Strong blast wave interaction with multiphase media

by

Sembian Sundarapandian

June 2018

Technical Reports
Royal Institute of Technology
Department of Mechanics
SE-100 44 Stockholm, Sweden
Akademisk avhandling som med tillstånd av Kungliga Tekniska Högskolan i Stockholm framlägges till offentlig granskning för avläggande av teknologi doktorsexamen torsdag den 7 Juni 2018 kl 10:00 i sal F3, Lindstedsvägen 26 Kungliga Tekniska Högskolan, Stockholm.

TRITA-SCI-FOU 2018:21
ISSN 0348-467X
ISRN KTH/MEK/TR-18/21-SE

©Sembian Sundarapandian 2018
Universitetsservice US–AB, Stockholm 2018
“If you only do what you think you can do, you will never be better than what you are.”

Anonymous
Strong blast wave interaction with multiphase media

Sembian Sundarapandian
KTH Mechanics, Royal Institute of Technology
SE-100 44 Stockholm, Sweden

Abstract
The interaction of a blast wave propagating in air with different fluids like water column, aqueous foam and thermal/density inhomogeneity have been studied both experimentally and numerically. The blast waves were generated at atmospheric conditions in a newly constructed exploding wire facility. For fixed capacitance and wire size, the intensity of the shock front (measured typically at 200 mm from the wire explosion plane) was varied by controlling the charges stored in the capacitor and the size of the test section. Qualitative features of the interaction were captured using shadowgraph technique. Numerical simulations were performed to better analyze and understand the flow features observed in experiments. The main points across each fluid interactions are as follow:

Water column: A new technique was implemented to create highly repeatable, properly shaped, large diameter water column. The impact of a blast wave with shock Mach number ranging from 1.75 to 2.4 on a 22 mm diameter water column resulted in a complex system of waves propagating inside the column. Due to the concave boundary of the downstream interface, the reflected expansion wave naturally focused at a point before travelling upstream resulting in the generation of large negative pressures leading to nucleation of cavitation bubbles. Through high speed photography, various aspects of the flow features were discussed qualitatively and quantitatively. With the aid of numerical simulation, the effect of size of water column and shock strength on the maximum attainable negative pressures in the absence of cavitation were quantified.

Aqueous foam: The performance of various aqueous foam barrier configurations on the attenuation of externally generated blast wave peak pressure was examined. Here a blast wave with shock Mach number 4.8 was allowed to interact with an aqueous foam barrier of initial liquid fraction 0.1. The dominant process responsible for reduction of peak pressure was the ‘catching up’ of the rarefaction wave with the wave front travelling in the foam barrier. Additional reduction was provided by the impedance mismatch factor at the foam-air interface which was further exploited to achieve greater reduction. A simple numerical model treating the foam by a pseudo-gas approach was used for re-constructing the experimental results.

Density inhomogeneity: The unstable evolution of a 2D elongated, elliptically-shaped inhomogeneity embedded in ambient air and aligned both normal and
at an angle to the incident plane blast wave of impact Mach number 2.15 was studied. The inhomogeneity was created on the basis of ‘Joule heating’ wherein heat produced by a current carrying wire was used to heat its surrounding air. Two counter-rotating vortices primarily due to Richtmyer-Meshkov instability (RMI) and a train of vortices primarily due to Kelvin-Helmholtz instability (KHI) were observed for two different inclination angles. Similarly circulation, calculated from numerical simulation solving Navier-Stokes equation, was also found to vary from a linear to a quadratic function when the inhomogeneity was inclined.

**Key words:** Blast waves, Negative pressure, Cavitation, Blast wave attenuation, RMI, KHI
Växelverkan av starka stötvågor med fleras medier

Sembian Sundarapandian
KTH Mekanik, Kungliga Tekniska Högskolan
SE-100 44 Stockholm, Sverige

Sammanfattning

Cylindrisk vattendroppe. Cylindriska vattendroppar med stor diameter och lämplig cirkulär form kunde skapas med hjälp av hydrofobisk ytbehandling. Växelverkan mellan stötvägor med Machtal från 1.75 till 2.4 och en cylindrisk vattendroppe med en diameter på 22 mm resulterar i ett komplext system av vågor inom droppen. En expansionsvåg reflekteras från den cirkulära ytan och fokuseras i ett område nedsträms ytan. Denna fokuseringsprocess leder till stora negativa tryck i droppen och ger upphov till ett område med kavitationsbubblor inom vattendroppen. Undersökningen utförs experimentellt och visualiseras med enstaka bilder och höghastighetsfotografering. Inverkan av droppens storlek och den inkommande stötvägens styrka på maximala negativa tryck utan kavitation undersöks numeriskt.


Domän med temperatur- och densitetsvariation. Växelverkan mellan en stötvåg

**Nyckelord:** Stötvägor, negativa tryck, kavitation, stötvägstdämpning, RMI, KHI
Preface

This thesis deals with strong blast wave interaction with heterogeneous media. A brief introduction on the basic concepts and methods is presented in the first part. The second part contains five articles. The papers are adjusted to comply with the present thesis format for consistency, but their contents have not been altered as compared with their original counterparts.


June 2018, Stockholm

Sembian Sundarapandian
Contents

Abstract v

Sammanfattning vii

Preface ix

Part I - Overview and summary

Chapter 1. Introduction 1

Chapter 2. Important concepts 3
  2.1. Shock waves 3
  2.2. Shock non-linear effects 8
  2.3. Richtmyer-Meshkov instability (RMI) and Kelvin-Helmholtz instability (KHI) 10
  2.4. Shock-bubble interaction 12
  2.5. Negative pressure induced cavitation 14

Chapter 3. Exploding wire technique 18

Chapter 4. Exploding wire facility 21
  4.1. High current generator 22
  4.2. Test cell unit 26
  4.3. Optical unit 28
  4.4. Trigger unit and sensors 29
  4.5. Blast wave formation 31

Chapter 5. Inhomogeneity creation techniques 35
  5.1. Water column 35
  5.2. Foam barrier(s) 37
Part II - Papers

Paper 1. Plane shock wave interaction with a cylindrical water column 63

Paper 2. An experimental time-based analysis and numerical parameter study on shock-water column interaction 89

Paper 3. Attenuation of strong external blast by foam barrier 103

Paper 4. Plane blast wave propagation in air with a transverse thermal inhomogeneity 133

Paper 5. Plane blast wave interaction with an elongated straight and inclined heat-generated inhomogeneity 159
Part I

Overview and summary
Chapter 1

Introduction

Blast waves are encountered in a wide variety of physical situations ranging from the pop of a balloon to supernovae explosions. Sudden release of energy in a finite area by any means results in the generation of strong blast waves with naturally destructive capabilities. As the wave propagates through the medium, damage will be caused to any object in its path by the combination of significant compression behind the shock front and high speed wind that follows it. Of course it can be harnessed to be beneficial to mankind with recent industrial applications like oil extraction enhancement, synthesis of diamonds from carbon, dust removal from surfaces, supersonic combustion, inertial confinement fusion etc.; and medical applications like extracorporeal shock wave lithotripsy, neovascularization and tissue regeneration, targeted drug delivery, tissue necrosis and hemostasis to name a few. They are also of natural interest in astrophysical flows. So not only the nature of the waves but also its interaction with other objects/media has a great impact that fuels further research. Due to the complex nature of blast waves, its interaction with random media can lead to strong coupling of a wide variety of fluid dynamic phenomena that are both beneficial and detrimental. With a surge of recent interest in using such waves across various fields, it is of importance to have a fundamental understanding of the occurring phenomena through basic research. Therefore the aim of the present study is to investigate and develop an understanding of the dynamical characteristics exhibited during blast wave interaction with various media.

This thesis is a collection of basic research on three class of problems - which can be construed as a foundation upon which complex interactions can be studied - that fall under the broad umbrella of blast wave interaction with multiphase media. The problems studied here involve cylindrical water columns (with density 1000 kg/m$^3$), aqueous foams (density 100 kg/m$^3$) and thermal/density inhomogeneities (density 0.39-0.5 kg/m$^3$) with each targeting different application area. A
continuity between the problems can be established when seen from the point of view of their corresponding density with respect to atmospheric conditions. The blast waves are generated using exploding wire technique. A new facility, named Exploding Wire facility, was built at the Shock Wave Lab in KTH Mechanics for this purpose. The experiments are conducted at atmospheric conditions in a specially designed test section. The interaction details and the resulting flow are captured using shadowgraph technique and are aided by numerical simulations.

**Thesis structure.** The thesis is organized as follows; In Part I, chapter 2 begins with a description of the difference between shock wave and blast wave, followed by descriptions of shock non-linear effects, interface induced and shear induced instabilities, and shock interaction with a bubble. The concept of negative pressure is introduced in the end of chapter 2. Chapter 3 is intended to provide an introduction to exploding wire technique while chapter 4 discusses the in-house built exploding wire facility in detail. An overview of the various techniques used for creating the inhomogeneities are described in chapter 5. Chapter 6 deals with numerical simulation method and the work is concluded with a summary of results in chapter 7. Part II consisting of 5 papers forms the nucleus of the thesis where the results are discussed in detail. The papers appear in the order of the problems studied: Paper 1 and 2 is a study investigating the water column problem; Paper 3 deals with the foam problem; and Paper 4 and 5 discusses the thermal inhomogeneity problem in detail.
2.1. Shock waves

Shock waves are very thin discontinuities, characterised as a wave propagating faster than the local speed of sound of the medium across which flow properties like pressure \( p \), velocity \( u \), density \( \rho \) and temperature \( T \) vary abruptly. The shock process is adiabatic wherein the temperature and velocity gradients internal to the shock provide complex viscous and heat conduction phenomena that increases entropy thereby rendering it irreversible. However for practical purposes, the primary interest is not on the interior of the shock but on the net changes in fluid properties across it. As these changes occur within a very small distance, of the order of a few hundred \( \text{nm} \) that is comparable with a few mean free path of the molecules, it can be simplified by ignoring the interior of the shock and treat it as a pure line or plane discontinuity. For a flow from right to left passing through a stationary normal shock with conditions ahead denoted by state 1 and behind by state 2 [figure 2.1(a)], pressure, density and temperature increases in state 2 while velocity and stagnation pressure \( (P_0) \) decreases. Stagnation temperature \( (T_0) \) remains constant due to the adiabatic nature of the process. For a calorically and thermally perfect gas, the ratio of properties across a normal shock determined by solving the following equations,

\[
\begin{align*}
\rho_1 u_1 &= \rho_2 u_2 \quad \text{(continuity)} \\
p_1 + \rho_1 u_1^2 &= p_2 + \rho_2 u_2^2 \quad \text{(momentum)} \\
h_1 + \frac{u_1^2}{2} &= h_2 + \frac{u_2^2}{2} \quad \text{(energy)} \\
p &= \rho RT \quad \text{(ideal gas law)} \\
h &= c_p T \quad \text{(enthalpy)}
\end{align*}
\]  

(2.1)
2. Important concepts

(a) Stationary normal shock
\[
\begin{align*}
p_2 & > p_1 \\
\rho_2 & > \rho_1 \\
T_2 & > T_1 \\
u_2 & < u_1 \\
P_{o2} & < P_{o1} \\
T_{o2} & = T_{o1}
\end{align*}
\]

(b) Moving normal shock
\[
\begin{align*}
p_2 & > p_1 \\
\rho_2 & > \rho_1 \\
T_2 & > T_1 \\
u_2 & > u_1 \\
P_{o2} & > P_{o1} \\
T_{o2} & > T_{o1}
\end{align*}
\]

\[u_i = 0\]

Figure 2.1: Properties on both sides of (a) stationary normal shock and (b) moving normal shock.

Here, the ratio of densities and pressures are
\[
\frac{\rho_2}{\rho_1} = \frac{u_1}{u_2} = \frac{(\gamma + 1)M_1^2}{2 + (\gamma - 1)M_1^2}
\]
\[
\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1}(M_1^2 - 1)
\]
\[
\frac{T_2}{T_1} = \frac{h_2}{h_1} = \frac{p_2}{p_1} \frac{\rho_2}{\rho_1}
\]

(2.2)

Here, \(M\) is the Mach number, defined as \(u/c\), where \(c\) is the local speed of sound of the medium. The ratio depends solely on the inlet Mach number \(M_1\) (assuming constant \(\gamma\)) which defines the strength of the shock. This implies stronger the shock, larger is the ratio. In many practical situations, like from an explosion or suddenly opening a valve, the shock wave is moving when viewed from a stationary reference frame. Assuming the gas ahead of the shock to be motionless for such cases as in figure 2.1(b), the gas behind will be travelling with the moving shock at subsonic or supersonic speeds. Since the jump conditions depend only on the strength of the shock, other flow properties like pressure, density, relative velocity (between the shock and gas behind) and temperature will be the same for both stationary and moving shocks.
2.1. Shock waves

2.1.1. Blast waves

Sudden deposition of energy in a very small volume, like detonation of explosives or lightening will result in a blast wave propagating outwards from the source. A shock with ‘blast profile’ or simply called blast wave is basically a shock wave that consists of a leading shock followed by continuous expansion waves which decays its strength at a steady rate. This decay occurs across all the properties behind the leading shock. For a typical shock wave with ‘step profile’ or ‘shock profile’, the properties behind the shock remain constant. Since blast waves were used throughout this thesis\(^1\), a more intuitive understanding between the two is essential and will be provided by means of a piston analogy (Shapiro 1953) and a shocktube analogy. Consider a piston moving from left to right into a motionless gas with a series of impulsive acceleration occurring at equal intervals of time as shown in figure 2.2(a). Each acceleration sends out an isentropic pressure wave of equal strength that slightly increases the pressure of air in the region it has propagated through. For instance, the first acceleration sends out a pressure wave travelling downstream that influences region ‘a’ marked in figure 2.2(a). The second pressure wave from the next acceleration now travels in the region where the flow properties are relatively larger. Since the ratio of increase across the wave is only dependant on its strength, the pressure in region ‘a’ is again slightly increased from its previous value. Also as the second wave travels with the local speed of sound \((c)\) of the fluid, it travels faster compared to the first wave and tends to overtake it. Meanwhile, a succession of accelerations sends out a series of waves that works on the same fluid over and over again until the pressure gradient becomes infinite as shown at step 5 in figure 2.2(a). This also means that all the waves coalesce together to form one single strong wave (a discontinuity) where the viscous and heat dissipation effects within are no longer negligible thereby resisting the infinite steepening of the wave. Thus a shock wave is formed with a constant pressure profile, marked as ‘shock profile’ in figure 2.2(c).

Now lets consider a fully formed shock with the piston position and the state of conditions as in step 5. Now instead of accelerating it rightward, let us accelerate it as before but leftwards [figure 2.2(b)]. This sends out a series of isentropic expansion wave which acts opposite to a compression wave i.e. across an expansion wave pressure, density and temperature decreases while velocity increases. So the propagation of the

---

\(^1\)Blast waves are typically accompanied by a negative pressure (in gauge) wind. But as will be shown in Sec 4.5, blast waves generated in this work will have a decaying profile but not the negative phase.
2. Important Concepts

Pressure

Piston accelerated rightwards in steps

Piston accelerated leftwards in steps

(a) Shock profile formation

(b) Blast profile formation

(c) Graphical representation of both profiles

Figure 2.2: The formation of (a) shock wave and (b) blast wave using a piston analogy. The corresponding pressure profiles are shown in (c).

First expansion wave slightly reduces the pressure in region ‘a’. Since it travels in the region worked by the shock wave where the local speed of sound is relatively higher, it travels faster and catches up with the shock
2.1. Shock waves

When waves of opposite families interact, they tend to cancel each other thereby weakening the shock. Meanwhile the second expansion wave travels in a slightly expanded and cooled region with lower speed of sound and will not be able to catch up with the shock until its strength is weakened. This results in a shock wave with a pressure profile that is gradually reducing. Such shock waves are usually defined as ‘shock with blast profile’ or simply ‘blast waves’. A blast profile is as shown in figure 2.2(c). For longer propagation distance, the strength of the wave gradually weakens until it becomes a Mach wave (a pressure wave travelling with the speed of sound).

![Driver Driven Shock wave Expansion waves](image1)

**Figure 2.3:** Schematic representation of x-t diagram for (a) shock profile and (b) blast profile

Shock tube is a device used extensively for the generation of shock waves in a controlled laboratory environment. It is a long tube divided into two regions: a high pressure region called ‘driver’ section and a low pressure region called ‘driven’ section, by a diaphragm of known thickness. When the diaphragm is suddenly removed, a shock wave propagates into the driven section and an expansion fan centered at the diaphragm position propagates into the driver section. A contact surface separating the fluids worked by the shock wave and the expansion wave travels into the driven section. The strength of the shock wave depends on the high pressure to the low pressure ratio while its profile depends
on the corresponding lengths of the driver and driven sections. An x-t diagram is constructed with the shock tube set-up shown in figure 2.3(a). When the diaphragm is removed, a shock wave (indicated by a bold solid line) travels to the right while the expansion fan head travel to the left. Within the domain set for constructing this diagram, the expansions waves reflecting from the driver end wall was never able to catch up with the shock wave. For this scenario, the shock wave propagates with a constant strength and the pressure profile will resemble the shock profile in figure 2.2(c). Now let us reduce the length of the driver section considerably and construct the x-t diagram as in figure 2.3(b). Due to its shorter length, the expansion waves are reflected much sooner and are able to catch up with the shock wave. When the expansion waves interact with the shock, it reshapes its profile thereby gradually weakening it along $x$.

To sum-up, blast wave is a type of shock wave whose strength and conditions behind it decay at a steady rate during its propagation. In this work only blast waves were used in experiments.

2.2. Shock non-linear effects

Whenever a shock wave encounters different medium it undergoes non-linear effects like reflection, refraction and diffraction at the interface. The incident shock will always be transmitted as a shock wave but will be reflected either as a shock wave or a rarefaction wave depending on the medium characteristics. Generally, for most cases involving gas, they can be classified on the basis of acoustic impedence mismatch as slow/fast or fast/slow configuration and the same on the basis of density are classified as heavy/light or light/heavy configurations. The acoustic impedence ($\mathcal{R}$), given as $\mathcal{R} = \rho c$, is a thermodynamic property of the medium that measures its stiffness. Consider a shock wave travelling in medium 1 with impedence $\mathcal{R}_1$ and encounters medium 2 with impedence $\mathcal{R}_2$. For a case where $\mathcal{R}_2 < \mathcal{R}_1$ as in figure 2.4(a), the transmitted shock will travel relatively faster in medium 2 as it offers less resistance to its propagation. Usually in such a scenario, although the absolute velocity of the shock wave is higher, its Mach number will be lower due to higher sound speed of the medium. As the flow properties are dependant only on Mach number, their corresponding ratios will also be relatively lower in medium 2. Therefore in order to maintain equilibrium of gases across the interface, the incident shock will be reflected as a rarefaction wave. For a reverse case where $\mathcal{R}_2 > \mathcal{R}_1$, the transmitted shock will travel relatively slower. This implies that the gas must contract and therefore the incident shock will be reflected as a shock wave [figure 2.4(b)].
2.2. SHOCK NON-LINEAR EFFECTS

Shock diffraction pattern is categorised into regular and irregular reflection. Generally regular reflection is a two shock system, consisting of the incident and reflected shock meeting at the reflecting surface. The irregular reflection encompasses various patterns that is subdivided into two major domains: Mach reflection and weak shock reflection domain that includes von Neumann, Guderley and Vasilev reflection. As the focus is only on Mach reflection, the weak shock reflection domain will not be considered here. A thorough treatment of these domains can be found in Ben-Dor (2007). The Mach reflection consists of the incident shock, reflected shock, Mach stem and a slip stream meeting at a single point, called triple point, located above the reflecting surface. Depending on the direction of propagation of the triple point from the reflecting surface, the Mach reflection is further classified as: (a) Direct Mach reflection, where the triple point moves away from the surface; (b) Stationary Mach reflection, where the triple point moves parallel to the surface; and (c) Inverse Mach reflection, where the triple point moves towards the surface. Consider shock wave interaction with a solid cylindrical convex surface shown in figure 2.5. This geometry is chosen as it involves both the regular and Mach reflection patterns and is closely reminiscent of the geometries investigated in this thesis. During this interaction, regular reflection occurs as long as the angle of incidence $\alpha$ measured between the incident shock and the tangent to the surface does not exceed a critical value, called detachment angle. When the angle of incidence exceeds the detachment angle, a transition from regular to Mach reflection occurs where the incident and the reflected shock no
longer meet at the surface. Typically regular reflection is observed during the initial phase of shock propagation, marked as phase (1) in figure 2.5 and Mach reflection is observed in phase (2). Usually the triple point moves further away from the surface as the shock proceeds through phase (2).

Figure 2.5: Schematic representation of Regular and Mach reflection of an incident shock passing over a solid cylinder at two successive time instants.

2.3. Richtmyer-Meshkov instability (RMI) and Kelvin-Helmholtz instability (KHI)

Richtmyer-Meshkov instability (RMI), named after Richtmyer (1960) who first provided the theoretical analysis and Meshkov (1969) who confirmed it experimentally, is a hydrodynamic instability that typically develops when a material interface separating two fluids is impulsively accelerated by passage of a shock wave. The instability develops as a result of baroclinic vorticity generation, due to the misalignment of pressure and density gradient across the interface, given by the following vorticity transport equation:

$$\frac{D\omega}{Dt} = \frac{1}{\rho^2} (\nabla \rho \times \nabla p)$$

(2.3)
2.3. Richtmyer-Meshkov instability (RMI) and Kelvin-Helmholtz instability (KHI)

The disturbances initially present in the interface, however small it may be, are amplified by the deposition of vorticity that finally develops into regions of intense mixing. RMI can be considered as the impulsive-acceleration limit of a constant acceleration environment as in Rayleigh-Taylor instability (RTI). The evolution of the instability can be easily understood following figure 2.6 taken from Brouillette (2002). Consider a sinusoidal interface separating a light from a dense fluid (with the light fluid on top) is impacted by a shock wave travelling in the light fluid. A distorted shock wave is both transmitted and reflected at the interface which is impulsively accelerated in the direction of the transmitted wave. The misalignment of pressure gradient of the shock and density gradient of the interface results in the formation of an unstable vortex sheet of varying strength due to the generation of counterclockwise and clockwise vorticity on the right and left side [figure 2.6(b)]. This leads
to a nonlinear regime with the penetration of heavy fluid into the light fluid and vice versa, termed as ‘spike’ and ‘bubble’ formation respectively. The vortices then roll-up due to the onset of Kelvin-Helmholtz instability (KHI) eventually resulting in intense mixing of the fluids.

![Cloud pattern depicting Kelvin-Helmholtz instability (KHI)](source: amusingplanet.com)

The KHI is also a hydrodynamic instability that is induced by shear between fluid layers or a velocity difference/discontinuity across an interface. It means that any sharp velocity gradient in a shear flow is unstable leading to the amplification of disturbances thereby resulting in rolling up of the layers/interface. KHI can be seen occurring naturally in real life scenarios, like the cloud pattern as shown in figure 2.7.

**2.4. Shock-bubble interaction**

When a shock wave interacts with an isolated gas bubble, strong coupling of several of the previously described fluid dynamics phenomena occur. Due to the plethora of information contained within a shock-bubble interaction, it has been researched extensively making it a fundamental yet vital configuration for studies pertaining to shock accelerated inhomogeneous flows. Shock refraction pattern over a geometrically well defined gaseous bubble, from the experiments of Haas & Sturtevant (1987), for both slow/fast and fast/slow cases are as shown in figure 2.8. For the slow/fast scenario, irregular refraction occurs - similar to that of a solid cylinder - with the addition of a faster travelling transmitted
shock through the bubble and a reflected rarefaction wave. The trans-
mittted shock is connected to the incident shock through a precursor
shock initiating a quadruple point at its intersection with the Mach stem
and reflected wave. The refraction follows a Direct Mach type as the
quadruple point moves away from the interface during shock passage.
In case of fast/slow scenario, the incident shock is directly connected
to the transmitted shock at the interface due to the slow nature of the
transmitted shock.

Figure 2.8: Sequence of images showing the shock structures and devel-
oped counter-rotating vortices for both slow/fast and fast/slow configu-
ration bubble impacted with a shock wave (Haas & Sturtevant 1987)

During this process, baroclinic vorticity is deposited at the interface
between the two fluids due to the misalignment of pressure and density
gradients. The circular shape of the bubble covers the complete range of
angles between the pressure and the density gradients with the maximum
misalignment at the diametral plane. This phenomenon is classified into
two phases: (a) vorticity deposition phase where the incident shock wave
is in contact with the interface; and (b) vorticity evolution phase which
drives the flow after the incident shock leaves the interface. Depending
on the outward or inward radial density gradient at the circumference
of the bubble, the sense of rotation of the vorticity on the top half and
Important concepts

bottom half is either clockwise (negative) or counter-clockwise (positive). After several shock-passage times, this leads to the formation of two counter-rotating vortices - the primary instability often referred to as RMI. The rotation of the vortices also induces ‘jetting’ in the channel between them. Further investigation by Tomkins et al. (2008) using planar laser-induced fluorescence measurements (figure 2.9) revealed evidence of secondary instabilities appearing on the structures causing rapid transition to turbulence enhancing mixing. Appearance of coherent structures of KH type instability is observed at 650 µs but it soon evolves into a well-mixed incoherent state at 1000 µs.

Figure 2.9: Concentration (mass fraction) maps with time obtained using planar laser-induced fluorescence (Tomkins et al. 2008)

2.5. Negative pressure induced cavitation

Cavitation is a phenomenon observed in liquids. In Engineering literature, cavitation in liquids is usually stated to occur when the pressure of the liquid falls to its vapour pressure. In a more general sense, this statement is true. But the more correct statement is that cavitation occurs when tiny bubbles are formed due to pressure reduction usually below 0 Pa (in absolute scale), termed as ‘negative pressures’. Any liquid can be brought to such states by either superheating it above its boiling temperature or by stretching it below its saturated vapour pressure. Water, unlike gas, is held together by cohesive forces that can sustain such positive stress or negative pressures when subjected to tension. Under tension, it behaves differently from those at compressed or ambient pressure as attractive forces, rather than repulsive forces, dominate their behaviour. When the tension in water exceeds a certain limiting value, it fractures resulting in cavitation. This statement becomes reasonable if thought
in terms of density where negative pressures corresponds to a reduced liquid density compared to equilibrium. Molecules are further away but their mutual attraction allows the system to remain in ‘metastable’ state. For too large intermolecular distance, the system becomes mechanically unstable resulting in cavitation. Theoretical treatment for prediction of the threshold for nucleation of bubbles in pure homogeneous water (without impurities) was provided by Classical nucleation theory (CNT). It is based on the fact that when the liquid reaches its vapour pressure, the nucleation of bubbles is delayed due to the cost associated with the surface energy required to form the liquid-vapour interface. This leads to the metastable state that sustains negative pressures. The competition between the energy gained from the formation of bubble and the energy cost associated with the creation of interface results in an energy barrier which decreases as the pressure is further reduced. Upon reaching a critical value, the barrier is low enough to be overcome thereby generating bubbles resulting in cavitation. CNT predicts that water can get upto -150 MPa at atmospheric conditions before it begins to cavitate (Caupin 2005). A few other theories predicting this threshold have been developed over the years and figure 2.10 - extracted from Caupin & Herbert (2006) - gives an overview of those limiting values. However experimentally determined values are usually lower than predicted by theory and scattered between various experiments because of the heterogeneous nature of nucleation at dissolved gases, solid impurities and tiny crevices in walls of the vessel containing the liquid which acts as nucleation sites.

Traditionally numerous experimental techniques have been developed for creating tension in water that broadly fall under three categories viz. (a) static (b) dynamic and (c) ultrasonic stressing. A summary of the techniques can be found in Trevena (1984). The methods falling under static stressing are, (a) Berthelot tube method (Worthington 1892): a vessel almost completely filled with water is first heated until it expands and fully fills it. Upon cooling, the liquid sticks to the wall of the vessel preventing it from contracting thereby inducing tension and sustaining negative pressure if the temperature is low enough; (b) Centrifugation (Briggs 1950): a Z-shaped capillary tube, open at both ends, is filled with water and rotated at high speeds resulting in negative pressures due to centrifugal forces; (c) mineral inclusion method (Zheng et al. 1991; Azouzi et al. 2013): this method is similar to Berthelot method wherein water is trapped inside crystals (like quartz, calcite and fluorite) in the 10-100 µm range. It is then heated to remove any vapour bubbles and cooled until nucleation occurs. Most of the negative pressure reported for homogenous cavitation in water were obtained using this method.
Figure 2.10: Graph showing theoretical - CNT (dotted line, Caupin 2005); Density functional theory (solid line, Speedy 1982); and Molecular dynamics simulations (short-dashed line, Yamada et al. 2002) - and some experimental - Mineral inclusions (filled diamonds, Zheng et al. 1991); and acoustic method (empty circle, Herbert et al. 2006) - cavitation threshold pressure as a function of temperature extracted from Caupin & Herbert (2006).

While static stressing takes a few minutes for nucleation, dynamic stressing utilizes shock waves for creating tension where nucleation is rather immediate. It can be generated on the basis of (a) reflection principle: Underwater explosion (Trevena 1967; Wilson et al. 1975) resulting in shock wave which gets reflected as an expansion (tension) wave at the free surface, bullet-piston method (Overton & Trevena 1982; Williams & Williams 2000) in which the mode of generation of shock is by a bullet fired at a cylinder containing water; (b) tube arrest technique (Overton & Trevena 1981): a tube filled with water is pushed upwards by springs and is stopped suddenly creating a shock wave; and (c) Water shock tube method (Richards et al. 1980): expansion wave is produced in the liquid either directly or by converting a shock wave by a reflection method. Usually low values of negative pressures were reported as the free surface of water was not a perfect reflector and it was also difficult to avoid heterogeneous cavitation. Ultrasound stressing (Lauterborn & Cramer 1981; Herbert et al. 2006) makes use of an acoustic wave that can
quench water to negative pressure during its negative swing. The waves were usually generated using a spherical or hemispherical piezoelectric resonator immersed in water.
Chapter 3

Exploding wire technique

The blast waves in this work are generated using ‘Exploding Wire (EW)’ technique which offers the following advantages: table-size experimental facility; high intensity blast generation with relatively less resources compared to a more predominantly used shock tube; ease of operation; and ability to generate shocks in any medium. This technique works on the principle of ‘Joule heating’ wherein passage of current through an electrically conducive wire produces heat. A much simplified version of EW mechanism can be observed when a fuse blows where a slight pop sound is accompanied with a bright flash of light. Owing to relatively low current, the fuse wire melts under the generated heat forming liquid droplets that arcs resulting in a bright flash. But when a few hundred kilo amps of current is suddenly deposited at a very high rate in the wire, it undergoes violent detonation accompanied by shock waves.

The study of wires exploded by means of large current pulses has a long history after Nairne (1774) published the first paper on exploding wires. Singer & Crosse (1815) reported later with interest in explosive force. Later large amount of contributions, like Pugh et al. (1951); Allen et al. (1953); Zarem et al. (1958) to name a few, were put towards the improvement of high speed photography during the mid and late 1950’s following the development of Kerr cell photography. Bennett (1958) studied the shock waves associated with EW, relating the shock wave, electrical and heat energy. A comprehensive review of EW phenomena research until 1963 can be found in McGrath (1966a). Vaughan (1963) used the exploding wire technique to generate shock waves in water following which works on underwater explosion appeared on a consistent basis. McGrath (1966b) discussed the similarity existing between EW and chemical underwater explosions (CUE). Alenichev (1972) compared the strength of the shock wave generated in water by wires of different sizes by virtue of the deformation of the diaphragm. More recent studies in underwater exploding wires include works of Krasik et al. (2008); Veksler et al. (2009); Fedotov-Gefen et al. (2010); Bazalitski et al. (2011).
3. Exploding wire technique

Usually the charges were stored in a capacitor with the conducting wire being made of aluminum, copper, constantan, iron or gold. The size of the wire used, like its length and diameter, are dependant on the rating of the capacitor. A typical EW process - occurring within the microsecond scale - comprises of the following steps:

- Heating of wire due to dumping of electrical energy at a very high rate.
- Liquid metal replaces the wire and in some cases develops an instability that forms a spherical broken series of the liquid metal.

Figure 3.1: Shock fronts and associated vapour cloud resulting from a wire explosion at two time instants.
These were responsible for the striated pattern observed in the vapour cloud resulting from wire explosion.

- The liquid metal vaporize after which a period of very low conductivity immediately follows, reported as ‘dwell time’ or ‘dark time’.
- Following restrike, a very high current again flows that converts the vapour into plasma. A sudden flash of light is also observed.
- One or more shock fronts are also produced during this interval.

Figure 3.1 shows the shock fronts and the associated vapour cloud striation patterns for two wire lengths of 20 mm and 40 mm respectively at two time instants. The tests were made with a 0.2 mm diameter copper wire suspended in the middle of a $80 \times 80 \times 8$ mm square test channel.
Chapter 4

Exploding wire facility

The experiments were conducted in a newly built Exploding Wire (EW) facility at the Shock Wave Lab, KTH Mechanics. The initial 30% of the total time in the Doctoral programme was devoted to the design and construction of the EW facility which consists of three main units, namely (a) high current generator, (b) test cell unit, and (c) optical unit. A general outline of the setup is shown in figure 4.1. The facility along with the associated equipments used are described in detail in this chapter.

Figure 4.1: General outline of the EW facility.
4. Exploding wire facility

4.1. High current generator

As the name suggests, the primary function of this unit is to generate and deliver high current to the wire to be exploded. This unit, as shown in figure 4.2, consists of a high voltage power supply, a capacitor, a spark gap assembly, 8 co-axial cable assembly, and a safety circuit consisting of a charging resistor, discharging resistor, manual and automatic safety switches. The high voltage power supply is a Spellman SL300 that provides well regulated, low ripple high voltage with a maximum output rating of 60 kV at 5 mA current in an efficient and compact design. The DC output voltage and current are adjustable from zero to maximum rating either through two front controls with status indicators or through remote programming. To preset the desired output voltage and current manually, the HIGH VOLTAGE OFF switch should be pressed and held and then the corresponding control dial is rotated while noting its meter reading. Once the voltage and current is set to a desired value, pressing the HIGH VOLTAGE ON switch will start the charging process. During this process, the light inside the switch will be lit green indicating that power is being generated. To terminate the power generation, the HIGH VOLTAGE OFF is pressed. All these operations and monitoring can also be performed remotely by sending signals through the 26 pin interface situated at the rear panel. Majority of the experiments in this work are
performed using a preset voltage of 11-13 kV unless otherwise specified. The power generated by the power supply is utilized to charge a 6 µF capacitor with a maximum voltage rating of 30 kV. A charging resistor of resistance 3 kΩ and a high voltage coil intercepts the line between the power supply and the capacitor, thereby completing the charging circuit.

![Figure 4.3: (a) Spark gap assembly; (b) Safety circuit for discharging left-over or stored charges in the capacitor.](image)

The discharging circuit is either through the spark gap and the wire or through a discharge resistor and the manual and automatic safety switch. The spark gap (Series T-670), shown in figure 4.3(a), acts as a fast switch consisting of two brass electrodes at opposite ends with a flat brass disc in the middle, all separated by a small gap between them. The T-670 is a compact pressurised spark gap originally designed to operate in the 20-100 kV range and can handle a peak current of 100 kA. But as the test unit is not designed to handle such high voltages, the distance between the electrodes are adjusted manually so that self-breakdown of air occurs around ≈9 kV at atmospheric conditions. Self-breakdown, also termed as dielectric breakdown, is said to occur when current flows through an
insulator when the voltage applied across exceeds the breakdown voltage. So the minimum voltage at which the spark gap is functional is 10 kV. The spark gap can be operated in two modes (a) External trigger mode, and (b) Pressure release mode. In the external trigger mode, a TM-11A Trigger module designed to provide a 30 kV trigger pulse of fast rise time is connected to the middle brass disc. Also two resistors of resistance 10 $M\Omega$ are connected between the charged electrode and the disc and between the disc and the uncharged (zero initial potential) electrode respectively to distribute the potential difference evenly. The trigger module can be operated using a pushbutton, remote female jack or the oscillator BNC input connector. Each output lead is fitted with a 500 pF capacitor rated for 30 kV DC providing D-C isolation to prevent damage to the module due to discharge from the disc. Since self-breakdown occurs at 9 kV at atmospheric pressure, the spark gap is slightly pressurised with 0.5-1 bar overpressure to operate at higher voltages. The pressure is controlled accurately by a pressure regulator. When a 30 kV trigger pulse is applied to the disc, the potential difference shoots up breaking the air insulation, initiating rapid discharge of the capacitor. In the pressure release mode, the middle disc is removed and the spark gap is initially pressurised to 3 bars to prevent self-breakdown during the charging process. When the experiment is to be made, the pressure is released through a one-way solenoid valve initiating rapid discharge of the capacitor making it a simple method for firing. Both the modes can be operated manually as well as remotely. When the capacitor is discharged via the spark gap, the current is delivered to the wire by 8-parallel, short RG-213 co-axial cables in order to reduce the circuit inductance.

Depending on the type and size of the wire used for explosion, there may be some left-over charge in the capacitor after the shot is made. Also occurrences where the shot has to be aborted after charging the capacitor cannot be ruled out. In order to short-circuit either the left-over charge (when the shot is made) or the stored charge (when it is aborted) in the capacitor, a safety circuit consisting of a resistor and safety switches housed in a wooden box with aluminium casing as shown in figure 4.3(b) is included. The discharging resistor with a resistance of 100 k$\Omega$ connects the main of the capacitor with the safety switches. The automatic switch, controlled by a 5-port solenoid valve, is a pneumatic controlled actuator connecting the main and the ground. Another line from the pressure regulator pressurizes the actuator piston in turn opening the circuit while releasing the pressure retracts it, thereby closing the circuit. The manual switch, again connecting the main and the ground, can only be
4.2. Test cell unit

opened and closed by flipping the lever located on the side of the box [figure 4.3(b)]. It cannot be operated remotely and hence the name. Both the manual switch and the actuator must be open in order to charge the capacitor while closing even one discharges it, providing two layers of safety.

![Diagram](image_url)

Figure 4.4: General outline of the automation process using NI-USB 6009 DAQ.

The entire process starting from setting the desired output voltage and current level, opening the actuator, charging the capacitor, firing the spark gap and then closing the actuator was at first automated using NI USB-6009 DAQ and LabVIEW software. A general outline of this automation set-up is shown in figure 4.4. After repeated operations, the DAQ was burnt presumably by the electromagnetic fields induced by the long closed circuit loops. Full automation was then discarded and a manual procedure that avoids the loop was followed thereafter. In the manual process, due to safety concerns two physical switches for opening/closing the actuator and firing the spark gap were set-up at the other end of the room far away from the facility.
4. Exploding wire facility

4.2. Test cell unit

The test cell unit is the heart of the facility where the experiments are performed. The test cell, shown in figure 4.5, consists of two chambers namely exploding chamber and a test chamber. Both the chambers are made of 12 mm thick steel wall and are rigidly connected to each other through flanges. A cut-section view of the test unit showing the construction is shown in figure 4.5(c). The chambers are designed in a way that a uniform channel of width 74 mm and height 5 mm runs throughout. The overall length of the channel is 360 mm. The exploding chamber [figure 4.6(a)], as the name suggests, is the chamber where the blast waves are generated by exploding the wire by subjecting it to the very high current from the capacitor. This chamber houses the electrodes which are 74 mm apart across which the wire to be exploded is screwed. The other end of one of the electrode is connected to the capacitor via 8-parallel co-axial cables while the second electrode is grounded (also through the co-axial cables). The inner walls of the exploding chamber are insulated with PolyOxyMethylene (POM) thermoplastic of varying
4.2. Test cell unit

Exploding chamber

- The exploding chamber (figure 4.6) runs throughout the sides of the channel providing a straight, uniform rectangular test area. When high current passes through the wire, it undergoes rapid Joule heating resulting in the generation of a cylindrical blast wave. Since the generated blast wave is confined within the narrow rectangular section, it undergoes multiple reflections from the top and bottom walls and eventually modifies itself into a plane blast wave. A description of the modification is dealt in Sec 4.5. This plane blast wave then propagates into the test chamber where the shock interactions are investigated.

Test chamber windows

- The test chamber is a section providing a field of view of about 200 mm in length and 74 mm in width. The top and bottom of the test chamber are either fitted with transparent plexiglas windows or glass windows for capturing shocks and shock interactions optically. While the plexiglas windows are relatively cheap, easy to fabricate and more importantly offer a degree of freedom with mounting of pressure sensors at desired locations, glass is superior in avoiding stress waves (Paper 1) with excellent optical properties. Owing to the construction of the...
test chamber, the glass windows were specially fabricated by gluing it into a steel housing as shown in figure 4.6(b). As the steel housing was designed to provide access for mounting the sensors on the sides, the glass windows only provide a field of view of 140 mm in length and 40 mm in width. Note that the overall dimensions for shock propagation is still $200 \times 74$ mm. The windows are secured rigidly in place by a steel rim screwed along the edges of the top and bottom wall of the channel. The entire test unit is properly sealed using o-rings wherever necessary.

4.3. Optical unit

The shock interaction and its subsequent flow field exhibiting density variations are captured using a Shadowgraph optical setup. The shadowgraph technique is based on the principle that when a light passes through a density gradient, it deflects due to the variations in refractive index of the medium. A shadow effect is generated due to this deflection resulting in a dark area at the recording plane. The optical unit, shown in figure 4.7, is set-up to pass light through the test chamber. The main components making up the set-up are laser light source, camera, plane mirrors and optical lens assembly. The general arrangement is that the beam from the laser head is expanded horizontally to 180 mm in diameter and rendered parallel by a combination of concave (L1) and convex lens (L2) of focal length -6 mm and 1350 mm respectively. The resulting parallel beam is then redirected vertically by the top plane mirror tilted $45^\circ$ to pass through the transparent test chamber and then redirected back to the horizontal plane by the bottom $45^\circ$ mirror. The image of the shadowgraph plane is focused on the camera sensor by convex lenses L3 and L4 of focal length 1350 mm and 100 mm respectively.

Two types of camera with either high temporal or spatial resolution are used for capturing the flow structures. For temporal resolution, a Shimadzu HPV-X2 high speed camera with the capability to capture at the rate of 60 - 10 million frames per second (fps) is used. It is equipped with a 10 bit, monochromatic FTCMOS2 sensor with pixel count of 100,000 (400 X 250 grid) and recording capacity of 128 frames (full pixel mode) or 256 frames (half pixel mode). The camera is triggered by an external TTL-level (5 V) signal from the digital delay generator. For spatial resolution, a 12.3 megapixel Nikon D90 camera is used. As the shutter of the Nikon D90 cannot be triggered without internal modification, it was simply left open for 30 s due to which it can only be operated with a pulsed (single-shot) light source. As it is left open for long, a neutral density filter was placed close to the camera sensor that is opaque enough to damp the light coming from the explosion flash, room
and other equipment illumination while transparent enough to pass the light pulse.

Two types of laser light source are used: one continuous and the other pulsed, to be used in conjunction with the high speed camera and the Nikon D90 camera respectively. The pulsed laser is an Orion Nd:YAG laser (532 nm) with a pulse energy of 17 mJ and a beam diameter of 2.75 mm. With a frequency of 1 Hz coupled with a very short pulse width of 4-6 ns, it’s ideally suited to be used with the Nikon D90. To trigger the laser, two TTL signals are sent: the first to fire the flash lamp at a preset time and after a time interval of about 200 µs, the second signal follows to fire the Q-switch. Upon receiving the second signal, the laser is fired typically around 850 ns. Two energy modes, LOW and HIGH, with variable scale range from 0-100 are available for controlling the maximum transmitted energy. Since the light reaches the camera via a strong neutral density filter of optical density 3, the laser is operated at its highest transmission i.e HIGH 100. The continuous laser used is a Spectra-Physics BeamLok 2060 argon-ion with an output power of 10 mW.

4.4. Trigger unit and sensors

Once the charges are released, the events that follow are very fast requiring an accurate and reliable system to trigger the equipments. The triggering unit consists of a high voltage probe and a digital delay generator (figure 4.8). The high frequency high voltage probe is a HVP-39 pro
model with a very short rise time of 1.6 ns and can handle up to 40 kV. It has a division ratio of 1000:1, meaning the output voltage at its maximum (40 kV) is 40 V. Since the capacitor is mostly charged around 10-13 kV, its output voltage of 13 V acts as the main trigger signal. One of the terminals of the probe is connected between the 8 coaxial cable assembly and the EW while the other is grounded. The probe’s sensor lead is connected to the trigger terminal of a Stanford Research Systems DG-535 delay generator, consisting of 4 digital delay outputs which in turn triggers the pulsed laser or the high speed camera depending on the set-up. Since the charges released from the capacitor reach the wire almost instantly, the only uncertainty revolves around the probe’s sensor to report the potential difference. But due to its very fast rise time, this way of triggering using the probe was found to be highly repeatable with an uncertainty of $< 1 \mu s$.

![Figure 4.8: (a) HVP-39 pro high voltage probe (b) Stanford Research Systems DG-535 digital delay generator.](image)

The shock pressure profiles are recorded by mounting three PCB piezoelectric sensors (113B24, PCB Piezotronics) at desired locations. The diaphragm diameter of the sensor is 5.54 mm and it has a response time of 1 $\mu$s with a frequency response of $\leq 100$ kHz. The sensors are driven by a 482C series signal conditioner, connected through a 10-32 coaxial jack. The signal conditioner has 4 individual channels and supply a regulated 26 VDC at 2-20 mA excitation voltage to the connected sensors. The output signal from each channel of the signal conditioner is read by connecting it to an oscilloscope.
4.5. Blast wave formation

The formation and transformation of the initially cylindrical wave to plane blast wave is demonstrated using numerical density gradient images in figure 4.9, taken along the exploding chamber cut-section view. The real wire explosion is numerically simulated by replacing it with an equivalent high pressure region. The value of this high pressure region here is chosen by trial-and-error method to represent a 12 kV, 74 mm channel width case by juxtaposing the obtained numerical pressure profiles with the experimental trace from the pressure sensors. The numerical simulation method will be presented in Sec 6. The wire is mounted 3 mm away from the walls and the generated cylindrical wave is captured in figure 4.9(a). Due to the confined rectangular geometry, this cylindrical wave undergoes multiple reflections at the top, bottom and back walls. This complicated process, also responsible for the absence of negative phase in the blast wave profile, is shown in figure 4.9(b)-(f). The emergence of the second wave reflected from the back wall can be seen clearly in figure 4.9(g)-(h). Since the second wave propagates in an environment worked by the first wave, it travels relatively faster and eventually both the waves coalesce into one plane wave as shown in figure 4.9(k). This plane wave then propagates into the test chamber.
Table 4.1: Shock Mach number measured at 200 mm from EW as a function of charging voltage and length of wire for a 0.4 mm diameter copper wire.

<table>
<thead>
<tr>
<th>Length of wire (mm)</th>
<th>Voltage (kV)</th>
<th>Energy stored (J)</th>
<th>$M_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>74</td>
<td>10</td>
<td>300</td>
<td>1.6</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>432</td>
<td>2.15</td>
</tr>
<tr>
<td></td>
<td>13</td>
<td>507</td>
<td>2.4</td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>675</td>
<td>3.4</td>
</tr>
<tr>
<td></td>
<td>17</td>
<td>867</td>
<td>3.8</td>
</tr>
</tbody>
</table>

Since this being a blast wave, its intensity reduces as it propagates through the channel. For a given capacitance, the strength of the wave at a particular location depends on the charging voltage, the length and diameter of the wire. Several tests were performed and based on the charge left in the capacitor after the explosion and physical observations of the exploded wire residues in the channel, a 0.4 mm diameter copper wire was deemed optimal for this set-up. The summary of charging voltages, the energies stored and the shock Mach number measured at
around 200 mm from the EW for a 0.4 mm diameter copper wire are as shown in Table 4.1.

Figure 4.11: Experimental pressure profiles measured by two sensors for (a) 74 mm wire length and (b) 10 mm wire length; (c) Numerical pressure drop for both wire lengths as a function of propagation distance \(x\) and (d) Corresponding shock Mach number as a function of \(x\).

The transformation for a 10 mm wire length is shown in figure 4.10. The corresponding shock profiles measured using two shock sensors for a 12 kV charged, 74 mm and 10 mm wire length is as shown in figure 4.11(a)-(b). Notable differences are the leading shock strength and the rate of decay of pressure behind it. While the 74 mm case is characterised with a relatively weaker leading shock with moderate decay, the 10 mm case is characterised with a strong shock and very steep decay. So for a constant initial voltage, the reduction in length of the wire induces two compromising effects: (a) increases the leading shock strength and, (b) increases the rate of decay. The rate of decay is directly related to the expansion waves ‘eating into’ the leading shock thereby reducing its
strength. This effect for the overall propagation distance \( x \) is plotted in figure 4.11(c)-(d) as a function of pressure and shock Mach number respectively.
Chapter 5

Inhomogeneity creation techniques

The crux of the present work relies on the ability to generate good, repeatable, geometrically well-defined inhomogeneities. Each problem investigated here required a new solution to break through the dominantly used techniques in their corresponding area. The procedure used to create the inhomogeneities in the test chamber are described in this chapter.

5.1. Water column

Although various techniques have been employed previously to produce water droplets/columns (Nicholls & Ranger 1969; Hirahara & Kawahashi 1992; Hsiang & Faeth 1995; Joseph et al. 1999; Igra & Takayama 2001), the diameter of the column is limited to a maximum of 7 mm. When the interest is to study in detail the wave motions inside the column along with its associated instabilities, columns of larger diameter are a necessity. Also with such small diameter, no measurements inside the column are possible. But creating such large columns posed a problem as water tend to splash sideways resulting in non-uniform shapes. Also as only one high spatial image per column can be obtained, it was also important to produce them with good repeatability. This required an external force that does not interfere with the flow or optics, to hold the column in place. To overcome this issue, a technique of using super-hydrophobic coating on the windows was implemented. Hydrophobicity is a property of a molecule to repel water. When such materials are coated on a surface, it becomes a superhydrophobic surface where the water contact angle exceeds $150^\circ$ primarily due to surface tension effects (eg. water on lotus leaf).

A superhydrophobic paint was specifically chosen with a hydrophobicity level that just about prevents the splashing of water and is also transparent enough to pass light. The coating is applied on a limited area on the bottom window and the coating procedure is as follows: An O-ring of 22 mm diameter is placed at a suitable location on the bottom window and is pressed firmly using a solid mass (called as O-ring mould).
and lever as shown in figure 5.1(a). The superhydrophobic paint is then coated around the area surrounding the O-ring using a brush. Since the O-ring is firmly pressed, excess paint collects on the outer circumference of the O-ring and does not seep through it. It is then allowed to dry rapidly using an air gun for approximately 30 minutes. Upon carefully removing the O-ring, a perfectly circular rim of hydrophobic coating is formed as shown in figure 5.1(b). Now the top window is placed in position and rigidly secured. Note that the top window is not treated with any coating. A syringe with its top end modified to fit a small, long tube is filled with water and then inserted through a small opening from the back end flange of the test chamber. The water is then carefully and very slowly injected into the uncoated area, thereby creating a cylindrical water column of 22 mm diameter as shown in figure 5.1(d). A slightly larger wet area is observed in figure 5.1(c) at the bottom due to weight of the water. Nevertheless, the variation in diameter is less than 2%
indicating a geometrically well-defined cylindrical water column. The thickness of the coating is negligible compared to the thickness of the channel. The paint just about holds the column in place and even slight vibrations to the test unit would allow the water to flow over the barrier. This procedure of creating columns worked really well that during the course of this experiment, a total of 150-200 columns were prepared with excellent repeatability.

5.2. Foam barrier(s)

Aqueous foam is a two-phase system where gas bubbles are enclosed within thin liquid (usually water) films (Weaire & Hutzler 2001). On the basis of volume liquid fraction ($\alpha$), defined as $\alpha = V_l/V_f$ where $V_l$ and $V_f$ are the volumes of liquid content and foam sample respectively, foams are generally classified as dry foams ($\alpha < 0.05$) where the shape of the gas bubbles resemble polyhedrons with very thin films; and wet foams ($0.05 < \alpha < 0.36$) where the bubble shape approaches a circle. For $\alpha > 0.36$, the bubbles will no longer be in contact with its neighbours thereby becoming a bubbly liquid (Britan et al. 2007). Mechanical properties of the foam mainly depend on the liquid volume fraction and size distribution of the bubbles. When being impacted by very strong shocks, dry foam gets shattered into fine liquid droplets with transfer of momentum and heat occurring between the gas and liquid phases. These processes result in flow non-equilibrium. But wet foams are characterised with smaller bubbles where the net momentum and heat transfer between the liquid and gaseous phases are negligible. The relaxation effect becomes less pronounced resulting in rapid flow equilibrium (Britan et al. 2013). This makes numerical modelling relatively simple as the flow can be represented by a single component approach since both the phases are in dynamic and thermal equilibrium which is in contrary to dry foams where multiphase models are a must for accurate prediction (Britan et al. 2014). Also for applications concerning blast mitigation by absorption of energy, wet aqueous foams comprised of fine bubbles with significantly larger liquid total surface area is advantageous as water, owing to its large specific heat and latent heat of vaporization, is one of the best substance for absorbing energy.

For creating the foam barrier(s), a conventional shaving foam from Gillette, Procter & Gamble was used for the following reasons (Liverts et al. 2015):

- Wet aqueous nature of the foam with a density of $\approx 100kg/m^3$ ($\alpha = 0.1$).
• No pre-preparation required as it was readily available commercially.
• Microscopic observation of the foam revealed a uniformly distributed, fine polydisperse bubbles with an average diameter of 50 µm.
• The foam was stable and $\alpha$ remained constant for close to 2 hour of foam aging.

Figure 5.2: (a) Shadowgraph image showing broken shock structure due to interface instability in a 74 mm width foam barrier; (b) Normal still camera image showing the reduction of width by insertion of POM rails and the created foam barrier; (c) Schematic drawing of the test section with POM rails; (d) The conditions before (top) and after (bottom) incident shock impact with a 10 mm width foam barrier. The stabilised straight shock wave structure is indicated in the bottom shadowgraph image.

In order to form a barrier of specified length $l$, two T-inserts of height 5 mm and length 10 mm are placed at a distance $l$ between them. Foam is then uniformly filled within the space and excess foam on the top is wiped away. Care was taken to properly fill the foam without any voids. The T-inserts are then lifted in one swift motion to ensure a plane uniform interface. The most important criteria for creating the foam barrier was to maintain straight, smooth interface particularly in the downstream to avoid RM instability. However initial efforts proved futile as to no matter how carefully the barrier was created, it still resulted in non-uniform broken shocks emerging from the foam due to RMI [figure 5.2(a)]. For the sake of analysis, a relatively uniform shock
was expected to emerge from the foam. Since this being a 1D study, this problem was then alleviated by reducing the initial width of the channel to 10 mm by inserting POM rails [figure 5.2(b)-(c)]. Due to this reduced width, the shock transmitted from the foam quickly stabilised to form a uniform shock front as seen in figure 5.2(d). This technique was also suitable for creating multiple barriers.

5.3. Density inhomogeneity

Shock accelerated inhomogeneous flows were traditionally explored using cylindrical/spherical bubbles (from herein referred to as circular inhomogeneity). Techniques like spark discharge (Rudinger & Somers 1960), soap film (Haas & Sturtevant 1987), fine jet (Jacobs 1992) etc., were predominantly used in creating single or multiple circular inhomogeneities. Despite the shear volume of progress, experimental investigation on shapes other than circle are very scarce.

What is an inhomogeneity? Simply put, any collection of matter which is not uniform or consistent with the surrounding mass is defined as an inhomogeneity. A simplest way to create such difference is just by introducing heat. For example when gas is heated, its density drops and becomes lighter thereby creating an inhomogeneous system. Markstein (1957) in his pioneering study, utilized a reactive flame front to create an oblong shaped bubble. Similarly Rudinger & Somers (1960) also used heat to create a density inhomogeneity. Inspired from their works, a new technique of heating the surrounding gas on the basis of ‘Joule heating’ is implemented here to create elliptically shaped inhomogeneities. The setup is as shown in figure 5.3(a)-(b). For creating straight inhomogeneities, two brass electrodes of 3 mm diameter with one being solid (anode) and the other hollow (cathode) are screwed into the bottom plexiglas window at a distance of 145 mm between them. The anode is mounted at 200 mm from the wire explosion plane and cuts through the entire 5 mm channel while the cathode is positioned at half height. A variable external power supply is connected across the electrodes. A 0.2 mm diameter copper wire is suspended in the middle of the 5 mm channel with its one end tied rigidly at the anode while the other end is connected to a freely suspended weight after passing it through a guide cylinder and the hollow cathode. As the wire will expand when subjected to heat, the guide cylinder and the freely suspended weight makes sure that it is always suspended in the middle of the channel by maintaining it in a state of tension. The wire is then subjected to 15-24 V, 2.8-3.6 A current resulting in heating of the wire. This in turn heats the surrounding air, producing a density inhomogeneity. Heat from the wire will be
emanating cylindrically until it reaches the top and bottom plexiglas windows. When the plexiglas surface comes into contact with heat, it becomes soft and pliable at the contact region. This affects its optical properties thereby making it partially opaque as indicated by the dark region (termed ‘Hot gas’) in figure 5.3(c). The effective thickness of the dark region (or the inhomogeneity) is influenced by the amount of time the wire is subjected to conduct current. Figure 5.3(c) shows heat spread as a function of current supplied time for a 24 V, 3.6 A current. After about 60 s, the heat has spread to a thickness of \( \approx 10 \text{ mm} \) with an overall shape resembling an ellipse. Inclined elliptic inhomogeneities are created using similar mechanism but by placing the the anode and cathode at an angle to the incoming shock front.
Chapter 6

Numerical simulation method

Numerical simulations, performed to complement the experimental data, play a vital role by providing assistance in interpreting and analyzing the flow features observed in experiments. It also provides practically unlimited level of details of the results which are difficult to achieve in experiments. The numerical simulations in this work are performed either using a commercial CFD package (StarCCM+) or one of the two in-house codes solving compressible Euler and Navier-Stokes equations respectively. The first in-house code solving the compressible Euler equation is based on an artificial upstream flux vector splitting scheme (AUFS) while the other is based on a hybridised AUFS scheme with Roe’s solver (AUFSR). Each method exhibit certain strengths suitable for the class of problems investigated in this work. A description of the in-house codes are provided in this chapter.

6.1. Foam - AUFS

Wet foams are characterised with smaller bubbles where the net momentum and heat transfer between the liquid and gaseous phases are negligible resulting in rapid flow equilibrium. Here the flow can be represented by a single component approach as both the phases are in dynamic and thermal equilibrium. As the main focus is on the macroscopic characteristics of the foam, the strategy to simulate foam problem is by treating it using a pseudo-gas approach in which it is considered as a perfect gas with high molecular weight incorporating an effective adiabatic index ($\gamma$) for simulating the real water content in foam (Liverts et al. 2015). The description of the flow was based on liquid volume fraction ($\alpha$) formulation based algorithm which provides an approximate treatment for mixture of fluid component within a grid cell (Shyue 2006). The full set of compressible Euler equations along with the $\alpha$ based equation solved are as follows:

$$\mathbf{U}_t + \mathbf{F}_x = 0 \quad (6.1)$$
where $\mathbf{U}$ and $\mathbf{F}$ are vectors of conservative quantities and fluxes, respectively,

$$
\begin{align*}
\mathbf{U} &= \begin{pmatrix} \rho \\ \rho u \\ \rho E \\ \rho \alpha \end{pmatrix}, \\
\mathbf{F} &= \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho E u + pu \\ \rho u \alpha \end{pmatrix}
\end{align*}
$$

(6.2)

$\rho$ is the density, $u$ is the particle velocity, $p$ is the pressure and $E$, the total energy per unit mass given as the sum of specific internal energy and the kinetic energy $E = e + u^2/2$. The foam density is a mixture of density of water ($\rho_w$) and air ($\rho_a$), and is calculated using $\rho = \alpha \rho_w + (1 - \alpha) \rho_a$. The water density is set to be incompressible with a constant value of 1000 kg/m$^3$. The system of equations are completed by modeling the following Noble-Abel equation of state:

$$
p(\rho, e) = \left( \frac{\gamma - 1}{1 - \alpha} \right) \rho e
$$

(6.3)

The adiabatic index ($\gamma$) of the fluid mixture is updated using the following equation:

$$
\frac{1}{\gamma - 1} = \frac{\alpha}{\gamma_w - 1} + \frac{1 - \alpha}{\gamma_a - 1}
$$

(6.4)

The equations are solved using the AUFS method, proposed by Sun & Takayama (2003), which is a special flux vector splitting scheme. It introduces two artificial wave speeds into the flux decomposition for adjusting the direction of the wave propagation. Its strength lies in exactly resolving pure entropy waves without the associated carbuncle problem (instabilities in shock capturing) and it is a simple, accurate and robust upwind scheme. The flux vector $\mathbf{F}$ can be rewritten as,

$$
\mathbf{F} = u \mathbf{U} + \mathbf{P}
$$

(6.5)

where $\mathbf{P} = (0, p, pu, 0)^T$. Marching explicitly through time, the discretised governing equation yields in conservation form:

$$
\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{\Delta t}{\Delta x} \left( \mathbf{F}_{i+\frac{1}{2}} - \mathbf{F}_{i-\frac{1}{2}} \right)
$$

(6.6)

where $\Delta t$ is the time step and $\Delta x$ is the grid size. The numerical flux vector $\mathbf{F}_{i+\frac{1}{2}}$ is defined by the left and right cells. The fundamental idea of the AUFS scheme is to split $\mathbf{F}$ into,
\[ F = (1 - S)F_1 + SF_2 \] (6.7)

Here \( F_1 \) and \( F_2 \) are the intermediate flux vectors. Eq. (6.7) reduces to Eq. (6.5) when \( S = s_1/(s_1 - s_2) \), where \( s_1 \) and \( s_2 \) are the artificially introduced wave speeds. Depending on the value and sign of \( s_1 \), \( F_1 \) and \( F_2 \) are calculated from,

\[ F_1 = \frac{1}{2}(P^L + P^R) + \delta U \] (6.8)

\[ F_2 = U^\zeta (u^\zeta - s_2) + P^\zeta, \zeta = \begin{cases} L & \text{for } s_1 > 0 \\ R & \text{for } s_1 \leq 0 \end{cases} \] (6.9)

The artificial viscosity \( \delta U \), determined from Steger-Warming formula is represented as,

\[ \delta U = \gamma \frac{2 \bar{c}}{2\bar{c}} \left( \frac{p^L - p^R}{(pu)^L - (pu)^R} + \frac{\frac{1}{2}((pu)^2)L - (pu)^2R}{\alpha(p^L - p^R)} \right) \] (6.10)

where \( \bar{c} \) is the intermediate speed of sound given by \( c^2 = \gamma p / \rho(1 - \alpha) \). The numerical values of \( s_1 \) and \( s_2 \) are chosen as

\[ s_1 = (u^L + u^R)/2 \]

\[ s_2 = \begin{cases} \min(0, \min(u^L, u^R) - \max(c^L, c^R)) & \text{for } s_1 > 0, \\ \max(0, \max(u^L, u^R) + \max(c^L, c^R)) & \text{for } s_1 \leq 0 \end{cases} \] (6.11)

The final intercell flux is given as,

\[ F = (1 - S)\left[ \frac{1}{2}(P^L + P^R) + \delta U \right] + S\left[ U^\zeta (u^\zeta - s_2) + P^\zeta \right] \] (6.12)

The numerical geometry creation, mesh generation and the solver were coded using Matlab. Since this being a 1D problem, the domain was meshed with 36000 grid points with a uniform spacing of \( \Delta x = 0.01 \text{ mm} \) and run on a single desktop computer. The calculations were quite cheap but since this being an explicit scheme with relatively small stability bounds, the time step was chosen to fall strictly within the Courant number limit. A detailed description of the domain, initial and boundary conditions along with the validation of the code are found in Paper 3.
6.2. Density inhomogeneity - AUFSR

Compressible flow problems often involve complex flow phenomena, like strong shocks, shock wave refraction and reflection, shock-shock interaction and shear layers. While the AUFS is an efficient method for solving compressible inviscid flows, the scheme yields large values of viscosity in the presence of shear waves, leading to poor resolution of boundary layers Kemm (2015). This is mainly due to the choice of artificial viscosity $\delta U$ in Eq. (6.10). For the density inhomogeneity problem where shear induced instability is expected, solving a full set of 2D compressible Navier-Stokes equations given in Eq. (6.13) using the AUFS scheme as such will produce unacceptable results.

\[
U_t + (F_a + F_b) - (G_a + G_b) = 0 \quad (6.13)
\]

\[
U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}; \quad F_a = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho E u + p u \end{bmatrix}; \quad F_b = \begin{bmatrix} \rho v \\ \rho v^2 + p \\ \rho u v \\ \rho E v + p v \end{bmatrix} \quad (6.14)
\]

\[
G_a = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u \tau_{xx} + v \tau_{xy} - q_x \end{bmatrix}; \quad G_b = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ u \tau_{xy} + v \tau_{yy} - q_y \end{bmatrix} \quad (6.15)
\]

where

\[
\tau_{xx} = 2\mu u_x - \frac{2}{3}\mu(u_x + v_y); \quad \tau_{yy} = 2\mu v_y - \frac{2}{3}\mu(u_x + v_y); \\
\tau_{xy} = \mu(u_y + v_x); \quad q_x = -\kappa T_x; \quad q_y = -\kappa T_y \quad (6.16)
\]

Here $U$ represent the vectors of conservative quantities, $F_a$ and $F_b$ the vectors of inviscid flux, and $G_a$ and $G_b$ the vectors of viscous flux in $x$ and $y$ direction respectively. $T$, $\mu$, $\kappa$ denote temperature, dynamic viscosity and thermal conductivity respectively. $\mu$ and $\kappa$ are temperature dependent and are calculated using the following relations:

\[
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{1.5} \frac{T_0 + 110}{T + 110} \quad (6.17)
\]

\[
\kappa = 1.527e^{-11}T^3 - 4.8574e^{-08}T^2 + 1.0184e^{-04}T - 0.00039333
\]

Given an interface with normal vector $n = (n_x, n_y)$, $U$ and $F$ can be expressed in the direction of normal face $n$ as,
6.2. Density inhomogeneity - AUFSR

\[ \begin{bmatrix} \rho \\ \rho q_n \\ \rho q_t \\ \rho E \end{bmatrix}; \begin{bmatrix} \rho q_n \\ \rho q_n^2 + p \\ \rho q_n q_t \\ \rho E q_n + pq_n \end{bmatrix} \]  

(6.18)

where the normal \((q_n)\) and tangential velocity \((q_t)\) through the interface is given as,

\[ q_n = u_n x + v n y; \]
\[ q_t = v n x - u n y \]  

(6.19)

The system of equations are closed by modeling the ideal gas law. Formulations of flux difference splitting (FDS) schemes, like Roe’s scheme, are proven to accurately resolve shock waves with the capability to correctly deal with density or shear layers but suffers from carbuncle problem (MacCormack 2011) and also sometimes lead to unphysical flow solutions. Tchuen et al. (2014) proposed a new scheme that combines the efficiency of AUFS with the accuracy of FDS approach by hybridising the AUFS flux and Roe flux. The scheme, termed AUFSR, utilizes the modified wave speed approach from AUFS and applies it to Roe solver, thereby introducing numerical dissipation to shear waves while negating the carbuncle problem. By this, the AUFSR scheme becomes not only robust for shock capturing but also accurate for resolving shear layers.

Basically the AUFSR follows a similar path until Eq. (6.7) where \(F_1\) and \(F_2\) for a 2D system are given as,

\[ F_1 = \frac{1}{2} (P^L + P^R) + \delta U \]  

(6.20)

\[ F_2 = U^\zeta (q_n^\zeta - s_2) + P^\zeta, \zeta = \begin{cases} L & \text{for } s_1 > 0 \\ R & \text{for } s_1 \leq 0 \end{cases} \]  

(6.21)

The significance of the scheme appears in the formulation of \(\delta U\) which is defined on the basis of Roe-averaged values,
\[ \rho = \sqrt{\rho_L \rho_R}; \quad \bar{u} = \frac{\sqrt{\rho_L u_L} + \sqrt{\rho_R u_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}; \]

\[ \bar{v} = \frac{\sqrt{\rho_L v_L} + \sqrt{\rho_R v_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}; \quad q_n = \frac{\sqrt{\rho_L q_n^L} + \sqrt{\rho_R q_n^R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}; \]

\[ \bar{H} = \frac{\sqrt{\rho_L H_L} + \sqrt{\rho_R H_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}; \quad \bar{c} = \left[ (\gamma - 1) \left( \bar{H} - \frac{1}{2} \bar{u}^2 \right) \right]^{\frac{1}{2}} \quad (6.22) \]

expressed as,

\[ \delta \mathbf{U}^{AUFSR} = \frac{1}{2} \left( \begin{bmatrix} (q_n^L - s_1)\rho_L^L + (q_n^R - s_1)\rho_R^R \\ (q_n^L - s_1)(\rho u)_L^L + (q_n^R - s_1)(\rho u)_R^R \\ (q_n^L - s_1)(\rho v)_L^L + (q_n^R - s_1)(\rho v)_R^R \\ (q_n^L - s_1)(\rho E)_L^L + (q_n^R - s_1)(\rho E)_R^R \end{bmatrix} \right) - \frac{1}{2} \sum_{k=1}^{4} |\lambda_k| \eta_k R_k \quad (6.23) \]

Here \( \lambda_k, \eta_k, \) and \( R_k \) are the eigenvalues, characteristic variables and eigenvectors respectively. For this case,

\[ \lambda_k = (\bar{c}, 0, 0, \bar{c}) \]

\[ \eta_{1,A} = \frac{(P_R^R - P_L^L) \mp \bar{c}(q_n^R - q_n^L)}{2 \bar{c}^2} \]

\[ R_{1,A} = \begin{pmatrix} 1 \\ \bar{u} \mp \bar{c} \\ \bar{v} \mp \bar{c} \\ \bar{H} \mp q_n \bar{c} \end{pmatrix} \quad (6.24) \]

The numerical values of \( s_1 \) and \( s_2 \) are set as,

\[ s_1 = \bar{q}_n \]

\[ s_2 = \begin{cases} \min(0, q_n^L - c^L, \bar{q}_n - \bar{c}) & \text{for } s_1 > 0, \\ \max(0, q_n^R + c^R, \bar{q}_n + \bar{c}) & \text{for } s_1 \leq 0 \end{cases} \quad (6.25) \]

The final intercell flux through the interface is expressed as,

\[ \mathbf{F} = (1 - S) \left[ \frac{1}{2} (\mathbf{P}^L + \mathbf{P}^R) + \delta \mathbf{U}^{AUFSR} \right] + S \left[ \mathbf{U}^\xi(q_n^\xi - s_2) + \mathbf{P}^\xi \right] \quad (6.26) \]

Depending on the case studied, the domain was meshed with 2.5 - 5.5 million unstructured triangular cells refined along the zone of interest
with the mesh size varying slowly and smoothly as we go outwards. The overall smallest and the largest cell size were 0.025 mm and 0.15 mm respectively. The calculations were expensive and memory consuming that it was run on resources provided by the Swedish National Infrastructure for Computing (SNIC) at High Performance Computing Center North (HPC2N) requesting a RAM allocation of 90 GB. A detailed description of the domain, initial and boundary conditions along with the validation of the code are found in Paper 5-6.
Contributions and Conclusions

The study began with the construction of an Exploding Wire (EW) facility in which the experiments presented in the papers in Part II were performed. The facility had the capacity to generate strong blast waves in atmospheric conditions with shock Mach number typically varying from 1.75 - 4.8 and also provided flexibility for altering the test geometry depending on the case investigated. The shock wave-multiphase interaction and its associated flow field was captured optically using shadowgraph technique. The experiments were always accompanied with numerical simulation, performed using Star-CCM+ CFD package and two in-house numerical schemes, to better understand and analyze the distinctive flow features observed. A short and concise summary of the contributions and conclusions drawn from the investigations are presented in this section.

7.1. Water column

The studies were conducted on a 22 mm diameter water column impacted with a blast wave of shock Mach number 1.75-2.4. Images were captured with both the Nikon D90 single-shot camera for spatial resolution and Shimadzu HPV-X2 high speed camera that was set to capture 1 million frames per second for temporal resolution. The concluding remarks are as follows:

- A new technique of treating the bottom transparent plate with super-hydrophobic coating proved powerful and effective in creating highly repeatable, properly defined, large diameter water columns. Normal tap water was used for creating the columns.
- Upon impact, the incident shock wave was reflected, diffracted and transmitted at the upstream interface boundary. The reflection and diffraction were similar in nature to that of a shock interaction with a solid cylindrical wall. The transmitted wave was a shock wave seen propagating through the water column.
On reaching the downstream boundary of the water column, the transmitted shock wave was reflected back as an expansion wave. Due to the convex nature of the reflection, the expansion wave naturally focused at a point before travelling upstream. This focusing of expansion wave resulted in negative pressures initiating heterogenous cavitation. The cavitation bubble cloud was observed for shock impact with Mach number 2.4 and not for Mach number 1.75 case suggesting that the threshold negative pressures were not reached.

High speed camera images obtained for Mach number 2.2 revealed that cavitation was initiated during the converging path of the expansion wave. Time taken to complete all dynamical processes like cavitation bubble nucleation, growth and its subsequent collapse was 35 $\mu$s.

7.2. Foam

The efficiency of well characterized aqueous foam barrier configurations on the attenuation of strong blast waves was investigated. The foam barriers were impacted with a blast wave of shock Mach number 4.8. A PCB piezoelectric pressure sensor was placed at the target location to measure face-on pressure. Numerical simulation was performed through an Euler equation model that treats the foam using a pseudo-gas approach. The concluding remarks are as follows:

- The dominant mechanism responsible for the reduction of the peak pressure of the shock wave propagating in foam was the ‘catching up’ of the rarefaction wave with the wave front resulting in constant re-shaping of its profile. This process proved particularly effective in foams due to its very low speed of sound of $\approx 40$ m/s. This provided sufficient time for the rarefaction wave to reduce the energy of the wave front at shorter distances. Additional reduction of peak pressure was also observed due to ‘impedance mismatch factor’ at the foam-air interface.

- For various configurations with single foam barrier, both the mechanisms provided a combined pressure reduction in the range of 86% - 96.5%. The advantageous effect of the impedance mismatch factor was exploited by splitting the barriers into two so as to provide additional interfaces. For same thickness, this resulted in relatively large pressure reduction as compared to single foam barrier. For various configurations with twin foam barriers, an overall peak pressure reduction in the range of 96.5% - 97.2% was obtained.
7.3. Density inhomogeneity

The unstable evolution of an elongated elliptically shaped inhomogeneity aligned both normal and at angle to the incoming blast wave of Mach 2.15 was investigated. The 2D inhomogeneities were generated by heating the air around a current conducting wire suspended between two electrodes. Through the positioning of electrodes, this technique provided the flexibility to generate inclined inhomogeneities. Images were again captured using both the single-shot and high speed camera that was set to capture at 0.5 million frames per second. Numerical simulation was performed by solving a 2D Navier-Stokes equation with air as test gas. The concluding remarks are as follows:

- Due to the high speed of sound in the inhomogeneity, the transmitted shock wave travelled relatively faster than the diffracted incident shock. Both shocks were connected to each other through a precursor shock (a wedge shaped disturbance). At the junction of the precursor shock and the incident shock, a quadruple point was initiated that additionally consisted of the Mach stem and a reflected shock.

- Concerning the straight case, the flow was symmetric about the inclination axis and the ‘broadening’ of the pressure profile along the center plane was identified to be caused by three physical features namely shock jump, precursor region and vorticity induced flow. The structures observed were similar to that of a 2D circular bubble where two counter-rotating vortices were formed primarily due to RMI. The variation of normalized width, length and circulation of the vortices with both distance and time were found to be linear. Also the normalized circulation was independent of inhomogeneity density and thickness thereby enabling formulation of a unique linear fit equation.

- Regarding the inclined case, although the transmitted shock adjusted itself to travel normal to the inhomogeneity, the precursor shock regions were asymmetric about the inclination axis. For the case studied here, the size of the top precursor region was larger than the bottom precursor region. Most striking observation was the vortex trains generated primarily due to KHI phenomenon. The initial velocity of each individual vortex in the train was dependent on the strength of the transmitted wave and it was observed to lose its velocity linearly with time as it grows in size. The normalized circulation followed a non-unique quadratic function influenced by two factors: the reduction in strength of the transmitted shock thereby generating vortices with reduced
vorticity along with the gradual loss of vorticity of the earlier generated vortices.

- The qualitative features observed in both cases were largely unaffected by temperature variations and shock curvature phenomenon.
Chapter 8

Papers and authors contribution

The main advisor for the project is Prof. Nicholas Apazidis (NA). Dr. Michael Liverts (ML) and Dr. Nills Tillmark (NT) acts as co-advisors.

**Paper 1**

*Plane shock wave interaction with a cylindrical water column.*

S. Sembian (SS), M. Liverts (ML), N. Tillmark (NT) & N. Apazidis (NA)


SS set-up and performed the experiments with assistance from ML and NT. Numerical calculations were planned and performed by SS. The obtained data was analysed by SS with guidance from ML, NT and NA. The paper was written by SS with feedback from ML, NT and NA. Parts of this work have been published in or presented at:

*On the refraction of shock wave by a cylindrical water droplet*

S. Sembian, M. Liverts, N. Tillmark, N. Apazidis

30th **International Symposium on Shock Waves**

19-24 July 2015, Tel Aviv, Israel.

**Paper 2**

*An experimental time-based analysis and numerical parameter study on shock-water column interaction.*

S. Sembian (SS), M. Liverts (ML) & N. Apazidis (NA)

*manuscript in preparation.*

The experiments and numerical simulations were set-up and performed by SS. The paper was written by SS with feedback from ML and NA. Parts of this work have been published in or presented at:

*On the focusing of expansion wave due to shock interaction with*
8. Papers and authors contribution

**cylindrical water column**

S. Sembian, M. Liverts and N. Apazidis

22nd **International Shock Interaction Symposium**

4-8 July 2016, Glasgow, Scotland.

**Paper 3**

*Attenuation of strong external blast by foam barriers.*

S. Sembian (SS), M. Liverts (ML) & N. Apazidis (NA)


SS set-up and performed the experiments with assistance from ML and NA. Numerical calculations were planned and performed by SS. Analysis of the data and the preparation of figures used in the paper was done by SS with guidance from ML and NA. The paper was written by SS with feedback from ML and NA.

**Paper 4**

*Plane blast wave propagation in air with a transverse thermal inhomogeneity.*

S. Sembian (SS), M. Liverts (ML) & N. Apazidis (NA)


The experiments and numerical simulations were set-up and performed by SS. The obtained data was analysed by SS with guidance from ML and NA. The paper was written by SS with feedback from ML and NA. Parts of this work have been published in or presented at:

*Revisiting shock propagation through temperature gradients*

S. Sembian, M. Liverts and N. Apazidis

31st **International Symposium on Shock Waves**

9-14 July 2017, Nagoya, Japan.

**Paper 5**

*Plane blast wave interaction with an elongated straight and inclined heat-generated inhomogeneity.*

S. Sembian (SS), M. Liverts (ML) & N. Apazidis (NA)


The experiments and numerical simulations were set-up and performed by SS. The results were analysed by SS with guidance from ML and NA. The paper was written by SS with feedback from ML and NA.
Acknowledgements

To my mom Mrs. Kalaiselvi: this is all because of your sacrifices. Many Thanks!

The project was financially supported by Vetenskapsrådet (The Swedish Research Council) and travel grant for participation in international conferences was provided by Stiftelsen Erik Petersohns minne. The numerical simulations were performed on resources provided by the Swedish National Infrastructure for Computing (SNIC).

I would like to express my sincere gratitude to my supervisor Prof. Nicholas Apazidis for giving me the opportunity to do research under his guidance. A PhD student’s life has its ups and downs and it was his motivation and positive thinking that pushed me forward through those hard times. Whenever a need for a discussion arose, personal or professional, he was always there and that includes some very late evenings as well. Thanks to him, I had complete freedom and never felt pressure at any point of time.

I owe a great many thanks to my co-supervisor Dr. Michael Liverts for moulding me into what I am today. We had many friendly arguments which played a huge role in my development. His critical comments made me dig a little deeper that made the work thorough. His assistance in reviewing many of my manuscripts is highly appreciated. I would also like to thank my previous co-supervisor Dr. Nills Tillmark for his invaluable assistance in the set-up of the experimental apparatus and also for the many discussions during my initial stages. I am grateful to him for sharing his broad knowledge and experience on experimental techniques.

I would like to thank Prof. Jens Fransson and Dr. Bengt Fallenius for providing a pleasant lab atmosphere. The technical and administrative staff have also been of great support. Every experimental work requires skilled technicians for crafting the components and in that respect I
thank Joakim Karlström, Jonas Vikström and Rune Lindfors for their expertise.

As a newbie coming from a region with different work culture, its natural to get intimidated or overawed at a new place. In that respect, my previous officemate Ramin and coworkers Renzo, Sohrab, Marco, Julie, Sissy, Karl, Mathias (all are titled Dr.) and Chetan are gratefully acknowledged for their warm and welcoming demeanour that made the transition instant. My current officemate Krishne also deserves a special mention for all those interesting discussions on anything under the sun and sometimes even beyond that during our spare time. I would also like to extend my thanks to current colleagues for their support and a delightful working environment.

We always save the best for the last and no words would suffice my heartfelt gratitude to my mom, sister, wife and daughter. Being my audience, they bore the brunt of a constantly nagging upcoming fluid dynamist! Without their unconditional love and support, this would not have been possible. Special thanks to my beloved wife Nandhini, for taking care of everything at home that gave me the liberty to focus on work alone. And my dear daughter Saashini, appa is now ready to play gugu!
Bibliography


Markstein, G. H. 1957 Flow disturbances induced near a slightly wavy contact surface, or flame front, traversed by a shock wave. J. Aerosp. Sci. 24, 238.


Nairne, E. 1774 Electrical experiments by mr. edward nairne, of london, mathematical instrument-maker, made with a machine of his own workmanship, a description of which is prefixed. Philosophical Transactions (1683-1775) 64, 79–89.


Singer, G. J. & Crosse, A. 1815 Xxi. account of some electrical experiments by m. de nelis, of malines in the netherlands: with an extension of them. The Philosophical Magazine 46 (209), 161–166.


Part II

Papers
Paper 1
Plane shock wave interaction with a cylindrical water column

S. Sembian, M. Liverts, N. Tillmark, N. Apazidis
Department of Mechanics, KTH - Royal Institute of Technology, SE-100 44 Stockholm, Sweden

Physics of Fluids (2016), vol. 28, 056102

A complex system of waves propagating inside a water column due to the impact of plane shock wave is investigated both experimentally and numerically. Flow features, such as, focusing of expansion waves generating large negative pressure, nucleation of cavitation bubbles and a re-circulation zone are observed and discussed qualitatively and quantitatively. Experiments are conducted on a 22 mm diametrical water column hit by shock waves with Mach numbers 1.75 and 2.4 in a newly constructed exploding wire facility. A new technique to create a properly shaped, repeatable, large diameter water column with straight walls is presented. Qualitative features of the flow are captured using the shadowgraph technique. With the aid of numerical simulations the wave motions inside the column are analyzed; the spatial location of the expansion wave focusing point and the corresponding negative peak pressures are estimated.

1. Introduction
The breakup of liquid droplets has a great variety of applications including rain erosion damage, combustion and detonation of multi-phase mixtures, liquid jet atomization, atmospheric dispersal of liquid agents released at supersonic speeds, geothermal waste heat recovery applications, etc. Classically, on the basis of Weber number ($We$), the mode of droplet breakup has been classified into five regimes denoted as vibrational, bag, bag and stamen, stripping and catastrophic breakup with the corresponding range of $We$ being $\leq 12$, 12-50, 50-100, 100-350 and $>350$ respectively (Pilch & Erdman 1987; Hsiang & Faeth 1992). Theofanous et al. (2004) then reclassified it into two regimes namely Rayleigh-Taylor piercing (RTP) occurring in the range $10 < We < 10^2$ and shear-induced entrainment (SIE) in the range $We > 10^3$, with transition occurring
in the range $10^2 < We < 10^3$. A large number of arguments were synthesized into the above classifications which included some classical works (Engel 1958; Nicholls & Ranger 1969; Simpkins & Bales 1972; Waldman et al. 1972; Harper et al. 1972) and recent investigations (Pilch & Erdman 1987; Wierzba & Takayama 1988; Hsiang & Faeth 1992, 1995; Chou & Faeth 1998; Dai & Faeth 2001; Joseph et al. 1999; Theofanous et al. 2004; Theofanous & Li 2008). While Engel (1958) studied the effects of varying droplet size and shock Mach number, investigations on calculating characteristic breakup times (Nicholls & Ranger 1969; Waldman et al. 1972; Hsiang & Faeth 1992) and dependence of breakup dynamics on parameters such as density and viscosity ratios of the fluids (Hirahara & Kawahashi 1992; Hsiang & Faeth 1992, 1995; Joseph et al. 1999; Theofanous & Li 2008; Theofanous et al. 2012, 2013) were also conducted.

The events occurring during the early stages of shock propagation inside the liquid droplet are an inherent part of the aerobreakup problem which has been addressed in various numerical works (Igra & Takayama 2001a,b,c; Igra & Sun 2010; Meng & Colonius 2015). Beside its direct application in the realm of liquid droplet aerobreakup, the early shock/droplet interaction also serves as a classical test problem for development of numerical methods for studying interface dynamics in compressible multi-fluid flows (Chang & Liou 2007; Terashima & Tryggvason 2009; Hu et al. 2009). Experimentally, the longer timescale breakup of spherical liquid droplets has been studied in many of the above mentioned works. However visualizing the complex system of waves propagating inside the liquid spherical droplet on shorter timescales undoubtedly presents a hard experimental challenge. Igra & Sun (2010) noted that on comparing droplet deformation and disintegration, a 2D cylindrical water column behaved similarly to a spherical droplet, while in the 2D case the visualization becomes more accessible applying the traditional techniques such as shadowgraph, schlieren and/or interferometry methods. Although recently the existence of the shock waves inside the water column were confirmed using holographic interferometry (Igra & Takayama 2001a,c), the detailed experimental analysis of the wave motion inside the column, that are also in demand for validating the available numerical data, are still lacking. One of the most important factors limiting the quality of the experimental data is the diameter of the water column. In the studies thus far, the diameter ranges from a minimum of 0.7 mm to a maximum of 6.4 mm. Irrespective of the visualization technique used, the column size is too small to effectively capture the details in sufficient spatial and temporal resolution. The probable reason for the small size columns is
the difficulty to create and maintain properly shaped, repeatable, large
diameter cylindrical water columns having straight walls.

In this work, we focus on the wave motion inside a water column
(2D) during the early stages of shock interaction with an emphasis on
experimental detailed visualization and pressure measurements. We
employ a new technique of specially treating the test chamber windows
on which symmetric cylindrical water columns of relatively large diameter
are produced. The water column is hit by a shock wave generated using
an exploding wire (EW) technique and the visualization is conducted
using the shadowgraph method. Numerical simulations are performed
using the Star-CCM+ CFD solver to assist in interpreting and analyzing
the flow features observed in experiments.

Most of the experiments conducted in the area of droplet aerobreakup
have been carried out in shock tubes. An EW facility is an effective
and suitable alternative offering the following advantages: table-size
experimental facility; generation (with less resources) of high intensity
shock waves compared to a shock tube; ease of operation; generation of
shock waves with blast-shape pressure profile as in real explosion scenarios.
Recently EW methods were used in underwater studies (Efimov et al.
2009; Fedotov-Gefen et al. 2010; Kozlov et al. 2013), validation (Doney
et al. 2010), blast-wave structure interaction (Ram & Sadot 2012), shock
mitigation studies (Liverts et al. 2015), etc. The newly built EW facility
has a capability of generating blast waves with Mach numbers in the
range $M_S = 2 - 6$, with a promising repeatability between the tests.

This paper is organised as follows. In section 2, we describe the
experimental setup including the detailed description of technique used
to create a large diameter water column (Sec. 2.4). Section 3 presents
approximations made and governing equations solved in numerical sim-
ulations. In section 4, the experimental data obtained for two incident
shock Mach numbers are presented, discussed and compared with the
numerical results. Finally in section V, this work is summarised.

2. Experimental setup

2.1. Electrical setup

The general outline of the electrical setup is shown in Fig. 1. The high-
current generator consists of a 6 $\mu$F, 30 kV capacitor charged by a high
voltage power supply (Spellman SL300) via a charging resistor. The
discharge of the capacitor is either through a spark gap and the EW or
through a discharge resistor and a safety switch in case of emergency.
The spark gap acts as a fast switch consisting of two brass electrodes
separated by a distance small enough to allow dielectric breakdown of
air at atmospheric pressure. The spark gap is initially pressurized to 3 bars to prevent the dielectric breakdown during the charging of the capacitor. The release of pressure through a remotely controlled solenoid valve initiates a rapid discharge of the capacitor, and results in a high current flowing through the wire via 8 parallel (decreasing circuit inductance) coaxial RG-213 cables (Efimov et al. 2013). Two safety switches, automatically controlled pneumatic actuator, and additional manual switch, are installed to short-circuit the capacitor. All the operations are automated using NI USB-6009 DAQ and LabVIEW software.

Figure 1: Electrical outline

2.2. Test unit

The experiments are conducted in a specially designed test unit consisting of two chambers viz exploding chamber and test chamber, as shown in Fig. 2a. The sidewalls of both chambers are made up of 12 mm thick steel plates and flanges rigidly connecting the chambers to each other. A uniform channel with a width of 74 mm and a height of 5 mm runs continuously across both chambers inside the assembled test unit.

The exploding chamber houses two electrodes which are 70 mm apart and across which a copper wire of 0.4 mm diameter is fixed. The other ends of the electrodes are connected to the 8 coaxial cables delivering electric current to the EW from the high-current generator. The inner walls of the exploding chamber are insulated with 10 mm thick
PolyOxyMethylene (POM) thermoplastic to avoid sparking between the electrodes and the steel body.

When high current passes through the copper wire, the latter undergoes rapid Joule heating during which the wire begins to melt and vaporize. The expansion of the vapour column is accompanied by generation of a cylindrically shaped blast wave moving outwards from the EW axis. Since the wire explosion is confined inside the narrow rectangular exploding chamber, the initially cylindrical blast wave is modified and rapidly transformed into a plane shock wave with a blast-shape profile that propagates towards and into the test chamber. The resulting shock intensity depends primarily, for a fixed capacitance, on the charging voltage and the diameter of wire used for explosion. A few wire diameters were tested for a particular voltage on the basis of the charge remaining in the capacitor after the explosion and observations of the EW residues inside the test unit. As a result, a copper wire with optimal 0.4 mm diameter was used in the tests.

The shock interaction with the water column was visualized in the test chamber, which provides a field of view of about 200 mm length and 74 mm width. The top and bottom of the test chamber are either plexiglas windows or glass windows separated by 5 mm distance (channel height). Although the glass is superior in terms of image quality, the plexiglas can be fabricated for mounting of pressure sensors.

Table I summarises the charging voltages, the energies stored and the shock Mach numbers obtained correspondingly. The Mach number was estimated based on the time-of-flight measurements between two pressure sensors flush-mounted inside the plexiglass window at a distance
of 200 mm from the EW origin which corresponds to the position of the water column.

<table>
<thead>
<tr>
<th>Voltage [kV]</th>
<th>Energy stored [J]</th>
<th>Mach number</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>300</td>
<td>1.6</td>
</tr>
<tr>
<td>12</td>
<td>432</td>
<td>2.4</td>
</tr>
<tr>
<td>15</td>
<td>675</td>
<td>3.4</td>
</tr>
<tr>
<td>17</td>
<td>867</td>
<td>3.8</td>
</tr>
</tbody>
</table>

Table 1: Charging voltage vs. shock Mach number for 0.4 mm copper wire at 200 mm distance from EW

2.3. Visualization setup

A shadowgraph system is employed for visualization of shock wave interaction with the water column. The test unit is placed horizontally between two 45° mirrors as shown in Fig. 2b. The light source is an Orion laser (Nd:YAG 532 nm) with a pulse energy of 17 mJ, a pulse width of 4-6 ns, and a beam diameter of 2.75 mm. Since this being a single pulsed laser, only one image can be obtained during each test run. At a preset time, the laser is triggered by a digital delay generator (Stanford Research Systems DG535) which in turn is triggered by a voltage probe connected across the EW.

The beam from the laser head is expanded by a combination of concave-convex lenses L1-L2. The resulting horizontal 180 mm diam. parallel beam is redirected vertically using the top plane mirror tilted 45°; it is then passed through the transparent test chamber; and finally it is redirected back horizontally using the bottom 45° mirror. The image of the shadowgraph plane is focused on the camera sensor (Nikon D90) with the aid of lens combination L3-L4. The advantage of using a still camera is that it offers high spatial resolution images. Since the image exposure time is governed by the laser pulse the camera shutter was open for long times (~30 s). A neutral density filter was placed close to the sensor opaque enough to damp the light coming from explosion flash, room and other equipment illumination, while transparent enough to pass the laser light pulse.

2.4. Creation of a large diameter water column

Various techniques have been employed in literature to produce water droplets/columns (Hirahara & Kawahashi 1992; Joseph et al. 1999; Igra & Takayama 2001c) but the diameter has always been limited to a
few millimeters. The creation of large diameter water column is vital for detailed studies of wave motions inside the droplet. Evidently it is hard to maintain a cylindrical shape of a large size water column, since the water will tend to splash sideways. To overcome this issue, an external force should be applied to hold the column walls in place. In addition, the method should not affect the flow and the test chamber transparency properties. In this work, we implement a technique of using super-hydrophobic coating.

Hydrophobic effect refers to the tendency of substances to repel water molecules. The coating was applied on a limited area on the bottom window of the test chamber. The hydrophobicity level was chosen to just about to prevent the water from sliding sideways on the bottom window at static unperturbed conditions. Even slight vibrations of the test unit would allow water to flow over the hydrophobic barrier. As only one image per coating can be obtained in each test run, it was necessary to employ repeatable coating routines. An O-ring of 22 mm diameter was pressed against the bottom window and the hydrophobic coating was carefully applied on the surrounding area. The thickness of the coating layer was negligible compared to the thickness of the channel. An air gun was used to speed up the drying process and after approximately 30 minutes, the O-ring was removed; the top window was placed in position and secured. A syringe was introduced through a small hole at the back end of the test chamber and water was injected into the uncoated area,

Figure 3: 22 mm water column created using hydrophobic coating. Images taken using normal still camera: (a) isometric view with pressure sensors and (b) side view. Shadowgraph image: (c) top view (without sensors)
thus creating a 22 mm diameter water column as shown in Fig. 3. The water column is in good contact with the top window. Figs. 3a and 3b show images taken with a still camera, while Fig. 3c demonstrates a shadowgraph image. Note that the top window is not treated with hydrophobic coating. The analysis of the sideview images (Fig. 3b) has shown that the variation in diameter along the column’s height was about 2% in average.

3. Numerical simulation method

3.1. Modeling geometry and problem description

The problem geometry, as shown in Fig. 4, is a two-dimensional rectangular domain. Only half the water column is simulated as the experimental visualizations show no significant asymmetries along the test chamber’s centerline. Hence a symmetry plane boundary condition is set at the domain’s bottom and wall boundary conditions are set at the remaining three boundaries. The real case is simulated numerically by replacing the wire with a high pressure region. The width of the high pressure region and the initial high pressure amplitude are obtained by matching both experimental and numerical pressure profiles at two different locations. While initializing, the entire domain is at rest with the following initial conditions:

\[
(\rho_g, u, p) = \begin{cases} 
(580.7 \text{ kg/m}^3, 0 \text{ m/s}, 500 \text{ bars}) & \text{for } 1.5 \text{ mm} < x \leq 2.5 \text{ mm} \\
(1.17 \text{ kg/m}^3, 0 \text{ m/s}, 1.01 \text{ bars}) & \text{for } 1.5 \text{ mm} \geq x > 2.5 \text{ mm}
\end{cases}
\]

and \(\rho_l = 1000 \text{ kg/m}^3\). Here \(\rho\), \(u\) and \(p\) denote density, velocity and pressure, and the subscripts \(l\) and \(g\) refer to liquid and gas phases respectively. The domain is meshed with a uniform grid of 5000 \(\times\) 740 points, with a grid resolution of 440 points along the water column diameter. An overview of equations modeled and solved using Star-CCM+ are described in the next section.

Figure 4: Computational domain
Table 2: Weber number and Reynolds number for two incident shock Mach numbers

<table>
<thead>
<tr>
<th>Shock Mach number</th>
<th>Weber number (We)</th>
<th>Reynolds number (Re)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.75</td>
<td>$95.22 \times 10^3$</td>
<td>$1.16 \times 10^6$</td>
</tr>
<tr>
<td>2.4</td>
<td>$379.66 \times 10^4$</td>
<td>$2.75 \times 10^6$</td>
</tr>
</tbody>
</table>

3.2. Physics modeling

The estimations of Reynolds $Re$ and Weber $We$ numbers for incident shock Mach numbers 1.75 and 2.4 are presented in table 2. $Re$ and $We$ were calculated using the following relations:

$$Re = \frac{\rho_2 u_2 d_0}{\mu_g}; \quad We = \frac{\rho_2 u_2^2 d_0}{\sigma}$$

where $\rho_2$ and $u_2$ are the density and velocity of the gas behind the incident shock wave, $\sigma$ is the surface tension coefficient, $d_0$ is the initial water column diameter (22 mm) and $\mu_g$ is the dynamic viscosity of the gas.

Following the data summarized in table 2, it was assumed in the model that the flow is inviscid and the surface tension is negligible, as $Re$ and $We$ have relatively high values, suggesting that the inertial forces dominate the flow over the viscous and the surface tension forces correspondingly. In addition, since the primary focus of this study is on the early stages of shock/water-column interaction and not on the droplet breakup process, neglecting viscosity and surface tension seems to be a reasonable assumption.

With these approximations, the flow is governed by 2D compressible Eulerian multiphase equations. The Eulerian multiphase model solves the conservation equations for mass, momentum and energy for each phase. Since the fluids are considered to be immiscible, Volume of Fluid (VOF) model is used which captures the movement of the interface between the fluid phases on the basis of their volume fractions, $\alpha_i$. The VOF model assumes that all immiscible fluid phases share velocity, pressure and temperature fields. The equations are solved for an equivalent fluid whose physical properties are calculated as functions of the physical properties of its constituents and their volume fractions. The governing equations in integral form that describe the transport of scalar quantity
φ and volume fractions $\alpha_i$ are respectively:

$$\frac{\partial}{\partial t} \int_V \rho \phi dV + \int_A \rho \phi \mathbf{v} \cdot d\mathbf{a} = 0 \quad (3)$$

$$\frac{\partial}{\partial t} \int_V \alpha_i dV + \int_S \alpha_i \mathbf{v} \cdot d\mathbf{a} = 0 \quad (4)$$

where $\rho = \Sigma_i \rho_i \alpha_i$; $\rho_i$ is the density of each constituent. The first and second terms in both the equations are the transient and the convective fluxes respectively. Substituting $\phi = [1, u_x, u_y, e + u^2/2]$, where $e$ is the internal energy, into Eq. (3) gives the conservation equations of mass, momentum and energy in integral form. The system of equations is completed by modeling ideal gas law for the gas phase and the following Tait’s equation of state for the liquid phase (Shyue 2004, 2006; Johnsen & Colonius 2006, 2009):

$$p = (p_0 + B) \left( \frac{\rho}{\rho_0} \right)^\gamma - B \quad (5)$$

where $p_0$ and $\rho_0$ are the reference pressure and density, $B$ is the pressure-like constant and $\gamma$ is the adiabatic index. The values of $B$ and $\gamma$ were chosen to be 3050 bars and 6.68 respectively (Cocchi et al. 1996). Since the simulations are set-up mainly to identify the wave structures observed in the experiments, cavitation dynamics is not modelled. The discretization is based on a finite volume method. The transient terms are marched implicitly through a first-order temporal scheme and a second order upwind scheme is employed to advance in space. Algebraic multigrid algorithm with a V-cycle of maximum 30 cycles is used for solving the linear system iteratively and to accelerate convergence.

### 3.3. Validation

To validate the numerical algorithm, a wire explosion without any water column was considered. Two sensors for measuring pressure were placed at 200 mm and 240 mm from the EW. In the numerical simulation, the EW was modelled as a high pressure region with initial conditions as given in Eq. (1). Figure 5 shows the pressure profiles obtained both in experiments and computations. As can be seen, the numerical simulation agrees well with the experimental data, accurately predicting the position, speed, and pressure profiles of the blast wave at each sensor. A grid dependence study was also conducted and the results showed to be grid independent.
4. Results and Discussion

4.1. Qualitative analysis

The interaction between an incident shock wave of Mach number 2.4 and a 22 mm diameter water column sandwiched between plexiglas windows is traced throughout a sequence of shadowgraph images presented in Figs. 6 and 7. When the incident shock wave (I) hits the water column (Fig. 6a), it gets reflected (R), diffracted and transmitted (T) at the upstream interface (Fig. 6b). When the incident is a shock/compression wave, the transmitted wave will also be a shock/compression wave; however the reflected wave may either be an expansion or a shock wave depending on the acoustic impedance, defined as \( Z = \rho c \) where \( c \) is the speed of sound, as the continuity in pressure and velocity across the interface should be satisfied (Henderson 1989). Since \( Z_{\text{air}} < Z_{\text{water}} \), the reflected wave is a shock wave. The reflection of the incident wave at the air/water interface is therefore similar to that of a reflection from a solid cylinder.

When a planar incident shock wave encounters a cylindrical convex surface a regular reflection occurs as long as the angle of incidence is above a certain critical value, the so-called detachment angle (Fig. 6b). When the angle of incidence becomes less than the detachment angle a
transition from regular to Mach reflection takes place (Ben-Dor 2007). A triple point is initiated consisting of incident shock, reflected shock and a Mach stem along with its corresponding slip surface (Bryson & Gross 1961; Heilig 1969) (Fig. 6d). Until the onset of stripping type breakup of the water column, the diffraction of the incident shock wave is almost identical to a solid cylinder case. The transmitted wave along with its propagation inside the water column is shown in Figs. 6b-6d. During the initial stages, the transmitted shock is directly connected to the incident and reflected shock waves at the air-water interface. Although in the images (e.g. Fig. 6c) the transmitted shock looks disconnected as it travels faster in water, it is indeed connected along the water column boundary (Igra & Takayama 2001c).

Figure 6: Sequence of shadowgraph images for a 22 mm diameter water column hit by a Mach 2.4 incident shock wave (plexiglas windows)
When reaching the downstream water-air interface (Fig. 6e), the transmitted shock gets reflected as an expansion wave, REx (Fig. 6f), since $Z_{\text{water}} > Z_{\text{air}}$. Due to the column’s downstream concave boundary the reflected expansion wave focuses at a point, FReX, (Fig. 6g) creating ‘negative pressures’. By negative pressure, we mean that the absolute pressure is below 0. As liquids are held together by cohesive forces, they can withstand tensile stresses or negative pressures (Pettersen et al. 1994) which are one of the anomalies of water. Classical Nucleation Theory (CNT) predicts that water can get to as low as -100 MPa due to its strong cohesion (Caupin 2005). The tension of thus stretched water is defined as negative pressure. Herbert & Caupin (2005) provide a review of various techniques to produce such negative pressures.
The REx resembles a ‘horse-shoe’ structure (Fig. 6h) after focusing and continues to propagate towards the upstream high pressure region. Figure 6i shows the bifurcation of the expansion wave consisting of a ‘head and a ‘tail’. The second reflection (SR) of the expansion wave’s ‘head’ from the upstream interface and its subsequent motion is shown in figures 7a - 7e. Mach-Mach collision (Fig. 7d) occurs behind the water column which initiates a secondary wave system. Meanwhile the flow behind the incident shock initiates the stripping of the liquid material from the surface of the column walls creating a micro-mist whose motion is immediately preceded by the slip surface. The micro-mist then ends up into a re-circulation zone (Fig. 7f), created by two counter-rotating vortices. The baroclinic vorticity term is responsible for generation of the two counter-rotating vortices. Negative vorticity is generated when the incident shock wave passes over the surface and positive vorticity is generated along the back side of the water column (Meng & Colonius 2015). The interaction of these two vortices results in the re-circulation zone. With time, more and more vortices are shed resulting in a chaotic behaviour of the wake. Also note two standing shocks (Figs. 7g and 7h) in the wake associated with the turning of locally supersonic flow.

The decision to use plexiglas windows was initially based on the fact that plexiglas is relatively soft, possesses high impact strength and offers a possibility for mounting pressure sensors at the required locations. However plexiglas was also responsible for the appearance of small bubbles inside the water column observed in every image of Figs. 6 and 7. This is presumably caused by the propagation of stress waves through plexiglas, resulting in local negative pressures at the water-window interface and consequent nucleation of cavitation bubbles.

Moreover there is also a naturally occurring negative pressure at the FReX inside the water (Fig. 6g) which leads to possible nucleation of cavitation bubbles (Petersen et al. 1994; Caupin 2005; Herbert & Caupin 2005; Zheng et al. 1991; Caupin & Herbert 2006; Herbert et al. 2006; Azouzi et al. 2013) at the focusing point. On the images, these bubbles can be merged with the bubbles generated at the water-window interface which complicates the analysis of naturally occurring cavitation. To negate this effect, a series of experiments with similar conditions were conducted with regular glass windows. Figure 8 shows a sequence of images captured during the interaction between a Mach 2.4 shock wave and a 22 mm diameter water column sandwiched between glass windows. The absence of bubbles immediately preceding the transmitted wave confirms that it was indeed the plexiglas effect inducing the nucleation of bubbles. At the focusing point, as seen in Figs. 8e and 8f, local dark
Figure 8: Sequence of shadowgraph images for a 22 mm diameter water column hit by a Mach 2.4 incident shock wave (glass windows)

clouds appear indicating the generation of cavitation bubbles due to negative pressures created by FREx.

In order to complement the experimental shadowgraph images and to better understand the wave propagation, numerical simulation results are considered. The numerical simulation represents an ideal case (as cavitation is not modelled) and apart from identifying the wave structures, it can also be used as a tool to identify where and how negative the pressures can be obtained in an ideal scenario. A few experimental shadowgraph (top half) and numerical density gradient (bottom half) images at different time instants are compared in Fig. 9 to confirm the prediction accuracy. As can be seen, the reflected, diffracted and transmitted shock wave structures are consistent with the experimental shadowgraphs at all instants. Establishing this fact is necessary for further analysis of numerical pressure contours.

Figure 10 demonstrates simultaneous computational density gradient (top half) and pressure contour (bottom half). An increase in pressure in Figs. 10b and 10c suggests that it is indeed a compression wave travelling towards the downstream interface. Fig. 10d confirms that it reflects as an expansion wave (REx) and the focusing (FREx) occurs at a point as shown in Fig. 10e. This expansion wave acts as a separating line between two regions of pressure during its further propagation towards
the upstream interface. The region behind the ‘head’ of the expansion wave is a high pressure region, while the region behind the ‘tail’ is a low pressure region (Fig. 10g). It can be seen from Fig. 10i that the second reflection (SR) of the ‘head’ from the upstream interface results in a compression wave while the crossing-over of the ‘tail’ near the upstream interface creates a local low pressure region, reversing the previous effect, as in Fig. 10j. It then propagates downstream as shown in Fig. 10k. When the tension in water exceeds a limiting value, it fractures resulting in cavitation. The negative pressure observed at Fig. 10e ideally acts as a trigger for cavitation nucleation. Thus generated cavitation bubbles can be seen in Figs. 8e and 8f. Once cavitation is triggered, the simulation predicted pressure values near the focusing region is weakened in comparison to reality as this being an ideal case simulation.

To study the effect of Mach number, another set of tests was conducted with an incident shock Mach number 1.75. Images taken only with glass windows are shown in Fig. 11. The flow pattern structures are similar to the experimental images obtained in Mach 2.4 tests. Apart from the existing similarities, the major difference is the absence of cavitation bubbles. This absence is attributed to less (in absolute value) negative pressures at the focusing point due to weaker waves as compared to Mach 2.4 case. In order to substantiate this reasoning a quantitative analysis of pressure development is performed in the following subsection.

4.2. Pressure measurement

Three piezoelectric dynamic pressure sensors (113B24, PCB), flush mounted on the top plexiglas window, were used for recording the pressure history. The pressure sensors have a response time of 1 µs with a
Figure 10: Numerical density gradient (top half) and pressure contour, in bars, (bottom half) for a 22 mm water column hit by a Mach 2.4 incident shock wave

high frequency response of $\leq 100$ kHz. The diaphragm diameter of the sensor is 5.54 mm and therefore the values obtained are averaged across the sensor’s face area. The sensing geometry is compression and hence
Figure 11: Shadowgraph images for a 22 mm diameter water column hit by a Mach 1.75 shock wave with glass windows.

the sensor is unable to qualitatively measure the negative pressure. The location of the sensors with respect to the water column is shown in Fig. 12.

Figure 12: Location of sensors

The pressure profiles recorded by the sensors in Mach 2.4 tests are shown in Fig. 13. The first and second rises in the pressure trace recorded by sensor 1 are due to the incident wave (I) and the reflected wave (R) from the air-water interface respectively. Since it takes about 6.4$\mu$s for the incident shock wave and even larger time for the reflected shock to travel across the sensor’s face, the corresponding jumps in pressure do not appear steep (Fig. 13). The pressure amplitude behind the incident shock is about 6.49 bars and the one behind the reflected shock is about 13.1 bars. The first jump in pressure behind the transmitted wave (T) (Fig. 6b) as registered by sensor 2 is steeper suggesting higher shock speeds in water. Corresponding calculations predict a speed of 1681 m/s which is a shock wave in water with Mach number 1.12 and peak pressure amplitude of 19.6 bars. A negative pressure region observed in experiments just before the arrival of the transmitted shock is due to
the stress waves propagating inside plexiglas (PL), which also supports the appearance of small bubbles in the shadowgraph images taken with plexiglas windows.

Figure 13: Experimental and numerical pressure profile for Mach 2.4. The locations of the three sensors are as given in Figure 12. Note that the diaphragm diameter of the sensor is 5.54 mm and the values obtained are already averaged by the sensor. Similarly numerical simulation results are also averaged.

A second rise from 6.81 bars to 10.7 bars, noted shortly after 75 µs on the pressure trace recorded by sensor 2, is caused by the second reflection from the upstream interface (SR) (Figs. 7a and 10i). In the sensor 3 pressure trace, the transmitted wave (T) peak pressure amplitude is 7.89 bars (Fig. 6d). The shock attenuation in water is evident as the peak pressure has dropped from 19.6 bars to 7.89 bars in a 13 mm distance (between centers of the sensors 2 and 3). Immediately after the initial rise, a sharp dip to negative pressure is observed, which is caused by the focusing of the reflected expansion wave (FREx) (Figs. 6g and 10e). Although the numbers cannot be relied upon as the sensors are not designed to measure negative pressure, the tendency assures that the negative pressure exists at the focusing point. The results demonstrate
Table 3: Peak pressure for Mach 1.75 and 2.4

<table>
<thead>
<tr>
<th>Wave type</th>
<th>Mach 1.75</th>
<th>Mach 2.4</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experiment</td>
<td>Computation</td>
</tr>
<tr>
<td>I</td>
<td>3.39</td>
<td>3.53</td>
</tr>
<tr>
<td>R</td>
<td>6.15</td>
<td>6.40</td>
</tr>
<tr>
<td>T (Sensor 2)</td>
<td>6.27</td>
<td>6.44</td>
</tr>
<tr>
<td>T (Sensor 3)</td>
<td>2.31</td>
<td>3.21</td>
</tr>
<tr>
<td>SR</td>
<td>4.98</td>
<td>5.54</td>
</tr>
</tbody>
</table>

Table 4: Information obtained from numerical simulation by analyzing pressure data at cell centers

<table>
<thead>
<tr>
<th></th>
<th>Mach 1.75</th>
<th>Mach 2.4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location of FREx</td>
<td>4.2 mm from downstream interface</td>
<td></td>
</tr>
<tr>
<td>Negative peak pressure</td>
<td>-4.24 bar</td>
<td>-23.29 bar</td>
</tr>
<tr>
<td>Δt</td>
<td>3.3 μs</td>
<td>3.6 μs</td>
</tr>
</tbody>
</table>

good agreement between experiments and computations on comparison of pressure traces registered by sensors 1 and 2. The prediction of the pressure trace registered by sensor 3 which is mounted very close to the focusing point is less consistent with the experimental data. The reason for this discrepancy presumably could be because of the following: (a) disturbances arising in plexiglass window and (b) cavitation not modelled in simulation. However the time of arrival of the transmitted wave matches and the pressure profiles follow a common trend. Table 3 summarizes peak pressures of various types of waves obtained in Mach 1.75 and Mach 2.4 tests.

Using numerical simulation, the exact spatial location of FREx along with its peak value can be obtained by recording the pressure traces at each cell center. By analyzing thus obtained data, the location of FREx for both Mach numbers are found to be at 4.2 mm (19.1% of the column diameter) along the centerline from the downstream column’s boundary. Thus, the location of FREx is determined by the column’s geometry and not by the shock wave intensity. The peak negative pressure and the time taken (Δt) for the pressure amplitude to drop from atmospheric pressure to negative peak pressure and then back to atmospheric pressure is -4.24 bar and 3.6 μs; -23.29 bar and 3.3 μs for Mach 1.75 and Mach 2.4 respectively (see Table 4). Assuming Δt constant in both cases, the
presence or absence of cavitation bubbles is determined by the amplitudes of negative pressures. Since in Mach 2.4 tests larger negative pressures were registered compared to Mach 1.75 tests, nucleation of bubbles are only observed inside the water column hit by a stronger Mach 2.4 incident shock wave.

5. Summary

In this work, a detailed experimental analysis of waves motion inside a liquid column at the early stages of its interaction with a shock wave was performed. A new approach of treating the test chamber transparent surfaces with super-hydrophobic coatings proved powerful and effective. Using this technique highly repeatable, properly shaped, 22 mm diameter water columns with straight walls were successfully created. This assisted in visualization of flow patterns inside the water droplet with higher resolution due to larger droplet size. The propagation of waves inside the water column for two incident shock Mach numbers 1.75 and 2.4 were effectively captured using shadowgraph technique and compared with numerical simulations demonstrating good agreement. The transmitted wave reflected at the downstream interface resulted in a focusing of an expansion wave creating negative pressures sufficient for initiating cavitation. The cavitation bubble clouds were visualized in Mach 2.4 while not in Mach 1.75 tests. The shock pressure traces inside the water column were registered in experiments, which with the aid of numerical calculations were analyzed and the corresponding flow features were quantified. Less negative pressures at the focusing point were recorded for Mach 1.75 compared to Mach 2.4 explaining the non-appearance of cavitation bubbles. Using numerical simulation, the exact location of the focusing point along with its peak absolute value was estimated.

Acknowledgement

The financial support of the Swedish Research Council (VR) is gratefully acknowledged. The simulations were performed on resources provided by the Swedish National Infrastructure for Computing (SNIC) at PDC Centre for High Performance Computing (PDC-HPC).

REFERENCES


88

S. Sembian et al.


Theofanous, T. G., Mitkin, V. V. & Ng, C. L. 2013 The physics of aerobreakup. iii. viscoelastic liquids. Physics of Fluids 25 (3), 032101.


Paper 2
An experimental time-based analysis and numerical parameter study on shock-water column interaction

S. Sembian, M. Liverts, N. Apazidis
Department of Mechanics, KTH - Royal Institute of Technology, SE-100 44 Stockholm, Sweden

manuscript in preparation

In the first part, the dynamic characteristics exhibited by a water column during its interaction with a blast wave of shock Mach number 2.2 is studied in detail through high speed imaging. The blast waves are generated using Exploding Wire (EW) technique and the images are captured optically using shadowgraph technique. By processing consecutive frames captured during a single experiment, the timeline of events occurring during the interaction process is presented. An in-depth analysis of the cavitation dynamics is also presented. In the second part, a numerical parameter study is conducted to investigate the effect of focusing of the reflected expansion wave on the peak negative pressure. Since the main interest is vested on the maximum negative pressure obtained, it is assumed that the water does not fracture at any amount of tension and therefore cavitation is not modelled in the simulation. The study is performed for four different water column diameters (6, 10, 16 and 22 mm) impacted by shock waves with Mach number 1.8, 2.2 and 2.6.

1. Introduction

In our previous work on water columns in Sembian et al. (2016), we studied the interaction of a blast wave with a 22 mm diameter water column with particular emphasis on the flow features like shock non-linear effects, focusing of expansion waves generating large negative pressures, nucleation of cavitation bubbles, and the re-circulation zone. The investigation was carried out both experimentally and numerically. The water columns were created by employing a special technique of coating the surface with superhydrophobic paint. Qualitative features of the interaction were captured as shadowgraph images using a Nikon
D90 camera. Owing to its high spatial resolution of 0.07 mm/pixel, the flow features were captured in good detail that provided a deep understanding of the interaction process. Although the repeatability of the shock generation and column creation process was good, we could not provide a proper quantitative time-based analysis of the processes due to the capture of single image per run. Also heterogenous cavitation dynamics is a complex problem and the accuracy of the results depends on other factors as well, like exact location of the water column altering the surface nucleation sites, air-water composition etc., which varies from run to run. With the recent acquisition of Shimadzu HPV-X2 high speed camera with the capability to capture 10 million frames per second, the problem is revisited to provide an in-depth analysis by processing consecutive frames obtained during one experiment. Besides quantifying the cavitation process right from nucleation to collapse of cavitation bubbles, this also serves as a classical test problem for development of numerical methods involving heterogenous cavitation. So in the first part of the present study, we extend our previous experimental work on water columns wherein the interaction from a time-based perspective is investigated.

Water can sustain positive stress or negative pressure (absolute pressure below 0 Pa) when it is subjected to tension. This in fact is one of the anomalies of water. Numerous techniques such as Berthelot method (Worthington 1892), mineral inclusion (Zheng et al. 1991; Azouzi et al. 2013), centrifugation (Briggs 1950), shock waves (Trevena 1984), acoustic cavitation method (Herbert et al. 2006) to name a few, have been used to create tension in water. All those techniques broadly fall under three categories viz. (a) static (b) dynamic and (c) ultrasonic stressing. When the tension in water exceeds a limiting value, it fractures thus leading to cavitation. Classical Nucleation Theory (CNT) predicts that water can get as low as -150 MPa due to its strong cohesion before it begins to cavitate (Caupin 2005). These works were dedicated towards estimating the limiting pressure before cavitation occurs. Dynamic stressing utilizes shock waves for creating tension. It can be generated on the basis of (a) reflection principle: Underwater explosion resulting in positive pulse which gets reflected as an expansion (tension) wave at the free surface, bullet-piston method in which the mode of generation of positive pulse is by a bullet fired at a cylinder containing water; (b) tube arrest technique: a tube filled with water is pushed upwards by springs and is stopped suddenly; and (c) Water shock tube method: expansion wave is produced in the liquid either directly or by converting a positive pulse by a reflection method.
In the second part of the present study, the effect of combination of the reflection principle with the focusing of the reflected expansion wave (from herein called as reflection-focusing method) on the peak negative pressure is investigated through a numerical parameter study. The advantage of this method compared to conventional reflection principle across a flat interface is that the negative pressure increases many folds due to focusing effect. In case of a cylindrical water column, the reflection of the transmitted wave at the free surface naturally focuses at a point due to its curved boundary at the downstream interface, thereby generating large negative pressure. Since we are only interested in the strength at the focusing point, numerical simulation is performed without modelling cavitation. Also, the study is further simplified by using shock wave with a step profile rather than blast profile. The parameters under study are the size of the column and the strength of the incident shock wave.

This paper is organised as follows: In section 2, the description of the experimental set-up along with the models used for numerical simulation are discussed; Part A of section 3 contains experimental time-based discussion of the presented high speed shadowgraph images and Part B contains the discussion of the results of the numerical parameter study; Finally in Section 4 this work is summarised.

2. Experimental setup and numerical simulation method

The experiments are conducted using a similar setup described in Sembian et al. (2016) with some changes to the components used in the visualization setup. Instead of the Nikon D90 digital still camera used previously, a Shimadzu HPV-X2 high speed camera is set to capture at 1 million frames per second for recording the images. This also necessitated employing a continuous mode Spectra-physics BeamLok 2060 argon-ion laser. As the interest is only on capturing the structures inside the 22 mm diameter water column coupled with the fact that the resolution of the high speed camera is only 0.1 MP, lens L4 is replaced with a convex lens of 100 cm focal length for optically zooming in close to the water column to maximise pixel detail. Lenses L1, L2 and L3 are left untouched [figure 1(a)]. The water column is created using the same technique of coating around the O-ring on the bottom plate with hydrophobic paint.

Owing to high Reynolds number, the flow is assumed to be inviscid and therefore the numerical simulations are performed by solving a 2D, compressible Eulerian Multiphase equation with ‘Volume of Fluid’ (VOF) model. The VOF model captures the movement of the interface between the fluid phases on the basis of their volume fraction. The system of equations are closed by user defining Tait’s equation of state for the
liquid phase and solved using Star-CCM+ commercial CFD package. Cavitation is not modelled here as the main goal of this parameter study is to quantify the maximum negative pressure obtained during the reflection swing of the shock wave. The problem geometry, shown in figure 1(b), is defined by a rectangular domain with the following boundary conditions: stagnation inlet on the left boundary; symmetry on the bottom boundary; and wall boundary condition on the remaining boundaries. The shock wave is initiated by patching a small region following the stagnation inlet with Hugoniot jump relations while the remaining region is set with atmospheric conditions. Time marching is done implicitly using a first-order scheme while a second order upwind scheme with segregated solver is implemented for advancing in space.

### 3. Results and Discussion

#### 3.1. Part A: High speed imaging

Time-stamped shadowgraph images depicting the interaction of a Mach 2.2 blast wave with a 22 mm diameter water column are as shown in figure 2. An x-t diagram detailing the events happening inside the water column is constructed with the help of all recorded images and is shown in figure 3. Time zero is set at the first point of contact between the blast wave and the upstream interface of the water column [figure 2(a)]. The black line corresponding to the path of the transmitted shock wave ($T$) indicates that it takes an overall time of 16.5 $\mu$s for $T$ to reach the column’s downstream interface. Since $T$ has to first propagate through a decelerating diverging section (upstream interface...
to the center of the column) followed by an accelerating converging section (center to the downstream interface), the overall time taken is reminiscent of an average time which translates to an average velocity of 1333 m/s. This also includes the impeding effect of the blast nature of the incident shock. Upon reflecting as an expansion wave ($RE_x$) from the downstream interface [figure 2(d)], it continues to travel upstream with its path indicated by the green line. Here the distance is measured with respect to the head of $RE_x$ and its path is almost linear. Surprisingly it takes an overall time of 17 $\mu$s to traverse the full length of the column.
Figure 3: x-t diagram providing information on the major events occurring during the shock-water column interaction.

Figure 4: Timeline of events.

(which is only 0.5 $\mu$s longer than that taken by $T$). Cavitation is observed at 21 $\mu$s, one frame before its focusing ($FRE_x$) at 22 $\mu$s. The head of $RE_x$ upon reaching the upstream interface gets reflected again, indicated as second reflection ($SR$) in figure 2(k), and its path is tracked by the blue line. It too follows a linear path with an overall time of 18 $\mu$s to reach the downstream interface.

The timeline of events from the onset of cavitation bubbles to its collapse is as given in figure 4. Due to the concave boundary of the downstream interface, the reflected expansion wave converge to focus
Experimental time-based analysis and numerical parameter study

at a point before travelling upstream. Nucleation of cavitation bubbles are observed during the converging path of the expansion wave at 21 µs, even before its focusing. This indicates that the threshold pressure for cavitation is reached not at the focusing point but rather before it. It is also to be noted that the cavitation region is not concentrated at the focusing point but takes the form of a triangle with its tip/apex pointed towards the upstream interface which further substantiates the above statement. Usually heterogenous cavitation have a relatively lower threshold cavitation limit due to the presence of large number of nucleation sites in the form of surface roughness and impurities present in water. Since the experiments here are conducted using normal tap water enclosed by glass windows, heterogenous nucleation of cavitation is the most likely scenario. Due to the limitation of the pressure sensors in measuring pressure below 0 atm, it is hard to quantify the exact pressure at which water fractured but it can be deduced that it fractured much lower than the theoretical maximum negative pressure that occurs at the focusing point. The bubbles are fully developed at 24 µs and the cavitation region is found to be stable until 32 µs. Bubble collapse begins shortly after with the disappearance of small bubbles surrounding the region. The collapse of cavitation region tip occurs at 37 µs. The collapse is then accelerated following the crossing of SR wave over the region at 47 µs. The bubbles near the tip region disappear at a faster rate compared to the base region with full collapse observed at 56 µs. So the complete cycle of cavitation bubble nucleation, growth and its subsequent collapse took an overall 35 µs for a 22 mm diameter column with a Mach 2.2 blast wave.

3.2. Part B: Numerical parameter study

Numerical simulation was performed by varying two parameters: (a) Strength of the shock wave $M_s = 1.8, 2.2$ and 2.6; (b) Water column diameter $d = 6, 10, 16$ and 22 mm; and measuring the corresponding maximum attainable negative pressure at the focusing point ($P_f$) during the reflection swing of the transmitted shock wave. It is to be noted that cavitation nucleation is assumed to be absent as this is an attempt to understand the relation between the parameters and $P_f$. In a more practical sense, this approach appeals more towards homogeneous cavitation which, for the present case, can be obtained by reducing nucleation sites and impurities by using very smoothly polished surfaces and ultrapure water. The pressure distribution along the centerline of the water column measured at the instant of focusing as a function of both $M_s$ and $d$ are as shown in figure 5. The pressure distribution for all simulated cases follow
similar profile: an upstream region where pressure decay at a steady rate until the critical point followed by a focusing region characterised by a sharp dip to negative pressures at the focusing point and a sharp raise to 1 bar (atmospheric pressure outside the interface) at the downstream interface. At the instant of focusing, $P_f$ attained for $d = 3$ mm is -4 bar, -10.74 bar and -19.65 bar for $M_s = 1.8$, 2.2 and 2.6 respectively. Similar values for each $d$ is tabulated in Table 1. Generally, increasing either $d$ or $M_s$ results in increase of $P_f$. Note that for the purpose of normalizing, all $P_f$ values corresponding to 22 mm diameter case are denoted as $P_{f-d}$ and values corresponding to $M_s = 2.2$ are denoted as $P_{f-M_s}$. The value of the combination of both $d = 22$ mm and $M_s = 2.6$ is denoted as $P_{f-max}$.

By keeping $M_s$ constant, for instance $M_s = 1.8$ and varying $d$ results in an increase in $P_f$ from -4 bar to -8.05 bar for $d = 6$ mm and 22 mm
Table 1: Negative peak pressure values \( (P_f) \) for various water column diameter \( (d) \) and shock Mach number \( M_s \)

<table>
<thead>
<tr>
<th>Diameter ( (d) ) (mm)</th>
<th>Negative peak pressure ( (P_f) ) (in bars)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.8</td>
<td>2.2</td>
</tr>
<tr>
<td>6</td>
<td>-4</td>
</tr>
<tr>
<td>10</td>
<td>-5.64</td>
</tr>
<tr>
<td>16</td>
<td>-6.96</td>
</tr>
<tr>
<td>22</td>
<td>-8.05</td>
</tr>
</tbody>
</table>

respectively. For the parameter values studied here, this indicates an increase of 100\% in \( P_f \). This increase can be realized through normalized plots, as shown in figure 6, for all combinations of \( M_s \) and \( d \). The nomenclature used for normalization is as follows:

- Normalized diameter \( P_d = P_f / P_{f-d} \); here for a fixed \( M_s \), the \( P_f \) values of all \( d \) are normalized with \( P_{f-d} \) which is the maximum value obtained for \( d = 22 \) mm case.
- Normalized Mach number \( P_{Ms} = P_f / P_{f-Ms} \); here for fixed \( d \), the \( P_f \) values of all \( M_s \) are normalized with \( P_{f-Ms} \) which is the maximum value obtained for \( M_s = 2.6 \).
- Normalized maximum \( P_{f-max} = P_f / P_{f-max} \); here all values are normalized with \( P_{f-max} = -37.8 \), which is the value for the combination of both \( d = 22 \) mm and \( M_s = 2.6 \).

In figure 6(a), the normalized diameter \( P_d \) is plotted as a function of \( d \) for different \( M_s \). It can be seen that the corresponding normalized value for each \( d \) collapses to one particular value making it independent of \( M_s \). As \( d \) is increased from 6 - 10 - 16 - 22 mm, \( P_f \) increases by 41\%, 74\% and 100\% respectively. Note that all increase are relative to the case with the lowest \( P_f \) examined in this study. This indicates that in order to double the obtained negative pressure, the diameter of the column has to to be increased from 6 mm to 22 mm. Similar plot for normalized Mach number \( P_{Ms} \), plotted as a function of \( M_s \) for different \( d \) is as shown in figure 6(b). Here too the corresponding normalized value for each \( M_s \) collapses to one particular value making it independent of \( d \). Increasing \( M_s \) from 1.8 in steps of 0.4 results in an increase of 146\% to a
substantial 370% respectively. Although $P_f$ can be doubled through $d$, it can be quadrupled by varying $M_s$ making it an important parameter for obtaining large negative pressures. Of course, $M_s$ cannot be increased unconditionally as generating shock waves with large values of $M_s$ are dependent on the capability of the facility used. The beneficial effect of the combination of both a large diameter and a strong shock wave is shown as a normalized maximum $P_{f_{\text{max}}}$ plot in figure 6(c). When this value is compared with that of a lowest case with $d = 6$ mm and $M_s = 1.8$, $P_f$ rises from -4 bar to -37.8 bar, an increase of 845%. So generating extremely large negative pressures are possible through a combination of large diameter column impacted with a strong shock wave. To this extent, the effect of reflection-focusing principle on $P_f$ was demonstrated and quantified.
4. Summary

The study was performed in two parts focussing on different aspects on shock-water column interaction. In the first part, with the aid of high speed imaging, the timeline of events occurring during the interaction of a Mach 2.2 blast wave with a 22 mm diameter water column was established by processing consecutive frames captured during a single experimental run. The flow features were captured using shadowgraph technique. It took an overall 16.5 $\mu$s for the transmitted wave to traverse the entire water column and reach the downstream interface. Corresponding numbers for the reflected expansion wave and second reflection ($SR$) were 17 $\mu$s and 18 $\mu$s respectively. Nucleation of cavitation was observed earlier to the focusing point, indicating that the water had fractured before reaching its peak negative pressure. The heterogenous cavitation region took the form of a triangle with the tip/apex pointing towards the upstream interface. The collapse of bubbles began 11 $\mu$s after the its nucleation, starting from the tip which disappeared 5 $\mu$s later. The collapse was accelerated following the crossing of $SR$ over the cavitation region. Full collapse was observed after 35 $\mu$s from its nucleation.

In the next part, the effect of focusing of expansion waves (reflected from the free surface of a cylindrical water column) on the generation of peak negative pressure was examined through a numerical parameter study. Cavitation was not modelled in the simulation as the interest was on quantifying the maximum obtainable negative pressure at the focusing point. The study was performed for four different water column diameters ($d = 6, 10, 16, 22$ mm) impacted by shock wave of Mach number ($M_s$) 1.8, 2.2 and 2.6. It was found that the negative pressure increased by 100% through variation of $d$, 370% through variation of $M_s$ and a substantial 845% through a combination of both $d$ and $M_s$.

The financial support of Vetenskapsrådet (The Swedish Research Council) is gratefully acknowledged.

References


Sembian, S., Liverts, M., Tillmark, N. & Apazidis, N. 2016 Plane shock


Paper 3
Attenuation of strong external blast by foam barrier

S. Sembian, M. Liverts, N. Apazidis

Department of Mechanics, KTH - Royal Institute of Technology, SE-100 44
Stockholm, Sweden

*Physics of Fluids* (2016), vol. 28, 096105

The mitigation of externally-generated strong blast waves by aqueous foam barrier of varying configurations within fixed distance between the the explosion origin and the object to be protected is investigated and quantified both experimentally and numerically. The blast waves of shock Mach number 4.8 at 190 mm from the explosion plane are generated using exploding wire technique. The initially cylindrical blast waves are transformed into a plane blast wave in a specially constructed test unit in which the experiments are performed. The shock waves emanating from the foam barrier are captured using shadowgraph technique. A simple numerical model treating the foam by a pseudo-gas approach is used in interpreting and re-constructing the experimental results. The additional contribution of the impedance mismatch factor is analysed with the aid of numerical simulation and exploited for achieving greater blast wave pressure reduction.

1. Introduction

Sudden release of large quantities of energy into the atmosphere by any means results in the generation and propagation of strong blast waves with destructive capabilities. The degree of destruction can be greatly reduced if the blast wave energy is sufficiently mitigated. Various solutions for blast wave attenuation include using water wall (Cheng *et al.* 2005), free standing plates (Kambouchev *et al.* 2007), mitigation device (Edberg & Schneider 2005), hydraulic energy redirection and release technology (Chen *et al.* 2012), granular filters (Britan & Levy 2001) etc. Apart from these, aqueous foam barrier has received considerable attention and proved to be a good protective system. Kudinov *et al.* (1977) and Borisov *et al.* (1980) reported on the shock waves in gas-liquid foams and water foams respectively; Raspet & Griffiths (1983) demonstrated
the reduction of blast noise with aqueous foam and Gel’fand et al. (1981) reported on the blast wave mitigation properties of aqueous foam as a function of liquid fraction. The important factors identified as affecting the mitigation properties are, (1) the arrangement of the barrier and its distance from the charge; (2) the type and the energy of the explosive; and (3) the foam density (Gelfand & Silnikov 2004). A series of large-scale experiments (Hartman et al. 2006; Prete et al. 2013) conducted with high explosive (HE) charges buried in aqueous foam demonstrated the mitigation capabilities of foam on the basis of propagation of blast waves. Britan et al. (1992) studied the processes involved in the breakup of dry foam, experimentally investigated the effects caused by additives on shock propagation in foam (Britan et al. 2011, 2012) and provided a review of available experiments and models (Britan et al. 2013b) along with the strength and weakness of pressure measurement in foam (Britan et al. 2013a). Jourdan et al. (2015) conducted a series of shock tube experiments, measuring pressure in foam at numerous stations for different incident shock Mach numbers highlighting the mitigation capability of foam. Liverts et al. (2015) reported on the decay of blast waves generated by exploding wire technique in wet foams.

When the foam barrier entirely covers the explosive charge, it is termed as internal blast and when a blast wave generated in surrounding air impacts a foam barrier placed away from the explosion origin, it is termed as external blast. Free-field experiments and exploding wires (EW) can achieve both internal and external blasts while shock tubes have been used extensively for studying external shock waves. In an internal blast scenario, the processes involved in blast wave mitigation varies depending on the position of the foam, usually termed near-field and far-field, with respect to the blast origin (Britan et al. 2013b). The near-field is a region very close to the explosion origin where the temperature is very high and energy dissipation occurs mainly due to liquid heating and evaporation. In the far-field, shock scattering at the numerous interfaces and viscous dissipation of the liquid flowing through plateau border and films are the mechanisms through which energy dissipates. Due to the additional mechanisms involved, the mitigation at near-field is extremely effective compared to the far-field. Liverts et al. (2015) observed a significant blast wave pressure reduction of 72.5% in the near-field.

In some cases, there exist certain scenarios when the explosion origin is unknown or unable to be covered with foam for internal blast. When an object is to be protected for such external blasts, suitable foam barriers of sufficient thickness is deemed necessary. Quantifying the effect of foam barrier thickness on blast wave pressure reduction is an area worth
exploring particularly when the distance between the explosion origin and the object to be protected is fixed. With the knowledge gained from previous works, this study reports on how various configuration of foam barriers further enhance the mitigation of strong blast waves (by far-field mechanisms) that affect the final pressure experienced by the object. Experimental studies of such nature require generation of strong blast waves. Although shock tubes are extensively used throughout, generating strong blast waves require extreme resources. In case of free-field experiments, foam decay is an important reason for the high scattering of data obtained. Therefore accurately controlling the foam transient features and the desired foam geometry is not a simple task. EW technique is an effective and suitable alternative offering following advantages: generation of high intensity blast waves as in real explosion scenarios with significantly less resources; foam properties, barrier geometry, and blast wave profile can be accurately controlled in the laboratory environment; table-size experimental facility; ease of operation. An EW facility has been developed in-house that has the capability of generating blast waves with Mach number in the range 2-7 with relative ease and good repeatability between each test run.

In order to numerically predict the blast wave behaviour in foam, a number of modelling strategies (Prete et al. 2013; Baer 1992; Vasil’ev et al. 1998; Faure & Ghidaglia 2011) have been developed. Britan et al. (2013b, 2014) provides a comprehensive review of the strength and weakness of most of the strategies. Numerical simulation in this study is performed using an in-house model which describes the macroscopic properties of the foam. The model treats the foam using a pseudo-gas approach in which it is considered as a perfect gas with high molecular weight incorporating an effective adiabatic index for simulating the real water content in the foam. Wet foams with initial liquid fraction $\alpha = V_l/V_f = 0.1$, where $V_l$ and $V_f$ are volumes of the liquid content and the foam sample respectively, are used in this study. Wet foams are characterised with smaller bubbles where the net momentum and heat transfer between the liquid and gaseous phases are negligible resulting in rapid flow equilibrium. Since both the phases are in dynamic and thermal equilibrium, the flow can be represented by a single component approach. In addition, as we are interested in far-field mitigation, the model advantageously excludes complicated processes like high-temperature effects and shattering mechanisms making it a simple model.

This paper is organised as follows. In section 2, we describe the overall experimental setup including the description of the technique used to create foam barriers. Section 3 presents the governing equations
solved in numerical simulations along with its validation. In section 4, the results obtained for various configurations are discussed and finally in section 5, this work is summarised.

2. Experimental setup

The EW facility consists of three main parts, namely (a) High current generator (b) Test unit and (c) Optical setup. The general outline is as shown in Fig. 1

![Figure 1: General outline of the experimental set-up](image)

The charging circuit of the high current generator consists of a 6 µF capacitor charged to 12 kV (432 J) by a high voltage power supply. The discharging circuit is either through a spark gap and the EW or through a discharge resistor and a safety switch. The spark gap consists of two electrodes separated by a small distance to avoid dielectric breakdown of air while the capacitor is charged. A 30 kV trigger pulse of fast rise time generated by a trigger module was applied to the spark gap, initiating the rapid discharge of the capacitor which delivers very high current to the wire. A pneumatic actuator and an additional manual switch are installed to short-circuit the capacitor.

The experiments are conducted in a test unit consisting of an exploding chamber and a test chamber as shown in Fig. 2. Both the chambers are made of 12 mm thick steel walls and are rigidly connected to each
other through flanges. The chambers are designed in a way that a uniform channel of width 10 mm and height 5 mm runs throughout. The overall length of the channel measured from the explosion plane is 358 mm. The 10 mm long, 0.4 mm diameter copper wire to be exploded is screwed to the electrodes which are housed in the exploding chamber. The other end of the electrodes are connected to the high current generator. The inner wall of the exploding chamber is insulated with PolyOxyMethylene (POM) thermoplastic to avoid sparking to the steel body. When high current passes through the wire, it undergoes rapid Joule heating resulting in generation of a cylindrical blast wave. Since the generated blast wave is confined inside a rectangular narrow channel, it modifies itself into a plane wave propagating into the test chamber.

The test chamber consists of two glass windows, which provides a field of view of length 200 mm, for optical observation. The downstream end of the channel is subjected to the blast wave.
of the test chamber is plugged with an aluminum box which houses a PCB piezoelectric pressure sensor S3 (113B24, PCB Piezotronics) for measuring the face-on pressure. The pressure sensor S3 is positioned at 358 mm from the explosion plane, thereby indicating the object/target position. The diaphragm diameter of the sensor is 5.54 mm and it has a response time of $1 \mu s$ with a high frequency response of $\leq 100$ kHz. A conventional shaving foam (Gillette, Procter & Gamble) was used for making the foam barrier. The density of the samples was $\rho_f \approx 100 kg/m^3$, referring to volume liquid fraction of $\alpha = 0.1$. The foam was stable and the liquid volume fraction remained constant for close to 2 h of foam aging. Liverts et al. (2015) microscopically observed the Gillette foam and reported that it consisted of fine poly-disperse bubbles with an average diameter of 50 $\mu$m. The bubble size distribution was also reported to be uniform when compared between different samples. To form a foam barrier, two T-inserts are placed at the desired location. Foam is then uniformly filled within the space and the excess foam on the top are wiped away. The T-inserts are then lifted carefully to form a foam barrier as in Fig. 3. Image (a) is taken using a normal still camera and image (b) is a shadowgraph image. The foam barrier thus created have an error of $\pm 2$ mm. When only one such foam barrier is formed, it is referred to as single barrier and when two barriers are formed, it is referred to as twin barrier. The nomenclature used throughout the paper for describing the configurations of both the foam barriers are as follows (Fig. 4):

- $l_{SOD}$: standoff distance between the foam position and the object/target position
- $l_{foam}$: thickness of the foam
- $\Delta l$: distance between foam in twin barrier

![Diagram of foam barriers](image)

(a) Single barrier

(b) Twin barrier

Figure 4: Nomenclature used for describing the configurations of foam barrier
The blast wave impacting the foam and the wave emanating from the foam are captured as shadowgraph images using the optical setup. The beam from a laser light source (532 nm wavelength with a pulse energy of 17 mJ and pulse width of 4-6 ns) is expanded and rendered parallel using a concave-convex lens assembly. The resulting 180 mm parallel beam is redirected to pass through the test chamber. The beam is then deflected by a density gradient in the chamber and the resulting beam is focused back on the camera sensor (Nikon D90). A neutral density filter was placed close to the sensor, opaque enough to damp the light coming from explosion flash, room and other equipment illumination, while transparent enough to pass the laser light pulse. Fig. 3(b) is an image taken using this set-up.

3. Numerical simulation

3.1. Physics modeling

The numerical simulations are performed using an in-house code, successfully tested for foam (Liverts et al. 2015), solving the full set of compressible Euler equations coupled with a complementary liquid volume fraction, $\alpha$, based equation:

$$U_t + F_x = 0$$ (1)

where $U$ and $F$ are vectors of conservative quantities and fluxes, respectively,

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho E \\ \rho \alpha \end{pmatrix}, \quad F = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho E u + pu \\ \rho u \alpha \end{pmatrix}$$ (2)

where $\rho, u, p$ and $E$ denote the density, the particle velocity, the pressure and the specific total energy, respectively. The volume fraction formulation based algorithm provides an approximate treatment for mixture of one or more fluid component within a grid cell (Shyue 2006). The foam is modeled as a pseudo-gas with density $\rho$, which is considered to be a mixture of density of water ($\rho_w$) and air ($\rho_a$), and is calculated from $\rho = \alpha \rho_w + (1 - \alpha) \rho_a$. Setting $\alpha = 0$ corresponds to density of air, $\alpha = 1$ corresponds to density of water and values in between correspond to density of pseudo-gas (foam). The initial conditions are set using a discontinuous distribution of $\alpha$, 0 where there is no foam and 0.1 in correspondence of foam barrier. Since the expected pressure amplitudes are low ($\approx 100$ bar), the water is assumed to be incompressible with constant density of $\rho_w = 1000 \text{kg/m}^3$. The system of equations are
completed by modeling the following Noble-Abel equation of state:

\[ p(\rho, e) = \left( \frac{\gamma - 1}{1 - \alpha} \right) \rho e \]  

(3)

where \( e \) and \( \gamma \) represents the specific internal energy and the adiabatic index respectively. The adiabatic index \( \gamma \) of the fluid mixture is updated using the following equation:

\[ \frac{1}{\gamma - 1} = \frac{\alpha}{\gamma_w - 1} + \frac{1 - \alpha}{\gamma_a - 1} \]  

(4)

The equations are solved numerically using a Godunov-type flux vector splitting scheme (Sun & Takayama 2003). The flux vector \( \mathbf{F} \) can be rewritten as,

\[ \mathbf{F} = \mathbf{uU} + \mathbf{P} \]  

(5)

where \( \mathbf{P} = (0, p, pu, 0)^T \). The basic idea of the AUFS scheme is to split flux vector \( \mathbf{F} \) into,

\[ \mathbf{F} = (1 - S)\mathbf{F}_1 + S\mathbf{F}_2 \]  

(6)

Here \( \mathbf{F}_1 \) and \( \mathbf{F}_2 \) are the intermediate flux vectors and Eq. (6) is exactly Eq. (5) when \( S = s_1/(s_1 - s_2) \), where \( s_1 \) and \( s_2 \) are the artificial wave speeds. The intermediate flux vectors, \( \mathbf{F}_1 \) and \( \mathbf{F}_2 \), are calculated from,

\[ \mathbf{F}_1 = \frac{1}{2}(\mathbf{P}^L + \mathbf{P}^R) + \delta \mathbf{U} \]  

(7)

\[ \mathbf{F}_2 = \mathbf{U}\zeta(u^\zeta - s_2) + \mathbf{P}\zeta, \zeta = \begin{cases} L & \text{for } s_1 > 0 \\ R & \text{for } s_1 \leq 0 \end{cases} \]  

(8)

The artificial viscosity, \( \delta \mathbf{U} \), is given as

\[ \delta \mathbf{U} = \frac{\gamma}{2\tilde{c}} \left( \frac{p^L - p^R}{\gamma-1} \right) \left( \frac{(pu)^L - (pu)^R}{\alpha(p^L - p^R)} \right) \]  

(9)

\( \tilde{c} \) is the intermediate speed of sound given by \( c^2 = \gamma p/\rho(1 - \alpha) \). The numerical values of \( s_1 \) and \( s_2 \) are computed from

\[ s_1 = (u^L + u^R)/2 \]  

(10)

\[ s_2 = \begin{cases} \min(0, \min(u^L, u^R) - \max(c^L, c^R)) & \text{for } s_1 > 0, \\ \max(0, \max(u^L, u^R) + \max(c^L, c^R)) & \text{for } s_1 \leq 0 \end{cases} \]  

(11)
Finally the solution for grid cell \( i \) is advanced between time-steps \( n \) and \( (n + 1) \) through

\[
U_i^{n+1} = U_i^n - \frac{\Delta t}{\Delta x} (F_{i+\frac{1}{2}} - F_{i-\frac{1}{2}}) \quad (12)
\]

The scheme has been previously tested and validated for a broad variety of both single phase (Eliasson et al. 2007; Kjellander et al. 2010, 2011, 2012; Apazidis et al. 2013) and multi phase (Apazidis 2016) compressible flows.

### 3.2. Validation

Data for determining the initial conditions for the numerical simulation were obtained by performing an experimental wire explosion test in ambient air. The glass windows in the test chamber are replaced with plexiglas windows on which two piezoelectric pressure sensors, S1 and S2 (same model as S3), are flush mounted at 190 mm and 234 mm respectively from the explosion plane. Note that sensor S3 is already in position measuring face-on pressure at 358 mm from the explosion plane. The initial conditions chosen by trial-and-error method to fit the experimental pressure traces obtained from the three sensors are:

\[
(\rho, u, p) = \begin{cases} 
(95.82 \text{ kg/m}^3, 0 \text{ m/s}, 1100 \text{ bar}) & \text{for } 1 \text{ mm} \leq x \leq 3 \text{ mm} \\
(1.17 \text{ kg/m}^3, 0 \text{ m/s}, 1.01 \text{ bar}) & \text{for } x \leq 1 \text{ mm} \& x \geq 3 \text{ mm}
\end{cases}
\]  

(13)

where \( \rho \), \( u \) and \( p \) denote density, velocity and pressure respectively. The numerical domain is meshed with 3600 grid points with a uniform spacing of 0.01 mm between them. A grid dependence study was conducted and the results shown are grid independent. Since 1D numerical simulation is performed, the numerical pressure profiles are approximated by averaging over the sensor diameter (5.54 mm) so as to be consistent with the experimental pressure profile. As can be seen in Fig. 5(a), the numerical simulation prediction in terms of position, speed and pressure profiles of the blast wave are in good agreement with the pressure signals registered by the sensors. The shock Mach number of the blast wave at the sensor positions are 4.8, 4.4 and 3.65 respectively indicating a strong blast wave. The face-on peak pressure registered at the sensor S3 is 88 bar.

Attempts made to measure the pressure development inside the foam to validate the problem proved futile due to two reasons: (a) pressure sensors are air-calibrated and are not designed for measuring in foam and (b) stress waves (Sembian et al. 2016) travelling in plexiglas windows affect the propagation in foam. To overcome these effects and to properly validate the algorithm with foam, glass windows are installed back and
Figure 5: (a) Experimental and numerical pressure trace at three sensor positions: S1 at 190 mm, S2 at 234 mm and S3 at 358 mm. Sensor S3 measures the face-on pressure (b) Time of arrival comparison at S3 for different foam thickness. A schematic of the arrangement is as shown inside the plot. The numerical profiles are averaged over a 5.54 mm length so as to be consistent with the pressure sensor.

A time of arrival study at sensor S3 was performed with several foam thickness. A schematic of the arrangement is as shown inside Fig. 5(b). The channel is filled with the foam of the test thickness with sufficient care to ensure that the foam is in good contact with the sensor S3 face and no air pockets exist. The foam thickness here refers to the distance between the sensor S3 face and the foam starting position. When the wire is exploded, a blast wave is generated which propagates downstream through the chamber. On impact with the air-foam (AF) interface, the incident blast wave gets reflected and transmitted. The transmitted shock wave propagates through the foam barrier until it reaches the end wall where sensor S3 is mounted. Fig. 5(b) shows very good agreement between experimental and numerical results. The model accurately predicts the time of arrival of the shock wave in foam for all tested foam thicknesses.

The experimental errors in the measurements are mainly due to: (a) sensor sensitivity ($\approx 1\%$); (b) sensor response time ($\approx 1\mu s$); and (c) triggering time instant dependent on a chain of relevant equipment, e.g. response times of voltage probe, oscilloscope, delay generator, laser pulse, sensors, etc. However the dominant uncertainty in the final values of pressure amplitudes and times of arrival is associated with the repeatability of blast wave generation and foam preparation. To ensure the repeatability a special test was conducted where a foam barrier of
configuration $l_{SOD} = 8$ cm and $l_{foam} = 4$ cm was placed and pressure sensors S1 and S3 installed at 190 mm (ahead the foam barrier) and 358 mm (behind the foam barrier) respectively. The test was repeated 5 times keeping all the initial conditions identical. Fig. 6 demonstrates the resultant pressure profiles recorded by the sensors. The pressure profiles registered by S1 demonstrate the effect of wire explosion and blast formation from test to test. The uncertainty (standard deviation) in determination of peak pressure amplitudes is $\pm 5.5\%$ and of time of arrival is $\pm 0.5\%$. As such repeatability of the blast generation can be considered as good, the pressure profiles registered by S3 mostly demonstrate the effect of foam sample preparation. As can be seen the uncertainty in determination of peak pressure amplitudes increases to overall $\pm 9\%$ and of time of arrival in foam to $\pm 2\%$. It should be noted that the latter are the maximum estimates of the corresponding uncertainties since the values were measured behind the blast that has propagated along the whole foam column.

4. Results and Discussion

The foam barrier used in the validation process can be thought of as one limiting case with $l_{SOD}$ set as 0 cm. When a barrier of foam is formed at a distance $l_{SOD}$ from the target position (Fig. 7(a)), the transmitted wave in foam on reaching the foam-air (FA) interface gets reflected and transmitted again due to the impedance mismatch across the interface.
The reflected wave here is an expansion wave while the transmitted wave in air is a shock wave. Fig. 7(b) shows the thus transmitted shock wave captured as a shadowgraph image. Meanwhile the foam barrier is compressed, fragmented and moves behind the shock wave. This shock induced foam flow following the shock wave can also be seen in Fig. 7(b).

Figure 7: Shock wave emanating from both single and twin barrier foam captured as shadowgraph images. (a) is an image taken before the shot and (b) corresponds to an image taken during the shot.

In the far field, the dominant underlying process responsible for the reduction of peak pressure of the shock wave propagating in foam is the ‘catching up’ of the rarefaction wave with the wave front, thereby effectively attenuating it. This process is also aided by the very low speed of sound in foam ($\sim 40m/s$), providing sufficient time for the rarefaction wave to catch up with the wave front in shorter distances. This can be effectively explained using a shock-tube type schematic representation of x-t diagram as given in Fig. 8. An ideal case with no foam is represented by Fig. 8(a) where the incident shock wave ($In$) propagates along the driven section while the expansion fan ($Ex$) propagates along the driver section. Within the domain set for constructing this diagram, the $Ex$ which was reflected off at the driver end wall was barely able to ‘catch up’ with the $In$. Meanwhile the $In$ reaches the driven end wall and is reflected back as a shock wave ($Ra$). By introducing foam as in Fig. 8(b), the $In$ gets transmitted ($Tr$) as well as reflected ($Rf$) at the air-foam
interface. The slope of $\textbf{Tr}$ indicates the low speed of sound in foam. Therefore the $\textbf{Ex}$ has sufficient time to 'catch up' with the transmitted shock wave, reshape its pressure profile, thereby effectively attenuating it. Generally shorter the driver section, the quicker the expansion fan can catch up with the upstream travelling shock front. The length of the driver section in the experiments conducted in this study is very small ($\approx 3-4$ mm) that the expansion fan almost immediately follows the shock front, thereby decaying it along its propagation. When such shock fronts propagate through foam, the rate of decay is higher compared to its decay in air following similar mechanisms as in Fig. 8.

Figure 8: Shock-tube type schematic representation of x-t diagram for (a) with foam and (b) without foam.

On reaching the FA interface, the intensity of the wave front is diminished even further due to impedance mismatch across the interface. This drop in intensity is a property of the medium and cannot be
considered as attenuation as there is no significant energy loss. But the drop across the interface is also an intrinsic effect and cannot be neglected of the analysis too. As the goal is to reduce the blast wave peak pressure at the target, including the interface effect makes the analysis complete. For the sake of present analysis, the drop in intensity at the FA interface will be termed as 'impedance mismatch factor'. The transmitted shock wave in air then propagates further downstream until it reaches the target position. The pressure trace registered at the target position by sensor S3 is shown in Fig. 9. The initial rise in pressure is due to the impact of the shock wave. The effectiveness of the barrier can be easily demonstrated by comparing the registered peak pressure due to shock impact with the peak pressure obtained in ambient air with no foam barrier (Fig. 5(a)). The peak pressure has dropped from a maximum of 88 bar to as low as 4 bar, as registered in Fig. 5(b). After a few microseconds, the foam following the shock wave impacts the target. As the foam has more momentum, foam impact results in greater pressure compared to shock impact. Good agreement exists between the experiment and simulation prediction until foam impact although the profiles are similar. The reason for this discrepancy after foam impact is two folds: (a) the assumption of the inviscid Euler equations for shock propagation is less suitable for the shock induced foam flow in the confined channel. (b) the values registered upon foam impact cannot be relied upon as the sensors are air calibrated (Britan et al. 2013a). Nevertheless, since the main focus is on blast wave mitigation, the pressure development after foam impact
is neglected in the barrier configuration study and only shock impact pressures will be reported from here after. For practical purposes, it is to be noted that the foam if left unattended may hit the target with a higher pressure than the mitigated shock wave.

4.1. Single barrier foam

The overall effect of $l_{SOD}$ on a single barrier foam of increasing $l_{foam}$ is plotted in Fig. 10. Introducing foam barrier even as thick as 2 cm drastically reduces the peak pressure experienced by the target. For constant $l_{SOD}$, the overall peak pressure reduces exponentially with increasing $l_{foam}$. For a particular $l_{SOD}$, increasing the thickness of the barrier results in strong reduction (as low as 3 bar) of the peak pressure whereas for a particular $l_{foam}$, increasing the standoff distance very minimally affects the peak pressure. In fact for constant $l_{foam}$, the peak pressure marginally decreases with decreasing $l_{SOD}$. In order to decouple and study these effects in detail, x-t plots with constant $l_{SOD}$ (Fig. 11(a)) and constant $l_{foam}$ (Fig. 11(b)) are plotted for one particular foam barrier.

![Figure 10: Peak pressure on the target as a function of $l_{foam}$ for varying $l_{SOD}$ for a single barrier foam. The error bars indicate the maximum estimated value following the procedure in section 3.2](image)

In case of Fig. 11(a), $l_{SOD}$ is maintained at 4 cm and the thickness of the foam barrier is increased from 2 to 12 cm. All solid lines are
the trajectories of the shock wave in air and dashed-coloured lines are trajectories in foam. The bottom most solid line is the trajectory of the blast wave in ambient air with no foam barrier. The time of arrival of the blast wave is 0.2 ms and the face-on pressure experienced by the target is 88 bar. The shock Mach number corresponding to 88 bar is 3.65. Note that the numerical results are averaged over a 5.54 mm length so as to be consistent with the experimental sensor. Since this being a blast wave, different foam barriers intercept at different shock Mach number as the distance between the blast origin and the front edge of the foam differs.

For constant $l_{SOD}$, a 2 cm thick foam barrier is impacted by a relatively weaker blast wave compared to a 12 cm thick barrier. On head-on collision with a foam barrier, the pressure at the immediate vicinity jumps to a very high value due to the reflected and transmitted shock waves. For a 2 cm thick barrier, this pressure rise is 64.8 bar which determines the initial strength of the transmitted shock wave. The rarefaction waves quickly catch up and decelerate the transmitted shock wave resulting in pressure reduction reaching 25 bar after travelling 2 cm inside foam. On reaching the FA interface, the intensity across the interface is reduced even further to 3.2 bar due to impedance mismatch factor. The face-on pressure measured at the target is 8.7 bar with time of arrival at 0.3 ms. The shock Mach number corresponding to 8.7 bar is 1.7. Introduction of a short 2 cm foam barrier resulted in almost 10 times (90.1%) pressure reduction as the corresponding blast wave strength is reduced greater than half as compared to ambient air without foam. The numerical pressure development at different time instants for a 2 cm barrier is as shown in Fig. 12(a). The initial foam location is marked with striped lines and the incident blast wave along with the reflected shock wave are clearly indicated. The pressure profile of the transmitted wave as it propagates through the barrier at different time instants can also be seen. It is to be noted that across the FA interface, the transmitted wave in air is a shock wave with a step profile. To explain this in terms of shock tube analogy, the foam barrier acts like a driver section with FA interface as a diaphragm. The transmitted wave reaching the FA interface implies diaphragm rupture resulting in shock wave and expansion waves propagating either side of the interface to maintain continuity in pressure and velocity. The values here are slightly inconsistent with Fig. 11(a) as no averaging is performed. Now on the other end is the larger 12 cm foam barrier. Since the impacted blast wave is relatively stronger, the transmitted shock wave registered a 88.8 bar rise in pressure across the interface. The numerical pressure
Figure 11: x-t plot with (a) constant $l_{SOD} = 4$ cm and (b) constant $l_{foam} = 4$ cm for a single barrier foam. The peak pressure values (in bar) are indicated along the trajectories in the figure. Note that the peak pressure values are averaged as in section 3.2.
Figure 12: Numerical pressure profile development for a constant $I_{SOD}$ case with thickness (a) 2 cm and (b) 12 cm single barrier foam.

Figure 13: Numerical pressure history inside a 12 cm thick foam barrier. The pressures are computed at an interval of 2 cm from the air-foam (AF) interface. The values are averaged at each x-location.

Development depicting the deceleration is as shown in Fig. 12(b). The pressure inside the foam barrier reduces exponentially with significant deceleration occurring close to the interface. This is to be expected due to the blast profile of the incident shock wave. The propagation of the transmitted wave is progressively slowed down (Fig. 12(b)) and
Attenuation of blast wave by foam barrier

registers a pressure of 8.3 bar on reaching the FA interface. Across the interface, a mere 1.8 bar strong shock wave is transmitted into air and the face-on pressure experienced by the target is 3.1 bar. The corresponding shock Mach number is 1.3, the time of arrival is 0.91 ms and the overall reduction in pressure is 96.5%. Although the bulk of the deceleration occurs close to the interface, additional thickness ensures even more deceleration albeit at a slower rate. Therefore increasing the thickness beyond a certain limit has very little effect on the expected final pressure.

The numerical pressure history inside the 12 cm foam barrier at different x-location is shown in Fig. 13. The pressures are computed at an interval of 2 cm from the air-foam (AF) interface.

Fig. 11(b) is a x-t plot showing the effect of $l_{SOD}$. For this case, a constant $l_{foam}$ of 4 cm is maintained throughout while the $l_{SOD}$ is varied from 2 to 12 cm. As can be seen, the deceleration across a 4 cm barrier is fairly similar for all $l_{SOD}$ plotted. The barrier that intercepts at an earlier point is impacted by a relatively stronger wave and therefore the target pressure is slightly higher than the value measured for a barrier at a later location. Nevertheless, the effect of $l_{SOD}$ on pressure reduction is very minimal compared to $l_{foam}$.

4.2. Twin barrier foam

Although foam exhibits good mitigation characteristics, the foam-air interface is also one of the contributors for pressure reduction. A comparison between pressure reduction by wave propagation inside foam (rate of decay mechanism) and pressure reduction across foam-air interface (impedance mismatch factor) for a single barrier with configuration as in Fig. 11(a) is as shown in Fig. 14. As demonstrated, the dominant mechanism responsible for peak pressure reduction is the rate of decay mechanism but the contribution of the impedance mismatch factor cannot be overlooked. In fact for small barriers, say for 2 cm foam, the pressure across the interface drops from 25 bar to 3.2 bar. As the barrier thickness increases, the pressure drop across the interface decreases. Nevertheless, the contribution of the impedance mismatch factor also plays a significant role in the overall pressure reduction.

The idea of using twin foam barrier stems from the fact that by separating the total thickness of the foam equally by a distance $\Delta l$, two foam-air interfaces are created ideally resulting in additional pressure reduction as opposed to a single barrier foam of same thickness. Fig. 15 is a peak pressure plot confirming the above mentioned effect. Here the peak pressure is plotted as a function of $l_{foam}$ for varying $\Delta l$ with constant $l_{SOD}$ where the black dashed lines represents the peak pressure.
Figure 14: Single barrier foam comparison between pressure reduction by wave propagation inside foam and pressure reduction across foam-air interface. The configuration is as in Fig. 11(a).

measured for a single barrier foam of same thickness. The single barrier can be thought of as one limiting case with ∆l set as 0. For all cases, as the ∆l increases, the reduction in peak pressure also increases for a particular $l_{foam}$. The reduction is relatively higher for shorter $l_{foam}$, while for larger $l_{foam}$ the reduction tends to approach similar values. Again the reduction is large for smaller $l_{SOD}$ but this is owing to the fact that smaller $l_{SOD}$ foams intercept a relatively weaker wave.

An x-t plot depicting the decoupled analysis of ∆l and $l_{foam}$ in detail are as shown in Fig. 16. For both the cases, $l_{SOD}$ is maintained constant at 4 cm. The first dashed lines indicate the first barrier and the next indicate the second barrier. All solid lines indicate shock propagation in air. In Fig. 16(a), $l_{foam}$ is varied from 4 cm to 8 cm while maintaining the ∆l at 4 cm. Until the shock wave reaches the second foam barrier, the analysis is the same as for a single barrier. When the shock wave emanating from the first barrier’s FA interface impacts the second barrier, it again gets reflected and transmitted. The pressure rise due to this impact at the second AF interface indicates that the impacted shock wave is of considerably lower strength, thereby propagating at a much slower pace. This provides enough time for the first foam barrier following the incident shock wave to also impact the second foam barrier effectively increasing the pressure and providing momentum to the flow. Before
Figure 15: Peak pressure as a function of $l_{\text{foam}}$ for varying $\Delta l$ with (a) $l_{\text{SOD}} = 2$ cm (b) $l_{\text{SOD}} = 4$ cm (c) $l_{\text{SOD}} = 6$ cm and (d) $l_{\text{SOD}} = 8$ cm for a twin barrier foam.
Figure 16: x-t plot with (a) constant $\Delta l = 4$ cm and (b) constant $l_{foam} = 4$ cm for a twin barrier foam. $l_{SOD}$ is maintained constant at 4 cm for both the cases.

waves inside the second foam barrier are absent as it is impacted by a shock wave with step profile. If not for the following foam impact, much
Figure 17: Numerical pressure profile development as a function of (a) x and (b) time for a twin barrier configuration of \( l_{SOD} = 4 \text{ cm}, \Delta l = 4 \text{ cm} \) and \( l_{foam} = 8 \text{ cm} \).

higher pressure reduction could have been obtained with smaller \( l_{foam} \). The foam impact pressure rise is indicated on both the plots.

In Fig. 16(b), \( \Delta l \) is varied from 2 cm to 8 cm while maintaining the \( l_{foam} \) at 4 cm. Ideally larger pressures are expected from barriers formed away from the target than those formed closer as they are impacted by relatively stronger blast wave. This holds true until impact with the second barrier. When the second barrier is placed with small \( \Delta l \) from the first barrier, the distance between the emerging shock wave from the first barrier and the foam following it is very small that it immediately catches up with the transmitted wave in the second barrier, thus transferring all its momentum. Whereas when \( \Delta l \) increases, the distance between the emerging shock wave and the foam following increases too. For large \( \Delta l \), the transmitted wave in the second barrier propagates longer distance compared to small \( \Delta l \) before foam impact. The foam impact loses some of its momentum in compressing the second barrier before catching up with the transmitted wave which reduces the pressure registered at the second barrier’s FA interface. Therefore increasing the \( \Delta l \) from 2 cm to 8 cm results in 31% higher pressure reduction.

On the basis of impulse developed at sensor S3, the overall effect of different configurations in both single and twin barrier has been presented in Fig. 18. \( I \) represents the impulse calculated by integrating the overpressure registered over \( t^* \) for the configurations considered, where \( t^* \) represents the time following the impact of the shock wave until foam impact. Different configurations have different \( t^* \) and are therefore
Figure 18: Normalized impulse developed at sensor S3 for both single barrier (SB) and twin barrier (TB) foam configuration. $I_o$ represents the impulse developed in ambient air without foam. The integration is performed following the impact of the transmitted shock wave at sensor S3 until foam impact.

normalized with $I_o$, the impulse developed without foam in ambient air, calculated by integrating over the corresponding $t^*$. The effects observed are similar to peak pressure analysis. For a particular $l_{foam}$ with same $l_{SOD}$, the impulse developed is reduced just by splitting the barrier into two. A definite effect of increasing $\Delta l$ can also be seen. In both single barrier and twin barrier, increasing $l_{SOD}$ increases the impulse too.

5. Summary

In this work, we have examined the far-field performance of well characterized aqueous foam barriers on blast wave attenuation with particular emphasis on various foam barrier configurations. The strong blast waves of shock Mach number 4.8 were generated using exploding wire technique. A relatively stable and homogeneous wet foam with $\alpha = 0.1$, eliminated any of the drainage-related effects with time. There was good agreement between experiment and numerical simulation and it was used successfully for interpreting and complementing the experimental results. On impact with the foam barrier, it was found that the rate of attenuation of the transmitted wave in foam was much steeper close to the air-foam interface and it gradually approached a plateau with additional thickness.
However, the overall rate of attenuation was much faster as compared to the rate of blast decay in air. The impedance mismatch factor also played a significant role, aiding the dominant rate of decay mechanism. The transmitted wave in air was a shock wave with step-profile. In case of single barrier, overall pressure reduction in the range of 86% - 96.5% was observed for various configurations. The advantageous effect of the impedance mismatch factor was further enhanced by dividing the barrier equally forming two foam-air interfaces. This resulted in relatively greater pressure reduction as opposed to single barrier of the same thickness. The distance between the foam barriers were identified to have a positive influence on the overall peak pressure reduction.

Acknowledgement

The financial support of Vetenskapsrådet (The Swedish Research Council) is gratefully acknowledged.

References


Britan, A., Liverts, M., Shapiro, H. & Ben-Dor, G. 2012 Blast wave mitigation by a particulate foam barrier. *Transport in porous media* 93 (2), 283–292.


Britan, A., Shapiro, H., Liverts, M., Ben-Dor, G., Chinnayya, A. &


Chen, Y., Huang, W. & Constantini, S. 2012 Blast shock wave mitigation using the hydraulic energy redirection and release technology. PloS one 7 (6), e39353.


Paper 4
Plane blast wave propagation in air with a transverse thermal inhomogeneity

S. Sembian, M. Liverts, N. Apazidis
Department of Mechanics, KTH - Royal Institute of Technology, SE-100 44 Stockholm, Sweden


An alternate mechanism explaining the shock broadening and splitting effects observed during its propagation through an elongated region with transverse thermal inhomogeneity is described. The shock wave is generated by exploding wire technique and its propagation is captured optically using shadowgraph method. Visualizing the flow provided distinct advantage not only for obtaining detailed information on the propagation characteristics but also for validating the numerical scheme used in the analysis. Three physical features namely shock jump, precursor region and vorticity induced flow, are identified to contribute to the shock structure with the latter two being responsible for the pressure profile ‘broadening’. The physical behaviour of the incident shock is also analysed along with other factors like temperature and curvature effects.

Key words: Thermal inhomogeneity, Shock curvature, Shock broadening/splitting, Exploding wire

1. Introduction

The propagation of a shock wave through thermally inhomogeneous media results in shock acceleration in the higher temperature region and consequent shock broadening and splitting. Early experiments (Klimov et al. 1982; Gorshkov et al. 1984; Basargin & Mishin 1985; Klimov & Mishin 1990; MishinI et al. 1991; Gridin et al. 1994; Bedin & Mishin 1995; Ganguly et al. 1997; Bletzinger & Ganguly 1999; Garscadden et al. 1999; Adamovich et al. 1998; Ionikh et al. 1999; Macheret et al. 1999; Aithal & Subramaniam 2000; White & Subramaniam 2001; Macheret et al. 2001; Soukhomlinov et al. 2002; Kremeyer et al. 2002; Bletzinger et al. 2003) sparked numerous opinions to explain the fundamental mechanism(s) that
Table 1: Discharge tube size and electrode diameter of some of the reported facilities.

<table>
<thead>
<tr>
<th>Tube material</th>
<th>Tube cross-sectional dimension [mm]</th>
<th>Electrode diameter [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Klimov et al. (1982)</td>
<td>Plastic 100 × 100</td>
<td>10-60</td>
</tr>
<tr>
<td>Gorshkov et al. (1984)</td>
<td>Quartz φ 35</td>
<td>30</td>
</tr>
<tr>
<td>Ganguly et al. (1997)</td>
<td>Pyrex φ 50</td>
<td>30</td>
</tr>
<tr>
<td>Bletzinger et al. (1999)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Garscadden et al. (1999)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bletzinger et al. (2003)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Princeton</td>
<td>Quartz φ 38</td>
<td>2-25</td>
</tr>
<tr>
<td>Ionikh et al. (1999)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Macheret et al. (2001)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>White et al. (2001)</td>
<td>Pyrex φ 50</td>
<td>25</td>
</tr>
</tbody>
</table>

cause these effects exhibited by the shock wave. Since those experiments were conducted in weakly ionised plasma, some researchers were inclined towards the inherent plasma-specific phenomena. For example, Klimov et al. (1982) reported that the measured velocity of the shock front in the plasma was higher than the calculated velocity in a thermal inhomogeneity and concluded that releasing the energy stored in the vibrational degree of freedom was causing the shock acceleration. They also proposed that the energy of the electron-ion recombination and the energy of the association of atoms into molecules may be released in this region. Gorshkov et al. (1984) reported that the measured velocity of the shock front reduced in the presence of a transverse magnetic field further supporting the plasma-specific mechanism. Ganguly et al. (1997); Bletzinger & Ganguly (1999); Garscadden et al. (1999) studied the effect of plasma polarity on shock propagation and reported that the formation of a travelling strong double layer near the shock front leads to local electron heating, excitation, ionization and local gas heating with the latter being responsible for shock front broadening and velocity changes in addition to the effect of overall discharge heating.

Adamovich et al. (1998) found that there was not enough energy stored because of the low ionization fraction in the discharges and concluded that such effects could hardly be explained by the plasma-specific mechanism. Aithal & Subramaniam (2000) reported that wall shear could also result in a ‘split’ signal provided that the shock is very weak
Blast wave interaction with thermal inhomogeneity

(Mach number < 1.025) and propagates longer distances. On the basis of detailed experimental and numerical data (Aithal & Subramaniam 2000; White & Subramaniam 2001; Macheret et al. 2001; Kremeyer et al. 2002), it was found that the plasma effects have a thermal nature and can be attributed to the longitudinal and transverse temperature gradients. They emphasised that the fundamental mechanism depends on the transverse temperature gradients resulting in a multi-dimensional nature of shock i.e curved shock and the shock curvature was misinterpreted as ‘broadened’ and ‘split’ one-dimensional structure in previous experiments. Therefore they concluded that the main mechanism behind the effects was the shock curvature resulting from spatial inhomogeneity of temperature.

The reported experimental setups at most were fairly similar to one another. Shock waves were produced by a spark discharge resulting in a blast shaped profile. Plasma, either continuous or pulsed, was generated by applying a large potential difference across two ring electrodes housed in a discharge tube. Table I summarises the size of the discharge tube and electrode diameter of some of the facilities.

Laser schlieren method (Kiefer & Lutz 1965) was employed for detecting the shock wave front in Ganguly et al. (1997); Bletzinger & Ganguly (1999); Garscadden et al. (1999); Adamovich et al. (1998); Ionikh et al. (1999); Macheret et al. (1999); White & Subramaniam (2001); Macheret et al. (2001); Bletzinger et al. (2003). The ‘split’ schlieren signals were recorded using this method. Macheret et al. (2001) measured the temperature profile of the discharge using ultraviolet filtered Rayleigh scattering. The profile was fitted with a gaussian curve with the steady-state centerline temperature varying from 440 - 830 K (± 8%) to a wall temperature of 300 K, ideally filling the entire tube with a transverse temperature gradient. White & Subramaniam (2001) also measured the centerline temperature and found it to be 1009 K. The experimental error in their temperature measurements was 20%. However, depending on the discharge strength and the ratio between the electrode diameter to the tube inner diameter, there is a probability for existence of a cold gas region near the walls. For instance, Klimov et al. (1982) used electrodes with diameters 10-60 mm to produce plasma in a 100x100 mm channel. Although no temperature profile measurements were made, the centerline temperature was reported to not exceed 1000 K. Evidently, with the smallest electrode diameter, there is a high probability for the hot region (plasma) to be surrounded by an unaffected cold gas region. In addition, to the knowledge of the authors, the oscilloscope traces from the piezoelectric pressure gauge (Klimov et al. 1982) showing the ‘broadened’ pressure profile has not been reproduced. Also lacking is an experimental
image capturing the shock structure in the inhomogeneity to put the discussion to rest.

In this paper, we describe an alternate mechanism, causing similar effect in a special case where the test area consists of a hot transverse gas region embedded in the cold surrounding air. The dynamical characteristics of the leading shock and the post-shock region are investigated combining both experimental and numerical efforts. Since it was shown that the effects have a thermal nature, we employ an alternative way to generate heat on the principle of Joule heating by subjecting a wire to conduct current. In a certain way this also provides an advantage of ‘clean’ heat generation without additional plasma effect. The shock wave is generated using exploding wire (EW) technique and the flow is visualized using shadowgraph method. Numerical simulations are performed by solving the full set of Navier-Stokes equations to assist in analyzing and understanding the observed flow features.

This paper is organised as follows. In section 2, we describe the overall experimental setup including the description of heat generation and optical system. Section 3 presents the governing equations solved along with its validation. In section 4, the results obtained along with other factors affecting it are discussed and finally in section 5, this work is summarised.

2. Experimental setup

The experiments were conducted in a test cell consisting of two chambers namely, exploding chamber and test chamber. A schematic of the test cell is shown in Fig. 1(a). A uniform channel of length 360 mm, width 74 mm and height 5 mm runs continuously across both chambers. The exploding chamber houses two electrodes across which a 74 mm long, 0.4 mm diameter copper wire is screwed. The electrodes are connected to a
Blast wave interaction with thermal inhomogeneity

6 µF, 30 kV capacitor via a spark gap. By triggering the spark gap, the capacitor initially charged to 12 kV is rapidly discharged through the connected wire. As a result of high current flowing through the wire, it undergoes rapid Joule heating during which the wire melts and vaporizes. A cylindrical blast wave moving outward of the EW axis is generated during the expansion phase of the vapour column. The cylindrical blast wave is then quickly modified by the narrow confined rectangular body of the test cell into a plane blast wave which propagates into the test chamber. The top and bottom wall of the test chamber are made up of 20 mm thick Plexiglas plates which can provide a maximum field of view of 200 mm in length for visual access.

Fig. 1(b) represents a cut section view of the test chamber. Two conducting rods, solid (anode) and hollow (cathode), of 3 mm diameter are secured in the bottom plate as shown in Fig. 1(b). The anode which is mounted at 200 mm from the wire explosion plane cuts across the entire 5 mm channel height while the cathode is adjusted to half the height of the channel. The electrodes are made up of brass and the distance between them is 145 mm. A 0.2 mm copper wire is wound around the anode on one end while the other end is passed through the hollow cathode and is attached to a freely suspended weight, making the wire continuously tensed. The wire is then subjected to 15 V, 2.8 A current resulting in heating of the wire. This in turn heats the surrounding gas resulting in hot gas concentrated near the wire and cold gas region around it. Note that the 0.4 mm wire in the exploding chamber generates blasts while the 0.2 mm wire in the test chamber generates heat. Hereinafter ‘wire’ always corresponds to generating heat unless otherwise stated.

The propagation of the shock wave through the inhomogeneity is captured as shadowgraph images by an optical setup as shown in Fig. 2. The light source is a single pulsed, 532 nm Nd:YAG laser with a pulse width of 4-6 ns and pulse energy of 17 mJ. Since this being a single pulsed laser, only one image per test run can be obtained. The beam from the source is expanded horizontally to 180 mm and rendered parallel by a concave (L1) and convex lens (L2) of focal length (f) -6 mm and 1350 mm respectively. It is then redirected vertically to pass through the test chamber and back to the horizontal plane by a set of two mirrors tilted 45°. The image of the shadowgraph plane is then focused on the camera sensor (Nikon D90) by convex lenses L3 (f=1350 mm) and L4 (f=100 mm). The images captured using this setup had a spatial resolution of 0.07 mm/pixel. A neutral density filter was placed close to the sensor, opaque enough to damp the intense flash coming from the explosion,
room and other equipment illumination, while transparent enough to pass the laser light pulse.

![Shadowgraph optics setup](image)

**Figure 2:** Arrangement of shadowgraph optics.

![Pressure profiles](image)

**Figure 3:** Pressure profiles recorded at 180 mm from the wire explosion plane for a sample population of 5 tests.
To ensure repeatability, a special test was conducted by flush mounting a piezoelectric pressure sensor (113B24 PCB piezotronics) 20 mm ahead of the anode. The test was conducted 5 times keeping the initial conditions identical. The pressure profiles recorded by the sensors demonstrating the effect of wire explosion and subsequent blast formation are as shown in Fig. 3. The uncertainty in determination of peak pressure amplitude and time of arrival is $\pm 5.5\%$ and $\pm 0.5\%$ respectively. As such, the repeatability of the blast generation can be considered good.

When the power supply is turned on, the heat from the wire will be emanating cylindrically heating up the nearby gas. In terms of time, the longer the current is supplied, the larger will be the heat spread. Shadowgraph images showing the effect of current supplied time to the spreading of heat is presented in Fig. 4. As the height of the channel is only 5 mm, the heat from the wire reaches the Plexiglas plates which also conduct heat. When the plexiglas surface comes into contact with the hot gas, it becomes soft and pliable at the contact region. The further the hot gas spreads, the further the plate gets affected. Ultimately, the plate heating results in affecting the optical properties at the surface making it partially opaque. This in fact is captured as the dark region representing the hot gas around the wire in the shadowgraph images. At
240 s [Fig. 4(g)], the heat has spread to 12.9±1 mm in the transverse direction.

3. Numerical simulation

3.1. Physics modeling

The numerical simulations are performed using an in-house code solving a full set of 2D compressible Navier-Stokes equations:

\[ U_t + (F_i + G_i) - (F_v + G_v) = 0 \]  \quad (1)

where \( U \) represent the vectors of conservative quantities, \( F_i \) and \( G_i \) the vectors of inviscid flux, and \( F_v \) and \( G_v \) the vectors of viscous flux in \( x \) and \( y \) direction respectively.

\[
U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, \quad F_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho E u + pu \end{bmatrix}, \quad G_i = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho E v + pv \end{bmatrix}
\]  \quad (2)

\[
F_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} - q_x \end{bmatrix}, \quad G_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{yy} \\ u\tau_{xy} + v\tau_{yy} - q_y \end{bmatrix}
\]  \quad (3)

where

\[
\tau_{xx} = 2\mu x - \frac{2}{3}\mu(u_x + v_y)
\]
\[
\tau_{yy} = 2\mu y - \frac{2}{3}\mu(u_x + v_y)
\]
\[
\tau_{xy} = \mu(u_y + v_x)
\]
\[
q_x = -\kappa T_x
\]
\[
q_y = -\kappa T_y
\]  \quad (4)

Here \( \rho, u, v, p, E, T \), denote density, particle velocity components in \( x \) and \( y \) direction, pressure, specific total energy, and temperature respectively. Dynamic viscosity (Johnson 1998) (\( \mu \)) and thermal conductivity (\( \kappa \)) are temperature dependent and are calculated using the following relations:

\[
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{1.5} \frac{T_0 + 110}{T + 110}
\]
\[
\kappa = 1.527e^{-11}T^3 - 4.8574e^{-08}T^2
\]
\[+ 1.0184e^{-04}T - 0.00039333
\]  \quad (5)
The above equation for $\kappa$ is obtained by curve fitting the data found in Stephan & Laesecke (1985). The system of equations are closed by modeling the ideal gas law. The equations are solved numerically using an improved version of the artificial upstream flux vector splitting (AUFS) scheme implemented on an unstructured triangular mesh. The scheme, named AUFSR, was proposed by Tchuen et al. (2011) and obtained by hybridising of the AUFS (Sun & Takayama 2003) and Roe solver. The AUFSR scheme is robust for shock-capturing and also allows for resolving shear layers. Detailed description of the scheme along with examples demonstrating computational accuracy can be found in Tchuen et al. (2011).

### 3.2. Problem geometry and validation

![Figure 5: Computational domain.](image)

The computational domain for the problem with hot gas region is as shown in Fig. 5. The initial conditions for the numerical simulation were obtained by performing an experimental wire explosion test in ambient air without hot gas. For this purpose, three piezoelectric pressure sensors, S1-S3 (113B24 PCB Piezotronics), were flush mounted on the Plexiglas window at 205 mm, 245 mm and 285 mm from the wire explosion plane, respectively. The experiment was simulated numerically by replacing the wire with a high pressure region. The initial conditions chosen by trial-and-error method to fit the experimental pressure traces obtained from the sensors were:

$$
(\rho, u, p) = \begin{cases} 
(140 \text{ kg/m}^3, 0 \text{ m/s}, 150 \text{ bar}) & \text{for } 1 \text{ mm} \leq x \leq 3 \text{ mm} \\
(1.17 \text{ kg/m}^3, 0 \text{ m/s}, 1.01 \text{ bar}) & \text{for } x \leq 1 \text{ mm} \& x \geq 3 \text{ mm}
\end{cases}
$$

The entire domain was meshed with 3.2 million cells refined along the centerline of the domain. The mesh size varied slowly and smoothly as we go outwards from the center. The smallest and the largest cell size were 0.05 mm and 0.15 mm respectively. The grid was found to be sufficiently fine to resolve the important structures observed in the
experiments. Also the values of the properties of the flow have stagnated exhibiting mesh independency. As can be seen in Fig. 6, the numerical simulation prediction in terms of position, speed and pressure profiles of the blast wave agree within ±6% at each sensor. The shock Mach number at each sensor is 2.15±0.05, 2.04±0.05 and 1.98±0.05 respectively. The disturbances due to blockage by the 3 mm diameter anode and 0.2 mm diameter wire for shock propagation are shown in Fig. 7. Since only one image can be captured per shot, all other sensors were removed and a reference sensor S4 (113B24 PCB Piezotronics) was mounted at 180 mm from the wire explosion plane (20 mm ahead of the anode). By registering the instant of laser pulse, the time stamp for each experimental image was obtained with reference to the instant of blast location at sensor S4. As known already, the shock diffraction around the solid cylinder results in an incident wave, reflected wave \(R_c\) and a Mach stem connected to each other at the triple point. The collision of the Mach-Mach stem \(MM_c\) behind the cylinder gives rise to shock patterns as shown in Fig. 7(a). As it travels downstream, the initial reflected waves get reflected from the side walls forming a ‘concave’ structure as in Fig. 7(b). Further downstream, the reflections are weakened as seen in Fig. 7(c). The results of numerical simulation performed with a 3 mm cylinder are shown as density gradient image beside each shadowgraph image.
magnitude of the density gradient field is calculated as (Quirk & Karni 1996),

\[ |\nabla \rho| = \left[ \left( \frac{\partial \rho}{\partial x} \right)^2 + \left( \frac{\partial \rho}{\partial y} \right)^2 \right]^{\frac{1}{2}} \]  

(7)

Note that the wire is not included in the simulation. The simulation captured important shock structures observed in experiments with good accuracy. It also suggests that the wire by itself does not induce any significant disturbances in the flow.

Figure 7: Experimental shadowgraph (left) and numerical density gradient (right) images captured at three time instants, (a) 75 \( \mu \)s (b) 150 \( \mu \)s and (c) 210 \( \mu \)s, depicting the disturbance effects for a case without hot gas. The dimensions along \( x \) are from the reference sensor location.
S. Sembian, M. Liverts, N. Apazidis

Figure 8: Shock propagation through a uniform hot gas region surrounded by a cold gas region are captured as experimental shadowgraph (left) and numerical density gradient (right) images at three time instant. (a) at 40 µs shows the marked distinction between the elliptic shaped hot gas (dark) and cold gas; (b) at 110 µs depicts the incident, reflected, transmitted, precursor shock, Mach stem and the quadruple point; (c) at a later time instant (150 µs), the vorticity induced flow is clearly visible along with the shock structure.

4. Results and Discussion

When the power supply delivering 2.8 A current is turned on, the heat generated by the wire heats the surrounding resulting in a hot gas concentrated near the wire as depicted by the dark region in Fig. 8(a).
Figure 9: (a) Experimental shadowgraph image; (b) Numerical schlieren with step-wise temperature variation; (c) Numerical schlieren with gradual temperature variation.

The area unaffected by heat is termed cold gas and is represented by the visible region. Note that a few dark ring like structures present in the shadowgraph images are interferometric fringes produced by the optical system and not by the gas heating. The hot gas region shape resembles an ellipse along the longitudinal direction with its maximum width occurring at the center. The peak width is measured to be $12.9\pm1$ mm before the passage of shock. The anode and cathode are thus located at the vertices of the ellipse. The cold gas is at room temperature (298 K). When an incident shock wave propagates through such a medium, it gets reflected and transmitted. Since the shock structures arising due to disturbances to the flow are already known, the effect induced by the hot gas can be easily separated. For instance, the shocks $R_c$ and $MM_c$ in Fig. 7 can be identified in Fig. 8 and neglected from the analysis. Since the speed of sound in the hot gas is higher compared to the cold gas, the absolute velocity of the transmitted shock wave is also higher and therefore it travels faster in the hot region initiating a quadruple point as shown in Fig. 8(b). The quadruple point consists of the incident shock, reflected shock, precursor shock and Mach stem. There is also a baroclinic vorticity generated flow field behind the transmitted shock wave which can be clearly distinguished at a later time instant [Fig. 8(c)]. Also at this instant, the distance between the transmitted shock wave and the quadruple point has increased with the quadruple point moving further away from the hot region.

4.1. Temperature distribution

It is important to determine the temperature in the hot gas region along with its distribution in the transverse direction. On close observation,
it is very interesting to note that the transmitted shock in Fig. 8(b)-(c) is very slightly curved. Also the transition between the transmitted and the precursor shock is rather immediate at the interface. In the presence of a temperature gradient along the transverse direction, the shock will always be curved (Aithal & Subramaniam 2000; Macheret et al. 2001; Kremeyer et al. 2002). But the shock structures observed here are not curved and it is possible only with step-wise temperature distribution. Fig. 9 shows comparison between an experimental and numerical shadowgraph images with step-wise and gradual temperature variation. The initial temperature conditions used for simulation will be described later.

Of course in reality, there will always be gradients in the presence of a heat source contradicting the experimental images obtained. In the present case, the plexiglas heating plays a dominant role in creating a step-wise temperature distribution. As the plate surface becomes soft and pliable due to heat, its material properties change resulting with the temperature across the affected region becoming uniform. Such temperature distribution results in unbowed shock structure seen in Fig. 9(a). This also works to our advantage as the mechanism is now void of shock curvature which was solely responsible for shock broadening and splitting in previous studies. But for the sake of completeness, simulations are performed with two separate temperature distribution along the transverse direction: (a) Step-wise distribution with a uniform hot gas temperature region surrounded by cold gas and (b) Exponential distribution (gradual temperature variation) case. This simplified model also provides an opportunity to study the influence of shock curvature on the mechanism under study.

In order to perform a numerical simulation, the following are the approximations made:

- The shape of the hot gas region is already known to be elliptic from experimental images with the peak width measured to be 12.9 mm. The distribution along the longitudinal direction is assumed to be uniform.
- As heat conduction also occurs in the plates coupled with the very small channel height/width ratio, the temperature distribution across the channel height is approximated to be uniform representing a 2D problem.

4.2. Step-wise temperature distribution

The temperature of the hot gas is determined by matching the distance travelled by the incident shock and transmitted shock from the reference
Figure 10: Numerical pressure profiles registered at six locations from the reference point along the centerline of the chamber. Time zero refers to the time at which the shock wave reached the reference location. The incident shock has a blast profile and the hot gas shape is elliptic.

S4 location. By trial-and-error method, the temperature is found to be 700 K. For each experimental image, the corresponding numerical density gradient for a 2D, elliptic shaped uniform hot region (700 K) surrounded by cold region (298 K) is plotted on the right side of Fig. 8. There is good agreement between experiments and numerical simulation. The qualitative features of the flow are captured accurately indicating that the approximations are reasonable. The numerical pressure profiles are plotted in Fig. 10. Time zero in the pressure profiles refers to the time at which the shock wave reached the reference location. As opposed to the blast pressure profiles in Fig. 6, significant broadening (A-E) of the profile (Fig. 10) in the presence of an inhomogeneity is observed. The broadening increases as the shock front propagates further downstream. Qualitatively this broadening is similar to the oscilloscope pressure trace recorded by Klimov et al. (1982).

During the initial rise of the profiles, for instance at the fourth profile (100 mm from the reference location), a definite jump (A-B) followed by a more gradual increase (B-C) in pressure can be seen. The line A-B behaves like a shock jump with a very short rise time. By comparing with the images in Fig. 8, the jump A-B corresponds to the transmitted shock wave propagating through the hot gas. Therefore the rise A-B can
be termed ‘shock jump’. The pressure value at point B is 2.63 bar and
the corresponding shock Mach number is 1.54. The trend of the short
dashed line which separates the two curves is decreasing implying that
the transmitted shock wave decelerates. The deceleration could also be
because of the decelerating blast profile of the incident wave.

Figure 11: Contour plots of numerical pressure, density and u-velocity
along with velocity vector plot at a particular time instant. All dimensions
are in mm.

The transmitted shock wave does not influence the curve B-C as it
occurs behind it. In order to determine the physics behind curve B-C, let
us consider the contour fields of pressure and density as in Fig. 11(a)-(b)
respectively. Below the quadruple point, pressure is continuous but an
interface separating the fluids worked by the transmitted shock and the
precursor shock can be noted in the density field [Fig. 11(b)]. Also it
is interesting to note the mushroom shaped vorticity induced region.
The value of density in the region behind the precursor shock is not
uniform as indicated by the contour lines, rather it increases along the
Figure 12: Numerical (a) schlieren signal and (b) pressure development computed at 20, 75, 115, 160 and 205 $\mu$s from the reference location. Time zero refers to the time at which the shock wave reached the reference location.

precursor shock. The pressure contour lines originating from the precursor shock also indicate a gradual rise in pressure as we move behind the transmitted shock from point B to point C in Fig. 11(a). Note that C is the maximum pressure point for this increase. The contour lines suggest that the strength of the precursor shock varies despite its oblique nature. Why does the strength of the precursor vary? Our argument is based on the relative velocity of the shocks. The precursor shock bridges the faster moving (low Mach number) transmitted shock to the relatively slower moving (high Mach number) incident shock through the quadruple point. On revisiting Fig. 8, it can be noted that the initially straight incident shock later bends, suggesting that the transmitted shock tends to pull the incident along with it by imparting more momentum. The precursor shock which connects both should be continuous and therefore varies smoothly in strength along its length. If it had been of constant strength, then there would be abrupt discontinuities in the thermodynamic variables at the connection points which will be unreal. Hence it is the precursor region comprising of the precursor shock, quadruple point and the Mach stem influencing the curve B-C.

The pressure gradient behind the shock and the density gradient are effectively perpendicular to each other. The baroclinic source term in the vorticity transport equation, given as:

$$\frac{D\omega}{Dt} = \frac{1}{\rho^2}(\nabla\rho \times \nabla p)$$ (8)
represent the rate of vorticity generation due to baroclinicity in the flow. The resulting two counter-rotating vortices, generated by the baroclinic vorticity, induces ‘jetting’ in the channel between the two vortices as indicated by the high velocity region in Fig. 11(c). The vorticity-generation mechanism is similar to Richtmyer-Meshkov instability and the mushroom-shaped structure is a result of interface instability, an inherent feature also observed in shock-bubble interaction problems (Johnsen & Colonius 2009; Ranjan et al. 2011; Apazidis 2016). The pressure begins to drop on entering the mushroom-shaped structure reaching a minimum at point D, where the jet velocity is at its maximum. It then increases and reaches an overall maximum at point E, located outside the vortices in the surrounding fluid. Therefore the curves C-D and D-E are due to baroclinic vorticity induced flow field.

The broadening of pressure profile is due to the three underlying physical features outlined above. Based on the evidence provided, using the term ‘shock broadening’ for the present case is misleading as the transmitted shock front does not ideally broaden or disperse. It has a definite jump and its strength can also be measured. It is in fact the 2D shock structure with the conditions imposed by the precursor region and the vorticity giving rise to pressure variations creating the post-shock broadened profile.

Fig. 12 shows the simulated schlieren signal and pressure development at different x-locations computed at 20, 75, 115, 160 and 205 µs from the reference location. The signals are obtained by averaging the density gradient along the transverse direction. Intensity fluctuations due to shock reflection from the boundary and the blockage are filtered out. These signals are similar to the measured signals in Ganguly et al. (1997); Macheret et al. (2001); White & Subramaniam (2001). Before entering the inhomogeneity, the incident shock wave recorded a delta-function like shape with a single peak. Upon entering, two-peak signal shapes appear. The weak first peak is due to the transmitted shock, the strong second peak is due to the incident shock and the signal following the first peak is due to the precursor shock. Previously, this was attributed to the curvature of the shock (Macheret et al. 2001; Aithal & Subramaniam 2000; White & Subramaniam 2001). Also as we move along the length of the channel, the distance between the peaks increases (Ganguly et al. 1997). Therefore the shock ‘splitting’ signatures are observed even in the absence of curvature effects and can be explained by the present mechanism.
Figure 13: Numerical pressure profiles for (a) without hot gas and (b) with hot gas conditions; registered at locations close to the wall along the length of the chamber from the reference point. Time zero refers to the time at which the shock wave reached the reference location. The incident shock has a blast profile and the hot gas shape is elliptic.

4.2.1. **Effect on Incident shock**

To investigate the bending of the initially straight incident shock, pressure profiles are plotted in Fig. 13 at points chosen near the wall so as to be far away from the hot gas region. The deceleration of the shock wave by the ‘catching-up’ of expansion waves in a simulation with the absence of hot gas is as plotted in Fig. 13(a). The small jumps in pressure following the peak in the first few profiles are due to the shocks $R_c$ and $M M_c$. Now on including the hot gas, it can be seen that the wave decelerates initially as in Fig. 13(a), momentarily accelerates and then starts to decelerate again. The profile following the peak also becomes much steeper. The increase in shock speed is due to the effect of the faster moving transmitted shock tending to pull it by imparting momentum against the expansion waves. The acceleration could be even higher than predicted if not for the expansion waves. Once a threshold is reached, the expansion waves then again decelerate the incident shock.

4.2.2. **Temperature effects**

As the flow field is dependant on temperature, it is important to look at the effects of varying the temperature of the hot gas. Note that the cold gas temperature is unchanged. Fig. 14(a) is a comparison between pressure traces recorded at 100 mm from the reference location for three different temperatures. As the temperature increases, the transmitted
shock travels much faster albeit with a much lower intensity as shown by the shock jump occurring location and pressure value. The precursor region effects are also more pronounced at higher temperature although the peak pressure value (corresponding to point C) decreases. Also higher temperatures are associated with large baroclinicity resulting in high jet velocities and large overall drop in pressure.

Figure 14: Pressure traces recorded for three different hot gas temperatures (500 K, 700 K, 1000 K) at 100 mm from the reference point.

4.3. Exponential distribution

The maximum temperature attained along the wire is determined by solving the Joule heating problem with COMSOL Multiphysics package. The problem is solved in 3D by including the convective and radiative effects. The modelling also assumes no thermal expansion of the wire. When the copper wire is subjected to 2.8 A current, the maximum temperature \( T_{\text{max}} \) calculated along the centerline is 1020 K. The distribution is approximately fitted by a gaussian curve: \( T(y) = T_{\text{max}} \exp[-(200y)^2] \). It is also to be noted that varying the distribution function only slightly affected the results. Fig. 15 shows the simulation results obtained for an initial condition with temperature gradient. The curvature of the transmitted shock along with a gradual transition to precursor shock can be seen in Fig. 15(a). The vortices are not as pronounced as with a step-wise temperature case along with the notable absence of rolling up
of the vortices. Contour lines can still be seen emerging from the pre-cursor shock in Fig. 15(b)-(e). The pressure profile plotted in Fig. 15(f) is largely unaffected. Evidently, the influence of shock curvature on the overall physical process is marginal. When a temperature gradient case without the cold gas region is considered, the laser schlieren signal will register a single peak structure followed by a tail when the shock reflection at the walls are neglected. In previous experiments, due to the rapid nature of the discharge process combined with the very slow heat conduction rate to the surrounding, the temperature between the discharge-heated gas and the surrounding could vary rather abruptly at a particular point along the radial direction (although gradients could be present in the discharge core). In such cases, the two-peak nature of the acquired signals were due to the effects presented in section IV.B.
5. Summary

The qualitative features of shock propagation through a thermal inhomogeneity was investigated both experimentally and numerically. Visualizing the flow using shadowgraph technique provided detailed information of the propagation characteristics and a concrete foundation to base our results. The simulation was performed by solving the compressible Navier-Stokes equation with an accurate AUFSR scheme. There was good agreement between the experimental and numerical results. Based on the above results, three physical features namely shock jump, precursor region and vorticity induced flow, were identified to govern the overall shock propagation. Apart from the shock jump, the ‘broadening’ of the pressure profile was solely due to the latter two processes occurring due to temperature difference in the field. It was also shown that laser schlieren signals with two-peak structure was not specific to shock curvature effect. The incident shock was provided momentum by the pulling effect of the transmitted shock. The observed features were largely unaffected by temperature variations and shock curvature phenomenon.

6. Acknowledgement

The financial support of Vetenskapsrådet (The Swedish Research Council) is gratefully acknowledged.

References


Ganguly, B. N., Bletzinger, P. & Garscadden, A. 1997 Shock wave


Paper 5
Plane blast wave interaction with an elongated straight and inclined heat-generated inhomogeneity

S. Sembian, M. Liverts, N. Apazidis
Department of Mechanics, KTH - Royal Institute of Technology, SE-100 44 Stockholm, Sweden

Journal of Fluid Mechanics (2018), accepted for publication

The unstable evolution of an elongated elliptically-shaped inhomogeneity that is embedded in ambient air and aligned both normal and at an angle to the incident plane blast wave of impact Mach number 2.15 is investigated both experimentally and numerically. The elliptic inhomogeneities and the blast waves are generated using gas heating and exploding wire technique and their interaction is captured optically using shadowgraph method. While two symmetric counter rotating vortices due to Richtmyer-Meshkov instability (RMI) are observed for the straight interaction, the formation of a train of vortices similar to Kelvin-Helmholtz instability (KHI), introducing asymmetry in the flow field are observed for an inclined interaction. During the early phase of interaction process in the straight case, the growth of the counter-rotating vortices (based on the sequence of images obtained from the high speed camera) and circulation (calculated with the aid of numerical data) are found to be linear in both space and time. Moreover, the normalized circulation is independent of the inhomogeneity density and the ellipse thickness, enabling the formulation of a unique linear fit equation. Conversely, the circulation for an inclined case follows a quadratic function, with each vortex in the train estimated to move with different velocity directly related to their size at that instant. Two factors influencing the quadratic nature are identified: the reduction in strength of the transmitted shock thereby generating vortices with reduced vorticity along with the gradual loss of vorticity of the earlier generated vortices.

1. Introduction
Strong coupling of several fluid dynamic phenomena occurs when a shock wave interacts with a density inhomogeneity. This interaction of pressure
discontinuity at an interface between two fluids results in the amplification of disturbances due to baroclinic deposition of vorticity that finally develops into regions of intense mixing, a phenomenon often referred to as Richtmyer-Meshkov instability (RMI). Secondary instabilities, like shear induced Kelvin-Helmholtz instability, develop at a later time as the interface becomes more distorted. This class of problems - having diverse applications in the fields of supersonic combustion systems (Scramjet), astrophysical environments of supernovas, supersonic/hypersonic flow control and inertial confinement fusions - has been studied extensively after Richtmyer (1960) first provided the theoretical analysis, and Meshkov (1969) confirmed it experimentally. Brouillette (2002) and Zabusky (1999) gave a comprehensive review on RMI environments and shock accelerated inhomogeneous flows respectively.

Rich literature in the realm of RMI are found for the interaction of a shock wave with an isolated gas bubble, which is a fundamental yet vital configuration for studying the shock accelerated inhomogeneous flows. When a shock wave impinges on a bubble, baroclinic vorticity deposition evolves the interface into two counter-rotating vortices entraining fluid from the surrounding flow thereby enabling stirring and mixing. Much earlier to all these events, the impacted incident shock undergoes non-linear shock effects, like reflection, refraction and diffraction at the gas bubble. Generally, depending on the acoustic impedance of the gas bubble and the surrounding fluid, they are classified as either slow/fast or fast/slow configurations. The same can be classified on the basis of density as heavy/light or light/heavy respectively. A wide variety of investigations covering theoretical, experimental and numerical aspects on light and heavy, cylindrical (2D) and spherical (3D) gas bubbles were previously performed. For instance, pioneering work by Markstein (1957) utilized a reactive flame front of an oblong shape to study the shock interaction. Rudinger & Somers (1960) created bubbles using either a spark discharge or fine jet technique to study the effects experimentally and theoretically. Haas & Sturtevant (1987) reported on the shock nonlinear acoustics and evolution of spherical bubbles produced by soap film and 2D gas cylinder by nitrocellulose membrane. Picone & Boris (1988) numerically modeled the experiments conducted by Haas & Sturtevant (1987) and presented an analytical model for calculating circulation, a measure of net strength of the developed vortices. Another analytical model for circulation was formulated by Yang et al. (1994) based on their numerical results. A detailed high-resolution numerical study capturing the nonlinear acoustics and the vortices were performed by Quirk & Karni (1996). By implementing planar laser-induced fluorescence and
a seeded round laminar test jet, Jacobs (1992) improved the quality of flow visualization enabling measurement of species concentration and mixing. Layes et al. (2003, 2005, 2009) utilized a high speed camera for obtaining the evolution from a single run as compared to single shot images captured previously. Giordano & Burtschell (2006) and Layes & LeMetayer (2007) numerically simulated the experimental results of Layes et al. (2003). The effects of high incident Mach number (M = 2.95) was investigated experimentally by Ranjan et al. (2005, 2007) and numerically by Niederhaus et al. (2007), where they reported a secondary baroclinic source of vorticity. Departure from single cylinder configuration was initiated by Yang et al. (1993) who numerically simulated two and three cylinder interactions in various configurations. Tomkins et al. (2003) investigated twin cylinder interactions experimentally and later it was increased to multiple cylinders by Kumar et al. (2007). An in-depth review by Ranjan et al. (2011) offers a comprehensive discussion on the shock bubble interaction problem. The common factor in the above stated investigations is the classic circular shape of the bubble where studies on different interface shapes are limited. Besides the circular shape, Ray et al. (2000) studied the interaction of a shock wave with a heavy prolate ellipse and provided a heuristic model for calculating interfacial circulation. Bates et al. (2007) studied the evolution of a rectangular block experimentally and numerically. Large-eddy simulation on rectangular interface was performed by Wang et al. (2010). Zou et al. (2010) studied a heavy elliptic cylinder with various aspect ratios, Zhai et al. (2014) investigated the interaction with three polygonal interface (square, triangle and diamond) and most recently Georgievskiy et al. (2015) numerically studied the shock focusing process in both light and heavy prolate ellipse.

Despite these achievements, the inhomogeneities studied thus far are aligned normal to the incident shock wave introducing symmetry along the center axis. The effect of inclining an isolated bubble embedded in the surrounding fluid to the incoming flow field is an area unexplored and could potentially open up interesting features driving future investigations. This paper is focused on exploring the effects of such inclined isolated bubble introducing asymmetry in the flow field. Geometrically, a circular shape is always symmetric along any axis of incidence of the incident shock and therefore not suitable for studies pertaining to our problem. The closest shape to a circle covering the complete range of angles between the pressure and density gradients, and still permit inclination is an ellipse. Here we define the angle of inclination ($\alpha$) as the angle between the normal to the incident shock and the ellipse major axis.
Moreover, we are interested in the dynamic characteristics of the early time development of the vortices necessitating the need for an elongated ellipse. By early time, we mean the time taken for the fastest shock in the system to reach the downstream interface. But information on such early time development even for an uninclined ellipse ($\alpha = 0^\circ$) is very scarce that a rigorous treatment of it is essential before studying the inclination effects. Therefore the motivation in this paper is two folds: (a) first to study the early time development of vortices in a straight elongated elliptic configuration ($\alpha = 0^\circ$) and (b) extend the knowledge obtained to analyze an inclined elongated elliptic configuration.

Experimentally generating an elliptic inhomogeneity that can also be inclined is a vital part of the investigation. Inspired from Markstein (1957) and Rudinger & Somers (1960) where they utilized heat to create a slow/fast (heavy/light) density inhomogeneity, we implement a new technique based on the principle of ‘Joule heating’. This technique, successfully used in our previous work (Sembian et al. 2018) to create straight elliptic inhomogeneities ($\alpha = 0^\circ$), is extended in this paper to generate inclined ones. This inhomogeneity is impacted by a plane wave of shock Mach number 2.15 with blast profile generated using an exploding wire technique. An existing in-house numerical code solving a full set of 2D compressible Navier-Stokes equations is used for interpreting and analyzing the experimentally observed flow features.

Such elongated inhomogeneities are also used in energy deposition problems that are mainly targeted towards flow control and drag reduction in supersonic and hypersonic flights. Energy deposition is a method of controlling the aerodynamic forces and moments through localized regions of thin elongated low density/heated channels (Mirels 1986; Artem’ev et al. 1989, 1993; Nemchinov et al. 1994; Aleksandrov 1993; Grun et al. 1998; Svetsov 2001; Georgievskii et al. 2010, 2016; Knight et al. 2009; Azarova et al. 2013), plasma/glow discharge (Klimov et al. 1982; Gordeev et al. 1996; Ganguly et al. 1997; Macheret et al. 2001; Zudov et al. 2003) or jets (Zudov 2010; Tretyakov et al. 2012) generated ahead of the vehicle body. A selected survey on drag reduction at high speeds can be found in Knight (2008). Generally when a shock wave propagates through such channels with high speed of sound, a thermal precursor shock (a wedge-shaped disturbance) forms as a result of the acceleration of the refracted shock from the incident shock (Mirels 1986; Artem’ev et al. 1993). This phenomenon was observed much earlier in a classical work by Jahn (1956) due to irregular refraction at the interface between two gas layers. While the influence of magnetic fields on the flow was examined by Aleksandrov (1993), the generation of an intense vortex
motion ahead of the vehicle body was investigated by Artem’ev et al. (1993). Grun et al. (1998) measured the precursor thermal layer with respect to its growth as a function of time. Later Knight et al. (2009) and Azarova et al. (2010, 2013) established three types of vortex related instabilities namely, RMI, KHI and instability of flat-parallel tangential discontinuity and investigated its influence on drag reduction. Although the elongated heat channels closely resemble the inhomogeneities generated in this work, they were primarily studied from drag reduction enhancement point of view with little to no emphasis on the vortex development. Also literature on shock interaction with inclined channels were notably meagre besides the work of Zudov (2010) and Tretyakov et al. (2012) who studied an oblique shock interaction with a subsonic and supersonic jet where the main focus was on the jet characteristics.

2. Experimental setup

The experimental setup and the generation of the inhomogeneity is similar to that in our previous work (Sembian et al. 2018). The shock waves are generated by exploding a 0.4 mm diameter copper wire in the test cell consisting of an exploding chamber and a test chamber [figure 1(a)]. Both the chambers are constructed in a way that a uniform channel of length 360 mm, width 74 mm and height 5 mm runs throughout. The test cell is maintained at an ambient pressure of 1 atm. The wire to be exploded, from herein called as ‘exploding wire’, is screwed to the electrodes housed in the exploding chamber. The other ends of the electrodes are connected to a 6 \( \mu \text{F} \) capacitor charged to 12 kV. Upon discharge, the exploding wire is subjected to very high current resulting in the generation of a cylindrical blast wave. Since the generated cylindrical wave is confined in a narrow rectangular section, it modifies itself into a plane wave with a blast profile. This plane wave then propagates into the test chamber where a 0.2 mm diameter copper wire (from herein called as ‘hot wire’ or simply ‘wire’) is suspended from two 3 mm diameter brass electrodes, one solid (anode) and the other hollow (cathode). Different inclination angles can be obtained by properly positioning the electrodes. The wire is always maintained in a state of tension by passing its other end through a guide cylinder and then connecting it to a freely suspended weight as shown in figure 1(b). The top and bottom wall of the test chamber are made up of Plexiglas plates providing a maximum field of view of 200 mm in length for visual access. The wire is subjected to 24 V, 3.6 A current resulting in heating up of the wire. This in turn heats the air surrounding the wire producing a density inhomogeneity.
Figure 1: (a) Test unit (top view) (b) Cut section view of the test chamber (side view).

Figure 2: Current supplied time versus heat spread

The propagation of the shock wave through the inhomogeneity is captured as shadowgraph images by an optical setup. Two types of camera are used in capturing the flow structures. A high speed camera (Shimadzu HPV-X2) set to capture at 0.5 million frames per second is used for temporal resolution, and a digital still camera (Nikon D90) with 13 MP is utilized for spatial resolution. Images captured using this setup had a resolution of 0.42 mm/pixel and 0.07 mm/pixel respectively. Depending of the camera used, the light source is either a continuous mode argon-ion laser (Spectra-physics BeamLok 2060) or a single pulsed Nd:YAG laser (New wave Orion).

The effective thickness (b) of the inhomogeneity is influenced by the amount of time the wire is subjected to conduct current. Single-shot
shadowgraph images showing the effect of current supplied time to the spreading of heat is presented in figure 2. As the wire is suspended in the middle of thin 5 mm channel, the heat from the wire reaches the Plexiglas plates making it soft and pliable at the contact region. This affects the optical properties at the surface making it partially opaque as depicted by the dark region, indicated as ‘Hot gas’ in figure 2(b). After about 60 s, the heat has spread to $b = 10 \pm 2$ mm with an overall shape resembling an ellipse along the inclination axis. Since the heat fills the 5 mm channel (wall to wall) coupled with the subsequent heating of the top and bottom plates, the resulting inhomogeneity is approximated to be a 2D configuration. Although there exists a gradient at the interface with a finite thickness, the distribution of the temperature in the transverse direction can be safely approximated to a step function based on the shape of the transmitted shock. In the presence of large gradients, the transmitted shock will be curved but as will be shown in Sec 4, the transmitted shock in our case is very slightly curved leading to the step function approximation. Also the observed features were found to be largely unaffected by the temperature gradients as discussed in Sembian et al. (2018).

3. Numerical Simulation

The numerical simulations were performed using an existing in-house code solving a full set of 2D compressible Navier-Stokes equations with air as the test gas.

$$U_t + (F_{ax} + F_{by}) - (G_{ax} + G_{by}) = 0$$

where $U$ represent the vectors of conservative quantities, $F_a$ and $F_b$ the vectors of inviscid flux, and $G_a$ and $G_b$ the vectors of viscous flux in $x$ and $y$ direction respectively.

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}; \quad F_a = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho E u + pu \end{bmatrix}; \quad F_b = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho E v + pv \end{bmatrix}$$

$$G_a = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} - q_x \end{bmatrix}; \quad G_b = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ u\tau_{xy} + v\tau_{yy} - q_y \end{bmatrix}$$

where

$$\tau_{xx} = 2\mu u_x - \frac{2}{3}\mu(u_x + v_y); \quad \tau_{yy} = 2\mu v_y - \frac{2}{3}\mu(u_x + v_y);$$

$$\tau_{xy} = \mu(u_y + v_x); \quad q_x = -\kappa T_x; \quad q_y = -\kappa T_y$$
Here $\rho$, $u$, $v$, $p$, $E$, $T$, denote density, particle velocity components in $x$ and $y$ direction, pressure, specific total energy, and temperature respectively. Dynamic viscosity ($\mu$) (Johnson 2016) and thermal conductivity ($\kappa$) for air are temperature dependent and are calculated using the following relations:

$$
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{1.5} \frac{T_0 + 110}{T + 110}
$$

$$
\kappa = 1.527e^{-11}T^3 - 4.8574e^{-08}T^2 + 1.0184e^{-04}T - 0.00039333
$$

The equation for $\kappa$ is obtained by curve fitting the data found in Stephan & Laesecke (1985). Other relevant thermodynamic properties of air at normal atmospheric conditions are used. The system of equations are closed by modelling the ideal gas law. Given an interface with normal vector $\mathbf{n} = (n_x, n_y)$, $\mathbf{U}$ and $\mathbf{F}$ can be expressed in the direction of normal face $n$ as,

$$
\begin{bmatrix}
\rho \\
\rho q_n \\
\rho q_t \\
\rho E
\end{bmatrix} + \begin{bmatrix}
\rho q_n \\
\rho q_n^2 + p \\
\rho q_n q_t \\
\rho Eq_n + pq_n
\end{bmatrix} - \mathbf{G}_{x,y} = 0
$$

where the normal ($q_n$) and tangential velocity ($q_t$) through the interface is given as,

$$
q_n = un_x + vn_y \\
q_t = vn_x - un_y
$$

The equations are solved numerically using a scheme termed AUFSR proposed by Tchuen et al. (2014), that combines the efficiency of Artificial Upstream Flux Vector Splitting (AUFS) scheme with the accuracy of Roe’s solver. The AUFS is a special Flux Vector Splitting (FVS) scheme proposed by Sun & Takayama (2003), that introduces two artificial wave speeds into the flux decomposition for adjusting the direction of wave propagation. While its an accurate and robust technique for resolving entropy waves without the carbuncle problem (MacCormack 2011), it has some shortcomings while resolving shear layers (Kemm 2015). The Roe’s scheme is a Flux Difference Splitting (FDS) scheme that is known to resolve shear layers accurately but suffers from carbuncle problem. The AUFSR overcomes these disadvantages by implementing the artificial wave speed technique with the Roe’s solver thereby introducing certain amount of numerical dissipation to shear waves. Hence, the AUFSR scheme is not only robust for shock capturing but also accurate for resolving shear layers. The basic idea is to split the flux vector $\mathbf{F}$ in
Eq. (6) as,

\[ \mathbf{F} = (1 - \mathbf{S})\mathbf{F}_1 + \mathbf{S}\mathbf{F}_2 \]  

where \( \mathbf{F}_1 \) and \( \mathbf{F}_2 \) are the intermediate flux vectors, and \( \mathbf{S} = s_1/(s_1 - s_2) \), where \( s_1 \) and \( s_2 \) are the artificial wave speeds. \( \mathbf{F}_1 \) and \( \mathbf{F}_2 \), are calculated from,

\[ \mathbf{F}_1 = \frac{1}{2}(\mathbf{P}^L + \mathbf{P}^R) + \delta \mathbf{U} \]  

\[ \mathbf{F}_2 = \mathbf{U}^\zeta (u^\zeta - s_2) + \mathbf{P}^\zeta, \zeta = \begin{cases} L & \text{for } s_1 > 0 \\ R & \text{for } s_1 \leq 0 \end{cases} \]

The artificial viscosity, \( \delta \mathbf{U} \), is given as

\[ \delta \mathbf{U} = \frac{1}{2} \left[ (q_n^L - s_1)\mathbf{U}^L + (q_n^R - s_1)\mathbf{U}^R \right] - \frac{1}{2} \sum_{k=1}^{4} |\lambda_k|\eta_k R_k \]

where \( \lambda_k, \eta_k, \) and \( R_k \) are the eigenvalues, characteristic variables and eigenvectors calculated using Roe averaged values respectively.

The numerical values of \( s_1 \) and \( s_2 \) are computed using,

\[ s_1 = \bar{q}_n \]

\[ s_2 = \begin{cases} \min(0, q_n^L - c^L, \bar{q}_n - \bar{c}) & \text{for } s_1 > 0, \\ \max(0, q_n^R + c^R, \bar{q}_n + \bar{c}) & \text{for } s_1 \leq 0 \end{cases} \]

where,

\[ \bar{q}_n = \sqrt{\rho^L q_n^L + \rho^R q_n^R} \]

\[ \bar{c} = \left[ (\gamma - 1) (\bar{H} - \frac{1}{2} \bar{u}^2) \right]^{\frac{1}{2}} \]

\[ \bar{H} = \frac{\sqrt{\rho^L H^L + \sqrt{\rho^R H^R}}}{\sqrt{\rho^L} + \sqrt{\rho^R}} \]

More detailed description can be found in Tchuen et al. (2014).

3.1. Domain description and initial conditions

The computational domain of the simulated problem is as shown in figure 3. All boundaries are set with wall boundary conditions. Depending on \( \alpha \), the entire domain was meshed with 2.5 - 5.5 million cells (unstructured triangle) refined along the axis of inclination. The mesh size varied slowly and smoothly as we go outwards from the axis with the smallest and the largest cell size being 0.025 mm and 0.15 mm respectively. The values of the properties of the flow have stagnated
exhibiting mesh independency. Due to the associated complexities in the problem coupled with the need for large computing resources required for resolving the finer structures, the simulations are performed assuming a 2D inhomogeneity even for small $b$ values. Although this is an approximation, it will be shown later in Sec 4 that the prediction results will be identical with experiments, justifying the assumption. The numerical analysis can be thought of as an accurate first order estimate. The experiment was simulated numerically by replacing the exploding wire with a high pressure region with the following initial conditions:

\[
\begin{cases}
(140 \text{ kg/m}^3, 0 \text{ m/s}, 15 \text{ MPa}) & \text{for } 1 \text{ mm} \leq x \leq 3 \text{ mm} \\
(1.17 \text{ kg/m}^3, 0 \text{ m/s}, 0.1 \text{ MPa}) & \text{for } x \leq 1 \text{ mm} \& x \geq 3 \text{ mm}
\end{cases}
\]  

(14)

The above initial conditions were chosen by trial-and-error method to match the experimental pressure traces obtained from three piezoelectric pressure sensors, S1-S3 (113B24 PCB Piezotronics), flush mounted on the windows at 205 mm, 245 mm and 285 mm from the exploding wire plane respectively. The uncertainty in determination of peak pressure amplitude and time of arrival measured from a set of 5 repeatability tests were ±5.5% and ±0.5% respectively. Apart from demonstrating the simulation prediction accuracy, figure 4 also shows the blast wave profile along with its associated shock Mach number at each sensor which were 2.15±0.05, 2.04±0.05 and 1.98±0.05 respectively.

It is important to know the disturbances in the flow due to the 3 mm anode prior to conducting the experiments with the inhomogeneity. By this way, the disturbances can be identified later and neglected from the analysis. As can be seen in figure 5, the shock diffraction around the anode results in a reflected wave ($R_c$) and a Mach stem. The collision of the Mach-Mach stem ($MM_c$) behind the cylinder gives rise to shock patterns as shown in figure 5(a). As it travels downstream, the initial reflected waves get reflected from the side walls forming a
Figure 4: Comparison between experimental and numerical pressure trace at three sensor positions: S1 at 205 mm, S2 at 245 mm and S3 at 285 mm from the exploding wire plane.

Figure 5: Experimental shadowgraph (left) and numerical density gradient (right) images depicting the disturbances due to anode. The times, (a) $t = 48 \mu s$ and (b) $t = 123 \mu s$ are relative to initial shock wave impact at the anode.
Figure 6: Single shot (high spatial resolution) experimental shadowgraph images (top) for straight inhomogeneities ($\alpha = 0^\circ$) along with their corresponding numerical density gradient (middle) and vorticity magnitude (bottom) for (a) $b = 3$ mm and (b) $b = 10$ mm case. ‘concave’ structure as in figure 5(b). The results of numerical simulation performed with a 3 mm cylinder are shown as density gradient image beside each shadowgraph image. The magnitude of the density gradient field is calculated as (Quirk & Karni 1996),

$$|\nabla \rho| = \left[ \left( \frac{\partial \rho}{\partial x} \right)^2 + \left( \frac{\partial \rho}{\partial y} \right)^2 \right]^{\frac{1}{2}}$$

(15)

4. Results and Discussion

The experiments are conducted for a set of two different inclination angles ($\alpha$) with each set comprising of two $b$ values. The evolution of two straight elliptic inhomogeneities ($\alpha = 0^\circ$) with $b = 3$ mm and 10
Blast wave interaction with straight and inclined inhomogeneity

Figure 7: Sequence of high-speed experimental images at normalized time $t/t_o = 0.14, 0.28, 0.42, 0.55, 0.69, 0.83$ for $b = 3$ mm case.

mm shocked by an initially plane blast wave of strength Mach 2.15 are as shown in figure 6-8. Figure 6(top) is a single shot shadowgraph image with high spatial resolution while figure 7-8 represent images taken with the high-speed camera. With the aid of numerical simulation, the hot gas density ($\rho_h$) for this case was determined to be $0.5\pm0.03\ kg/m^3$ by trial-and-error method i.e by adjusting $\rho_h$ in the simulation until the distance travelled by the incident shock and the transmitted shock matches with the experimental measurement. The corresponding numerical simulation results are shown as density gradient images (middle) and vorticity magnitude in figure 6. By juxtaposing experimental and numerical images (figure 6), it can be seen that the simulation captured important structures observed in experiments with sufficient accuracy. As expected, when a shock wave encounters such a medium, it gets reflected and transmitted. Since this being a slow/fast case, the absolute velocity of the transmitted shock wave is higher and tends to travel faster than the incident shock. Also the transmitted shock curvature is very small justifying the step function approximation for temperature distribution (Sembian et al. 2018). It is connected to the incident shock through a precursor shock initiating a quadruple point at its intersection with the Mach stem and reflected shock. The flow field is symmetric along the
Figure 8: Sequence of high-speed experimental images at normalized time $t/t_o = 0.13, 0.27, 0.41, 0.54, 0.68, 0.81$ for $b = 10$ mm case.

inclination axis. The transmitted shock propagation speed also varies slightly corresponding to $b$. For instance, to traverse the 145 mm long inhomogeneity, it takes $180 \pm 5 \mu s$ and $185 \pm 5 \mu s$ for $b = 3$ mm and 10 mm respectively. The propagation distance $x$, defined as the distance travelled by the transmitted shock wave through the inhomogeneity [figure 9(a)] is measured at an interval of 10 $\mu s$ and the non-dimensionalized $x - t$ diagram for both thicknesses is plotted in figure 9(b). The propagation distance $x$ is normalized with the overall length of the inhomogeneity ($x_o = 145$ mm) and the time $t$ is normalized with $t_o$, the time taken to traverse ($x_o$). The variation is almost linear across both thicknesses and the speed difference is very small compared to the overall propagation time scale.

During this period of shock interaction, vorticity is deposited locally on the interface due to the misalignment between density and pressure gradients resulting in the formation of two counter-rotating vortices. The sense of rotation of the vortex is clockwise (negative) on the bottom half and counter-clockwise (positive) on the top half as shown in figure 6 (bottom). The vortices are observed experimentally from figure 7(b)-8(b) and grow in both length ($l$) and width ($d$) as the shock propagates...
Figure 9: (a) Schematic drawing illustrating the notations associated with the test configuration; (b) Normalized propagation distance ($x/x_0$) as a function of $t/t_0$; (c) Normalized width ($d/b$) and (d) Normalized length ($l/x_0$) of the vortices as a function of $x/x_0$. The corresponding numerical results are also plotted against their experimental values.

Further into the region. The corresponding vorticity magnitude (top) and density gradient (bottom) images are shown in figure 10. The size of the vortices is directly related to the mixing characteristics and therefore is an important parameter to analyze. The width of the vortex is scaled with $b$, the length with $x_0$ and both are plotted against $x/x_0$ as in figure 9(c)-(d). For $b = 3$ mm, the $d/b$ increases and reaches a factor of 5.5 at the downstream interface while it reduces to 2 for $b = 10$ mm. Large factor indicates more fluid being entrained by the vortices from the surrounding. On close observation, it is interesting to note that irrespective of initial thickness, the location of quadruple point is fairly similar and the vortices always almost fills up the entire region below it. For example, the quadruple point distance measured from the center axis
in figure 7(f)-8(f) is \( \sim 10.9 \) mm and \( \sim 11.4 \) mm respectively providing relatively more fluid for entrainment by vortices resulting in large \( d/b \) for small \( b \) values. Conversely, this also implies minor variations in \( d \) across various thicknesses and is governed by the shock dynamics in the precursor shock region. Similarly, the length (\( l \)) which is measured as the distance between the transmitted shock and the downstream edge of the vortex boundary is also dependent on the precursor shock region and varies linearly with \( x \).

Since this being a slow/fast case, the reflected wave was supposed to be an expansion wave. But due to the presence of a 3 mm solid cylinder (anode), it was reflected as a shock wave. In order to determine this effect, numerical simulation with no anode is performed and the results are compared in figure 11. The shock structures are observed to be identical in both cases with no variation in either \( l \) or \( d \). Although the anode introduces unwanted shock reflections, this proves that the end results are not affected by it.

Circulation \( \Gamma \) defined as the line integral of velocity around a closed curve is an important scalar quantity for describing and understanding vortical flows. It quantifies the net strength of the vortices generated by the shock interaction with the inhomogeneity. Several analytical models, like Kelvin’s model (Haas & Sturtevant 1987), PB model (Picone & Boris 1988; Picone \textit{et al.} 1984), YKZ model (Yang \textit{et al.} 1994), SZ model (Samtaney & Zabusky 1994), Niederhaus 1D model (Niederhaus \textit{et al.} 2007), were formulated for calculating the circulation in shock-bubble interactions. Although the models predict the total circulation (\( \Gamma_o \)) present at the instant of shock passage, the approach and the underlying assumptions made in arriving at the expressions vary. The PB and YKZ model assume that the shape of the bubble and the density ratio do not change significantly during shock passage so that the effects associated with shock refraction are neglected. The SZ model uses the extension of shock polar analysis on planar interfaces by scaling laws to include the refraction effects on circular interfaces. Niederhaus \textit{et al.} (2007), on the basis of a series of 3D simulations formulated a heuristic model reconstructed using 1D gasdynamic parameters obtained from modelling. But, the common factor in all these models is that they were formulated for a circular interface with a defined radius (\( R \)) where the density gradient is normal to the interface as we move radially outward. Ray \textit{et al.} (2000), extended the SZ model to an elliptic interface but it is applicable only to heavy prolate gaseous ellipses with major axis along the vertical direction. In the present study, we use a slow/fast elliptic interface - where the density gradient normal to the interface is no longer
Figure 10: Vorticity magnitude (top) and density gradient (bottom) for (a) b = 3 mm and (b) b = 10 mm straight inhomogeneities. The times $t/t_o$ correspond to their respective high speed experimental images in figure 7-8.

radial - with major axis along the horizontal direction for which models are non-existent. Deriving a formulation for such cases is beyond the
Figure 11: Images showing the effects due to the presence of anode. Apart from the initial shock reflections, the anode does not affect the evolution of the inhomogeneity. Due to symmetry, only top half is presented.

Figure 12: (a) Numerically computed circulation ($\Gamma$) as a function of $x$ for both $b = 3$ and 10 mm case; (b) Normalized circulation ($\Gamma/\Gamma_0$) for different combinations of $b$ and $\rho_h$ plotted against $x/x_0$.

Scope of this work, and only numerically determined circulation will be presented. Additionally, the models provide the total circulation ($\Gamma_0$) only after the fastest shock in the system reaches the downstream interface whereas numerically it can be obtained at every shock instant along $x$.

Numerically it is calculated following Stoke’s theorem that the circulation around a closed curve is equal to the flux of vorticity ($\omega$) through a closed surface $S$.

$$\Gamma = \oint_S \omega \cdot dS \quad (16)$$

where

$$\omega = \nabla \times U \quad (17)$$

and $U$ is the velocity vector. Figure 12(a) plots the circulation, calculated along the top-half of the inhomogeneity, for $b = 3$ mm and
Table 1: Total circulation ($\Gamma_o$) for different combination of $b$ and $\rho_h$.

<table>
<thead>
<tr>
<th>$b$ (mm)</th>
<th>$\rho_h$ (kg/m$^3$)</th>
<th>$\Gamma_o$ (m$^2$/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>0.5</td>
<td>22.92</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>25.34</td>
</tr>
<tr>
<td>6</td>
<td>0.5</td>
<td>27.5</td>
</tr>
<tr>
<td>10</td>
<td>0.5</td>
<td>28.54</td>
</tr>
<tr>
<td>3</td>
<td>0.39</td>
<td>35.48</td>
</tr>
<tr>
<td>10</td>
<td>0.39</td>
<td>38.4</td>
</tr>
</tbody>
</table>

Figure 13: Single shot experimental shadowgraph (left) and numerical density gradient (right) corresponding to $\alpha = 24^\circ$ case for $b = 1.5$ mm depicting the vortex train structure and bound precursor refraction (BPR).

10 mm for every 10 mm transmitted shock passage with the dotted line representing a linear fit. As expected, the circulation value is higher for the 10 mm case at all instants owing to the large initial energy from the hot gas coupled with the relatively small energy dissipation through entrainment compared with the 3 mm thickness. $\Gamma$ at $x = 145$ mm represent the total circulation ($\Gamma_o$) corresponding to each $b$ and the normalized $\Gamma/\Gamma_0$ is plotted against $x/x_o$ as in figure 12(b). Evidently $\Gamma/\Gamma_0$ falls along the same line represented by one linear fit equation. To examine the effects of thickness and density, simulations were performed for different combinations of $b$ and $\rho_h$, and the normalized circulation is plotted in figure 12(b). It is interesting to note that all data points follow a unique linear fit equation with a measured growth rate of 0.9715 conveniently decoupling the normalized circulation from thickness and density. The $\Gamma_o$ for all combinations is tabulated in Table 1. Since $x/x_o$ varies linearly with $t/t_o$, the variation of $\Gamma/\Gamma_0$ with $t/t_o$ will also be linear.
The second part of the analysis is done for an inclined inhomogeneity with $\alpha = 24^\circ$. The single shot and high-speed experimental shadowgraph images for $b = 1.5$ mm and $10$ mm are as shown in figure 13-15. As with the straight case, the general structures observed here are also similar for both thicknesses. This indicates that the qualitative behaviour is dependant on the inclination angle and not on the thicknesses investigated in this paper. The corresponding numerical density gradient and vorticity magnitude with $\rho_h = 0.39$ for $b = 1.5$ mm are as shown in figure 16-17. The first and foremost feature observed is the asymmetric flow field along the inclination axis. The transmitted shock rotates itself by $\alpha$ in the clockwise direction and propagates normal to the inclination axis making it symmetric. But this introduces an asymmetry in the precursor shock region about the inclination axis. The location of quadruple point above the axis is far away from the interface creating an elongated precursor region compared to the bottom. The incident shock is largely unaffected by the inclination as it is seen propagating along the same plane. So
the quadruple points can be thought of to act as a hinge, providing leverage for the transmitted shock rotation by varying the precursor shock length without hindering the incident shock path. Also additional refraction pattern corresponding to bound precursor refraction (BPR) (Abd-El-Fattah & Henderson 1978) is observed along the lower part.

The most striking feature observed is the appearance of a train of vortices particularly distinct in figure 13. The vortices are similar in nature to Kelvin-Helmholtz instability (KHI). During shock passage through the inhomogeneity, baroclinic mechanism deposits vorticity initiating an external perturbation at the interface as seen clearly in figure 14(b)-15(b). Similar to $\alpha = 0^\circ$, the sense of rotation of vorticity is clockwise (negative) on the bottom interface and counter-clockwise (positive) on the top interface. In addition, they are also embedded in a high velocity region flowing normal to the incident shock which provides additional momentum and sweeps it along its path. During this process, there are effectively two regions of different velocity i.e the flow due the
incident shock and the interface interacting with each other at an angle. This velocity discontinuity due to vorticity causes the amplitude of the initial perturbation to grow resulting in the rolling-up phenomenon of the interface as seen in figure 14(c-f)-15(c-f). As the shock traverses the full length, a train of vortices are generated similar to KHI. As time progresses, the rolled-up vortices continue to grow in size but eventually slow down.

Incidentally, the vortices are also translated normal to the incident shock and can be easily traced to its origin point. In order to estimate its average translational velocity, two points in the train (marked A and B in figure 14(f)-15(f)) are identified and its path is tracked from the series of images obtained from experiments. The distance travelled from its point of origin ($x_{\text{origin}}$) is fitted against time in figure 18(a) while its corresponding velocity averaged every $10 \mu s$ is plotted in figure 18(b) for
Figure 17: (contn) Density gradient (left) and vorticity magnitude (right) for \( b = 1.5 \) mm case. The times \( t/t_o \) correspond to their respective high speed experimental images in figure 14.

\( b = 1.5 \) mm case. Similar plots for \( b = 10 \) mm is shown in figure 18(c-d). They are tracked as soon as it was discernible up until \( t_o \). It can been seen that their path follows a quadratic function consistent with a linear drop in velocity. This is attributed to the energy expenditure associated with the growth of vortex that entrains more fluid from the surrounding flow. The velocity close to its point of origin (initial velocity) of the generated vortex is dependant on the transmitted shock strength, the source of baroclinicity. But the strength of the transmitted shock gradually reduces as it propagates through the inhomogeneity resulting in a reduction of initial velocity of the vortices generated at a later time. This is the reason behind the reduced initial velocity of B compared to A after which its velocity drop at a constant rate. Also this effectively means that each vortex along the train move with a different velocity with the lowest farther away, and highest close to the transmitted shock.
Figure 18: (a) Distance travelled by points A and B in the vortex train from their point of origin \( (x_{\text{origin}}) \) and (b) their corresponding average velocity plotted against \( t \) for \( b = 1.5 \) mm. Similar plots for \( b = 10 \) mm is shown in (c) and (d).

respectively. This results in a slight bending of the vortex train as can be clearly seen in figure 14(f)-15(f).

The evolution of \( \Gamma \) for \( \rho_h = 0.39 \) and 0.5 with respect to \( t/t_o \) and \( x/x_o \) is shown in figure 19(a)-(b). As expected, \( (\Gamma)_{\rho_h=0.39} \) is larger than \( (\Gamma)_{\rho_h=0.5} \) at every point. Also both the positive \( (\Gamma_+) \) and negative \( (\Gamma-) \) component of circulation, calculated by integrating only the counterclockwise and clockwise vorticity respectively, are nearly equal as can be seen in figure 19(a). Unlike the \( \alpha = 0^\circ \) case, the variation follows a second order function. This quadratic nature is influenced by two factors: the reduction in strength of the transmitted shock thereby generating vortices with reduced vorticity coupled with the gradual loss
Figure 19: (a) Numerically computed positive ($\Gamma_+$) and negative ($\Gamma_-$) circulation for $\rho_h = 0.39$ and 0.5 plotted against $t/t_0$; Due to the symmetric nature of both $\Gamma_+$ and $\Gamma_-$, only one component will be plotted to be consistent with the $\alpha = 0^\circ$ results. (b) Circulation ($\Gamma$) plotted against $x/x_0$; Normalized circulation ($\Gamma/\Gamma_0$) plotted against (c) $t/t_0$ and (d) $x/x_0$ respectively. Also included in (c) and (d) are the results from $\alpha = 0^\circ$ for comparison.

of vorticity of the earlier generated vortices. These effects are particularly amplified during the late stage of shock propagation, in this case after $t/t_o, x/x_0 = 0.7$. $\Gamma/\Gamma_0$ for different densities is plotted in figure 19(c)-(d). Also included is $\Gamma/\Gamma_0$ from $\alpha = 0^\circ$ for comparison. The normalized circulation too follows a second order function but is non-unique with the value at each corresponding point larger than that of ($\Gamma/\Gamma_0)_{\alpha=0^\circ}$. It is interesting to note that ($\Gamma/\Gamma_0)_{\rho_h=0.5}$ is slightly larger than ($\Gamma/\Gamma_0)_{\rho_h=0.39}$ despite the opposite trend in $\Gamma$ variation. This is due to either the lower than expected value of $\Gamma_0$ for $\rho_h = 0.5$, or higher than expected value
for $\rho_h = 0.39$ for the function to be unique. Therefore the normalized circulation is dependent on $\rho_h$ and a universal fit equation is non-existent.

5. Summary

The results presented here describe the early time interaction of a plane blast wave with an elongated elliptic inhomogeneity, aligned both normal as well as at an angle to the incoming wave. The inhomogeneities were generated on the basis of ‘Joule heating’ by supplying current to a small conducting wire suspended between two electrodes. This technique provided freedom in the positioning of the electrodes, enabling production of inclined inhomogeneities. The study was performed for two cases: $\alpha = 0^\circ$ and $\alpha = 24^\circ$ with two $b$ values. The structures observed in $\alpha = 0^\circ$ were similar to that of a circular bubble, where two counter-rotating vortices were formed due to RMI. The flow field was symmetric about the center axis. Since this being an elongated inhomogeneity along the horizontal axis, the baroclinic vorticity deposited along each point at the interface lead to a much faster rolling of vortices compared to a circular bubble, enabling early time (time until the transmitted shock reached the downstream interface) study. The variation of the normalized width, length and circulation of the vortices with distance was found to be linear. Also with the aid of numerical simulation, a unique linear fit equation for normalized circulation was given as it was found to be independent of the inhomogeneity density and thickness. For the inclined case, the transmitted shock was seen to adjust itself to travel normal to the inclined inhomogeneity with uneven precursor shock regions about the inclination axis. Also local flow corresponding to a BPR shock-interface interaction pattern was found on the lower part of the inhomogeneity. Unlike $\alpha = 0^\circ$, a vortex train similar to KHI was observed for $\alpha = 24^\circ$ case. The initial velocity of each individual vortex in the train is dependent on the strength of the transmitted wave and it was observed to lose its velocity linearly with time as it grows in size. The normalized circulation at each location along $x$ was found to be larger than that of $\alpha = 0^\circ$ case. Also it follows a quadratic function influenced by two factors: the reduction in strength of the transmitted shock thereby generating vortices with reduced vorticity along with the gradual loss of vorticity of the earlier generated vortices.

The financial support of Vetenskapsrådet (The Swedish Research Council) is gratefully acknowledged.

References
Blow wave interaction with straight and inclined inhomogeneity


Blast wave interaction with straight and inclined inhomogeneity


