Analyzing the Effect of Diverging Geometry in Shock Tube Configurations Through Computational Simulation.

Steven C. Lewis

Georgia Institute of Technology

Shock Tube and Advanced Mixing Laboratory

ENGAGES

December 4th, 2023

Abstract

The effects on radial shockwaves on fluid interfaces has been studied with explosives, but not in a shock tube configuration that uses test sections high-pressure air. This research explores that concept in the form of fluid dynamics simulation that models an unconventional shock tube assembly. The use of high-pressure experimental apparatuses rather than explosive-oriented approaches to modeling the Blast Driven Instability can prove beneficial to operator safety, ease of experiment conduction, and post-experiment cleanup. Where industry-standard shock tube construction implores a cylindrical design, designs modeled resemble a trumpet-like test domain in order to mimic radial expansion of a blast wave in the form of a shock wave. Results concluded from these simulations include an exponential drop off of shock wave velocity as it propagates throughout the driven section. Results also show an accentuated low-pressure zone beginning from the dissection point and extending much further into the tube.

Table of Contents

Introduction

In today's technological world, there is a growing need for cleaner and more efficient energy sources. A promising contender for these much more efficient fuel sources is fusion technology, which uses the fusion of various light atoms (usually isotopes of hydrogen like deuterium) to generate significant amounts of energy with minimal waste and environmental impact.

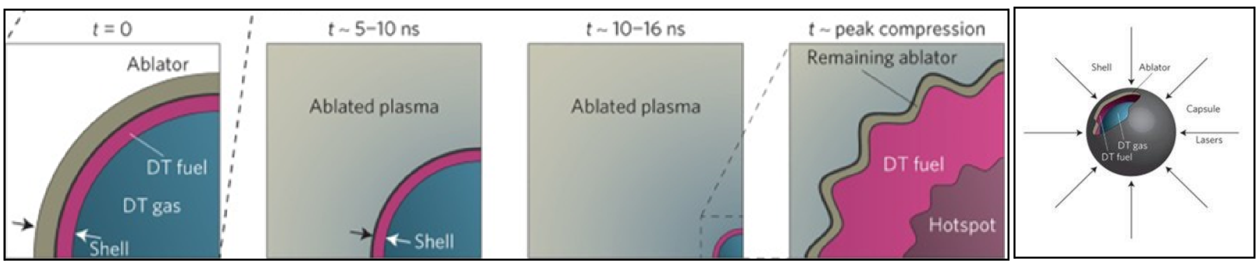

Fig.1: Diagram of fuel pellet and stages of ablation once superheated

One of the drawbacks to this technology is the fact that it is still in its experimental stages, with unoptimized properties such as inefficient mixing of the fuel source. In a fusion reactor, a small plastic pellet filled with a hydrogen isotope is superheated to temperatures as hot as the surface of the sun with multiple high-energy lasers. This rapid and extreme heating causes the plastic shell of the pellet to liquify, sublimate, and mix with the hydrogen fuel inside, causing inefficient fuel usage and suboptimal energy output [Fig.1]. Research is currently being done to minimize this mixture phenomenon through the use of computational fluid dynamics (CFD) and supersonic shock tube experiments.

A conventional shock tube has three major spaces: A high-pressure (driver) section that takes up about one-fourth of the entire assembly, a low-pressure (driven) section that occupies the rest of the space in the assembly, and a diaphragm that separates them, usually made of a rigid metal such as steel. [Fig.2]

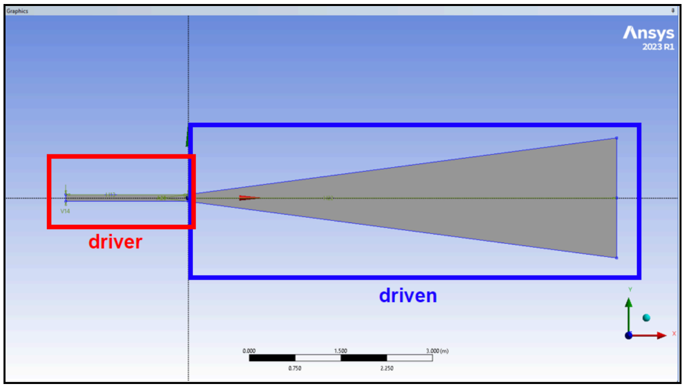

Fig.2: Driver & Driven sections in geometry

When running an experiment, the diaphragm is broken or removed, causing a supersonic shockwave to propagate through the length of the assembly, from the high-pressure driver section to the low-pressure driven section. At the far end of the shock tube resides an interface between two gasses; one light and one heavy. These gasses sit atop one another, and when struck with the supersonic shockwave, cause turbulent mixing between them [S. Petter].

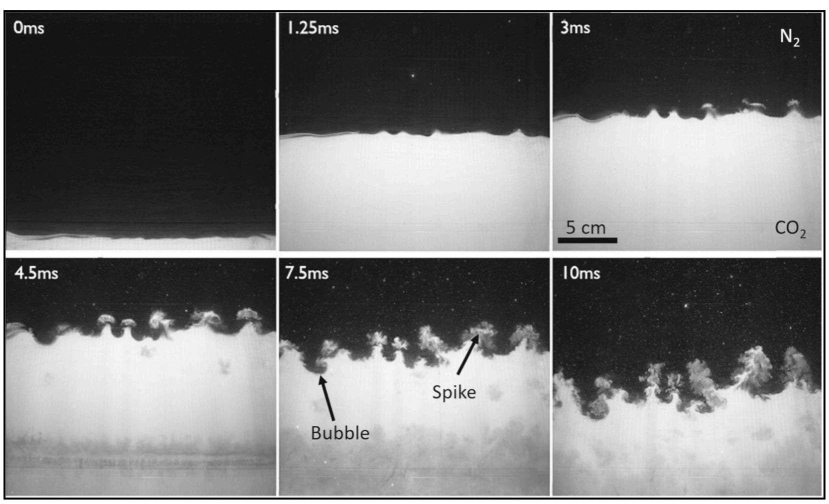

Fig. 3: Shocked Fluid Interface

The mixing of these two gasses mimic what happens when the sublimed plastic shell and gaseous hydrogen of a fuel pellet mix together [Fig. 3]. With these experiments, phenomena can be observed that can be exploited in order to mitigate (or perhaps accentuate) the mixing between two substances.

In conventional shock tube assemblies, the geometry is linear and has an unchanging cross-section down the length of the tube [Fig.4].

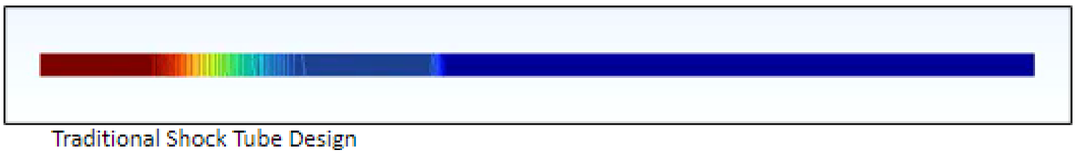

Fig.4: Traditional Shock Tube Design

What if the area of the perpendicular cross-section increases down the length of the shock tube? A geometry like this would better mimic naturally occurring shockwaves (which for the most part expand radially), such as large explosions, supernovae, and even the fuel pellet in fusion reactors. Research on radially expanding shockwaves' effects on gaseous interfaces has already been done, but said experiments use explosives to generate a shockwave rather than high-pressure air that a shock tube would. Advantages to using high-pressure air is the elimination of shrapnel that could interfere with results, safer conduction of the experiment, and more consistent outcomes. The ultimate goal of this experimentation is to prove the validity of using shock tubes to simulate radial blast waves through the use of high-fidelity simulations before actual real-world testing.

Methodology

Simulation is conducted in a computational fluid dynamics software known as ANSYS Fluent. Fluent is a CFD software package used to simulate fluids' flow, heat transfer, and fluid interactions. It can be used to analyze a wide range of problems, including the flow of gasses and liquids [1]. Fluent uses the finite volume method to solve the governing equations of fluid flow and heat transfer. This is when the domain is into a series of small and interconnected cells. The values of the variables (such as velocity and temperature) are then calculated at the center of each cell [Fig.5].

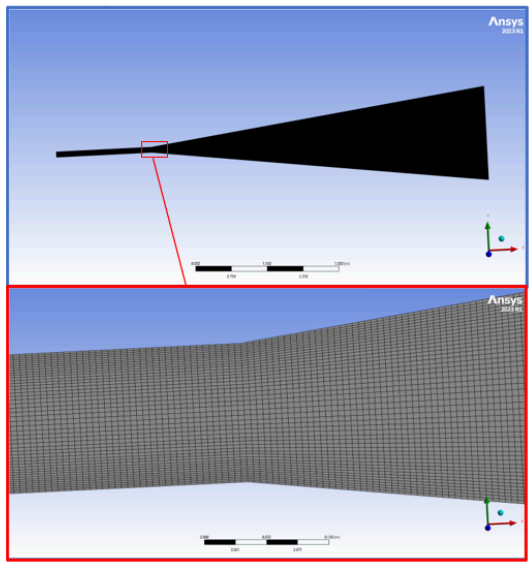

Fig.5: Enhanced Image of Mesh Showing Individual Cells

The domains being tested include a geometry of a conventional shock tube, with a uniform cross section down the length of the tube. This translates in 2D to an elongated rectangle-like shape. Also like a traditional shock tube, this geometry has two sections that are reserved for high-pressure and low-pressure air. The Driver section is defined as being 2 meters long, and 0.2 meters in diameter. The Driven section is 7 meters long and 0.2 meters in diameter, for a total assembly length of 9 meters. The second domain being tested is nearly identical, but with an increasing cross-sectional diameter down the length of the driven section, resulting in a trumpet-like geometry. The back wall diameter is not explicitly defined, although the angle in which the top and bottom walls of the geometry diverge from control diameter is. This angle is 7.5 degrees from each wall, resulting in a 15-degree divergence factor.

Constant Values include the driver pressure (25 atm), maximum iterations per timestep (1000), and the total simulation time steps (2500 timesteps). A method known as Adaptive Time Stepping is used in this experimentation. This is when individual timesteps will lengthen or shorten based on active solution conditions. The advantage to using this method as opposed to a static timestep size is that the solution will not over or under-define certain areas in the simulation, which saves on computational time and increases solution detail. A method known as Adaptive Meshing is also used in this experimentation. This is when cells in the mesh will multiply and shrink in certain areas of the solution in reaction to the local intensity of phenomena in a region. Vice versa also occurs, where the mesh will coarse in areas of low importance. The advantage to using this method as opposed to a static mesh is that the solution will not over or under-define certain areas in the simulation, which saves on computational time and increases solution detail. Simulations run in double precision mode. This means that the solvers used in the simulations can handle much larger numbers (64-bit integers) and in turn, have much less error probability. A density-based solver is also used. Density-based solvers in CFD, as opposed to pressure-based solvers, excel in high-speed supersonic flows, which is why it is being used in this experiment. The simulation also uses “Implicit Formulation”. Implicit formulation uses many different equations per step and provides convergence much more reliably as opposed to an explicit formulation. The spatial discretization gradient used is the “Point Linear Interpolation” method. This method splits cells into smaller regions, calculating values for the cell based on an average of said regions. This method provides better accuracy in skewed meshes (such as the one used) and reduces error . A flow scheme determines how many points in a cell are evaluated for calculations. In this experiment, a second order flow scheme is used, which evaluates values based on two points, as opposed to a first order scheme which uses one point. This can drastically increase simulation accuracy.

Results/Discussion

In a conventional shock tube design, simulation shows that there is minimal drop off in shockwave velocity as the shock wave propagates down the length of the tube. There is also no evidence of a "suction" effect occurring at the diaphragm release point due to a low pressure section created just past the interface between driver and driven sections. The expansion fan of a traditional shock tube does not seem to adapt a triangular shape, or even pronounce itself at all. In the 15- degree diverging shock tube design, simulation shows that there is a significant drop off in shockwave velocity as the shock wave propagates down the length of the tube. This drop off in velocity seems to coincide with the gradually expanding surface area of the driven section. The shockwave itself also seems to get less defined and fall apart as its speed decreases, eventually losing its shape. The expansion fan that is created just in front of the diaphragm upon release is much more noticeable in a diverging tube rather than in a traditional linear tube. This fan gradually grows over time as well. Pressure contours show that just in front of the expansion fan, there is a reverse flow of fluid, indicative of the relatively large low-pressure zone just in front of the diaphragm release point [Fig.6].

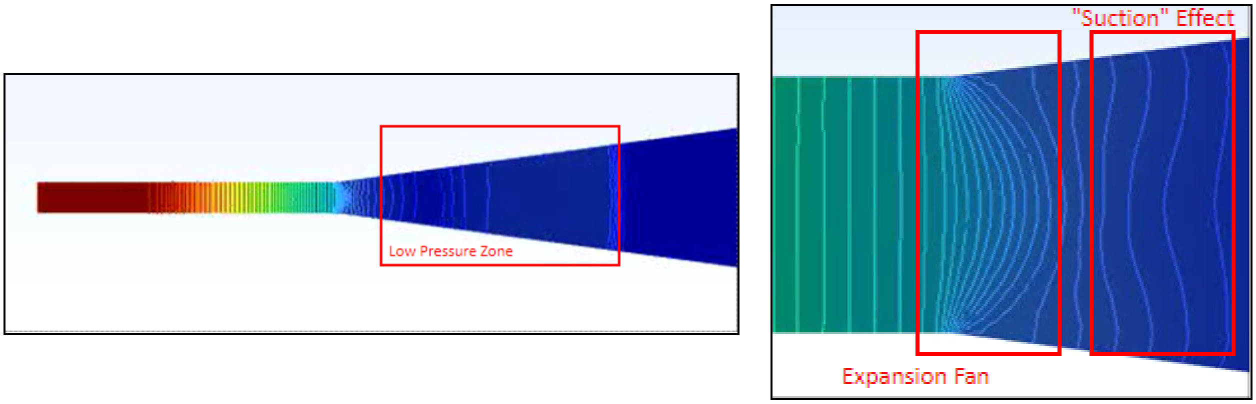

Fig.6: Low-pressure zone, expansion fan, and “suction” effect

These simulations took ~3 hours to run each. This computational time was significantly cut down from a previous average simulation total run time of ~36 hours. This is mostly due to the Adaptive Time Stepping and Adaptive meshing methods that provided the hardware to use only as much computational power as absolutely necessary. Error residual plots show an average error of around 1e-06 percent, which is plenty accurate for applications such as this [Fig.7].

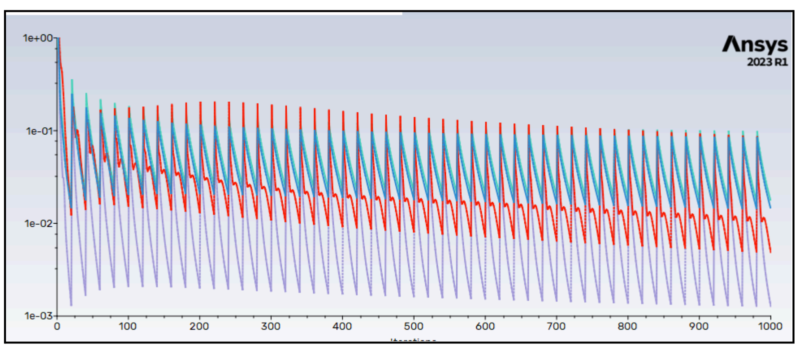

Fig.7: Example Residual Plot (no direct relation to experiment)

Residual upticks occurred at diaphragm release. The average amount of iterations used to converge within each timestep was ~ 150 out of 1000.

An initial driver pressure of 2500 kPa in the 15-degree diverging shock configuration produced a mach number of ~1.9 or an initial wave speed at ~650 m/s. This velocity was quickly reduced to ~400 m/s within the first meter of the driven portion of the tube.

Shock velocity was observed to exponentially drop as it traveled through the domain. The traditional case produced similar statistics, although shock velocity did not drop as violently; only to ~500 m/s in the first meter of the tube with minimal signs of slowing.

Conclusions

Findings conclude that in a partially radially expanding environment such as the one constructed, shockwave velocity decreases rapidly with more cross-sectional area to cover. The low-pressure zone created adjacent to the diaphragm release is significantly more pronounced in an expanding driver than in a traditional shock tube geometry. This pronounced low-pressure area causes an accentuated suction-like effect towards the back of the tube, also playing a role in the slowed velocity of a shockwave propagation. With a higher angle of divergence, it can be inferred that the expansion fan (the triangle-like protrusion of high pressure emanating from the diaphragm release) will also grow and be more pronounced. What this means for accurately recreating a blast wave in such a fashion is that extremely high pressure is needed to accurately recreate a chemical explosion. As initial nuclear detonation waves have typical velocities of around 7 km/s (7000 m/s), and the fact that this experiment concluded that at a pressure of 2500 kPa and initial shock velocity is ~660 m/s, a closed container made of any conventional metal alloy would not be able to withstand the pressures necessary to provide a shock of nuclear velocity. But for shockwaves with reasonable mach numbers, it is completely possible to simulate chemical explosions of smaller scale using a radial shock tube. Future studies include reading temperature data as well as testing higher divergence factor geometries.

Acknowledgements

Thank you Project ENGAGES, Georgia Institute of Technology, Shock Tube and Advanced Mixing Laboratory, Carbon Neutral Energy Solutions Laboratory, Samuel Petter, Stephen Johnston, Quinton Dzurny.

References

[1] Musci, Benjamin, et al. "Supernova hydrodynamics: A lab-scale study of the blast-driven instability using high-speed diagnostics." The Astrophysical Journal 896.2 (2020): 92.

[2] Jeong-Yeol Choi, In-Seuck Jeung, and Youngbin Yoon. “Computational fluid dynamics algorithms for unsteady shock-induced combustion, part 2: comparison”. In: AIAA journal 38.7 (2000), pp. 1188–1195.

[3] Juan Cheng and Chi-Wang Shu. “High order schemes for CFD: A review”. In: Chinese Journal of Computational Physics 26.5 (2009), p. 633.

[4] Luca Mangani, Wolfgang Sanz, and Marwan Darwish. “Comparing the performance and accuracy of a pressure-based and a density- based coupled solver”. In: 16th International Symposium on Trans- port Phenomena and Dynamics of Rotating Machinery. Honolulu, United States, Apr. 2016. url: https://hal.science/hal-01894391.

[5] Lewis, Elmer E. Fundamentals of nuclear reactor physics. Elsevier, 2008.

[6] Hamerly, Ryan. "Inertial Confinement Fusion." Inertial Confinement Fusion (2010).

[7] https://www.mr-cfd.com/introduction-to-ansys-fluent-a-beginners-guide/x