Title: CFD Analysis of Laminar Oscillating Flows

Author(s): Charles W. Booten, ESA-DE
Snezana Konecni, NMI SNS-3
Barton L. Smith MST-10
Richard A. Martin ESA-DE

Submitted to: 2001 ASME Fluids Engineering Division Summer Meeting (FEDSM01)
New Orleans, Louisiana
May 29-June 1, 2001
CFD Analysis of Laminar Oscillating Flows
Charles W. Booten, Snezana Konecni*, Barton L. Smith**, and Richard A. Martin
Los Alamos National Laboratory
P.O. Box 1663
Los Alamos, NM 87544

ABSTRACT
This paper describes a numerical simulations of oscillating flow in a constricted duct and compares the results with experimental and theoretical data. The numerical simulations were performed using the computational fluid dynamics (CFD) code CFX4.2. The numerical model simulates an experimental oscillating flow facility that was designed to test the properties and characteristics of oscillating flow in tapered ducts, also known as jet pumps. Jet pumps are useful devices in thermoacoustic machinery because they produce a secondary pressure that can counteract an unwanted effect called streaming, and significantly enhance engine efficiency. The simulations revealed that CFX could accurately model velocity, shear stress and pressure variations in laminar oscillating flow. The numerical results were compared to experimental data and theoretical predictions with varying success. The least accurate numerical results were obtained when laminar flow approached transition to turbulent flow.

INTRODUCTION
Thermoacoustic engines and refrigerators are promising devices, which have been under development at Los Alamos for over 20 years (Swift, 1988) and that have the potential of running with no moving parts and no close tolerances. They can be designed to operate on a Stirling thermodynamic cycle. Consequently, these engines provide increased potential for reliability and efficiency over conventional engines. (Swift, 1995)

One serious obstacle to achieving high efficiency in thermoacoustic engines is caused by the streaming flow that is created from the thermal expansion and contraction of the working fluid as it undergoes a Stirling cycle. This streaming motion causes heat to be transferred by convection from the hot to the cold heat exchangers without doing any work, thereby reducing the efficiency of the engine. Presently this "ratcheting" effect is countered by employing a jet pump. The jet pump is a tapered section of tube that extracts a small amount of the oscillatory power and creates a secondary pressure in the flow direction opposite to the direction generated by the normal working cycle. Optimization of jet pumps is therefore an important step in making thermoacoustic devices competitive with conventional technologies (Backhaus, 2000).

One of the major difficulties in evaluating and understanding jet pumps is that oscillatory flow is not understood as well as steady flow. Measurements are often harder to make because of periodic, low velocity, and fluctuations in properties during a cycle. It is therefore desirable to be able to model jet pumps numerically to help understand and evaluate separation and transition to turbulent flow as well as other variables that are difficult to measure experimentally. Therefore, CFD modeling was employed to obtain data to compare with experiments performed at Los Alamos National Laboratory to validate the software capability to model oscillating flow in jet pumps.

NOMENCLATURE

K Constant in Schlichting’s equation, eqn. 4
r Radial position
R Hydraulic radius
Re_{crit,osc} Critical Reynolds Number in oscillating flow
ρ Density
τw Shear stress at the wall
u Velocity
u* Friction Velocity
y Distance away from wall
y' Dimensionless distance from wall
μ Dynamic viscosity
ν Kinematic viscosity
ω Frequency, Hz
ω Dimensionless frequency

THEORETICAL ANALYSIS
Oscillating flow in ducts is very different from steady internal flow. One major difference occurs during transition
from laminar to turbulent flow. In steady flow the critical Reynolds number can range from 2000-4000. Generally 2300 is regarded as a "rule-of-thumb" for the critical Reynolds number. However, in oscillating flow the critical Reynolds number is dependent on the frequency of oscillation in the flow and has been approximated by Ohmi et. al. (1982).

\[ \text{Re}_{\text{crit, osc}} = 882 \sqrt{\omega} \quad \sqrt{\omega} > 7 \quad (1) \]

\[ \text{Re}_{\text{crit, osc}} = 2450 \quad \sqrt{\omega} < 1 \quad (2) \]

The dimensionless frequency, \( \omega' \), is defined as

\[ \omega' = \frac{R^2 \omega}{v} \quad (3) \]

Another important characteristic of oscillating flow that is very different from steady flow is the velocity profile. In steady flow the velocity profile is parabolic for laminar flow and can be approximated by the \((1/7)^{th}\) power law profile in turbulent flow. Adverse pressure gradients in oscillating flow cause features that are not present in steady flow, regardless of the Re number. The velocity profile in oscillatory flow is given in Schlichting (1979) by:

\[ u(r,t) = \frac{K}{\omega} \left( R^2 - r^2 \right) \cos(\omega) \quad (4) \]

\[ u(r,t) = \frac{K}{\omega} \left[ \sin(\omega) - \frac{R}{r} \exp \left( -\frac{\omega}{2v} (R-r) \right) \sin \left( \omega - \frac{\omega}{2v} (R-r) \right) \right] \quad (5) \]

Equation (4) represents the velocity profile for very slow oscillations and Equation (5) applies when the oscillations are large. Equations (4) and (5) were used to determine a velocity gradient near the wall for laminar cases so that a theoretical wall shear stress could be calculated.

The computer code that was used here employs a hybrid upwind differencing scheme to solve the momentum and mass equations. This differencing scheme is first order accurate except in regions of low flow where central differencing is used for second order accuracy (AEA Technologies, 1997). Solving the governing equations yields the velocity profile in the flow as well as the shear stress at the wall and the pressure distribution in the flow. For laminar flow of a Newtonian fluid, CFX4 calculates shear stress by using

\[ \tau = \mu \frac{du}{dy} \quad (6) \]

\[ \omega \]

where \( du/\text{dy} \) is obtained from the momentum equation, and \( \mu \) is the dynamic viscosity of the fluid.

### APPARATUS AND EXPERIMENTS

A two-dimensional flow apparatus was constructed at Los Alamos National Laboratory.

![Figure 1. Experimental Apparatus and Test Section.](image)

Figure 1 shows the configuration and dimensions of the apparatus. The cross sections of the inlet and outlet cavities are square with a width of 6.125" per side. The center narrow section, referred to as "the slot," was designed to be of variable height and to have the option of being tapered axially. Glass covers were fitted over the front and rear of the slot to allow for flow visualization. For all the laminar cases the slot was 0.1" (vertically in Fig. 1) by 6.125" to ensure an oscillating flow Reynolds number (Re_{oc}) in the laminar regime. For these cases no taper was used in the slot.

Table 1 summarizes the oscillatory flow cases considered here. All cases had a large \( \omega \) and so Equation (1) was used to calculate the critical Reynolds number. The Reynolds numbers were calculated based on the hydraulic diameter and the average velocity amplitude. The two higher frequency cases imply that a laminar model is more accurate. The lower frequency case (10 Hz) suggests a turbulent flow model.

| Table 1: Critical Reynolds Numbers in Oscillatory Flow |
|--------------------------------------|---|---|---|---|
| Freq. (Hz) | \( \omega \) cavity | \( \omega \) test | Re_{oc, cavity} & slot | Re_{oc, crit, cavity} | Re_{oc, crit, slot} |
|----------|------|------|----------------|-----------------|-----------------|-----------------|
| 10       | 26x10^3 | 6.93 | 2024 | 142x10^3 | 2321 |
| 30       | 78x10^3 | 20.8 | 894 | 246x10^3 | 4021 |
| 100      | 260x10^3 | 69.3 | 868 | 450x10^3 | 7341 |

Copyright © #### by ASME
All measurements were taken in the span-wise center of the apparatus (in the direction normal to the page) to eliminate any edge effects. Loud speakers located at the base of the inlet cavity created the oscillating flow. The speakers ran between 10 and 100 Hz for the laminar tests. The outlet cavity was exhausted to atmospheric pressure. The centerline velocity was measured using a hot-wire anemometer mounted on the outlet cavity and directed into the slot. Centerline velocity measurements were taken 0.5 inches into the slot from the outlet cavity. Shear stress sensors were located on the sides of the slot at the same axial position as the hot-wire anemometer measurements and pressure transducers were placed 1.0 inch (2.54 cm) along the axial direction into the cavities from the slot on both sides of the inlet and outlet cavities.

NUMERICAL METHOD AND MODEL

The flow field in the slot studied here was simulated using CFX4.2, a commercially available CFD package. Such CFD models provide a simultaneous numerical solution of continuity, Navier-Stokes, and energy equations for a flow-field geometry with specified boundary conditions. CFX4.2 uses a finite volume (finite difference) scheme based on the SIMPLEC mathematical algorithm discussed by Van Dormaal and Raithby (1984). A predecessor of this algorithm, SIMPLE is given by Patankar (1980).

The meshing in this version of CFX is structured, which is advantageous for modeling a two-dimensional geometry as in Fig. 1. The symmetry of the experiment was helpful in reducing the computational complexity of this problem. The centerline was used as a symmetry plane so that only the top half of the geometry in Fig. 1 was modeled. The two-dimensional nature of this problem meant that only one cell thickness needed to be modeled, in effect neglecting any gradient in properties into or out of the page in Fig. 1. The large volume in the cavities was of little concern because of the lack of gradients in properties far from the slot. Therefore, most of the actual physical space of the cavities was not modeled. The limiting factor in how little of the cavities could be modeled was how far the vortices traveled from the slot before breaking up. The characteristic length that the vortex traveled was proportional to the length of the slug of fluid that passed through the slot during one half-cycle. It was determined by trial and error that allowing for 6 inches from the end of the slot was sufficient distance for the vortices not to interfere with the boundary conditions at either the inlet or the outlet.

The velocities produced in the test chamber were on the order of 10 m/s, so compressibility was neglected, as were thermal variations. The details of the grid structure are shown in Fig. 2. Figure 2 shows the mesh for the region near the outlet cavity for one half of the test apparatus that was modeled using CFX4.2. The radius of curvature in this figure is 0.25 inch and it shows the wall between the slot and the outlet cavity. This figure gives an idea as to how the mesh varied at the sudden expansion into the cavities. The height of the slot in Fig. 2 is 0.05 inch. The aspect ratio of the cells in the slot is 10:1.

This ratio was chosen because much larger gradients perpendicular to the flow than along the flow direction allowed a smaller overall number of cells to be used. When developing the mesh, care was taken to ensure that it was not severely distorted; the mesh regularity is apparent in Fig. 2.

![Figure 2. Mesh along slot and beginning of outlet cavity.](attachment:image)

BOUNDARY CONDITIONS

The inlet boundary condition was specified as uniform cross-sectional flow with sinusoidal amplitude. The correct amplitude was determined by using experimental data for the peak centerline velocity. An estimate was made, using mass conservation, based on the incompressible, isothermal model of the required amplitude to achieve the desired peak centerline velocity. The resulting velocity profile produced in the pipe was integrated and averaged, and the ratio of the experimental to numerical peak velocities was used to estimate a new amplitude that achieved the desired peak velocity. The outlet was modeled as a pressure boundary. The pressure at the outlet was equal to atmospheric (11.4 psia at Los Alamos, NM) at all times. This was very important in determining how much of the cavity to model because a coherent vortex (which contains pressure gradients) crossing the constant pressure boundary could disrupt the code and produce non-physical results or divergence of the (convergence) residuals. All other boundary conditions were left as the default no-slip, adiabatic boundaries.

TIME-STEP INDEPENDENCE, STEADY STATE STUDY

It was necessary to do several extra simulations to ensure that the duration, the number of time-steps, and decaying transient effects did not affect the numerical results. The 100 Hz laminar case was used to establish the periodic steady state. Four cycles were run and the velocity profiles in the slot were compared. Steady state was achieved after one cycle.
The appropriate time-step duration was determined by performing several runs of 2 cycles each with the number of time steps varying from 40 to 160 per cycle. The velocity profiles were then compared in the outlet cavity region, because the presence of vortices makes the behavior more chaotic and hence harder to duplicate. This revealed that 80 time steps per cycle are enough to capture all relevant transient behavior.

RESULTS

Laminar simulations were performed with inlet velocity frequencies of 10, 30 and 100 Hz and compared to experimental and theoretical results. Comparisons were performed using the 100 Hz case to test the time-step independence and steady state behavior of the simulations. Figure 3 below shows the velocity profile in the outlet cavity at various times during the cycle for the 80 and 160 time-steps per cycle simulations. Even in this region with the vortices propagating through, the agreement between the 80 and 160 time-step simulations is within 0.1% at all positions and time-steps. Therefore 80 simulations per time-step were deemed acceptable to save on computational time.

The comparisons with experimental data and theoretical predictions consisted of comparing the centerline velocity at one half inch in the slot to the top cavity and the shear stress at the wall at the same distance. The pressure 1 inch into the top and bottom cavities from the slot was also compared for one cycle. Figures 4 through 6 show these comparisons for the 10 Hz laminar case.

Disagreement between the experimental velocity curve and the numerical and theoretical curves on the down-stroke of the flow is observed in Fig. 4. This is an experimental artifact of the hot-wire anemometer probe interfering with the flow. When the flow reverses and goes toward the inlet cavity the measurements are taken in the wake of the probe and are therefore not valuable. The experimental values agree better with the theoretical values on the up-stroke than the numerical values. The maximum difference of 16% between the numerical values and the theoretical/experimental values is at the point of maximum velocity. The agreement is much better between the numerical and theoretical results on the down-stroke, where the difference at the minimum velocity is less than 0.1%.

Figure 5 shows a comparison of velocity gradient at the wall. The quantity du/dy is directly proportional to the shear stress. The experimental data is only valid for the higher values because at lower velocities (where the lower velocity gradients are measured) the shear stress sensors are not accurate. The sensors match well with the numerical results on the down-stroke; however, the numerical velocity gradient does not match well with the experimental results on the up-stroke.

The 100 Hz simulations in Figs. 6 through 8 produced the best agreement between results. The centerline velocity profiles in Fig. 6 matched almost perfectly for all three methods, aside from the down-stroke probe interference.

The velocity gradient comparisons in Fig. 7 agree within 15% for the experimental and numerical results using only one shear stress sensor, which is within the experimental uncertainty established from Fig. 9. The theoretical and numerical gradients match within 5% at their peak values.

The pressure fluctuation comparison is shown in Fig. 8. The agreement is surprisingly good throughout the entire cycle. This is especially interesting because of the poor agreement between experimental and numerical results in the other two cases.

Figure 3. Velocity Profile Comparison for 80 and 160 Time Steps Per Cycle, 100 Hz.

Figure 4. Peak Velocity at Centerline .5" from End of Slot, 10 Hz.
SUMMARY

The CFD code used to model the experiments presented here, CFX4.2, has been shown to have the ability to accurately simulate some oscillatory flow characteristics in ducts with laminar flow. The best results have been for the wall shear stress and for velocity measurements. Higher frequencies (up to at least 100 Hz) have been shown to give results for these parameters that match theoretical and experimental results better than at lower frequencies (down to at least 10 Hz) under the modeling conditions used. The pressure fluctuations were not accurately modeled except for the 100 Hz case, showing that the code is capable of modeling the pressure, but also demonstrating that more investigation is needed to have any confidence in future numerical pressure calculations.

It was also demonstrated that CFX4.2 could be useful in providing information that is not attainable experimentally. For example, the velocity profiles at various points in the slot at arbitrary times during the cycle are predicted by the code.

Future work should begin by investigating why the results are generally better for higher frequencies. It is possible that...
the lower frequencies, which achieve higher velocities, may actually include turbulent fluctuations that are difficult to model with CFX4.2 if the flow is mostly laminar. The discrepancy between the peak gauge pressures in the cavities also needs more study. The pressure boundary condition on the outlet cavity dictates that all of the fluid that passes through that boundary has ambient atmospheric pressure. If the vortices generated by the flow exiting from the slot are not fully dissipated by the time they cross that boundary the code may artificially adjust the pressure to make sure that the pressure is uniform across the exit boundary, thus creating an inaccurate pressure field calculation. One method to solve this problem would be to extend the cavities. This requires more memory and simulation time than was available for this study.

REFERENCES


