EXPERIMENTAL VALIDATION OF A PULSE TUBE CFD MODEL

R.P. Taylor[1], G.F. Nellis[2], S.A. Klein[2],
R. Radebaugh[3], M. Lewis[3], and P. Bradley[3]

[1]Department of Mechanical Engineering
Virginia Military Institute
Lexington, VA 24450 USA

[2]Cryogenic Engineering Lab
University of Wisconsin-Madison
Madison, WI 53706 USA

[3]Cryogenic Technologies Group
National Institute of Standards and Technology
Boulder, CO 80305 USA

ABSTRACT

Computational fluid dynamic (CFD) analysis has been applied by various authors to study the processes occurring in the pulse tube cryocooler and carry out parametric design and optimization. However, a thorough and quantitative validation of the CFD model predications against experimental data has not been accomplished. This is in part due to the difficulty associated with measuring the specific quantities of interest (e.g., internal enthalpy flows and acoustic power) rather than generic system performance (e.g., cooling power). This paper presents the experimental validation of a previously published two-dimensional, axisymmetric CFD model of the pulse tube and its associated flow transitions. The test facility designed for this purpose is unique in that it allows the precise measurement of the cold end acoustic power, regenerator loss, and cooling power. Therefore, it allows the separate and precise measurement of both the pulse tube loss and the regenerator loss. The experimental results are presented for various pulse tube and flow transition configurations operating at a cold end temperature of 80 K over a range of pressure ratios. The comparison of the model prediction to the experimental data is presented with discussion.

KEYWORDS: Pulse Tube, Cryocooler, Computational Fluid Dynamics, Design Model.

CP1218, *Advances in Cryogenic Engineering: Transactions of the Cryogenic Engineering Conference - CEC, Vol. 55,* edited by J. G. Weisend II

INTRODUCTION

In recent years there has been increased application of computational fluid dynamic (CFD) models to pulse tube cryocooler (PTC) design. This has been guided by the lack of precise information available from zeroth- and first-order PTC design models regarding multi-dimensional flow effects that strongly influence system performance. Researchers such as Cha et al.[1], Hozumi [2], and Flake and Razani [3] have presented multi-dimensional CFD analyses of the entire PTC using the commercial CFD software FLUENT. The consistent conclusion of these researchers is that two-dimensional flows in a PTC substantially influence system performance and must be accounted for in the design process.

Recently the authors have developed a multi-dimensional CFD model that is focused on the pulse tube and flow transitioning components in a PTC, rather than the entire PTC. Initial modeling results are consistent with those achieved by other researchers and demonstrate the usefulness of component-level CFD analysis in PTC design. In order to have the necessary confidence in the quantitative predictions from the CFD model required for a design tool, it is necessary to experimentally validate the model. To date there have been few experimental validation studies of CFD simulations that go beyond comparing gross energy quantities such as the heat rejected at the hot heat exchanger, heat accepted at the cold heat exchanger, and the input power to the system. In this paper, we present the experimental validation of a multi-dimensional CFD model for the pulse tube and flow transitions in a PTC.

PULSE TUBE CFD MODEL

The CFD model is focused on the pulse tube and flow transitioning components within a PTC (rather than simulating the entire PTC). The benefit of this approach is that useful results can be obtained in a computationally efficient manner by focusing the computational analysis on the area of the PTC where the multi-dimensional character of the flow may result in substantial refrigeration losses. The modeling methodology and specific details are discussed by Taylor [4] and briefly summarized here. The CFD model is a two-dimensional (2-D) axi-symmetric representation of the pulse tube and flow transitioning components implemented using the commercial CFD solver package FLUENT [5]. The notable features of the model include:

1. use of a porous media model that employs empirical data to represent the inertial and viscous flow resistances in the axial and radial directions for the packed wire mesh screens that are used in the flow straightening components,
2. the simulation of the turbulence associated with the warm end, high velocity gas flows using the Renormalization Group Theory k-ε model,
3. the ability to model two working fluids, ^{4}He and ^{3}He, via the use of the ideal gas equation of state (for high temperature simulation) or the real gas properties (for low temperature simulations) by coupling the NIST REFPROP package to the CFD simulation [5, 6], and
4. simulation times that range from 6 to 48 hours (i.e., days) compared to other models of the entire PTC that require simulation times on the order of multiple weeks to reach a converged solution with sufficient resolution for quantitative predictions.

The inputs required for a model simulation are the cold end mass flow rate, the phase angle between pressure and flow at the cold end of the pulse tube, the mean pressure and pressure ratio at the cold end of the pulse tube. The outputs from the model are both quantitative and qualitative. The quantitative predictions include the pulse-tube enthalpy flow, the acoustic power flow, and the effectiveness of the pulse-tube component with respect to converting acoustic power into useful cooling. Qualitative results include the visualization of the temperature contours and velocity vectors which can be used to identify areas and sources of flow non-uniformity.

EXPERIMENTAL VALIDATION METHODOLOGY

The quantities which must be measured experimentally in order to validate the CFD model can be understood using an energy balance applied to the cold end of a PTC, as illustrated in FIGURE 1. This energy balance is expressed as,

$$\dot{E}_{REG,c} + \dot{Q}_{net} = \dot{E}_{PT,c} \tag{1}$$

where $\dot{E}_{REG,c}$ is the regenerator energy flow term (also called the regenerator loss), $\dot{Q}_{net}$ is the net cooling power provided by the cooler, and $\dot{E}_{PT,c}$ is the net energy flow through the pulse-tube and flow transitioning components. An additional term illustrated in FIGURE 1 (but not part of the energy balance) is, $\dot{W}_{PV,c}$, the acoustic power flow at the cold end of the system. This term is the maximum theoretical refrigeration power that can be attained from the flow and is used to define the figure of merit (i.e., the efficiency) of the pulse tube component by normalization of the pulse tube enthalpy flow.

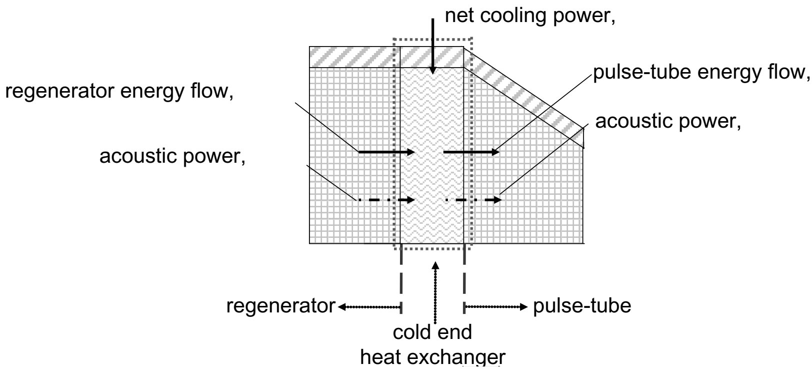


FIGURE 1. An energy balance applied to the cold end heat exchanger in a PTC showing the energy and power flows.

The pulse tube enthalpy flow and the acoustic power flow are the primary quantities predicted by the CFD model; therefore, these are the two quantities that must be measured in order to verify the CFD model. In an experimental system, the quantity that is most directly measurable is the net cooling power. The energy balance presented in Eqn. (1) shows that the net cooling power is equal to the pulse tube enthalpy flow less the regenerator loss. Therefore, the regenerator loss must be separately measured in order to infer the pulse tube enthalpy flow from the net cooling power. The acoustic power must also be measured in order to determine the figure of merit from the pulse tube enthalpy flow.

Based on this discussion, the quantities that must be measured for a complete verification of the CFD model include the regenerator loss, the net cooling power, and the acoustic power flow. The pulse-tube enthalpy flow and pulse tube figure of merit can be determined based on these measurements. It was necessary to develop specialized experimental methods in order to measure these quantities in an operational PTC. The experimental methodology that was developed is composed of five distinct parts,

1. Calibration of a thermal bus that allows the rate of heat transfer from/to the cold end to be measured,
2. Calibration of a custom mass flow meter for use under oscillatory flow conditions at cryogenic temperatures,
3. Measurement of the regenerator loss, independent of the pulse-tube component,
4. Measurement of the net cooling power with a pulse tube, and
5. Measurement of the acoustic power flow at the cold end of the system.

The reader is directed to Taylor et al. [4, 7] for a more thorough discussion of the test facility and experimental method.

EXPERIMENTAL RESULTS

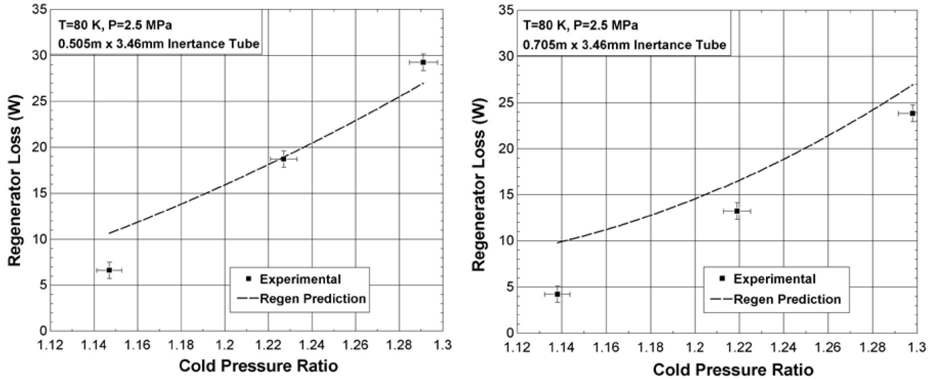
Regenerator Loss Characterization

The regenerator for the experimental test facility was designed to allow for the measurement of the regenerator loss over a range of cold end phase angles and pressure ratios. The cold end phase angle between the flow and pressure was varied by installing different inertance tubes. The pressure ratio was varied by adjusting the stroke of the compressor. The operating conditions and geometric specifications associated with the regenerator experimental measurements are listed in TABLE 1.

TABLE 1. Nominal Regenerator Design Parameters

Parameter	Symbol	Nominal Value
Matrix material	-	400 mesh SS304
Length	L	32.3 mm
Diameter	D	34.9 mm
Mean system pressure	$\overline{P}$	2.5 MPa
Pressure ratio (cold end)	PR	1.3
Frequency	f	60 Hz
Cold end temperature	T_c	80 K
Warm end temperature	T_h	300 K
Mass flow rate	$\dot{m}$	16 g/s
Cold end phase angle	θ	variable

For all experimental test permutations, the regenerator loss, mass flow rate, and cold end phase angle were determined from the raw experimental measurements using the data reduction process discussed by Taylor [4]. The measured mass flow rate, phase angle, and the pressure ratio at the cold end associated with each of the test conditions are used as inputs to the numerical model REGEN3.3 developed by Gary et al [8]. The experimentally measured regenerator loss is compared to the predicted loss from REGEN3.3. FIGURES 2 (a) and (b) illustrate the measured and predicted regenerator loss as a function of pressure ratio for a mean pressure of 2.5 MPa and a cold temperature of 80 K with two, different inertance tubes.



(a) (b)

FIGURE 2. Comparison between the measured regenerator loss and predicted loss for an inertance tube length and diameter of (a) 0.505 m and 3.46 mm respectively, and (b) 0.705 m and 3.46 mm respectively.

The results presented in FIGURE 2 show good agreement between the experimental measurements and the corresponding predictions obtained using REGEN3.3. The experimental results and model predictions clearly follow the same trends and the measurements and predictions agree to within 20% at moderate to high pressure ratios. The discrepancy between the measurements and model predictions does grow to approximately 50% as the pressure ratio is reduced. The experimental uncertainty in the regenerator loss measurement is relatively constant over the range of tested pressure ratios. However, as the pressure ratio decreases so does the regenerator loss and therefore the error becomes a larger fraction of the measured loss and it becomes difficult to accurately resolve this quantity. Also, at low pressure ratios the mass flow rate is reduced and therefore the measurement of the pressure difference across the thermal intercept, from which the mass flow is determined, becomes difficult due to a low signal-to-noise ratio. This second effect is likely the main source of the discrepancy between the experimental results and the model predictions at low pressure ratios. The mass flow rate is a primary input to the REGEN3.3 models and therefore any error in this measurement will have a large impact on the predicted regenerator loss.

Pulse Tube Loss Characterization

Two different pulse tubes were tested, each with multiple flow transition configurations. Each geometric configuration was tested over a range of pressure ratios. The two pulse tube configurations had different aspect ratios but the same overall pulse tube volume. The non-dimensional pulse tube diameter, D_{ND}, is a convenient scaling parameter for the pulse tube aspect ratio. The value of D_{ND} is zero if the diameter of the pulse tube is the same as the diameter of the inertance tube (very long and skinny) and unity if the pulse tube has the diameter of the regenerator (pancake shaped). The two regenerators that were tested had dimensionless pulse tube diameters of 0.25 and 0.40. The volume of a conical flow transition component at the warm end of the pulse tube was also changed. Two different hot end flow transitions were manufactured for each pulse tube design. The void volume associated with the two flow transitions were 5% and 15% of the pulse tube internal volume. The geometry and operating conditions utilized for the pulse tube experimental measurements are listed in TABLE 2.

TABLE 2. Nominal Pulse Tube Parameters

Parameter	Symbol	Design 1	Design 2
Gas Volume	V	50 cc	50 cc
Non-dimensional diameter	D_{ND}	0.25	0.4
Aspect Ratio	AR	10	4.5
Length	L	132.1 mm	78.7 mm
Diameter	D	13.8 mm	18 mm
Mean system pressure	$\overline{P}$	2.5 MPa	
Pressure ratio (cold end)	PR	1.3	
Frequency	f	60 Hz	
Cold end temperature	T_c	80 K	
Warm end temperature	T_h	300 K	
Mass flow rate	$\dot{m}$	16 g/s	
Cold end phase angle	θ	variable	

For all experimental test permutations, the net cooling power, acoustic power, mass flow rate, and cold end phase angle (between pressure and flow) were determined from the raw experimental measurements using the data reduction process discussed by Taylor [4]. The enthalpy flow was determined by adding the regenerator loss predicted by REGEN3.3 at the experimentally measured operating conditions to the experimentally measured cooling power. The predicted regenerator loss was used because it was not possible to exactly match the test conditions that were used to measure the regenerator loss in isolation; this might have been possible for a single case, but not for each permutation of the pulse tube design, flow transition design, and pressure ratio. The regenerator loss measurements showed that REGEN3.3 is capable of simulating the regenerator under these experimental conditions. The acoustic power flow at the cold end of the pulse tube component was determined using the measured mass flow rate, the measured phase angle between pressure and flow, and the pressure amplitude at the cold end. The measured mass flow rate, phase angle, and the cold end pressure ratio associated with each of the test conditions were then used as inputs to the CFD model. The measured and predicted enthalpy flow and the acoustic power as a function of cold end pressure ratio are illustrated in FIGURE 3 (a) through 3 (d) for each permutation of pulse tube and hot end flow transition.

FIGURE 3 shows that there is excellent agreement between the experimental measurements and the corresponding predictions using the CFD model. Note that in all cases, the experimentally measured acoustic power and the acoustic power used for the CFD model are identical because the parameters that determine the acoustic power correspond to the input parameters for the CFD model; therefore, only the experimental acoustic power is plotted. For all test cases, the CFD model predicts the correct enthalpy flow to within 15% of the measured value. The model predicts the correct value of the enthalpy flow (to within experimental error) even at relatively low pressure ratios (1.1 to 1.2). There is some error at low pressure ratios that is again attributable to the reduced mass flow sensor resolution at low pressure ratios. At high pressure ratios, the model seems to be offset slightly above the experimental predictions. This offset may be explained by the shuttle heat transfer loss was not accounted for in the CFD model for these simulations and will tend to increase with pressure ratio.

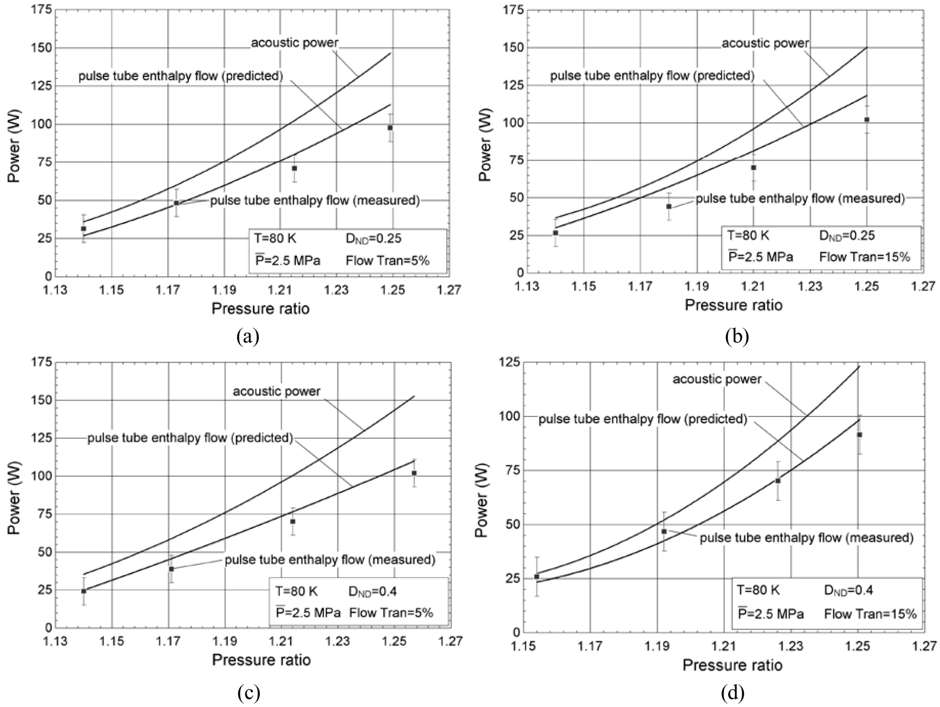


FIGURE 3. Experimental measurement of the pulse tube enthalpy flow overlaid on the CFD model prediction as a function of pressure ratio for each pulse tube flow transition permutation, indicated by the dimensionless pulse tube and the ratio of the hot flow transition volume to the pulse tube volume (a) D_{ND} = 0.25 and 5% transition volume, (b) D_{ND} = 0.25 and 15% transition volume, (c) D_{ND} = 0.4 and 5% transition volume, and (d) D_{ND} = 0.4 and 15% transition volume.

The simulation and experimental work suggest that the dominant source of loss in the pulse tube is related to flow mal-distribution induced at the warm end of the system. This was not an unexpected conclusion due to relatively high velocity gas jet exiting from the small diameter inertance tube. The results show that a flow transition that consists of an open conical section terminated by a short screen pack helps the flow to radially equilibrate before it enters the pulse tube. The transition performance is nearly independent of the volume, provided that there is some finite volume such that the flow can expand sufficiently before entering the heat exchanger and pulse tube component. This is a highly useful result for the pulse tube designer as it identifies one attractive flow transition configuration for the flow ranges tested and validated herein.

CONCLUSION

The CFD modeling tool and component level modeling methodology has been shown to be useful for understanding and designing pulse tube and flow transitioning components in a PTC. The model is capable of predicting the actual enthalpy flow of the pulse tube component with an error no larger than 15% for all of the experimental test cases. The model and the overall methodology are flexible and applicable to a wide range of pulse tube cooler applications and operating conditions. This work has illustrated that advanced computational analysis, when applied carefully to a specific component within a highly complex system such as a PTC, can be utilized in an efficient and useful manner in order to enhance the design and system level modeling required to deploy high efficiency PTCs.

ACKNOWLEDGEMENTS

This work was supported by the Office of Naval Research under contract N00014-06-1-0007 and the CEC Timmerhaus Scholarship. Additionally, the authors would like to thank the Mike Rybowiak for careful machining of parts used in the experimental measurements.

REFERENCES

1. Cha, J.S., Ghiaasiaan, S.M., Desai, P.V., Harvey, J.P., and Kirkconnell, C.S., "CFD Simulation of Multi-Dimensional Effects in an Inertance Tube Pulse Tube Refrigerator," *Proc. 13th Cryocooler Conf.*, R. G. Ross, Jr., ed., Springer Science, New York, NY, pp. 285-292, (2005)..
2. Flake B. and Razani, A., "Modeling Pulse Tube Cryocoolers with CFD," *Adv. Cryogenic Eng.*, Vol. 49, pp. 1493-1499, (2004).
3. Hozumi, Y., Murakami, M., and Iida, T., "Numerical Study of Pulse Tube Flow," *Proc. 10th Int. Cryocoolers Conf.*, R. G. Ross, Jr., ed., Kluwer Academic/Plenum Publishers, New York, NY, pp. 321-328, (1999).
4. Taylor, R.P., "Optimal Pulse Tube Design Using Computational Fluid Dynamics," *PhD Dissertation.*, The University of Wisconsin-Madison, (2009).
5. FLUENT, "FLUENT 6.3 Users Manual," Fluent INC, New Haven, Connecticut, (2007).
6. Lemmon, E., "Personal Communication", National Institute of Standards and Technology, (2008).
7. Taylor, R.P., Nellis, G.F., Klein, S.A., Lewis, M., Bradley, P., and Radebaugh, R., "Design of an Experimental Test Facility for Measurement of Pulse Tube Energy Flows," *Proc. 15th Int. Cryocoolers Conf.*, S.D. Miller and R. G. Ross, Jr., ed., Kluwer Academic/Plenum Publishers, New York, NY, pp. 191-199, (2009).
8. Gary, J., O'Gallagher, A., Radebaugh, R., Marquardt, E., and Huang, Y., "REGEN3.3 User Manual," National Institute of Standards and Technology, (2008).