World Journal of Science and Technology 2012, 2(4):46-49
ISSN: 2231 – 2587
Available Online: www.worldjournalofscience.com

Proceedings of " A National Conference on Advances in Mechanical Engineering (NCAME-2012)"
Held at R.C.Patel Institute of Technology, Shirpur, Dist. Dhule,Maharastra,India.
April 21, 2012

Study of the oscillating flow in the circular tube using CFD

J. J. Patel[1] and U. V. Joshl[2].

[1]Post graduate student, Mechanical engineering Department, S.V.M. Institute of Technology, Bharuch, India
[2]Asst. Professor, Mechanical Engineering Department Government Engineering College, Surat, India

Abstract
Pulse tube refrigerator (PTR) is one of the most promising low temperature refrigerators due to the absence of moving parts in the cold region and associated advantages of better reliability, long life and simplicity. Phenomenon of oscillating flow through the pulse tube, a typical long circular tube section, is least concerned part in the earlier research work published. A Computational Fluid Dynamics (CFD) analysis is done to study fluid flow in oscillating flow condition for air as working fluid. The results are compared with the experimental predictions recently published in the literature. A good agreement is observed between CFD predictions and experimental results. The findings discussed here are useful while designing a pulse tube refrigerator.

Keywords: CFD, Circular tube, Oscillating flow.

INTRODUCTION

Pulse tube refrigerator is a device for cooling to low temperatures. It is a closed system that uses an oscillating pressure at one end (typically produced by a compressor) to generate an oscillating gas flow in the rest of the system. This gas flow can carry heat away from a low temperature point (cold heat exchanger) if the conditions are right. An orifice controlling the flow at the other end of the cooler can provide the right condition for cooling to occur.

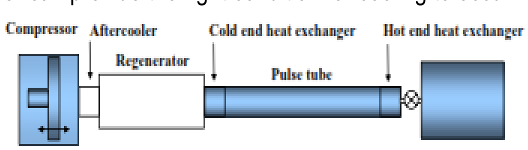

Fig 1.Basic pulse tube refrigerator

Pulse tube is the unique component in this refrigerator that has not appeared previously in any other system. It is simply an open tube. But the thermo-hydrodynamics of the processes involved in it are extremely complex and still not well understood or modeled. The overall function of the pulse tube is to transmit hydrodynamic or acoustic power in an oscillating gas system from one end to the other across a temperature gradient with a minimum of power dissipation and entropy generation. One function of the pulse tube is to insulate the processes at its two ends. That is, it must be large enough that gas flowing from the warm end traverses only part way through the pulse tube before flow is reversed. Likewise, Flow in from the cold end never reaches the warm end. Gas in the middle portion of the pulse tube never leaves the pulse tube and forms a temperature gradient that insulates the two ends. Roughly, the gas in the pulse tube is divided into three segments, with the middle segment acting like a displacer but consisting of gas rather than a solid material. For this gas displacer to effectively insulate the two ends of the pulse tube, turbulence in the pulse tube must be minimized. Thus, flow straightening at the two ends is crucial to the successful operation of the pulse tube refrigerator.

Although much theoretical & experimental work has been reported on the study of a periodically oscillating flow in a pipe, it appears that previous research has been confined to the study of pressure drop in a fully developed oscillating flow. Zhao and Cheng[2] obtained the following exact solutions for the experimental solution for pressure drop in oscillating flow in a circular pipe with diameter D. pressure drops were measured in the fully- developed flow region. Therefore, the reduction of experimental data will be based on a hydrodynamically fully-developed flow whose momentum equation is given by Eq. (1).

$$\frac{\Delta p}{L} = \rho \frac{\partial u}{\partial t} + \frac{4\tau}{D} \quad (1)$$

A hydrodynamically developing oscillating flow in a short pipe occurs frequently in engineering applications, such as in internal combustion engines, Stirling engines, cryocoolers and other periodical processes in thermal and chemical systems. However, relatively few papers have been published on the study of a hydrodynamically developing oscillating flow in tubes because of its complexity.

The purpose of the present study is to gain insights into this

*Corresponding Author

J. J. Patel
Post graduate student, Mechanical engineering Department, S.V.M. Institute of Technology, Bharuch.

Email: jigneshpatel29586@gmail.com

complicated flow phenomena, and to validate the result of experimental and CFD of pressure drop for oscillating flow in a short circular pipe.

Contribution of Zhao and Cheng

S. Zhao and P. Cheng [1] at The Hong Kong University of science & technology have published analytical expression for predicting the friction coefficient of a fully developed oscillating pipe flow (with zero mean velocity) for the axial velocity profile of a fully developed oscillating flow.

Three parameters are identified that govern the oscillating flow phenomenon which is,

1. Kinetic Reynolds number (Reω) defined as:

$$Re_{\omega} = \frac{\omega D^2}{\nu} \qquad (2)$$

2. Dimensionless oscillation amplitude (Ao) defined as:

$$A_0 = \frac{X_{max}}{D} \qquad (3)$$

3. Length to diameter ratio of tube (L/D).

Numerical solution is developed for incompressible fully-developed reciprocating flow in a circular pipe driven by a sinusoidal varying pressure gradient given by

$$\frac{1}{\rho}\frac{\partial p}{\partial x} = k\cos\omega t \qquad (4)$$

The instantaneous friction coefficient, $C_{f,\infty}$ and the cycle-averaged friction coefficient, $\bar{C}_{f,\infty}$ of a oscillating flow is obtained analytically [2] as

$$c_{f,\infty}(\tau) = \frac{\tau_w(\tau)}{\frac{1}{2}\rho u_{max}^2} \qquad (5)$$

$$\bar{c}_{f,\infty} = \frac{1}{2\pi}\int_0^{2\pi} \left|c_{f,\infty}(\tau)\right| d\tau \qquad (6)$$

A correlation equation of the space-cycle averaged friction coefficient for a laminar developing reciprocating pipe flow has been proposed in terms of the three non-dimensional parameters as follows,

$$\bar{C}_{f,\infty} = \frac{3.272}{A_o\left(Re_{\omega}^{0.548} - 2.039\right)} \qquad (7)$$

The analytical expression was compared with experimental data obtained by measuring temporal variations of axial cross-sectional mean velocity and pressure drops downstream of the pipe.

The phenomenon of producing cold effect in a pulse tube refrigerator is dependent on reciprocating flow through all of its major components. The pulse tube can be considered as the most resembled case. It is pointed out in earlier investigations proved by Cheng and Zhao [1] that the presence of "annular flow" can dominate pressure drop for the cases of oscillating flow in tube. An attempt is made in this paper to estimate pressure drop and to study insights of flow phenomenon in a pulse tube using Computational Fluid Dynamics (CFD) techniques.

A CFD based simulation model is developed based on the experimental investigation reported by Cheng and Zhao [2] and analysis is conducted to find out actual flow phenomenon in the pulse tube subjected to oscillating flow and to predict the pressure drop. The results obtained show good agreement with numerical and experimental predictions reported in literature [1] and [2].

CFD modelling of oscillating flow in a circular tube

The entire document should be in Times New Roman or Times font. Type 3 fonts must not be used. Other font types may be used if needed for special purposes. CFD programs like Fluent6.3® allow researchers to solve the volume-averaged conservation equations that govern fluid flow without making any unnecessary simplifying assumptions. This capability becomes especially valuable when seeking complete solutions for flow fields of complex flow definition. For the purposes of this investigation, the entire experimental test section from reference [2] was modeled based on its exact dimensions and used along with the mentioned boundary conditions to obtain a full simulated model of the flow behavior throughout the long pipe test section.

The first step in creating the CFD model was to accurately model the exact geometry of the test section being used to conduct the experiments. Based on the internal dimensions the axisymmetric model shown in Fig. 2, was created using GAMBIT. The model was then meshed to a total node count of about 2100.

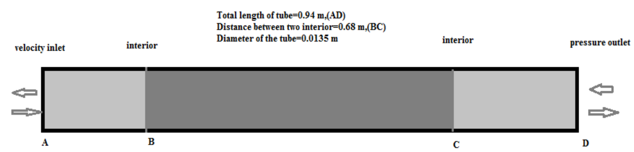

Fig 2. Test Section Model of a circular tube with Meshing

Upon successful meshing of the test section model, the file was exported to like Fluent6.3® where the problem could be fully defined. The model was scaled in terms of SI units and the solver was set to solve a two-dimensional unsteady flow, axisymmetric, laminar flow model. Air was selected as the working fluid and defined as a compressible ideal gas, copper was selected as pipe material, and the convergence criteria for continuity, x-velocity, y-velocity, and energy residuals were then set to 1.0e-5. The model was then completely defined except for the two required boundary conditions. These boundary conditions, the mass input at inlet, and the gauge pressure, P2, at the outlet were set as atmospheric pressure based on the experimental values [1].

A User Defined Function (UDF) was developed for the mass input at inlet of the pipe. The pressure oscillations are set according to equation (2).Continuum-based conservation equations are applied everywhere in the model. This is appropriate since the mean free path of gas molecules is typically much smaller than the characteristic dimension of the test section. The general governing equations used by the FLUENT code assuming a negligible asymmetry caused by gravity, in 2-dimensional cylindrical polar coordinate system are,

Continuity equation:

$$\frac{\partial\rho}{\partial t} + \frac{1}{r}\frac{\partial}{\partial r}(r\rho V_r) + \frac{\partial}{\partial x}(\rho V_x) = 0 \qquad (8)$$

Momentum Equation:

$$\frac{\partial}{\partial t}(\rho v_x) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_x) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_x) = -\frac{\partial p}{\partial x} \qquad (9)$$

The body forces (gravity forces) and any other external forces have been neglected in the above equations.

Energy equation:

$$\frac{\partial}{\partial t}(\rho E)+\nabla\bullet\left(\vec{V}(\rho E+p)\right)=\nabla\bullet\left(K\nabla T+\left(\tau\bullet\vec{V}\right)\right) \qquad (10)$$

RESULT AND DISCUSSION

Seven test runs are conducted at selected values of kinetic Reynolds number (Reω) and dimensionless oscillation amplitude (Ao) as illustrated in Table 1.

Table 1. List of Selected Values of Non Dimensional Parameters for Cfd Analysis

case	A_o	Re_ω
1	16.5	64
2	16.5	208.2
3	26.42	208.2
4	26.42	144.1
5	26.42	324.3
6	16.5	256.1
7	26.42	256.1

A typical transient velocity profiles during a cycle at different locations along the pipe length for Ao=10 and Reω=100 is reproduced in Fig. 3, from reference [2]. The fluid enters the tube as a plug flow from the left during the first half cycle, and a viscous layer caused by wall friction begins at the inlet and grows in thickness downstream. Because of viscous and inertial effects, the velocity profiles at downstream change from a rectangular shape to a parabola-like shape with velocity overshoot occurring near the walls, i.e., the so-called "annular effect" becomes pronounced. At the instant when the cross-sectional mean velocity of fluid is zero, the velocity near the wall is positive while the core flow remains in the negative x-direction due to the inertia effect from the previous half cycle. During the second half cycle, the fluid flow reverses and repeats the behavior similar to those in the first half cycle. It should be added that unlike a unidirectional laminar steady flow, the viscous layers in a reciprocating flow may not coalesce at the fully developed region because velocity near the wall and in the core is out of phase due to the inertial effect (at high Reω).

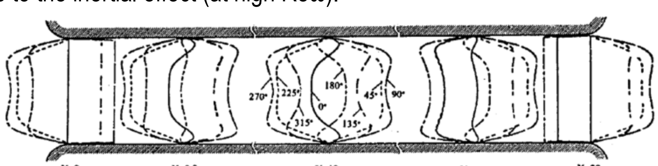

Fig 3. Transient velocity profiles at different locations along the pipe [3] for Ao=10, Reω=100 and L/D=20

The CFD simulation of oscillating flow in a circular tube also predicts the similar results and shows the presence of "annular effect". A small segment of the result is produced here as shown in Fig. 4.The annulus developed varies according to different values of Ao and Reω.

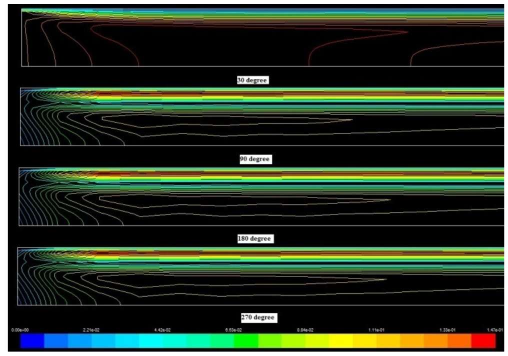

Fig 4. Transient velocity profile predicted by CFD analysis for Test run 1.

Two main reasons may be attributed to the increase of the pressure drops under these conditions. First, the increase of the kinetic Reynolds number leads to more significant "annular effect" and thus the radial velocity gradients adjacent to the pipe wall become steeper; consequently, the friction force increases with the increase of the kinetic Reynolds number. Second, the inertia component in the momentum balance increases with the increase of the kinetic Reynolds number [2].

Fig. 5 shows measurement from the experimental data compared with the CFD simulation. A good agreement is observed with the experimental results.

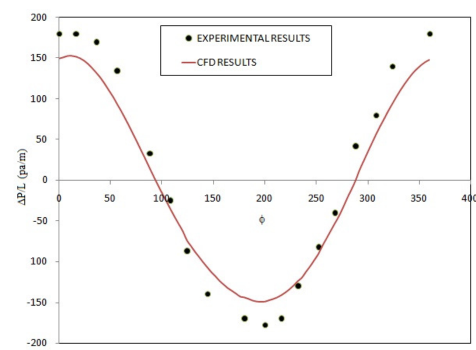

Fig 5. Typical variations of the ensemble-averaged pressure drops for Reω = 324.3 at Ao=26.42

CONCLUSION

In this paper, a CFD analysis of oscillating flow in a circular pipe is presented. Predicted results for pressure drop are in good agreement with experimental results published elsewhere. Presence of "annular effect" is also observed. The detailed study of flow features is found useful for the case of pulse tube.

Nomenclature

A. Notation

- A_0 Dimensionless oscillation amplitude of fluid
- $C_{f,\infty}$ Analytical friction coefficient
- $\bar{C}_{f,\infty}$ Cycle averaged friction coefficient
- D Diameter of the pipe
- K Amplitude of the imposed pressure gradient
- L Distance of the two taps of the pressure transducer
- Re_w Kinetic Reynolds number
- t, Dimensional and dimensionless time

u, U Dimensional and dimensionless axial velocity
u_m Cross-sectional mean velocity
U_{max} Maximum cross-sectional mean velocity
x Axial distance
V_r Radial velocity
V_x Axial velocity
Δp pressure drops

B. Greek Symbols

Φ Phase angle
ρ Density of fluid
T Shearing stress at the wall
ν Kinematic viscosity of fluid
ω Oscillatory frequency

C. Subscripts

∞ Fully-developed flow
exp Measured data
f Friction
r Radial coordinate
x Axial coordinate
m Cross-sectional mean value
max Maximum value

REFERENCES

[1] T. S. Zhao and P. Cheng, 1998."A numerical study of laminar reciprocating flow in a pipe of finite length", *Applied Scientific Research,* Vol. 59 (1), pp. 11-25.
[2] T. S. Zhao and P. Cheng, 1996."The friction coefficient of a fully-developed laminar reciprocating flow in a circular pipe", *Int. J. Heat and Fluid Flow* Vol.17, pp.167-172.