Computational Fluid Dynamic Investigation of Loss Mechanisms in a Pulse-Tube Refrigerator

K. Martin\textsuperscript{1}, J. Esguerra\textsuperscript{2}, C. Dodson\textsuperscript{3} and A. Razani\textsuperscript{4}
\textsuperscript{1}Applied Technology Associates 1300 Britt St. SE Albuquerque, NM 87123-3353
\textsuperscript{2}Glacier Technical Solutions, LLC, El Paso, TX 79925
\textsuperscript{3}Spacecraft Component Thermal Research Group at Air Force Research Labs Kirtland AFB, NM 87117-5776
\textsuperscript{4}The University of New Mexico, Albuquerque, NM 87131

Abstract. In predicting Pulse-Tube Cryocooler (PTC) performance, One-Dimensional (1-D) PTR design and analysis tools such as Gedeon Associates SAGE\textsuperscript{®} typically include models for performance degradation due to thermodynamically irreversible processes. SAGE\textsuperscript{®}, in particular, accounts for convective loss, turbulent conductive loss and numerical diffusion “loss” via correlation functions based on analysis and empirical testing.

In this study, we compare CFD and SAGE\textsuperscript{®} estimates of PTR refrigeration performance for four distinct pulse-tube lengths. Performance predictions from PTR CFD models are compared to SAGE\textsuperscript{®} predictions for all four cases. Then, to further demonstrate the benefits of higher-fidelity and multidimensional CFD simulation, the PTR loss mechanisms are characterized in terms of their spatial and temporal locations.

1. Introduction

Pulse tube cryocoolers (PTCs) play an important role for cryogenic cooling of space-based infrared detectors as well as many other applications requiring coolers with high reliability, low vibration, and high efficiency\textsuperscript{1}. Due to the complex fluid and heat transfer occurring in the PTCs, cryogenic system designers utilize simulation to predict cooler performance and design parameters for prototype models. Typically, quick first-order models are used which utilize empirical data from experiment, such as Gedeon Associate’s Sage\textsuperscript{3}. These simulations do a good job of predicting cooler performance without having to resort to the more time consuming computation fluid dynamics (CFD) simulations. Several investigators have utilized the CFD simulation of oscillatory heat and mass flows for PTRs\textsuperscript{5-11}.

While the simplicity of 1-D simulation tools facilitates PTR design and analysis, this convenience comes at the cost of modelling detail. An investigator wanting to drill-down into the constitutive relationships or governing principles can be shielded from low-level physical details that may otherwise lead to design insights—or details which may help to avoid design mistakes. In these types of investigations, a higher-order Computational Fluid Dynamics (CFD) simulation complements a 1-D simulation. Whereas 1-D simulation is a sufficient starting point for PTR design, Two-Dimensional (2-D) and Three-Dimensional (3-D) CFD models enable an investigator to refine the design—to explore and visualize “real” physical heat-transfer and fluid flow behavior that has been condensed, simplified or omitted in 1-D modelling tools. In a 2-D or 3-D CFD model, the system dynamics and complexity between the input and output of a particular PTR component are not hidden.
In this regard, higher order CFD is also a means of validating 1-D models, or of tuning lower-order design tools to new performance spaces before physical functional validation or prototyping.

The purpose of this study was to compare the results of changing the length of the pulse tube from a baseline cryocooler described in an optimized design paper by Zhang et. al.\textsuperscript{2} The results are shown for many different lengths as simulated by Sage\textsuperscript{2} and results are compared to four of those same pulse tube lengths using the 2D axisymmetric model solved by the CFD code built by Ansys Fluent\textsuperscript{4}.

2. Reduced order modelling and computational fluid dynamics

Model parameters were in general obtained from Zhang\textsuperscript{2}, with those not in this paper obtained via Sage\textsuperscript{2}. It is well known that a cryocooler can be designed for optimal cooling load or for optimal efficiency\textsuperscript{14}. This complication exists when choosing many of the cryocooler component lengths and diameters. For this study we show this by varying only the pulse tube length and keeping all other component parameters constant. Sage\textsuperscript{2} simulations are shown in Figure 1, it can be easily seen that there is an optimal length for maximum efficiency and maximum cooling load and that these design points are not the same. It should be noted that the full length pulse tube is the one from Zhang and that the values at about 2/3 of that length represent the maximum cooling load length. These two were chosen for CFD simulations along with the 1/3 length and 1.875 times the baseline length, as the represented points on the curve that had similar cooling loads but wildly different efficiencies.

These simulations were thus done with four varying pulse tube lengths of 2.4, 4.8, 7.2 and 13.5 cm. Simulations were run with a hot reject temperature of 300 K, cold side temperature of 90 K and helium charge pressure of 3 MPa. These values where used for the diameters and lengths of the aftercooler (AC), regenerator (REG), cold heat exchanger (CHX), pulse tube (PT), hot heat exchanger (HHX), inertance tube 1 (IT1) and inertance tube 2 (IT2), respectively: 2 cm by 0.5 cm, 2 cm by 6 cm, 2 cm by 0.5 cm, 1.49 cm by 7.1 cm, 1.49 cm by 0.5 cm, 3 mm by 1.09 m and 4.5 mm by 3.2 m. The reservoir (RES) volume was 250 cubic centimeters whose diameter was 4.3 cm by 4.3 cm. The hot, cold and aftercooler heat exchangers are copper woven screen matrix with porosity of 0.68 with wire diameter of 0.025 mm, in the CFD they are modelled as porous media with the viscous resistance permeability and inertial resistance coefficients respectively as $\beta=1.345e^{-9}$, $C=8147$, porosity=.68 \cite{7} with the walls having isothermal boundary conditions. The regenerator is 400 mesh stainless-steel, which in Sage is modelled as a Random Fiber Matrix SS304 and in the CFD using the porous media axial and radial viscous resistance permeability and inertial resistance coefficients respectively $\beta_x=2.5295e^{-11}$, $C_x=1.2e5$, porosity=.692 and $\beta_r=5.348e^{-12}$, $C_r=240000$ (Ref. \cite{7}). The compressor (CP) in Sage was modelled as a 8.2 cubic centimeter stroke and in the CFD the piston has a diameter of 2.5 cm and a stroke length of 5.6 mm operating at 48 Hz, for which the step size was chosen to be 2.60543 microseconds. The dead volume (DV) between the compressor and aftercooler had a length of of 1 cm and diameter of 2.5 cm.
Sage® simulations varied pulse tube length from 2 cm to 19.8 cm for a 90 K cryocooler. Values not given in the paper were estimated using Sage optimization simulation using Sage’s pulse tube model and ideal helium for the working fluid. The compressor volume swept in Sage was adjusted such that both Sage and CFD had similar input powers. The cold head simulated in Sage is shown in [13].

The commercial CFD code Ansys Fluent (version 14.5.7)\(^4\) was used to numerically solve the time dependent mass, momentum, and energy equations for a compressible ideal gas helium working fluid. The compressor piston was modelled as a dynamic mesh that utilized a user-defined function (UDF) to describe the sinusoidal velocity of the piston wall as a function of time. The heat exchangers of the aftercooler, regenerator, cold heat exchanger (HX) and the hot HX were modelled using Fluent’s porous media models\(^7\). Two inertance tubes are modelled and are attached to a fully meshed reservoir. All of the walls are modelled as adiabatic, except for the HXs which are isothermal. The simulations are run using the unsteady-second-order time method that solves the Navier-Stokes equations utilizing the PRESTO pressure solver, the PV-coupling scheme, first-order methods for momentum, density, temperature and \(k-\omega\) turbulence model. The regenerator and pulse tube were initialized with gradients from hot to cold temperatures, while the cold HX was initialized with the cold temperature so that the simulations would reach quasi-steady state more rapidly than if initialized from room temperature. The boundary layer along the wall was adapted for the initial time-step of all component walls except the moving piston face. Monitors were placed at each component junction to generate output files for the typically area-weighted averaged CFD data at each time step. Each cycle consisted of 800 individual time steps. The baseline simulation was run for about 400 cycles and the others about 120 cycles. The data is close to quasi-steady state and thus these results are considered preliminary in nature.

### 3. Discretized Exergy Analysis

To further interrogate the data provided by the CFD simulations a novel technique is utilized that is in essence a spatially discretized exergy analysis. It differs from standard exergy analysis such that the exergy is spatially calculated at various axial and radial positions, where the standard technique calculates exergy at component control volumes. This technique should enable designers to pinpoint areas were the pressure and the thermal exergies have large changes. The specific exergy is:

![Figure 1: Plot showing how cooling load and efficiency change as a function of pulse tube length. Sage® simulations varied pulse tube length from 2 cm to 19.8 cm for a 90 K cryocooler.](image)
\[ e = (h - h_0) - T_0(s - s_0) \]  

(1)

For an ideal gas with constant \(c_p\), it can be shown that the thermal exergy at each specific axial position \(x\) can be integrated from \(r\) to \(\Delta r\), as \(r\) goes from 0 to the full radius \(R\) and is given as:

\[
E_{th} \left( r + \frac{\Delta r}{2}, x, t \right) = \int_r^{r+\Delta r} m_x(r', x, t) c_p(T(r', x, t) - T_0) 2\pi r' dr' 
\]

\[
- T_0 \int_r^{r+\Delta r} m_x(r', x, t) c_p \ln \frac{T(r', x, t)}{T_0} 2\pi r' dr' 
\]

(2)

And the pressure exergy as:

\[
E_p (r, x, t) = T_0 \int_r^{r+\Delta r} m_x(r', x, t) R \ln \frac{p(r', x, t)}{p_0} 2\pi r' dr' 
\]

(3)

If one were to sum the discretized exergy in the radial direction at the junction between two components one would obtain the standard exergy at the output of the first component. The discretized exergy thus yields exergy in the direction of the mass flow rate as given in the \(x\) direction of the velocity component by:

\[
m_x(r, x, t) = \rho(r, x, t) A(r, x, t) v_x(r, x, t) 
\]

(4)

Using the \(x\)-direction of velocity the discretized exergy component then flows along the axial (\(x\)) direction and the ‘discretized components’ are the control volumes as one goes along this axial direction.

4. Results

The CFD provides a wealth of data at the discretized spatial and temporal points of the various components of the cryocooler. For instance, for the baseline cooler (or any of the other configurations) one can obtain the mass flow rates and pressure in the pulse tube as averaged at an axial position as shown in the left-side of Figure 2, or the pressure at several axial positions in the pulse tube in the right-side of Figure 2. The area-weighted pressure is shown at various ‘nodes’ or axial positions and it can be seen that little pressure drop occurs in the pulse tube as expected. The CFD provides discrete data along a radius (as well as at axial positions), interesting information can be obtained for the fluid variables. In Figure 3 is shown some of this type of data for temperature, on the

![Figure 2](image-url): Mass flow rate and pressure at the cold side of the pulse tube at various axial positions in the pulse tube.
left is the baseline cooler and on the right is the 1/3 length pulse tube configuration. Shown at each axial position in each component is the minimum, maximum, average and average plus/minus 1-σ (over time) values for all radial data. This shows how much larger the temperature oscillation is in the pulse tube compared to the rest of the components that are not heat exchangers. The temperature swing in the third length configuration shows a much larger temperature swing than the baseline cooler and from a design viewpoint would probably not be desired.

Contour plots of the spatially discretized thermal, pressure and total exergies are shown respectively in Figures 4, 5 and 6. The spacing between contours is shown in the figures as ΔW. Quantifying exergy components to thermal component and pressure components helps designer of cryocoolers to see how the components change as the gas oscillates in the cryocoolers. For example the variation in thermal exergy in regenerator is mainly due to exergy destruction as a result of heat transfer through the temperature difference between the working fluid and the regenerator material. On the other hand the variation in the pressure component is mainly due to fluid friction as working fluid passes through the regenerator. The exergy analysis helps in the design of more effective regenerator for cryocoolers. These plots show that the thermal exergy is most important in the pulse tube analysis and that the pressure exergy is dominant in the rest of the components. Junctions between different radial-sized components show that pressure exergy is high around the junction edges.

Figure 7 shows the irreversibilities in each of the components and the results when the irreversibility is calculated at 5 different axial positions in the pulse tube. It can be seen that the majority of the irreversibility in the pulse tube is very close to the cold heat exchanger. The negative values of irreversibility are currently being attributed to lack of mass (or energy) conservation of the CFD simulations as has been previously discussed [12], in which the lack of mass conservation at the cold side is more sensitive than at the hot side. Another potential culprit could be that there is a phenomenon at a lower frequency that would require multiple cycles of exergy calculations to properly resolve, such as some direct flow that is lower in frequency than the compressor operating frequency. The performance of the four coolers is shown in Table 1. In the CFD simulations the baseline cooler and the 2/3 pulse tube length configuration both provided cooling, but the other two did not as compared to Sage ®.
Figure 4: Thermal Exergy as a function of axial and radial coordinate for the components of a PTC, note how the pulse tube is dominated by thermal exergy compared to other components.

Figure 5: Pressure Exergy as a function of axial and radial coordinate for the components of a PTC.

Figure 6: Total Exergy as a function of axial and radial coordinate for the components of a PTC.
Figure 7: Irreversibility distribution in the various components and at five equidistant positions in the pulse tube for the baseline cooler.

Table 1: Cooling performance of cooler configurations and estimated exergy destruction in 5 equal sections of a pulse tube

| Pulse Tube Length | Load CFD/Sage (Watts) @ 90K | Estimated Exergy Destruction In Pulse Tube (Watts) |
|-------------------|-----------------------------|---------------------------------------------|
| 1/3 Length (2.4-cm) | -17.3/2.1 (heating) | Segment 0 (near CHX) 16.95 Segment 1 25.81 Segment 2 -3.09 Segment 3 -0.72 Segment 4 0.94 |
| 2/3 Length (4.8-cm) | 0.5/8.0 (cooling) | Segment 0 (near CHX) 31.31 Segment 1 -3.23 Segment 2 -0.66 Segment 3 -0.0041 Segment 4 0.15 |
| Baseline (7.2-cm) | 1.5/7.5 (cooling) | Segment 0 (near CHX) 25.66 Segment 1 -1.51 Segment 2 -0.37 Segment 3 -0.027 Segment 4 0.09 |
| 1.875 times Baseline (13.5-cm) | -1.7/3.0 (heating) | Segment 0 (near CHX) 16.38 Segment 1 -0.43 Segment 2 -0.17 Segment 3 -0.059 Segment 4 0.04 |

5. Conclusions
A new discretized exergy method has been shown that can yield insights into where losses exist as a function of spatial position in the PTC. It has been shown that the majority of the losses in the pulse tube potentially occur near the cold heat exchanger side of the tube where a significant change in the thermal component of exergy exists. There are complex losses near junctions, including areas where little potential for exergy change exists. To calculate the irreversibility due to fluid friction and heat transfer in the regenerator, a thermal non-equilibrium model of regenerator is required and is not considered in this study. The irreversibilities in the pulse tube that are negative, obviously violate the second law of thermodynamics and may be due to numerical issues or require more cycles to resolve. The effect of variation of the length of pulse tube on the irreversibility distribution in the pulse tube is reported in this study.

The differences between Sage and the CFD cooling results are even harder to qualify. They could be any of the dimensional effects that CFD simulates, that the Sage empirical values do not. This includes geometrical values like wall roughness, junction transitions, or heat exchanger geometries and simulation parameters like the turbulence model used. Of the many parameters in the CFD, the heat exchanger modelling may be the most likely culprit of the large differences. The heat
exchanger porous media model parameters for the viscous resistance permeability and inertial resistance coefficient in Cha’s paper [7] were iteratively derived between the CFD and his experimental setup. These values from the porous media models in the CFD have a large effect on the mass flow rates and pressure drops through the heat exchanger and have an effect on the exergy calculations of all of the components of a PTC.

References
1. Radebaugh, R., “Development of the pulse tube refrigerator as efficient and reliable cryocoolers,” *Proceedings of Institute of Refrigeration*, 2000, pp. 11-31.
2. Zhang, A., Chen, X., Wu, Y., Zhang, H. and Yang, K., “Study on 10 W/90K in-line pulse tube cryocooler,” *Advances in Cryoegenic Engineering*, Vol. 52, 2012, pp.800-804.
3. Gedeon Associates Sage, version 8.0
4. ANSYS® Fluent, Release 14.5.7
5. Dodson, C. *The comparison of CFD simulations and a first order numerical model of inertance tubes*, Master’s Thesis, University of New Mexico, 2007.
6. Taylor, R., Nellis, G. and Klein, S., “Optimal Pulse-Tube Design Using Computational Fluid Dynamics”, *Advances in Cryoegenic Engineering*, vol. 53, pp.1445-1453, 2008.
7. Cha, J.S., Ghiaasiaan, S.M., Desai, P.V., Harvey, and J.P., Kirkconnell, C.S., ‘Multi-dimensional flow in pulse tube refrigerators,” *Cryogenics*, 46, (2006), 658-665.
8. Chen, L., Zhang, Yu, Luo, E., Li, T., Wei, X., “CFD analysis of thermodynamic cycles in a pulse tube refrigerator,” *Cryogenics*, 50, (2010), 743-749.
9. Ashwin, T.R., Narasimham, G.S.V.L., and Jacob, S., “CFD analysis of high frequency miniature pulse tube refrigerators for space applications with thermal non-equilibrium model,” *Applied Thermal Engineering*, 30, (2010), 152-166.
10. Taylor, R.P., Nellis, G.F., Klein, S.A., Radebaugh, R., Lewis, M., Bradley, P., “Experimental validation of a pulse tube CFD model,” *Advances in Cryoegenic Engineering*, vol. 55, pp. 76-83, 2010.
11. Flake, B and Razani, A., “Modeling pulse tube cryocoolers with CFD”, *Advances in Cryogenics Engineering*, Vol. 49B, pp.1493-1499.
12. Dodson, C., Razani, A. and Roberts, T., ‘Numerical simulation of oscillating fluid flow in inertance tubes’, Cryocoolers 15, 2008
13. Martin, K., Dodson C., Razani, A., ‘Comparison of Fluid Flow and Heat Transfer for 1D and 2D models of an In-line Pulse Tube Refrigerator’, Proceedings of 18th International Cryocooler Conference, pp. 221-227, 2014.
14. Razani, A., Roberts, T. and Flake, B., ”The Second-Law Based Thermodynamic Optimization Criteria for Pulse Tube Refrigerators,” Proceedings of 14th International Cryocooler Conference, pp. 285-292, 2006.