data generated by experiments and CFD simulations. Their work confirms that experiments and CFD simulations have similar trends and are in general agreement with respect to the effect of gravitational misalignment on the performance of PTCs. However, they also noted that CFD simulations are somewhat less sensitive to misalignment with respect to gravity orientation than experiments. The reason behind this disagreement between experimental data and CFD simulations is under investigation. Conrad et al. [11] studied the convection caused by geometrically diameter change between different component using system-level CFD model.

However, simulating a PTC in its entirety can be computational expensive. Only a transient solver can be used for these simulations due to the oscillating flow in a PTC. The simulations can typically take hundreds of periods for the flow to become fully developed. The system model developed by Mulcahey [9], for example, modeled an entire single-stage PTC, including inertia tube in 3D CFD. The total number of mesh was very large. The model requires accurately resolving pressure wave propagation and resistance in the inertia tube. As a result, the time step has to be very small to ensure that Courant number is close to one and the boundary layer has to be sufficiently resolved to yield accurate resistance. The inertia tube and compliance tank in system-level simulations are 1D-like components. A multitude of 1D models have shown very good prediction of the flow in inertia tubes when compared with experiments [12-14].

A component-level model [9] only models a critical component with small dimension, possibly in addition to its two adjacent components, for example the pulse tube as well as the cold and warm heat exchangers, and thus does not require very small time step to resolve pressure wave propagation. As a result, a component-level model can be orders of magnitude faster than a system-level model. However, there are several troubling issues encountered in component-level modeling. Normally a CFD model requires a pressure boundary and mass flow boundary to yield a stable solution. The boundary conditions are often computational predictions or experimental measurements. However, measuring pressure or mass flow rate in most PTC components can be very difficult. Computational predictions of boundary condition can result in large propagating error. More importantly, the impedance of the modeled component is not coupled with other components in component-level models.

To reduce the computational expense and to preserve the integrity of the impedance, a 1D and 2D/3D CFD coupling model was developed in this work. Thus, 1D-like components, e.g. inertia tube and compliance tank are modeled using 1D formulation. The cold-tip assembly is modeled using CFD in order to resolve multi-dimensional features. The 1D model and 2D/3D CFD model are coupled using ANSYS FLUENT User Defined Functions (UDFs). The predictions of the 1D model and the coupled model are compared with experimental data as well as the predictions of other models.

2. 1D model
Inertia tube and compliance tank are modeled using one-dimensional distributed impedance network model, as described by Schunk et al. [13] with the entrance effects neglected. The impedance of the inertia tube and compliance tank is analogous to the electric circuit shown in figure 1. The inertia tube is discretized into $n$ segments. Each segment is represented by a flow resistance ($R_i$), compliance ($C_i$) and inertia ($I_i$) where $i$ presents the $i$th segment. The corresponding mass flow rate ($\dot{m}_i$) presents
the flow rate through \( R_i \) and \( C_i \). The pressure \( (P) \) represents the pressure at the right hand side end of the \( i \)th segment. The compliance tank is represented by a compliance \( (C_{\text{com}}) \).

![Impedance network of inertance tube and compliance tank](image)

**Figure 1.** Impedance network of inertance tube and compliance tank

The resistance \( (R_i) \), compliance \( (C_i) \) and inertance \( (I_i) \) of each segment can be calculated separately. The resistance \( (R_i) \) is given by

\[
R_i = 64 f_i m_i (L/n)/\pi^3 \bar{\rho} D^5
\]  

(1)

where \( \bar{\rho} \) is the average density of the fluid in the segment, \( L \) is the total length of the inertance tube, \( D \) is the inner diameter of the inertance tube, \( f_i \) is the Fanning friction factor. Since the flow in the inertance tubes of PTCs is mostly turbulent, \( f_i \) is calculated from

\[
f_i = 0.046 Re_i^{-0.2}
\]  

(2)

where \( Re_i \) is the Reynolds number in the segment. The compliance \( (C_i) \) is given by

\[
C_i = \pi D^2 (L/n) / 4 R \bar{T}
\]  

(3)

where \( \gamma \) is the specific heat ratio, \( R \) is the gas constant, and \( \bar{T} \) is the average temperature of the fluid in the segment. The inertance \( (I_i) \) is given by

\[
I_i = 4 L / n \pi D^2
\]  

(4)

The associated impedances of resistance \( (R_i) \), compliance \( (C_i) \) and inertance \( (I_i) \) are as follow, respectively

\[
Z_{C,i} = 1 / j \omega C_i
\]  

(5)

\[
Z_{I,i} = j \omega I_i
\]  

(6)

\[
Z_{R,i} = R_i
\]  

(7)

Therefore, the impedance of the network can be calculated as follow:

\[
Z_{S,1} = Z_{\text{com}} + Z_{R,1} + Z_{I,1}
\]  

(8)

\[
Z_{P,i} = \left( \frac{1}{Z_{S,i}} + \frac{1}{Z_{C,i}} \right)^{-1}, i = 1, 2, ..., n
\]  

(9)

\[
Z_{S,i} = Z_{P,i-1} + Z_{R,i} + Z_{I,i}, i = 2, 3, ..., n
\]  

(10)

where \( Z_{P,i} \) is the sum of \( Z_{S,i} \) and \( Z_{C,i} \) which are in parallel, \( Z_{S,i} \) is the total impedance of \( Z_{P,i-1}, Z_{R,i} \) and \( Z_{I,i} \) which are in series. Thus, the mass flow entering the inertance tube is given by

\[
\dot{m}_{in} = P_{in} / |Z_{P,i}|
\]  

(11)

where \( P_{in} \) is the pressure at the beginning, left-hand end, of the inertance tube. As a result, individual mass flow through every component and pressure at each point can be calculated by
where $\dot{m}_{C,i}$ is the mass flow through the $i$th compliance ($C_i$).

3. Comparison of the 1D model against published results

The predictions of the 1D model were compared with the experimental data and calculated result from Luo et al. [12]. The model in [12] included a turbulence model to estimate the effect of turbulence rather than assuming fully turbulent flow. Two inertance tubes were modeled. The inner diameter and length of tube 1 are 1.016 mm and 1.689 m, respectively. The inner diameter and length of tube 2 are 1.547 mm and 1.219 m, respectively. The oscillating frequencies are from 30 Hz to 90 Hz. The charging pressure is 2.5 MPa. The pressure ratios are from 1.05 to 1.4.

![Figure 2](image)

**Figure 2.** Comparison between the 1D model, experimental data and Luo et al.’s model [12]: a) the effect of oscillation frequency on the phase angle between pressure mass flow, b) the effect of pressure ratio on the phase angle between pressure mass flow.

The calculated phase angle, by which the pressure leads the mass flow entering the inertance tube, is shown in figure 2. The 1D model is compared with the experiments and simulations published in [12]. The overall prediction is close to Luo et al.’s experiments and simulations. There are three simplifications in the 1D model used in this work: 1) assuming the flow in inertance tube is fully turbulent, 2) neglecting the effects of junction and entrance regions and 3) assuming that the process is isentropic. These simplifications can be responsible for the discrepancy between model and experiment. In figure 2 (b), the disagreement between 1D model and experiment is larger at the lower pressure ratios. Pressure ratio has a direct effect on flow amplitude and therefore may change the turbulence intensity. Simplification 1 can be less likely to apply when pressure ratio is low which explains figure 2 (b). Luo et al.’s model uses a turbulence transport model to estimate the effect of turbulence and as a result performs better when pressure ratio and turbulence intensity are low. In figure 2 (a) the effect of frequency is displayed. The disagreement between 1D model and experiment is larger when frequency is too high or too low. The effect of frequency can be more complex due to two competing trends. On one hand, higher frequency leads to lower flow oscillation amplitude, whereby the simplification 1 will be inaccurate. On the other hand, higher frequency implies lower thermal penetration depth, whereby simplification 3 will be more accurate.
4. 1D and 2D/3D CFD coupling model

The 1D model in Section 2 was implemented in 2D/3D FLUENT CFD models using ANSYS FLUENT UDFs. The computational domain and boundary conditions are shown in Figure 3. The mass flow boundary condition at the junction between inertance tube and cold-tip is defined by UDFs and calculated using the 1D model. The other boundary is the experimentally measured pressure profile. In this way, the impedances of the inertance tube, compliance tank and cold-tip assembly are all coupled. There is only one imposed boundary condition which can be easily measured.

![Figure 3. CFD computational domain and boundary conditions.](image)

The geometry of the computational domain is the same as the PTC tested in this work. The experimental data is therefore used to validate the model. Two-dimensional axis-symmetrical domain was used to reduce the computational expense since the cold-tip was coaxial. As shown in Figure 3, the regenerator, cold heat exchanger (CHX) and flow straighteners were treated as porous media in the model. All solid parts were omitted to reduce the overall computational expense. This is a reasonable approximation because in comparison with convection, the conduction in the solid material is small and can be neglected [9]. Isothermal boundary conditions were used on the surfaces of the cold and warm ends of the cold-tip assembly. As shown in Figure 3, the working temperature of the cooler varied from 313 K (warm end) to 77 or 100 K (cold end). The key performance indicator, cycle-averaged cooling power, was calculated by averaging the instantaneous heat flux on the cold-end isothermal surface. Note that although the computational domain modeled here is 2D, using the developed method the 1D model can be easily coupled with 3D CFD as well.

![Figure 4. Computational mesh.](image)

Figure 4 shows the mesh used in the simulations. The flow regime inside the pulse tube was conditionally turbulent according to Iguchi et al. [15]. Therefore small and fine mesh was used. The size of grid cells inside the pulse tube was equal to 1/127 of the diameter of the pulse tube. A minimum of 30 layers of inflation mesh was used on pulse tube wall. The height of the first cell layer was equivalent to about one wall distance unit, $y^+ \approx 1$. A coarser mesh was used for other regions, e.g., regenerator and CHX, where the flow details were neither complex nor critically important.

The pressure-velocity scheme was coupled. Spatial discretization methods for density, momentum and energy were second-order upwind. The transient formulation was second order implicit. The convergence criteria for continuity, velocities and energy were $10^{-6}$, $10^{-6}$ and $10^{-9}$, respectively. Equilibrium thermal model was used for porous media, e.g. CHX and regenerator.
5. Comparison between the coupled 1D - 2D/3D CFD model and experimental measurements

The coupled 1D - 2D/3D CFD model was used to simulate a PTC prototype that has been tested in [16]. Four conditions were simulated. The cold end temperatures were 77 K and 100 K. The driving powers of the PTC were 400 W and 600 W which can be modeled by changing pressure amplitude of the boundary condition.

In figure 5, the cycle-average cooling power predicted by the coupled model is compared with experiment. There is fair agreement between the predictions of the coupled model with the experiment. The smallest discrepancy (about 2%) is obtained for the 100K 400W case, and the largest discrepancy (about 20%) occurs for the 77K 400W case. The main reason the calculated cooling powers are higher than experiment is that the conductive and radiative losses are not accounted for in the model. To assess the contribution of losses, the conductive loss was calculated using the computer code SAGE [17]. The calculated conductive losses through solid shells amounted to 4.068W and 4.343W, respectively, for 100K and 77K cold tip temperatures. The surface of the cold head in the experiments is slightly oxidized. The emissivity of the copper CHX depends on its surface conditions, and can vary from 0.07 (scoured to a shine) to 0.7 (oxidized). The surface area of the CHX is approximately 60 cm\(^2\), and the cold head is covered by two layers of radiation shield. The estimated radiative losses are in the range from 0.075W - 0.75W and 0.076W - 0.76W, respectively, for the 100K and 77K cold head temperatures. The differences of cooling power between coupled model and experiment are around 1W and 4W for the 100K and 77K cold head temperatures, respectively, which are in the same order of magnitude of estimated conductive and radiative losses.

![Figure 5](image-url)

Figure 5. Comparison of cooling power between coupled model and experiment.

6. Conclusions
In this work, a coupled 1D - 2D/3D CFD model was developed using ANSYS FLUENT UDFs. 1D-like components, such as inertance tube and compliance tank, were modeled using a 1D code. The cold-tip assembly was modeled using CFD to resolve the multi-dimensional features. The coupled model is orders of magnitude faster than its equivalent system-level models, such as the model in [9], because it models fewer components and does not require small time steps to resolve pressure wave propagation in the inertance tube. Furthermore, unlike the model used in [16] or the component-level model used in [9], the boundary conditions required by the present model can be in terms of easily measured parameters. No imposed calculated boundary condition is needed.

The predictions of the 1D model were compared with published experimental data and model predictions. The 1D model yielded good agreements with the data and simulations of Luo et al. [12]. The coupled 1D - 2D/3D CFD model also showed fair agreement with our own experimental data obtained with a co-axial pulse tube cryocooler with respect to the cooling power, with discrepancies ranging from 2 to 20% that are caused by conductive and radiative losses.

7. References

[1] D. L. Gardner and G. W. Swift, "Use of inertance in orifice pulse tube refrigerators," Cryogenics, vol. 37, no. 2, pp. 117-121, 1997.
[2] S. W. Zhu, S. L. Zhou, N. Yoshimura and Y. Matsubara, "Phase shift effect of the long neck tube for the pulse tube refrigerator," in Cryocoolers 9, Springer US, 1997, pp. 269-278.
[3] J. R. Olson and G. W. Swift, "Acoustic streaming in pulse tube refrigerators: tapered pulse tubes," Cryogenics, vol. 37, no. 12, pp. 769-776, 1997.
[4] D. Gedoen, "DC gas flows in Stirling and pulse tube cryocoolers," in Cryocoolers 9, Springer US, 1997, pp. 385-392.
[5] R. P. Taylor, "Development and Experimental Validation of a Pulse-Tube Design Tool Using Computational Fluid Dynamics," Doctor of Philosophy Dissertation, The University of Wisconsin-Madison, 2009.
[6] G. Thummes, M. Schreiber, R. Landgraf and C. Heiden, "Convective heat losses in pulse tube coolers: effect of pulse tube inclination," in Cryocooler 9, Springer US, 1997, pp. 393-402.
[7] G. Swift and S. Backhaus, "Why high-frequency pulse tubes can be tipped," in Cryocoolers 16, 2008.
[8] G. W. Swift and S. Backhaus, "The pulse tube and the pendulum," The Journal of the Acoustical Society of America, vol. 126, no. 5, pp. 2273-2284, 2009.
[9] T. I. Mulcahey, "Convective Instability of Oscillatory Flow in Pulse Tube Cryocoolers due to Asymmetric Gravitational Body Force," Doctor of Philosophy Dissertation, Georgia Institute of Technology, 2014.
[10] T. Fang, T. I. Mulcahey, R. P. Taylor, P. S. Spoor, T. J. Conrad and S. M. Ghiaasiaan, "Method for Estimating Off-Axis Pulse Tube Losses," in review, Cryogenics, 2017.
[11] T. Conrad, M. G. Pathak, S. M. Ghiaasiaan and C. Kirkconnell, "The effect of component junction tapering on miniature cryocooler performance," AIP Conference Proceedings, vol. 1434, no. 1, pp. 435-442, 2012.
[12] E. Luo, R. Radebaugh and M. Lewis, "Inertance tube models and their experimental verification," AIP Conference Proceedings, vol. 710, no. 1, pp. 1485-1492, 2004.
[13] L. O. Schunk, G. F. Nellis and J. M. Pfotenhauer, "Experimental investigation and modeling of inertance tubes," Journal of fluids engineering, vol. 5, no. 127, pp. 1029-1037, 2005.
[14] R. Radebaugh, M. Lewis, E. Luo, J. M. Pfotenhauer, G. F. Nellis and L. A. Schunk, "Inertance Tube Optimization for Pulse Tube Refrigerators," AIP Conference Proceedings, vol. 823, no. 1, pp. 59-67, 2006.

[15] M. IGUCHI, M. OHMI and K. MAEGAWA, “Analysis of free oscillating flow in a U-shaped tube,” Bulletin of JSME, vol. 25, no. 207, pp. 1398-1405, 1982.

[16] T. Fang, P. S. Spoor, S. M. Ghiaasiaan and M. Perrella, "Influence of minor geometric features on Stirling pulse tube," to be presented in 2017 Cryogenic Engineering Conference.

[17] D. Gedeon, Sage User’s Guide v11 Edition, 16922 South Canaan Rd.: Gedeon Associates, 2016.