

Investigation of Thermo-Acoustic Excitations in a Rijke Tube Geometry

Devadatta Mukutmoni, Robert Powell, and L.A.Raghu Mutnuri
Exa Corp.

ABSTRACT

Flow generated acoustic sources are of significant import for automotive applications since perception of noise is a critical customer satisfaction issue. High temperature acoustic sources known as thermo-acoustics such as those occurring inside an exhaust system of a vehicle, an important subset of acoustic sources, is the subject of the investigation.

In this article, we study a Rijke tube configuration that consists of a vertical and hollow cylindrical tube open at both ends where sound is generated by buoyancy driven flow as a result of a heated wire gauze placed in the bottom half of the tube. This configuration captures the essence of the thermo-acoustic phenomena and was investigated both numerically and experimentally and good agreement was observed between the two.

CITATION: Mukutmoni, D., Powell, R., and Mutnuri, L., "Investigation of Thermo-Acoustic Excitations in a Rijke Tube Geometry," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(3):2014, doi:10.4271/2014-01-1981.

INTRODUCTION

Flow generated acoustic sources are important design considerations for automotive applications, since noise perception is a critical customer satisfaction issue that must be accounted for in any automotive design process. It is therefore not surprising that considerable amount of work has gone into tracking down and reducing these noise sources via numerical simulations in order to reduce overall noise level for automobiles [1,2,3].

A type of acoustic source occurring primarily in high temperature regions, such as those inside an exhaust system of a vehicle, has not received that level of attention from researchers and is the topic of investigation in this article. These thermally driven acoustic sources or thermo-acoustic excitations are caused by density fluctuations as a result of unsteady heat fluxes from the high temperature surfaces leading to pressure waves and sound generation.

Rijke Tube and Rayleigh Criterion

As a first step to understanding of the thermo-acoustic phenomena, we limit the scope of the study to simulation of a simple geometrical configuration of a vertical, hollow cylindrical tube open at both ends where a strong sound is generated by flow driven by buoyancy as a result of a heated wire gauze placed in the bottom half of the tube.

This configuration, known as the Rijke tube, was first experimentally documented in [4], where it was noted that self-sustained sound waves are produced only when the heated gauze is placed in the lower half of the hollow tube.

An explanation of self-sustained oscillations in a Rijke Tube was provided by Lord Rayleigh in [5], in which he said that the required condition for self-sustained sound generation in thermo-acoustics is for the oscillating heat flux for the heat source to be in phase with the corresponding oscillating pressure field. This condition known as the "Rayleigh Criterion" also applies to any configuration where thermally generated sound occurs and is not limited to the Rijke Tube. Later investigations have shown that the heat flux and pressure oscillations may be at most ninety degrees out of phase for self-excited oscillations to occur [6].

Prior Work on Thermo-Acoustics

Previous investigations on the Rijke Tube and thermo-acoustics have been highlighted in [7] and [8]. It was seen in [7] and [8] that despite extensive applications and interest in thermo-acoustics, not limited to automotive applications alone, very little effort has been reported on numerical simulation of thermo-acoustic applications that models detailed geometry and physics.

In [9], a numerical simulation of Rijke Tube configuration limited to two-dimensional axisymmetric domain and artificial pressure boundary conditions to simulate the phenomena of self-excited thermal oscillations was reported.

In [10], a stability analysis of a horizontal Rijke Tube using a one-dimensional formulation was used to investigate thermo-acoustics parametrically and calculate the bounds for self-excited oscillations.

In [11], an experimental and numerical study of a horizontal Rijke Tube was carried out where the heat rate and the air flow was independently varied and stability zones were documented.

Scope of Current Investigation

A survey of previous work on thermo-acoustics does indicate that a simulation of the classic Rijke Tube Experiment that takes into account all the detailed physics has not been reported. Furthermore, detailed measurements on the classic vertical Rijke tube with open ends are not available. Given the significance of the Rijke tube as a classic problem in thermo-acoustics, the scope of the current study is an investigation of the Rijke tube using flow and a thermal simulation methodology and an experiment to provide some validation to the numerical model.

The simulations were performed using a coupling between the flow solver PowerFLOW version 5.0 and thermal solver PowerTHERM version 11.0 that accounts for conduction and radiation effects.

In the experiments, measurements were performed to record the frequency of the sound generation using a hollow tube with a wire gauze and a propane torch to heat the gauze as shown in Figure 1.



Figure 1. Experimental tube exterior and heating torch

In the next section, the Lattice Boltzmann Methodology (LBM) and turbulence modeling used in the flow solver is outlined. Following that, the test bench configuration and simulation methodology are described in some detail. We conclude with a discussion on simulation results.

LATTICE BOLTZMANN METHODOLOGY

The flow solver is a general purpose CFD code that uses the Boltzmann Equation to solve for the velocity and pressure field [12]. The dependent variable is the distribution function $f(\mathbf{x}, \mathbf{v}, t)$, the number density of molecules at position $\mathbf{x}$ and speed $\mathbf{v}$. The Boltzmann equation can be written in the following form:

$$\frac{\partial f}{\partial t} + \mathbf{v} \cdot \nabla f = C(\mathbf{x}, \mathbf{v}, t) \quad (1)$$

The left hand side represents the convective motion of particles and the right hand side the collision term that accounts for intermolecular interactions. The flow solver uses the BGK collision operator [13] that can be expressed as follows:

$$C(\mathbf{x}, \mathbf{v}, t) = -\frac{f - f_{eq}}{\tau} \quad (2)$$

Where τ is the relaxation time of the fluid and f_{eq} is the Equilibrium Maxwellian Distribution Function. The Boltzmann Equation along with the BGK operator is discretized on a three dimensional cubic lattice using a D3Q19 model as shown in Figure 2.

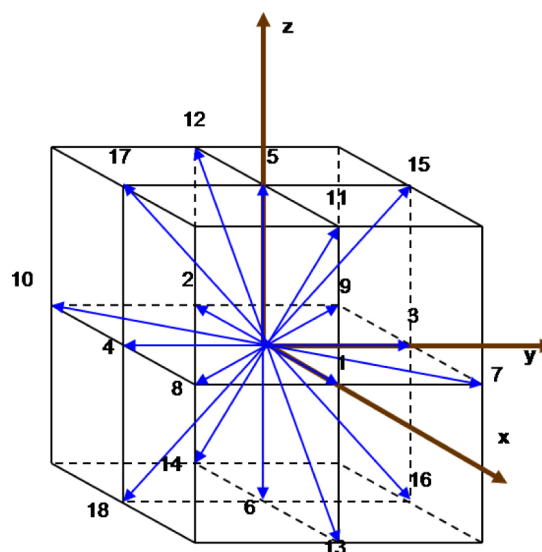


Figure 2. Finite Difference Scheme for Boltzmann Equation

It can be shown rigorously that the Lattice Boltzmann formulation thus outlined is equivalent to solving the time dependent compressible Navier-Stokes Equation. A review of the fundamentals of the Lattice Boltzmann Method can be found in [14].

The turbulence model employed in the flow solver is conceptually similar to a large eddy approach to turbulence modeling; further details on turbulence modeling are available in [15].

Recent improvements in modeling the temperature scalar equation have further enhanced the accuracy of the temperature calculations in the flow solver [16].

TEST BENCH

A physical reproduction of the simulated Rijke tube geometry was constructed to investigate correlation of acoustic frequencies and the onset of the thermally-excited resonance. A steel tube of length 0.5 m and diameter 60 mm, very close to the simulation geometry, was cut from conduit stock at a hardware store. Tube ends were deburred with a file and an outside handle was attached to the middle of the tube with hose clamps. A steel mesh screen was cut from a kitchen strainer, flattened and trimmed to fit snugly across the inner tube diameter. The screen was pushed up from the bottom opening to be located at a fifth of the tube length from this opening.

Initial attempts to match the 3 mm wire mesh used in the simulation failed to reach sufficient temperature to excite resonance, when heated by a hand-held propane torch. The torch could only produce a warm red color in limited areas of the mesh. This may have been caused by excessive conduction losses to the tube wall. But, the propane torch was adequate to produce the desired yellow-orange color temperature in the thinner strainer mesh (as shown in Figure 3) and the acoustic resonance was strongly excited. In order to approximate the turn-on transient of the simulation - zero initial fluid velocity - the interior mesh was heated with the tube axis horizontal. After removing the torch tip, the tube was rotated to the vertical orientation and the natural buoyancy allowed the vertical mean flow to develop.



Figure 3. Heating of mesh screen with propane torch

Pressure recording was taken at a point to the side of the bottom opening and about 500 mm from the tube centerline. For the simulation, the probe location is 500 mm from the tube centerline and at the top of the tube opening. Pressure time history was captured from a laptop computer microphone,

where the amplitude is un-calibrated, but the time sampling is accurately clocked by the onboard sound hardware. Audacity sound editing software[17] was used to capture the pressure time history and then to construct a spectrogram, illustrating the time-frequency transient behavior.

The directivity of the laptop microphone was not investigated, and this could lead to amplitude differences between different frequencies, as does positioning the sensor relative to the tube.

SIMULATION METHODOLOGY

The simulation methodology used in this study involves a combination of the flow and thermal solver. The flow solver calculates the flow and temperature fields in a volumetric flow domain and the thermal solver calculates the temperature in the solid surfaces by taking into account radiation from the surfaces, conduction within the solid surfaces and convection at the surface.

This process is enabled by a coupling process methodology shown in Figure 4.

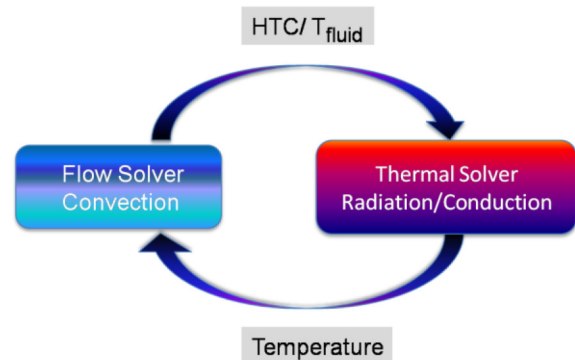


Figure 4. Overview of Coupled Simulation Methodology

The flow solver calculates the near wall temperatures T_f and heat transfer coefficients H and exports them to the thermal solver. The thermal solver calculates the wall temperatures of the solid surfaces T_w by taking into account radiation, conduction in solid surfaces and convection from the fluid and exports them back to the flow solver. This "coupled process" continues until convergence is achieved for the solid surface temperatures. The coupling process uses two different geometrical models one for the flow solver and the other for the thermal solver that have the same underlying geometry but different element types and size. The transfer of T_w , H and T_f between the flow and thermal solver is achieved by interpolation between different geometries.

The thermal solver employs a finite-volume formulation that exactly conserves energy. The surface to surface radiation is calculated by solving a grey body radiation enclosure problem and calculating the radiation view factors by a ray-tracing algorithm. The conduction is calculated in the context of a finite volume formulation. More details of the methodology can be found in [18].