Application of Computational Fluid Dynamics in Discontinuous Unsteady Flow with Large Amplitude Changes; the Shock Tube Problem

KHALED ALHUSSAN
Research Assistant Professor, Space Research Institute, King Abdulaziz City for Science and Technology, P.O. Box 6086 Riyadh 11442, KSA

Abstract:- Unsteady flow phenomena occur in all types of engineering applications. The present paper is concerned with unsteady flows in which large amplitude changes in the properties of the gas occur. Two types of discontinuities may occur in unsteady flows: contact surfaces and shock waves. Shock waves and contact surfaces are generally treated as mathematical discontinuities that separate regions of continuous flow.

A computational model that illustrates the physics of flow in shock tube was developed. The flow is non-steady and compressible. In this situation, one should expect oblique shock waves, expansion fans, shock wave interactions, and slip surface generation. The results including the contour plot of static pressure, static temperature and velocity plot from the numerical simulations show an agreement with the analytical solution.

Key- Words: Non-steady Flow, Shock Waves, Expansion Fans, Contact Surface, CFD.

Nomenclature

Symbol	Meaning
P	Static pressure
T	Static temperature
U	Velocity

1 Introduction

The work to be presented herein is a Computational Fluid Dynamics investigation of the complex fluid mechanisms that occur inside a non-steady flow region, the shock tube [1].

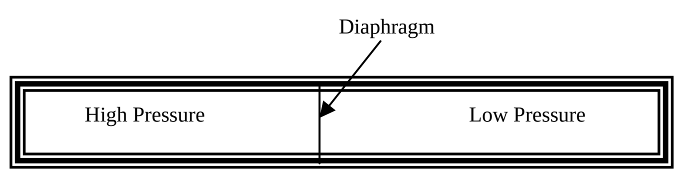
The shock tube is an important practical application of unsteady flow. A shock tube is a long constant-area duct divided into two sections separated by a diaphragm. The section to the left of the diaphragm is filled with a high-pressure gas, called the driver gas. The section to the right of the diaphragm is filled with a low- pressure gas, called the driven gas. The driver gas and the driven gas may be the same gas or different gases [2-3].

The diaphragm separating the different sections of the shock tube may be ruptured in two ways. The pressure difference across the diaphragm may be slowly increased until the material breaks spontaneously or the pressure may be kept at a little below this pressure and the rupture induced by mechanical or electrical means.

In the driving the properties of shock waves, one should assume that the change in the physical state of the fluid occurred so quickly that the front could be considered as mathematical discontinuity. Although this assumption is valid for treating the regions on each side of the front, it remains an approximation [4-7].

A shock tube is a device in which a normal shock waves generated by the rupture of a diaphragm separating a gas at high pressure from another gas at low pressure, as shown in Figure 1. A shock tube is a useful tool for investigating not only shock phenomena but also the behavior of materials and objects when they are subjected to the extreme conditions of pressure and temperature which exist in the gas flow behind a normal shock wave [8-14].

Fig.1 Arrangement of shock tube.



2 Computational Fluid Dynamics Analysis

The flow in the shock tube is so complex that there exist direct fluid-fluid interactions, oblique shock waves, expansion fans, slip surfaces, and shock wave interactions and reflections. The flow is non-steady, viscous, and compressible.

The governing equations are a set of coupled nonlinear, partial differential equations. In order to formulate or approximate a valid solution for these equations they must be solved using computational fluid dynamics techniques. To solve the equations numerically they must be discretized. That is, the continuous control volume equations must be applied to each discrete control volume that is formed by the computational grid. The integral equations are replaced with a set of linear algebraic equations solved at a discrete set of points.

CFX is used in the current research to model the flow in the shock tube. CFD code is an integrated software system capable of solving diverse and complex multidimensional fluid flow problems. The fluid flow solver provides solutions for incompressible or compressible, steady-state or transient, laminar or turbulent single-phase fluid flow in complex geometries. The code uses block-structured, non-orthogonal grids with grid embedding and grid attaching to discretize the domain. The code system has additional capabilities that can predict subsonic, transonic and supersonic compressible flows, including temperature solutions in solid regions of the domain for laminar or turbulent flow.

CFX is a finite volume method, but is based on a finite element approach of representing the geometry. Thus, the method used here possesses much of the geometric flexibility of finite element methods as well as the important conservation properties of the finite volume method.

It should be possible to model the interaction of the shock waves and expansion fans in the shock tube using the CFD analysis.

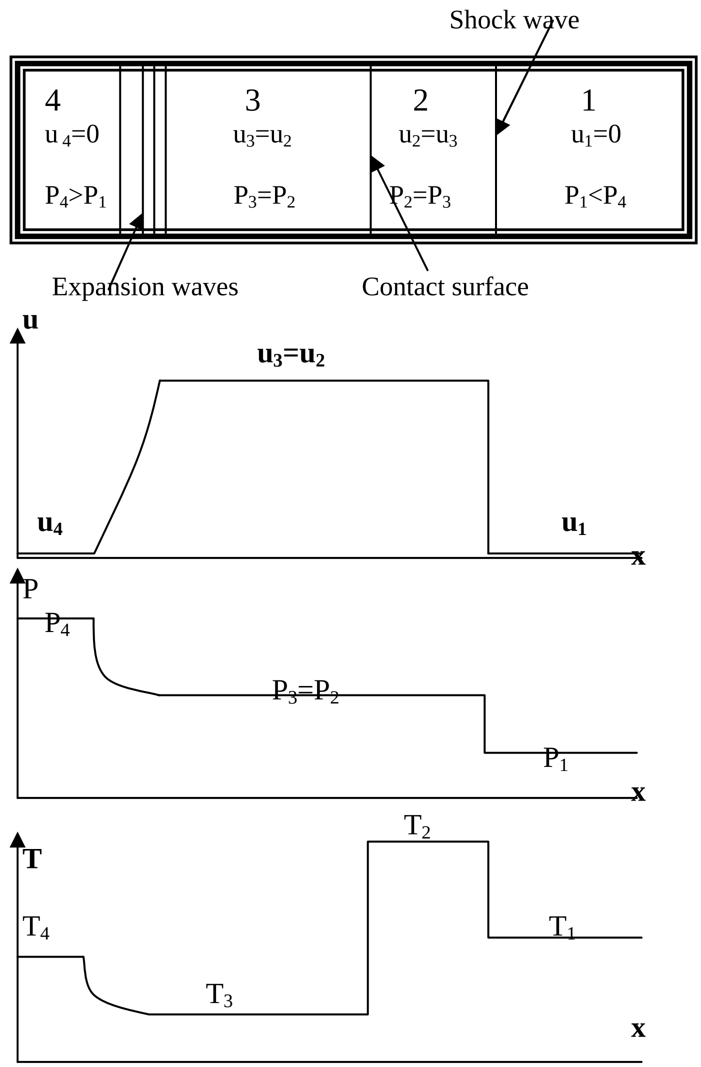
A numerical analysis must start with breaking the computational domain into discrete sub-domains, which is the grid generation process. A grid must be provided in terms of the spatial coordinates of grid nodes distributed throughout the computational domain. At each node in the domain, the numerical analysis will determine values for all dependent variables such as pressure and velocity components.

Creating the grid is the first step in calculating a flow. Solution parameters and fluid properties are defined in the parameter file. The advection discretization scheme selected is the Modified Linear Profile Skew scheme with the Physical Correction. The convergence criterion is 10E+04 and the solver is left to run until a converged solution is found [15-19].

3 Discussions and Results

After the rupture of the diaphragm in the shock tube, shown in figure 2, the system eventually approaches thermodynamic equilibrium such that the final state in the closed tube can be determined from the first law of thermodynamics. When the flow is adiabatic and there is no external work, the total internal energy of the mixture of gases at the final state is equal to the sum of the internal energies of the gases which are present initially on each side of the diaphragm. The primary interest is not the final equilibrium state of gases, however, but the transient shock phenomena after the diaphragm is ruptured. When it ruptures, a normal shock wave moves into the low-pressure side to increase the pressure, while a series of expansion waves propagate into the high-pressure side to decrease the pressure. The variation of the pressure, temperature and velocity with distance, x, shortly after the rupture of the diaphragm is shown in Figure 2, [4-8].

Fig.2 Velocity, pressure, and temperature variations in shock tube.



3.1 Shock Waves

Continuous compression waves always converge and the waves may coalesce and form a shock front. As more and more of the compression waves coalesce, the wave steepens and becomes more shock fronted. Discontinuities exist in the properties of the fluid as it flows through the shock wave, which may be treated as boundary for the continuous flow regions located on each side of it. Shock waves are also formed when the velocity of the fluid at the solid boundary of the flow field is discontinuous, as in the instantaneous acceleration of a piston.

A moving shock wave may be transformed into a stationary shock wave by a relative coordinate transformation wherein the observer moves at the same velocity as the shock wave. The resulting stationary shock wave may, therefore, be analyzed as a steady state case [4,9].

When the diaphragm is ruptured, a right-facing shock wave moves into the low- pressure gas and left-facing expansion waves move into the high-pressure gas, and a new state is formed such that $P_4 > P_2 > P_1$ and $u_2 > u_1 = u_4 = 0$. Figure 3 shows contour plot of static pressure. One can see in this figure the compression wave and the expansion fans.

Figure 4 shows pressure variations in shock tube, from the numerical results. This figure shows the static pressure variation across the shock wave and the expansion waves. This figure shows the different pressure regions, namely where $P_4 > P_3 = P_2 > P_1$. The strength of the normal shock wave and the gas velocities are dependent on the initial pressure ratio across the diaphragm, the properties of the gases that are involved and the initial temperature of the gases.

Fig.3 Contour plot of static pressure showing normal shock wave and expansion waves inside shock tube.

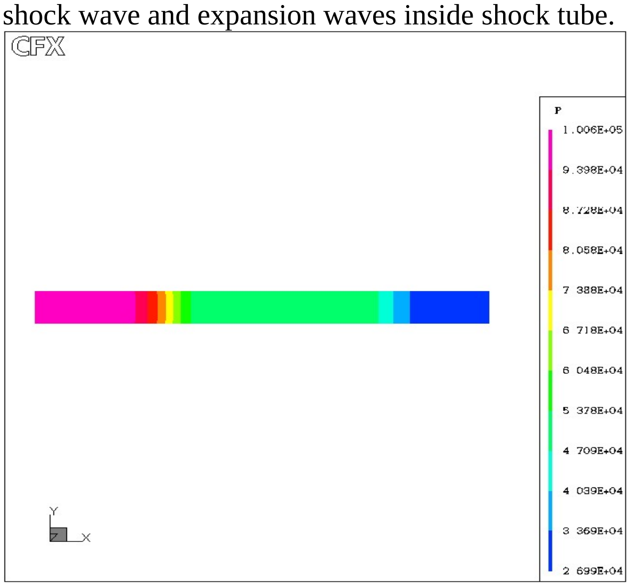
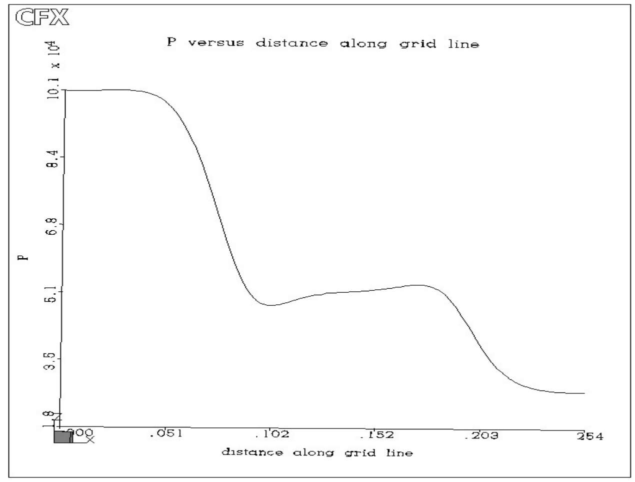


Fig.4 Pressure variations in shock tube, from the numerical results.

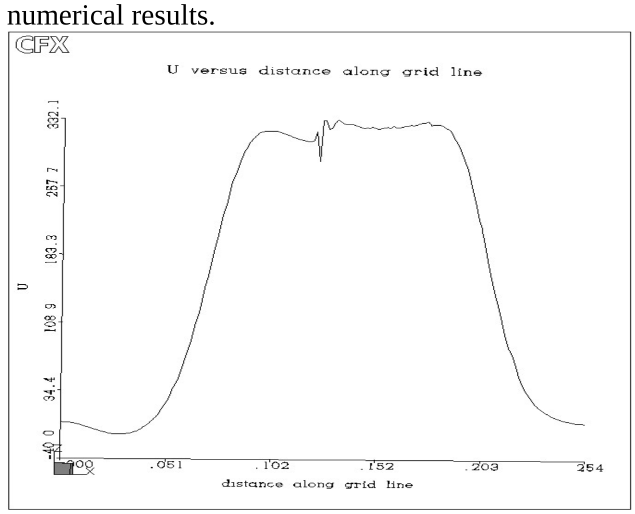


3.2 Contact Surface

In addition to the shock wave, there is another type of discontinuity termed a contact surface. The contact surface is an interface that separates two flow regions, but moves with those regions. The velocity and the pressure of the gas on each side of the contact surface are the same, but the other thermodynamic properties may be different. Unlike the shock wave, there is no flow of gas across a contact surface. Figure 5 shows the calculated values for the velocity in the shock tub. One can see in this graph that the velocity in regions 2 and 3 are equal.

It is clear that nothing is learned about the possibility of the formation of a contact surface from the u and P, because u and P are equal across the contact surface as shown in figures 3, 4 and 5. Its existence is deduced from entropy considerations of the shock and expansion waves [4-5,9].

Fig.5 Velocity variations in shock tube, from the numerical results.

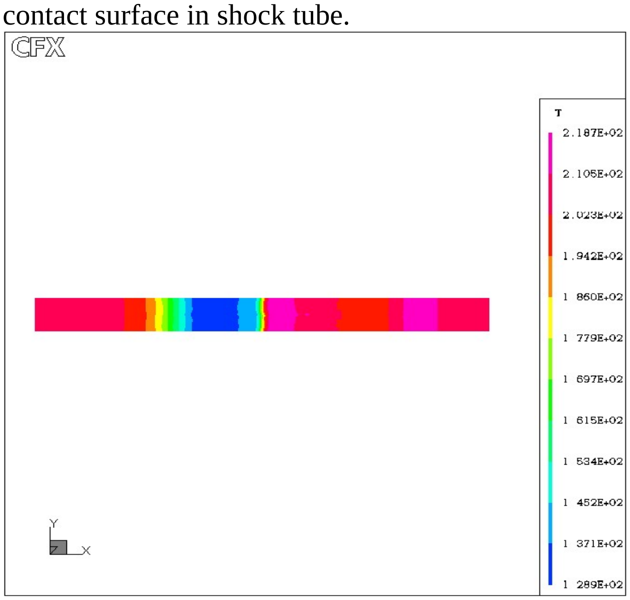


A contact surface arises at the rupture of the diaphragm separating two regions containing different gases, or two regions containing the same gas but at different thermodynamic states. Such a contact surface is generated in shock tube. Contact surfaces also arise at the interface between the regions in a combustion chamber, and at the point of intersection of two shock waves.

In the following discussion, it is assumed that a contact surface, once formed, retains its identity throughout the process. It is, therefore, assumed that such transport processes as viscous mixing, diffusion, and heat conduction are negligible.

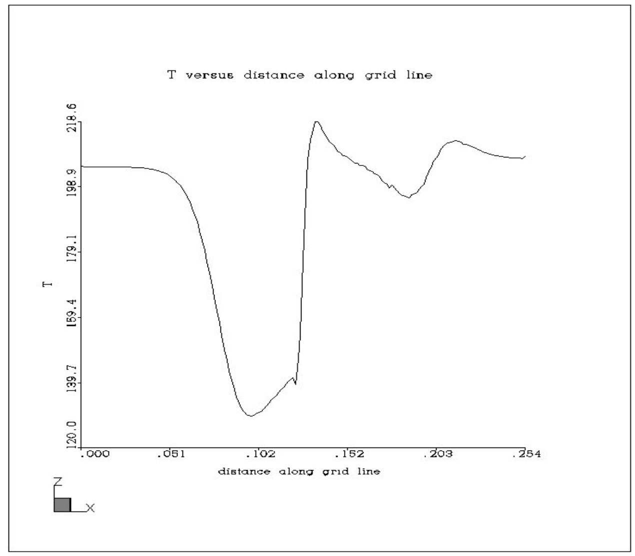
At the contact surface only the particle velocity and the pressure are continuous, and all other parameters are discontinuous, and this was shown in figures 4 & 5. Since the temperature and the density are discontinuous at the contact surface, there is a molecular transfer of heat between region 2 and 3. It should be noted that the greatest change in entropy exists at the contact front and not at the shock front, and arises from the fact that state 4, being a region of high pressure which is also at the same temperature as state 1, has a significantly lower entropy. A contact surface is formed between the two gases originally separated into the high and low pressure regions by the diaphragm and this is shown in figure 6. Figure 6 shows contour plot of static temperature.

Fig.6 Contour plot of static temperature showing the contact surface in shock tube.



A contact surface follows the initial normal shock wave down the tube. This contact surface is a temperature discontinuity, separating the gases which are compressed by the normal waves from those which are cooled by the expansion waves. Figure 7 shows temperature variations in shock tube, from the numerical results. One can see the discontinuity in temperature in the shock tube between regions 2 and 3 in figure 7.

Fig.7 Temperature variations in shock tube, from the numerical results.



4 Conclusion

A computational model that illustrates the physics of flow through non-steady shock waves and expansion fans was developed. The flow is non-steady and compressible. In this situation, one should expect oblique shock waves, expansion fans, shock wave interactions, and slip surface generation.

The results of the numerical data from this section, such as velocity, pressure and temperature were used to show the good agreement between the numerical and the analytical solution.

Through this computational analysis, a better interpretation of the physical phenomenon of the non-steady can be achieved. The results from the numerical analysis are used to study the flow structure of the shock tube and compared it to the analytical solution.

5 Acknowledge

The author gratefully acknowledges the support of King Abdulaziz City for Science and Technology.

References:

[1] Alhussan, K.: "Direct Fluid-Fluid Interaction in Three Dimensional Supersonic Non-steady Flow", The George Washington University, D.Sc. Dissertation, August 2002.

[2] Gaydon, A.G., and Hurle, I. R., The Shock Tube in High-Temperature Chemical Physics, Reinhold, New York, 1963.

[3] Henshall, B.D., The Use of Multiple Diaphragms in Shock Tubes, A.R.C. National Physical Laboratory, England, 1955.

[4] Anderson, J. D., Jr., Modern Compressible Flow, McGraw-Hill, New York, 1990.

Saad, M. A., Compressible Fluid Flow, 2nd ed., 1993.

[5] Shapiro, A. E., The Dynamics and Thermodynamics of Compressible Fluid Flow, 2nd vol., Ronald, New York, 1953.

[6] Zucrow, M. J., and Hoffman, J. D., Gas Dynamics, 2nd vol., Wiley, New York, 1977.

Cheers, F., Elements of Compressible Flow, Wiley, London, 1963.

[7] Bradley, J., Shock Waves in Chemistry and Physics, Wiley, London, 1962.

[8] Liepmann, H. W., and Roshko, A., Element of Gas Dynamics, Wiley, New York 1957.

[9] Thompson, P., Compressible-Fluid dynamics, McGraw-Hill, 1972.

[10] Glarke, T. F., and McChesney, M., The Dynamics of Real Gases, Butterworth, London, 1964.

[11] Oosthuizen, P. H., and Carscallen, W. E., Compressible Fluid Flow, McGraw-Hill, 1997.

[12] Schreier, S., Compressible Flow, Wiley, New York, 1982.

[13] Courant, R., and Friedrichs, K. O., Supersonic Flow and Shock Waves, Interscience Publishers, Inc. New York, 1948.

[14] Haluk, A., Gas Dynamics, McGraw-Hill, New York, 1986.

[15] Alhussan, K., Hong, W., Garris, C. A.: "Non Steady Pressure-Exchange Ejector", Fluids Engineering Summer Conference Montreal, Canada, July 2002.

[16] Alhussan, K., Garris, C. A.: "Non-Steady Three-Dimensional Flow Field Analysis in Supersonic Flow Induction", Fluids Engineering Summer Conference Montreal, Canada, Paper No. FEDSM2002-13088, July 2002.

[17] Alhussan, K., Garris, C., "Computational Study of Three- Dimensional Non-Steady Steam Supersonic Pressure Exchange Ejectors" IASME Transactions Issue 3, Volume 1, pp 504-512, ISSN 1790-031X, July 2004.

[18] Alhussan, K., Garris, C., " Comparison of Cylindrical and 3-D B-Spline Curve of Shroud-Diffuser for a Supersonic Pressure Exchange Ejector in 3-D, Non-Steady, Viscous Flow" IASME Transactions Issue 3, Volume 1, pp 474-481, ISSN 1790-031X, July 2004.

[19] Alhussan, K., Garris, C. 2005, "Study the Effect of Changing Area Inlet Ratiio of a Supersonic Pressure-Exchange Ejector" 43rd AIAA Aerospace Science Meeting and Exhibit, Paper No. AIAA-2005-519 , Reno, NV, USA.