Review on CFD analysis of a single stage Stirling type Pulse tube cryocooler

Ankitsinh J. Rajput[1]

[1] M.E. (Pursuing), Thermal Engineering, A.D.I.T., New V. V. Nagar, Gujarat, India

Abstract

This paper presents review of numerical analysis of a single stage Stirling type Pulse tube cryocooler using commercial available computational fluid dynamic (CFD) software package ANSYS FLUENT®. Pulse tube cryocooler is device used to produce cryogenic temperature and because of absence of moving parts in cold end region, it gives lesser vibration and reliable service over Stirling cryocooler. Due higher reliability, lesser vibration ,no displacer in cold end side, it finds applications in aerospace application, military application for cooling detectors, cooling superconducting magnets, gas liquefaction plants etc. Nodal analysis, Phasor analysis, isothermal analysis gives results with different level of accuracy that lower than numerical one. But simulation of exact phenomenon occurs in pulse tube cryocooler with given level of accuracy is possible by CFD software packages. Complex periodic oscillatory flow occurring in pulse tube can accurately analyzed using CFD analysis. The software enables modeling and simulation of complex transient, multidimensional fluid flow and heat transfer problems occurs in complex geometries. This paper gives idea of work that has been carried out using CFD software for different performance and parametric studies of pulse tube cryocooler. Paper shows modeling of various components like regenerator, pulse tube, heat exchangers, phase shift mechanism and reservoir; parametric study on Pulse tube cryocooler like effect of operating condition and physical dimension; cool down behavior, pressure and velocity variation. All heat exchanger, after cooler and regenerator are modeled as porous media. Thermal equilibrium and non-thermal equilibrium scheme can be used to simulated heat transfer between working gas and solid matrix. Idea of User defined function (UDF) to simulate piston cylinder effect is also presented.

Keywords: *Pulse tube Cryocooler, UDF, CFD*

---

1. Introduction

Pulse tube cryocooler is mechanical device capable of producing cryogenic temperature below 123 K. Due to no moving part in colder region, lesser vibration, lesser maintenance and higher reliability, pulse tube cryocooler is current area of research and suitably used in aerospace application , military application, semiconductor cooling, gas liquefaction plant etc. Gifford and Longsworth [1] first time give idea of basic pulse tube cryocooler in 1960s.They proposed that compression and expansion of working gas in closed volume produced temperature gradient over boundary of that closed volume. Mikulin et al. [2] introduce orifice in Basic pulse tube cryocooler and improved its performance. Thereafter lot of research and improvement are noticed for improvement of Pulse tube cryocoolers performance like double inlet type, Inertance tube type, tapered, U type, Coaxial etc. Presently research shows that Inertance tube type pulse tube cryocooler gives best result compared to other type of phase shift mechanism. The exact physical phenomenon undergoing in the operation of PTC is not well understood. Periodic compression and expansion of working gas, heat transfer process between solid body and working gas and phase lag phenomenon are not well understood. Based on simplicity and accuracy level there are different analysis has been carried out like heat pumping phenomenon, Phasor analysis, enthalpy entropy flow model , isothermal model etc. All are gives different level of accuracy but exact phenomenon is not yet well analyzed. Computational fluid dynamic is one of the ways of getting accurate results and understanding exact phenomenon undergoing in PTC. The main objective of this paper is to present the previous work carried out by different researchers related to CFD analysis. Commercial CFD software package ANSYS Workbench FLUENT®. Commercial CFD software package FLUENT® has facilities of dynamic meshing that can used to simulate piston cylinder effect .This paper shows use of CFD software FLUENT® for parametric studies of PTC, cold down behavior, pressure and velocity distribution, effect of various parameters like frequency, porosity of regenerator material etc. on performance of PTC.

Nomenclature

Symbols

X Piston head displacement (m)
X_a Piston head displacement amplitude (m)
V Piston head velocity
ω Angular frequency (rad/sec)
t Time (sec)
$\bar{\bar{C}}$ Inertial drag coefficient tensors (m^{-1})
h Enthalpy (J/kg)
j Superficial velocity (m/s)
k Thermal Conductivity (W/mK)
p Pressure (N/m^2)
T Temperature (K)
v Intrinsic velocity (m/s)
$\bar{\bar{\beta}}$ Permeability tensors (m^2)
ε Porosity
μ Absolute viscosity (Kg/ms)
ρ Density (Kg/m^3)
$\bar{\bar{\tau}}$ Stress tensors (N/m^2)

Subscripts

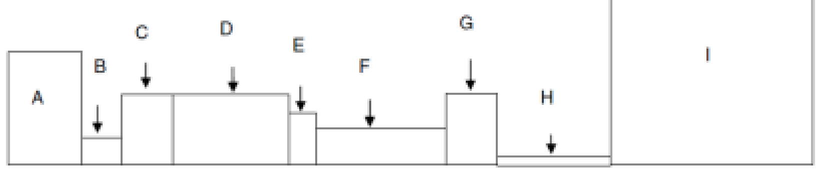
a Amplitude of displacement
f Fluid
s Solid
r Radial coordinate
x Axial coordinate

Abbreviation

CFD Computational Fluid Dynamic
UDF User Defined Function
PTC Pulse Tube Cryocooler

2. Modeling of pulse tube cryocooler

Mostly pulse tube cryocooler are modeled as 2D axis symmetric model to reduce computational time.



A: Compression Space
B: Transfer line
C: After cooler
D: Regenerator
E: Cold end Heat exchanger
F: Pulse tube
G: Hot end Heat exchanger
H: Inertance tube
I: Reservoir

Figure 2: 2-D axis symmetric model of Inertance tube pulse tube cryocooler [3]

Figure 2 shows 2-D axis symmetric model of Inertance tube pulse tube cryocooler [3]. This geometry can be change according to different type and phase shift mechanism. 3-D model can be used for CFD analysis but it takes to much computational time. All heat exchangers like after cooler, cold end and hot end heat exchanger and regenerator are modeled as porous media by giving porosity, permeability and inertial resistance in Fluent®. Cha et al. [3], Ashwin et al. [4], Banjare et al. [5] used porosity, permeability and inertial resistance 0.69, 1.06 E-10 m^2 and 76090 m^{-1} respectively. One can change the property according to different mesh size.

2.1 Dynamic meshing

ANSYS workbench FLUENT® has facilities of dynamic meshing which can be used for simulating reciprocating compressor. Dynamic meshing needs User Defined Function (UDF) that must be written in C program file. UDF guide piston motion and deformable mesh treated as compressible volume of compressor.

Hence virtually piston cylinder effect of compressor can be simulated using UDF dynamic meshing [6]. UDF must be compiled in FLUENT® set up. Piston head motion is given as below [3][4][5],

$$X = X_a * \sin(\omega * t) \tag{1}$$

By taking differentiation to get piston head velocity,

$$V = X_a * \omega * \cos(\omega * t) \tag{2}$$

3. Governing equation

Mass, momentum and energy equation solved by FLUENT® is given as below [3],

$$\frac{\partial \rho_f}{\partial t} + \frac{1}{r}\frac{\partial}{\partial r}\left[r\rho_f v_r\right] + \frac{\partial}{\partial x}\left[\rho_f v_x\right] = 0 \tag{3}$$

$$\frac{\partial}{\partial t}\left[\rho_f \vec{v}\right] + \nabla\left[\rho_f \vec{v}\vec{v}\right] = -\nabla p + \nabla[\bar{\bar{\tau}}] \tag{4}$$

$$\frac{\partial}{\partial t}\left[\rho_f E\right] + \nabla\left[\vec{v}(\rho_f E + p)\right] = \nabla[k_f \nabla T + \bar{\bar{\tau}}\vec{v}] \tag{5}$$

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{6}$$

All properties are considered for working gas helium. All above equation are valid for all component of PTC except porous region like regenerator and all heat exchangers. Mass, momentum equation for porous medium is given as [3],

$$\frac{\partial}{\partial t}\left[\varepsilon\rho_f + \frac{1}{r}\frac{\partial}{\partial r}\left[\varepsilon r\rho_f v_r\right] + \frac{\partial}{\partial x}\left[\varepsilon\rho_f v_x\right] = 0 \tag{7}$$

$$\frac{\partial}{\partial t}\left[\varepsilon\rho_f \vec{v}\right] + \nabla\left[\varepsilon\rho_f \vec{v}\vec{v}\right] = -\varepsilon\nabla p + \nabla[\overline{\overline{\varepsilon\tau}}] - [\mu\overline{\overline{\beta^{-1}}}\vec{j} + \frac{1}{2}\bar{\bar{C}}\rho_f\left|\vec{j}\right|\vec{j}] \tag{8}$$

$$\frac{\partial}{\partial t}\left[\varepsilon\rho_f E_s + (1-\varepsilon)k_s\right] + \nabla\left[\vec{v}(\rho_f E_f + p)\right] = \nabla[(\varepsilon k_f + (1-\varepsilon)k_s)\nabla T + \varepsilon\bar{\bar{\tau}}\vec{v}] \tag{9}$$

4. CFD analysis

Various literature sources are considered in order to get idea of computation fluid dynamic analysis of pulse tube cryocooler. Literature shows modeling of various components of pulse tube cryocooler, parametric analysis which includes various geometrical parameter and performance parameter, cool down behavior, pressure and velocity distribution of pulse tube cryocooler.

Flake and Razani [8] have modeled PTC using CFD. 2D axis-symmetric model was prepared using CFD software. They concluded that transient CFD model successfully predict PTC performance. Mane et al**.** [9] have conducted numerical analysis of pulse tube cryocooler. They used CFD FLUENT® for numerical analysis. Boundary conditions are taken as compressor; regenerator, cold end heat exchanger, Inertance tube and reservoir are adiabatic. After cooler, hot end heat exchanger and transfer line are at 300K. Cool down behavior and phase angle at cold end measured. Temperature at cold end is 132 K. Cool down time is 172s to reach 132K. Phase angle between pressure and mass flow rate is 40^0C.

Cha et al. [3] studied multidimensional flow effect in various Inertance tube PTC. They demonstrate feasibility of CFD software package FLUENT®. They concluded that 1D model is suitable only when system has larger length to diameter ratio. Flow recirculation and multi-dimensional effect occurs when one or more components have relatively small L/D ratio. Recirculation effects on performance of system. When total numbers of nodes are 4200 for system, it gives good results. They shows that second order upwind scheme differential scheme give lesser numerical error. Hot end heat exchanger and after cooler is maintained at 293 K and other components are act as adiabatic. Results show no load temperature is 87 K at cold end of PTC with time increment of 7.3529e-4 sec.

Rout et al.[10] have optimized Pulse tube refrigerator using CFD simulation. In their analysis, they vary the length of pulse tube keeping all other parameter constant. They optimized length of pulse tube that is 125mm for 5 mm constant diameter and 34 Hz frequency. They took 2D axis symmetric model with transient model. PISO algorithm with pressure setting of PRESTO (pressure staggered option) scheme was used. Quadrilateral cells were used for entire computational region.

Banjare et al**.**[5][6] developed 2-D axis symmetric model for ITPTC and OPTC. They numerically simulated the model by considering different frequencies. They used dual opposed piston compressor in their simulation. Piston motion is compiled as UDF facilities in ANSYS FLUENT® package. They observed that at higher frequency, the overall performance of system is deteriorated due to turbulence and recirculation fluid in the system. They found that there is optimum frequency exists for each model for which there is maximum refrigeration capacity occurs.

Rout et al**.** [11] has conducted numerical study and analysis of Inertance type pulse tube cryocooler. Analysis is done by CFD software package FLUENT®. Boundary condition is taken as compressor, transfer-line, regenerator, pulse tube; cold end heat exchanger, Inertance tube and reservoir are adiabatic. After cooler and hot end heat exchanger are at 300 K. Initial pressure and temperature is set as 30 bar and 300K. Cool down behavior, pressure variation inside the system and effect of varying porosity of regenerator matrix on cooling temperature and time is studied. Porosity 0.6 showed good results over other.

Gu et al. [12] did CFD analysis of nonlinear flow and multidimensional transport phenomenon that occurs in PTC. They studied streaming induced by vortices. They concluded that streaming induced by vortices affect the performance of PTC. It caused pressure drop and heat losses. They found streaming is very sensitive with operating frequency and temperature gradient. Nonlinear effect deteriorated the performance of the system.

Ashwin et al.[4] performed numerical simulation of high frequency miniature pulse tube cryocooler. They simulated the result by considering different length to diameter ratio of pulse tube by using FLUENT® software of ANSYS. For modeling the porous media like regenerator, all heat exchanger, thermal non equilibrium of gas and solid matrix were considered. They examined the mechanism of heat transfer and dynamic characteristic of gas flow in pulse tube. They concluded that using thermal non equilibrium of gas and matrix produced much lower temperature at cold end of the pulse tube compared to thermal equilibrium of gas and matrix. UDF function is used to simulated piston-cylinder effect of compressor. They have also provided information of computational time that was 500 seconds of real time needed to reach periodic steady state with time step size of 0.0005 sec. At each time step, 50inner iterations were used for better convergence. They observed that over 25 day of CPU time required with 3 GHz processor and 2 GB ram.

5. Conclusion

Complex periodic oscillating fluid flow in pulse tube cryocooler can effectively analyzed using CFD software package FLUENT®. Effect of various parameters like pressure, porosity, operating frequency etc. can be analyzed. Cool down behavior, pressure and velocity variation inside system, temperature gradient, phase difference between pressure and velocity can be simulated using FLUENT®. Optimization and parametric study are done with minimum cost over experimental analysis. Dynamic meshing using UDF gives facility to simulate actual piston cylinder operation. Time step size and number of iteration play important role on convergence of analysis. 2D axis symmetric model saves computation time to reach results. Various scheme like thermal non equilibrium between gas and porous matrix, pressure scheme like PRESTO, modeling of porous media by setting porosity, permeability and inertial resistance are used to simulated actual process of pulse tube cryocooler.

6. Acknowledgment

I express my gratitude to my friend Nilam Soni for her encouragement, suggestion and guidance for preparing this review work. I am also grateful to my parents, my friends and all supporting members. I am thankful to Dr Vishal N. Singh (HOD & Professor, Mechanical Department, ADIT) and Mr Anurag S. Verma (SCI-ENG,'SF', SAC, ISRO) for giving me this valuable opportunity to work on pulse tube cryocooler under their guidance.

7. References

[1] Gifford W, Longsworth R, “Pulse tube refrigeration”, Trans ASME B J Eng Industry (Series B),(1964), 86:264-8.

[2] Mikulin E, Tarasov A, Shrebyonock M,”Low temperature expansion tubes”, Advances in Cryogenics Engineering, (1984), 29:629-637.

[3] Cha J.S. Ghiaasiaan S.M, Desai P.V. Harvey J.P and Kirkconnell C.S.,” Multi- Dimensional flow effects in pulse tube refrigerators.”, Cryogenics 46, (2006), p 658-665.

[4] Ashwin T. R., G. S. V. L. Narasimham, and Subhash Jacob.,” CFD analysis of High frequency miniature pulse tube refrigerators for space applications with Thermal non-equilibrium model.”, Applied Thermal Engineering 30.2, (2010), p 152-166.

[5] Banjare Y. P., R. K. Sahoo, and S. K. Sarangi.,” CFD simulation of orifice pulse Tube refrigerator.”, ICRTME, (2007), p 1-12.

[6] Banjare Y.P., Sahoo R.K., Sarangi S. K.,” CFD simulation of Inertance pulse Tube refrigerator.”, ISHMT-ASME Heat and Mass Transfer Conference 8,(2008), Paper no. EXM-7 , p 34.

[7] Fluent INC. Fluent 6 User Manual. Fluent INC; 2003. p. 8-1.

[8] Barrett Flake and Arsalan Razani, “Modelling pulse tube cryocooler with CFD”, Advance in Cryogenic engineering AIP proceeding vol.49, (2004), p 1493-1499.

[9] Mane Pravin , Dasare Ashutosh , Deshmukh Ganesh , Bhuyar Pratik, Deshmukh Kshiteej, Barve Aditya,” Numerical analysis of pulse tube cryocooler”,IJIRCET, vol.2,(2013),p 590-598.

[10] Rout S.K., Mukare R., Choudhury B.K., Sahoo R.K., Sarangi S.K.,” CFD simu-lation to optimize single stage pulse tube refrigerator temperature below 60 K.”, International Conference on modelling, optimization and computing ICMOC (2012), p 1524-1530.

[11] Rout Sachindra Kumar, Choudhury Balaji Kumar, Sahoo Ranjit Kumar, Sarangi Sunil Kumar, “Numerical Study and Analysis of Inertance-Type Pulse tube Refrigerator”, WSEAS transactions on heat & mass transfer, vol.9,(2014).

[12] Chao Gu, Yuan Zhou, Junjie Wang, Wei Ji, Qiang Zhou, “ CFD analysis of nonlinear processes in pulse tube refrigerators : Streaming induced by vortices”, International Journal of Heat and Mass transfer ,vol. 55,(2012), p 7410-7418.

.