Mitigation of Moving Shocks in an Expanding Duct

Veraun Chipman

University of Nevada, Las Vegas, chipmanv@unlv.nevada.edu

Follow this and additional works at: https://digitalscholarship.unlv.edu/thesesdissertations

Part of the Aerodynamics and Fluid Mechanics Commons, Fluid Dynamics Commons, and the Mechanical Engineering Commons

Repository Citation
https://digitalscholarship.unlv.edu/thesesdissertations/2251

This Dissertation is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Theses, Dissertations, Professional Papers, and Capstones by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
MITIGATION OF MOVING SHOCKS IN AN EXPANDING DUCT

By

Veraun Dale Chipman Jr.

Bachelor of Science in Civil and Environmental Engineering
Brigham Young University
1995

Master of Science in Civil and Environmental Engineering
Brigham Young University
1996

A dissertation submitted in partial fulfillment of the requirements for the

Doctor of Philosophy – Mechanical Engineering

Department of Mechanical Engineering
Howard R. Hughes College of Engineering
The Graduate College

University of Nevada, Las Vegas
December 2014
We recommend the dissertation prepared under our supervision by

Veraun Chipman

entitled

Mitigation of Moving Shocks in an Expanding Duct

is approved in partial fulfillment of the requirements for the degree of

Doctor of Philosophy in Engineering - Mechanical Engineering

Department of Mechanical Engineering

William Culbreth, Ph.D., Committee Chair
Brendan O’Toole, Ph.D., Committee Member
Darrell Pepper, Ph.D., Committee Member
Evangelos Yfantis, Ph.D., Committee Member
Sahjendra Singh, Ph.D., Graduate College Representative
Kathryn Hausbeck Korgan, Ph.D., Interim Dean of the Graduate College

December 2014
Inviscid flow theory governs the bulk motion of a gas at some distance away from the walls (i.e. outside the boundary layer). That is to say, there are no viscous forces in the bulk flow, which is modeled using the Euler equations. The Euler equations are simply the Navier-Stokes equations with zero viscosity terms. An ideal inviscid fluid, when brought into contact with a surface or wall, would naturally slip right past it since the fluid has no viscosity. In real life, however, a thin boundary layer forms between the wall or surface and the bulk flow. Shock wave boundary layer theory governs this flow. That boundary layer naturally starts as laminar, but grows in thickness over the length of the boundary until it either separates (due to an adverse pressure gradient) or becomes turbulent. Generally, a turbulent boundary layer is thicker (or reaches further into the bulk flow) than a laminar boundary layer. The flow regime is even more complicated when moving supersonically, where shocks and boundary layer interact to cause even greater turbulence and unsteadiness.

For most engineering applications involving supersonic flow, a turbulence and unsteadiness is undesirable. However, for the application of presented herein it was postulated that the turbulence and unsteadiness would help mitigate the propagation of a blast/shock wave traveling in an expanding duct or laser beam tube. It was also postulated that small wall obstructions in the flow could enhance those effects to the point of mitigating the impulsive forces of the blast/shock wave on a thin laser focusing optic. Three questions were asked and answered:
1. Will the blast/shock wave generated from fusion burn propagate from the target chamber to the final optic? Yes, it will.

2. If the blast/shock wave does propagate to the final optic, is it strong enough to damage the final optic? Yes, it does.

3. If the advancing blast/shock wave is strong enough to damage to final optic, what types of mitigation strategies can be deployed to lessen or eliminate the impacts of the blast/shock wave on the final optic? Yes, they can.

By purposely tripping the boundary layer using small wall obstructions in a short section of beam tube, the turbulent boundary layer may grow in thickness to point where it reaches far enough into the bulk flow to cause the bulk to flow to lose it's parallel streamlined looking profile. The turbulent boundary layer may also reach far enough into the bulk flow that it "sees" the turbulent boundary layer from the opposite side of the wall, thus really knocking the bulk flow out of its streamlined pattern. Upon exiting the short section of beam tube, this turbulent and unsteady flow is not directly in-line with an opening to a longer beam tube section, and therefore does not supersonically jet across but enters the longer section of diverging beam tube subsonically and naturally slows.
ACKNOWLEDGEMENTS

I would like to express my heartfelt thanks to my Ph.D. Committee Chair and Advisor, Dr. William Culbreth, for his guidance and direction, without which I would not have been able to complete this work. I’ll long remember discussions with him in his office and laboratory, and notes scratched on paper. Thanks also to my committee members, Dr. Darrell Pepper, Dr. Brendan O’Toole, Dr. Sahendra Singh, and Dr. Evangelos Yfantis, and to Joan Conway, who provided logistical support, without which no Mechanical Engineering graduate student at UNLV would ever graduate.

I would also like to thank my colleagues at Lawrence Livermore National Laboratory for their support, and especially to Jim Blink who got me started on this path, Mike Gerhard and Tarabay Antoun who encouraged me kept me on the path, and Jeff Latkowski and Gwen Loosmore for giving me the opportunity to take on such a project. Thanks also Ilya Lomov (may he rest in peace) who mentored me in the use of GEODYN, and Andy Cook and Britton Olson for mentoring me in the use of Miranda.

Sincere thanks also for the prayers and support of my many friends and family members as I juggled the responsibilities of being a husband, a father, a provider, a student, a climber, and a bishop. Most importantly, my most sincere thanks and love to my devoted wife, Denise, for believing in me and what I was trying to accomplish, and who has always been so encouraging and supportive. And to my wonderful children Damyn, Hannah, Clayton, Ivy and Breanne, this work is dedicated to you. May you follow in the footsteps of your ancestors and the Lord.
TABLE OF CONTENTS

ABSTRACT.................................................................................................................. iii
ACKNOWLEDGEMENTS.............................................................................................. v
LIST OF TABLES ........................................................................................................... ix
LIST OF FIGURES ......................................................................................................... x
LIST OF SYMBOLS ...................................................................................................... xiii
CHAPTER 1 LASER INERTIAL FUSION-FISSION ENERGY ............................. 1
  Background.............................................................................................................. 1
  The Final Optic ...................................................................................................... 3
  Three Posed Questions ........................................................................................ 4
CHAPTER 2 WILL THE BLAST WAVE PROPAGATE TO THE FINAL
  OPTIC? ..................................................................................................................... 6
  Conceptual Model .................................................................................................. 6
  Modeling Approach ............................................................................................... 7
    Analytical Model for Shock/Blast Wave Propagation and Optic Response 8
    Bucky Results as Input to the Analytical Model ................................................. 8
    Short Beam Tube Section .................................................................................. 10
    Open Space of Vacuum Chamber ..................................................................... 11
    Main Beam tube Section .................................................................................... 13
  Numerical Models for Shock/Blast Wave Propagation .................................... 13
    Governing Equations ........................................................................................ 14
    Methodology ....................................................................................................... 15
  Results ................................................................................................................... 17
  Conclusions ........................................................................................................... 20
CHAPTER 3 DOES THE PROPAGATED BLAST/SHOCK WAVE DAMAGE
  THE FINAL OPTIC? ................................................................................................. 22
  Conceptual Model ................................................................................................. 22
  Modeling Approach ............................................................................................... 23
    Analytical Model for Deflection Angle and Displacement of the Final
      Optic .................................................................................................................. 23
    Numerical Model for Deflection Angle and Distance of the Final Optic .......... 27
      Governing Equations ....................................................................................... 27
      Methodology .................................................................................................... 28
      Results ............................................................................................................... 28
  Summary ............................................................................................................... 32
CHAPTER 4 CAN THE EFFECTS OF THE PROPAGATED BLAST BE
  MITIGATED? ............................................................................................................ 33
  Outline ................................................................................................................... 33
CHAPTER 5 THEORY OF NORMAL AND MOVING SHOCK BEHAVIOR ...... 34
  The Governing Equations ..................................................................................... 34
    Continuity ........................................................................................................... 34
    Momentum .......................................................................................................... 35
    Energy .................................................................................................................. 36
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fusion Burn Energy Deposition</td>
<td>109</td>
</tr>
<tr>
<td>Time Step Control</td>
<td>110</td>
</tr>
<tr>
<td>BIBLIOGRAPHY</td>
<td>112</td>
</tr>
<tr>
<td>VITA</td>
<td>114</td>
</tr>
</tbody>
</table>
LIST OF TABLES

Table 2.1  Input parameters to a BUCKY simulation provided by LLNL .......... 8
Table 2.2  Properties of xenon gas. ...................................................... 8
Table 3.1  Material properties of a Fresnel (SiO$_2$) optic ...................... 28
Table 7.1  Parametric modeling case descriptions ................................. 70
**LIST OF FIGURES**

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Depiction of a LIFE Power Plant</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>Depictions of the LIFE Target Chamber and Surrounding Components</td>
<td>3</td>
</tr>
<tr>
<td>2.1</td>
<td>Simplified rendition of LIFE target chamber surrounded by high-pressure gas plenum and single laser beam tube.</td>
<td>6</td>
</tr>
<tr>
<td>2.2</td>
<td>In the analytical approach, the propagation of the blast/shock wave and its impact on the final optic was decoupled into separate first principle models.</td>
<td>8</td>
</tr>
<tr>
<td>2.3</td>
<td>(a) RT contours from a BUCKY simulation. (b) Pressure versus distance for various times from a BUCKY simulation.</td>
<td>10</td>
</tr>
<tr>
<td>2.4</td>
<td>(a) Beam tube mesh for the GEODYN model. (b) Zoomed-in portion of the mesh.</td>
<td>16</td>
</tr>
<tr>
<td>2.5</td>
<td>Input conditions for density and specific energy from the BUCKY model.</td>
<td>17</td>
</tr>
<tr>
<td>2.6</td>
<td>Results of the analytical model in terms of Mach number.</td>
<td>18</td>
</tr>
<tr>
<td>2.7</td>
<td>Overpressure and Mach number contours at time = 0, 4, 15, 30, and 60 ms.</td>
<td>19</td>
</tr>
<tr>
<td>2.8</td>
<td>Pressure history at the would-be center of the final optic.</td>
<td>20</td>
</tr>
<tr>
<td>3.1</td>
<td>Analytical model for deflection angle and displacement of the final optic.</td>
<td>23</td>
</tr>
<tr>
<td>3.2</td>
<td>Zoomed-in view of the pressure history showing the impulsive load as the area under the curve highlighted in yellow that was used as for the $pt$ term in the Moir analytical model.</td>
<td>26</td>
</tr>
<tr>
<td>3.3</td>
<td>Contours of displacement (mm) at 42 ms for the optic as calculated by the numerical.</td>
<td>30</td>
</tr>
<tr>
<td>3.4</td>
<td>Contours of maximum principle stress (MBar) at 42 ms for the optic as calculated by the numerical LS-DYNA model.</td>
<td>31</td>
</tr>
<tr>
<td>3.5</td>
<td>Time-history of maximum principle stress plotted against the rupture modulus and the input pressure.</td>
<td>32</td>
</tr>
<tr>
<td>5.1</td>
<td>Rectangular control volume for 1-D fluid flow (Anderson, 2003).</td>
<td>34</td>
</tr>
<tr>
<td>5.2</td>
<td>Subsonic and supersonic flow over a flat-faced cylinder (Anderson, 2003).</td>
<td>39</td>
</tr>
<tr>
<td>5.3</td>
<td>(a) Stationary and (b) moving normal shocks (Anderson, 2003).</td>
<td>42</td>
</tr>
<tr>
<td>5.4</td>
<td>Incremental volume (Anderson, 2003).</td>
<td>46</td>
</tr>
<tr>
<td>6.1</td>
<td>Schlieren photography of a shock wave propagating through pipes of increasing diameters (top to bottom) and time (right to left) (Scharlin and Reichenbach, 1965, Figure 2a-2c).</td>
<td>49</td>
</tr>
<tr>
<td>6.2</td>
<td>Schlieren photography of a shock wave propagating through pipes of increasing roughness (top to bottom) and time (left to right) (Scharlin and Reichenbach, 1965, Figure 4a-4b).</td>
<td>49</td>
</tr>
<tr>
<td>6.3</td>
<td>Test section showing the vertically symmetric and oblique and staggered baffle plate arrangements (Ohtomo et. al., 2005, Figure 3).</td>
<td>50</td>
</tr>
<tr>
<td>6.4</td>
<td>Interferogram for $M_s = 1.5$ at 2.40 ms for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 8e).</td>
<td>51</td>
</tr>
</tbody>
</table>
Figure 6.5 Interferogram for $M_s = 1.5$ at 2.51 ms for the oblique and staggered baffle plates (Ohtomo et. al., 2005, Figure 12d)

Figure 6.6 Interferogram for $M_s = 3.0$ at 2.34 ms for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 10f)

Figure 6.7 Interferogram for $M_s = 3.0$ at 1.83 ms for the oblique and staggered baffle plates (Ohtomo et. al., 2005, Figure 14e)

Figure 6.8 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 1.5$ for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 9)

Figure 6.9 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 1.5$ for the oblique staggered baffle plates (Ohtomo et. al., 2005, Figure 13)

Figure 6.10 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 3.0$ for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 11)

Figure 6.11 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 3.0$ for the oblique staggered baffle plates (Ohtomo et. al., 2005, Figure 13)

Figure 6.12 Experimental shock tube set-up for Berger et. al. investigation of shock-wave load attenuation by geometrical obstructions (Berger et. al., 2009, Figure 1).

Figure 6.13 Schematic of a typical test-section (Berger et. al., 2009, Figure 3).

Figure 6.14 Schlieren images from a single obstacle test at 45° with an ROF=0.375 at $M=1.2$ (Berger et. al., 2005, Figure 4).

Figure 6.15 Pressure histories from a single obstacle test at 45° with an ROF=0.375 at $M=1.2$. from piezoelectric pressure transducers on the test-section walls upstream of the obstacle ($P_a$), downstream ($P_b$), and at the center of the end-wall ($P_{ew}$) (Berger et. al., 2005, Figure 5).

Figure 6.16 Schlieren images from multi-obstacle tests at 45°, 90°, and 135° with an ROF=0.375 at $M=1.2$ (Berger et. al., 2005, Figure 9).

Figure 6.17 Impulse linear slopes for ROF for ROFs of 0.375 and 0.625, inclination angles of 45°, 90°, and 135°, number of obstacles from 1 to 5, and integration times of 0.25, 1 ms, 2 ms, and 5 ms (Berger et. al., 2005, Figure 12).

Figure 6.18 Complex flow of shock wave/boundary layer interactions.

Figure 7.1 Input conditions for density and specific energy from the BUCKY model.

Figure 7.2 Comparison of the basecase (Case 1) to Case 2.

Figure 7.3 Comparison of the basecase (Case 1) to Case 3.

Figure 7.4 Comparison of the basecase (Case 1) to Case 4.

Figure 7.5 Comparison of the basecase (Case 1) to Case 5.

Figure 7.6 Comparison of the basecase (Case 1) to Case 6.

Figure 7.7 Comparison of the basecase (Case 1) to Case 7.

Figure 7.8 Comparison of the Case 2 to Case 7.

Figure 7.9 Comparison of the Case 3 to Case 9.
Figure 7.10  Model domain showing the boundary conditions used for the Miranda modeling. ................................................................. 84
Figure 7.11  Time-lapse of the simulated flow from the target chamber to the final optic................................................................. 87
Figure 7.12  Bulk viscosity of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in red) and ~2 degrees (shown in gray). ................................................................. 89
Figure 7.13  Temperature of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in red) and ~2 degrees (shown in blue). ................................................................. 90
Figure 7.14  Pressure of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in green) and ~2 degrees (shown in blue). ................................................................. 90
Figure 7.15  Velocity of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in orange) and ~2 degrees (shown in pink). ................................................................. 91
Figure 7.16  Generalized depiction of free flow separation (FSS - Left) and restricted shock separation (RSS – Right) for a rocket engine (Oslund and Muhammad-Klingmann, 2005)................................................................. 92
Figure 7.17  Timelapse of the flow through the small beam tube with no notches. ............................................................................. 95
Figure 7.18  Timelapse of the flow through the small beam tube with notches. ............................................................................. 96
Figure 7.19  Timelapse of the flow through the small beam tube with multiple notches. ............................................................................. 97
Figure 7.20  Density and Mach number contours at 3.2 ms for the short beam tube section without and with notches ............................................................................. 98
Figure 7.21  Plots of pressure versus time at the final optic for the case without and with notches. ............................................................................. 98
# LIST OF SYMBOLS

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_e$</td>
<td>Exit area</td>
</tr>
<tr>
<td>$A_s$</td>
<td>Spherical shock area</td>
</tr>
<tr>
<td>$A_t$</td>
<td>Throat area</td>
</tr>
<tr>
<td>$a$</td>
<td>Sound speed</td>
</tr>
<tr>
<td>$c_s$</td>
<td>Ratio of shock strength to area</td>
</tr>
<tr>
<td>$c_0$</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat</td>
</tr>
<tr>
<td>$D$</td>
<td>Diffusion coefficient</td>
</tr>
<tr>
<td>$d$</td>
<td>Diameter</td>
</tr>
<tr>
<td>$E$</td>
<td>Total energy</td>
</tr>
<tr>
<td>$e$</td>
<td>Energy</td>
</tr>
<tr>
<td>$e_{allow}$</td>
<td>Pointing error</td>
</tr>
<tr>
<td>$Fr$</td>
<td>Froude number</td>
</tr>
<tr>
<td>$f$</td>
<td>Body force</td>
</tr>
<tr>
<td>$f(x)$</td>
<td>Chisnell function</td>
</tr>
<tr>
<td>$h$</td>
<td>Enthalpy</td>
</tr>
<tr>
<td>$L$</td>
<td>Standoff distance</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$M_e$</td>
<td>Exit Mach number</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$n$</td>
<td>Normal</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$P_t$</td>
<td>Throat pressure</td>
</tr>
<tr>
<td>$R$</td>
<td>Gas constant</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
</tr>
<tr>
<td>$q$</td>
<td>Flux of bulk viscosity</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Heat generation</td>
</tr>
<tr>
<td>$s$</td>
<td>Deviatoric stress</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$u$</td>
<td>Fluid velocity in x-direction</td>
</tr>
<tr>
<td>$V$</td>
<td>Volume</td>
</tr>
<tr>
<td>$v$</td>
<td>Fluid velocity in y-direction</td>
</tr>
<tr>
<td>$W_s$</td>
<td>Wave or shock speed</td>
</tr>
<tr>
<td>$W$</td>
<td>Wave speed</td>
</tr>
<tr>
<td>$X_s$</td>
<td>Distance for critical shock formation</td>
</tr>
<tr>
<td>$x$</td>
<td>Component position or direction, or optic deflection</td>
</tr>
<tr>
<td>$\ddot{x}$</td>
<td>Acceleration</td>
</tr>
<tr>
<td>$y$</td>
<td>Component position or direction</td>
</tr>
<tr>
<td>$z$</td>
<td>Component position or direction, or Chisnell pressure ratio</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Angle of propagation</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Bulk viscosity or oblique shock angle</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Kronecker delta ($1$ if $i=j$, else $0$), or thickness</td>
</tr>
<tr>
<td>$\dot{e}_{ij}$</td>
<td>Strain rate tensor</td>
</tr>
<tr>
<td>$\delta t$</td>
<td>Time step</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stress</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Deflection</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Ratio of specific heats</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Ratio of specific heats</td>
</tr>
</tbody>
</table>
\(0, 1, 2, \ldots\) Subscripts denoting location or timestep

\(i, j, k\)  Subscripts denoting indices
CHAPTER 1
LASER INERTIAL FUSION-FISSION ENERGY

Background

Laser Inertial Fusion-Fission Energy (LIFE) is a hybrid fusion-fission energy concept under development at Lawrence Livermore National Laboratory (LLNL). It uses fusion energy, which is clean, inherently safe, and virtually unlimited to generate carbon free fission burn of spent nuclear fuel to generate usable power. It is estimated that a LIFE power plant, depicted in Figure 1.1, could generate gigawatts of power on an hourly basis for as long as 50 years without the need for refueling, all while avoiding carbon dioxide emissions, easing nuclear proliferation and nuclear safety concerns, and reducing the volume of nuclear waste to be stored long-term in a deep geologic repository. A LIFE power plant would require about half the energy of a pure fusion plant, and would produce 100 to 300 times more net energy due to the extra gain from the fission.

Figure 1.1 Depiction of a LIFE Power Plant
The physics and technology behind the nuclear fusion aspect of LIFE have been, and are currently being developed under the National Ignition Facility (NIF) project, funded by the U.S. Department of Energy’s (DOE) National Nuclear Security Administration (NNSA) and built at LLNL. NIF is the world’s largest and highest energy laser, and represents the culmination of nearly 60 years of research into controlled fusion. NIF fusion ignition experiments, specifically inertial confinement fusion (ICF), with the goal of net energy gain, began in 2010. Success of NIF will serve as a springboard for the LIFE concept.

The LIFE “engine” would use an ICF laser system similar to the one currently under development at NIF to ignite fusion targets. As depicted in Figure 1.2, the fusion targets are centered in a spherical like target chamber. Surrounding the target chamber is a cylindrical pressure vessel used to clear the system of “dirty gas” and debris between successive shots. Banks of lasers contained on opposite sides of the pressure vessel bend and focus their beams through final optic assemblies approximately 20 to 25 meters away from the fusion target. The laser beams pass through cylindrically converging beam tubes that intersect the external surface of the cylindrical pressure vessel, pass through the open space between the pressure vessel and the target chamber, pass through two circular openings on opposite sides of the target chamber, and deposit their incident energy on a cryogenically-frozen deuterium fusion target. Surrounding the target chamber is a blanket of subcritical reprocessed fission fuel. Laser induced ICF will produce a point source of 14.1 MeV neutrons at the center of the target chamber that travel spherically outward through the various structural and coolant layers surrounding the target chamber, from where they will be absorbed by the fission blanket, promoting neutron capture and fission reactions. These fission reactions, in turn, will release enormous amounts of heat to drive steam turbines.
Critical to ICF is the protection and survival of the final optic that resides approximately 20 to 25 meters from the target, but sits in direct line of sight of target fusion emissions which consist of high-energy (14.1 MeV) neutrons, X-rays, charged burn product and debris ions, as well as an advancing blast/shock wave of “dirty gas.”

Radiation incident on the final optic causes optical absorption lessening the ability of the optic to focus the laser energy on the target. To mitigate the effects of the radiation absorption, a thin transmissive Fresnel lens composed of fused silica is being considered. Past experimental work and ongoing numerical modeling by scientists and engineers at LLNL suggest that radiation damage to the lens tends to saturate, and even produces a “radiation annealing” effect when using a very thin (approximately 0.5 mm)
fused silica Fresnel lens for an ultraviolet beam wavelength of 351 nm (Latkowski, et. al., 2003).

To mitigate the effects of damaging X-rays, or high-energy photons, produced during fusion-burn, a low pressure xenon gas environment (approximately 1/100 of an atmosphere) is created within the target chamber, external pressure vessel, and laser-beam-tubes. In general, the ability of a gas to absorb high-energy photons increases as the gas becomes heavier, making xenon gas ($Z = 54$), the heaviest of the nonradioactive noble gases, a good candidate. In the LIFE design, xenon gas is introduced into the target chamber around 4 $\mu$g/cc to absorb X-rays in order to prevent those high energy photons from damaging the first wall of the target chamber, and to prevent them propagating outside the target chamber where they might damage other structures including the final optic. Xenon gas is also introduced into the system at varying levels of low pressure and density so as to create choked flow from the target chamber to the external pressure vessel, and from the laser-beam-lines to the external pressure vessel, for the clearing of charged fusion burn product and debris ions in between shots.

Three Posed Questions

With the promise that these mitigation strategies offer in protecting the final optic from the damaging effects of neutron and X-ray radiation (i.e. the use of a fused silica Fresnel lens for the final optic, and the use of xenon gas for X-ray absorption and target chamber clearing), attention is now focused on the advancing blast/shock wave created from the initiation of fusion burn. In particular, the following questions arise:

1. Will the blast/shock wave generated from fusion burn propagate from the target chamber to the final optic?

2. If the blast/shock wave does propagate to the final optic, is it strong enough to damage the final optic?
3. If the advancing blast/shock wave is strong enough to damage to final optic, what types of mitigation strategies can be deployed to lessen or eliminate the impacts of the blast/shock wave on the final optic?

The answers to these posed questions provide the basis for the original research contained in this dissertation.
CHAPTER 2

WILL THE BLAST WAVE PROPAGATE TO THE FINAL OPTIC?

Conceptual Model

Figure 2.1 shows a simplified rendition of various system components of a LIFE reactor. Final optic assemblies, with stand-off distances around 25 meters, focus high-energy ultraviolet laser beams down a beam tube to a deuterium/tritium fusion target located at the center of target core. A structural layer and fission blanket surround the target chamber (shown in light blue in Figure 2.1). A plenum surrounding the structural layer and fission blanket provides a continuous high pressure, low temperature xenon gas flow to the system to clear the chamber of debris between shots. A low-pressure external cylindrical vessel surrounds this entire assembly.

![Figure 2.1 Simplified rendition of LIFE target chamber surrounded by high-pressure gas plenum and single laser beam tube.](image)

Fusion burn initiates at the center of the target chamber, causing a spherical blast/shock wave of charged burn product, debris ions, and xenon gas to form. The blast/shock wave spreads spherically outward at supersonic velocity until it reaches the first wall of the structural layer confining the target chamber, where it reflects back onto itself. Researchers at the University of Wisconsin using a code called BUCKY are
modeling this process. Since the laser beam lines are in direct line-of-sight from the final optic to the target chamber, small diameter openings in the first wall are necessary, and give way to short cylindrically diverging tubes that act like diverging nozzles for the blast/shock wave to pass from the inner wall of the target chamber to the external pressure vessel. Upon exiting the short section of beam tube, a hemispherical blast/shock wave develops in the open space of the external pressure vessel. The hemispherical blast/shock wave then propagates across the gap to the wall of the external pressure vessel. If the blast/shock wave is able to overcome entrance effects to the small diameter opening of the cylindrically diverging main section of beam tube, the blast/shock wave may then propagate all the way to the final optic. If the blast/shock wave enters the main section of beam tube at subsonic velocity, then the diverging nature of the beam tube will cause the subsonic wave to attenuate. However, if the blast/shock wave enters the main section of beam tube at sonic velocity, then the diverging shape of the beam tube will cause the wave to speed up.

Due to the near vacuum environment within the system the blast/shock wave will be relatively low in pressure, and in and of itself, not very harmful to the optic. However, the relatively low force(s) applied over very short periods of time (micro to milliseconds) will transfer an impulsive momentum to the thin lens (0.5 millimeters) of the final optic that could result in significant displacement and stress. If that displacement is large enough, and/or if the stress exceeds the modulus of rupture, than the lens will be irreparably damaged.

Modeling Approach

Both analytical and numerical models were developed to track the propagation of the blast/shock wave from the target chamber to the final optic, and to predict the response of the optic in terms of displacements and stresses. A numerical model was developed using an LLNL proprietary code called GEODYN to model the basic physics
of compressible gas flow with real equation-of-state behavior. The numerical results were then fed to an LLNL structural/mechanical code called LS-DYNA to model the behavior of the final optic in terms of stresses, strains, and deflections.

Analytical Model for Shock/Blast Wave Propagation and Optic Response

The analytical approach is conceptually shown in the diagram of Figure 2.2. The system was decoupled, and used the governing equations for steady state compressible gas flow, nozzles, and shock tubes to estimate the propagation of the shock/blast wave. Using the results from the decoupled analytical gas dynamics, a separate analytical approach was employed to estimate the deflection of the final optic.

Figure 2.2 In the analytical approach, the propagation of the blast/shock wave and its impact on the final optic was decoupled into separate first principle models.

Bucky Results as Input to the Analytical Model

The University of Wisconsin performed a BUCKY simulation of the conditions within the target chamber after fusion initiation. The input parameters, shown in Table 2.1, were provided to them by LLNL. Table 2.2 summarizes the properties of xenon gas.

Table 2.1 Input parameters to a BUCKY simulation provided by LLNL.

<table>
<thead>
<tr>
<th>Input Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Target Chamber Radius</td>
<td>250 cm</td>
</tr>
<tr>
<td>Xenon Gas Specific Volume</td>
<td>$1.88 \times 10^{16}$ cm$^{-3}$</td>
</tr>
<tr>
<td>X-Ray Energy</td>
<td>4.5 MJ</td>
</tr>
<tr>
<td>Ions</td>
<td>3.8 MJ</td>
</tr>
</tbody>
</table>

Table 2.2 Properties of xenon gas.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ratio of specific heats</td>
<td>1.67</td>
</tr>
<tr>
<td>Gas constant</td>
<td>63.328 J/kgK</td>
</tr>
</tbody>
</table>
Some results of that simulation are shown in Figure 2.3, and include radius versus time (RT) contours (a) and total pressure versus radius time histories (b). In Figure 2.3a, the first sharp change slope of the RT contours (depicted as the dark line trending up and to the right beginning at time zero and a radius of 50 cm) indicates the arrival of the blast front as it shocks at the inner wall of the target chamber around 650 µs. The successive changes in slope after 650 µs indicate subsequent shocking of the blast waves as they bounces off the inner wall of the target chamber and themselves. From the first change of RT contour slopes, the velocity of the shock front can be estimated by determining the slope of the line of discontinuity. The Mach number was calculated to be approximately 3.4.

As an aside, in 1950 Sir Geoffrey Taylor published, “The Formation of a Blast Wave by a Very Intense Explosion. I. Theoretical Discussion,” and “The Formation of a Blast Wave by a Very Intense Explosion. II. The Atomic Explosion of 1945.” In these papers, Taylor develops a methodology for calculating the Mach number of an atmospheric nuclear explosion. In Part II, he presents an equation (Equation 8) that relates the energy released as a function of the properties of the gas, the blast radius, and time as:

\[ E = K \rho_0 r^5 t^{-2} \]  

(2-1)

Given the energy released from Table 2.1 as 8.3 MJ (X-ray energy plus the Ion energy), the gas properties of xenon in Table 2.2, and a value for K of 0.487 from Table 3 of Taylor (Part II), a Mach number of 2.3 is calculated in comparison to the Mach number calculated from the results of the BUCKY simulation.

Going back to results of the BUCKY simulation, the pressure of the blast wave as it strikes the inner wall was taken from Figure 2.3b by using the 600 µs pressure curve
to be $2.25 \times 10^4$ Pa. These conditions were used as input in the diverging nozzle calculations representing the short beam tube section.

Figure 2.3 (a) RT contours from a BUCKY simulation. (b) Pressure versus distance for various times from a BUCKY simulation.

**Short Beam Tube Section**

The results of the BUCKY simulation, specifically the pressure and Mach number of the shock as it reaches the inner wall of the target chamber, were used as input to a diverging nozzle calculation that represented the short section of beam tube between the inner and outer walls of the target chamber. The Mach number of the shock wave at the exit of the short section of beam tube was calculated using the following equation for a convergent-divergent nozzle (Anderson, 2003):

$$M_e = \sqrt{-\left(\frac{1}{\gamma - 1}\right) + \frac{1}{(\gamma - 1)^2} + \left(\frac{2}{\gamma - 1}\right) \left(\frac{2}{\gamma + 1}\right)^{\gamma - 1} \left(\frac{P_e A_e}{P_i A_i}\right)^2}$$  \hspace{1cm} (2-2)

Where

$M =$ \hspace{0.5cm} Mach number of the flow at the nozzle exit

$\gamma =$ \hspace{0.5cm} ratio of specific heats of the fluid = 1.67 for xenon gas
P_t = nozzle throat pressure = 2.25 \times 10^4 \text{ Pa from BUCKY results at 600 \(\mu\)s and radius of 2.5 m}

A_t = nozzle throat area = inlet area of the short beam tube section = 0.2 m

P_e = nozzle exit pressure = system background pressure = 266.645 Pa

A_e = nozzle exit area = outlet area of the short beam tube section = 0.4 m

Open Space of Vacuum Chamber

Upon exiting the short section of beam tube, the shock wave encounters an abrupt change in area where it undergoes a spherical expansion into the open space of the external pressure vessel. A literature search yielded a step-wise analytical model for the spherical expansion of the shock into open space. The approach was initially developed by Chisnell in 1957, later modified and validated by others including Sloan and Nettleton in the 1970s, and investigated further by Abate in his doctoral research in 2002. The approach is as follows:

(1) Use the Mach number of the shock obtained from the convergent-divergent nozzle calculation, \(M_e\), specific heat ratio, \(\gamma\), and speed of sound, \(c_0\) to calculate the shock speed, \(W_s\), and pressure ratio, \(z\):

\[
W_s = c_0 M_e \tag{2-3}
\]

\[
z = \frac{P_2}{P_1} = \frac{2\gamma M_e - 2\gamma + \gamma + 1}{\gamma + 1} \tag{2-4}
\]

(2) Calculate the Chisnell function, \(f(z)\):
\[ f(z) = \frac{1}{z}(z-1) \left( \frac{\gamma - 1}{\gamma + 1} \right)^{\frac{1}{2}} \left( 1 + \left( \frac{\gamma - 1}{\gamma + 1} \right) \right)^{\frac{1}{2}} \left( 1 - \left( \frac{\gamma - 1}{\gamma + 1} \right) \right)^{\frac{1}{2}} \]

\[ \times \left[ 1 + \left( \frac{\gamma - 1}{\gamma + 1} \right)^{\frac{1}{2}} - \left( \frac{\gamma - 1}{2\gamma} \right) \right] \]

\[ \times \left[ 1 - \left( \frac{\gamma - 1}{\gamma + 1} \right)^{\frac{1}{2}} + \left( \frac{\gamma - 1}{2\gamma} \right) \right] \]

\[ \times \exp \left( \frac{2}{\gamma - 1} \tan^{-1} \left( \frac{2}{\gamma - 1} \right) \left( \frac{\gamma z}{\gamma + 1} \right) \right) \]

(2-5)

(3) Calculate the angle of propagation, \( \alpha \):

\[ \tan^2 \alpha = \frac{(\gamma - 1)(M_e^2 - 1)(M_e^2 + 2)}{(\gamma + 1)M_e^4} \]  

(2-6)

(4) Calculate the distance, \( X_s \), for critical shock formation for an axisymmetric shock expanding at a sharp corner from an opening with diameter, \( d \):

\[ X_s = \frac{d}{2} \cot \alpha \]

(2-7)

(5) Calculate the surface area of the expanding spherical shock, \( A_s \):

\[ A_s = 2\pi X_s^2 \]

(2-8)

(6) Calculate the constant ratio of shock strength to area, \( C_s \):

\[ C_s = A_s f(z) \]

(2-9)

(7) Use the shock speed from step (1), \( W_s \), and choose an arbitrary time step, \( \delta t \), to calculate a new position of the shock, \( X_{s2} \):
\[ X_{s2} = W_s \delta t \]  

(2-10)

(8) Calculate the new surface area of the expanding shock, \( A_{s2} \):

\[ A_{s2} = 2\pi X_{s2}^2 \]  

(2-11)

(9) Using the constant, \( C_s \), obtained from step (6), calculate a new Chisnell function, \( f(z) \):

\[ f(z)_2 = \frac{C_s}{A_{s2}} \]  

(2-12)

(10) Calculate a new pressure ratio, \( z_2 \), using the equation shown in step (1).

(11) Calculate a new Mach number, \( M_{s2} \):

\[ M_{s2} = \sqrt{\frac{\gamma + 1}{2\gamma} (z_2 - 1) + 1} \]  

(2-13)

(12) Calculate a new shock speed, \( W_{s2} \):

\[ W_{s2} = M_{s2} c_0 \]  

(2-14)

(13) Repeat Steps 7 through 12 until the desired shock position has been reached, or until the shock strength asymptotes at Mach 1.

**Main Beam tube Section**

Once the front of the shock/blast wave has spherically expanded to reach the opening of the main beam tube section, the Mach number of the shock wave at the exit of this section of beam tube was calculated in a similar way to the short beam tube section using the equation for a convergent-divergent nozzle. The flow was treated as isentropic, and entrance effects and friction were neglected.

**Numerical Models for Shock/Blast Wave Propagation**

An LLNL proprietary code called GEODYN was used to numerically model the propagation of the shock/blast wave from the target chamber to the final optic. GEODYN is a multidimensional, multiphysics, parallel, Eulerian, adaptive mesh
refinement code capable of multi-fluid hydrodynamics with real equation-of-state behavior.  

_Governing Equations_

Taking the fluid to be inviscid and compressible, the governing equations for continuity, momentum, and energy implemented in the model are, respectively (see Chapter 5 for a complete development of the governing equations):

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0 \tag{2-15}
\]

\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j + p \delta_{ij}) = 0 \tag{2-16}
\]

\[
\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j}((\rho E + p) u_j) = 0 \tag{2-17}
\]

With

\[
E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho |u|^2 \tag{2-18}
\]

The governing equations are closed using real equation-of-state behavior, which for an ideal gas is given by:

\[
p = \rho RT \tag{2-19}
\]

Where

- \(\rho\) = density
- \(u\) = fluid velocity
- \(p\) = pressure
- \(\delta\) = Kronecker delta \((\delta_{ij} = 1\) if \(i = j\); otherwise \(\delta_{ij} = 0\))
- \(E\) = total energy per unit mass of the fluid
- \(\gamma\) = ratio of specific heats of the fluid
- \(R\) = gas constant
\[ T = \text{fluid temperature} \]
\[ t = \text{time} \]
\[ ij = \text{indices ranging from 1 to 3 for three component directions, x, y, and z} \]

**Methodology**

The domain was modeled using a 2D axisymmetric mesh centered along the axis of the beam tube. Figure 2.4a shows the entire r-z mesh, while Figure 2.4b zooms in on the left most part to better show the quadrilateral elements. In Figure 2.4, the red elements represent the fluid domain, and the green elements represent solids. All interfaces between the fluid domain and the solids are represented as walls. The boundary at the final optic is represented as a no-flow condition. The upper boundary at \( r = 0 \) is the axis of symmetry. It should be noted that due to the possibility of turbulent flow conditions, an axisymmetric domain is not entirely accurate. However, for simplicity and reasonable simulation requirements (i.e. computer time and the number of needed computer processors), this approach will suffice as a first cut. Subsequent numerical modeling documented in Chapter 7 will abandon asymmetry.
Figure 2.4  (a) Beam tube mesh for the GEODYN model. (b) Zoomed-in portion of the mesh.

The initial conditions within the target chamber are obtained from the results of BUCKY simulations performed by Greg Moses at the University of Wisconsin. BUCKY is a 1-D radiation hydrodynamic code used to simulate the behavior of high energy density plasmas typical in inertial confinement fusion and target chambers. Given an initial input energy of 8.3 MJ at the center of the target chamber, it takes approximately 1 ms for the front of the plasma shock to reach the inner wall of the target chamber. At that point, the specific energy and density as a function of radial distance (see Figure 2.5) are extracted from the BUCKY output and used as the initial conditions for within the target chamber for the GEODYN shock propagation model. Everywhere else within the flow
regime, the xenon gas density and temperature are initially set to 2 \( \mu g/cm^3 \) and 1000 K. The GEODYN simulation is then allowed to proceed for 100 ms.

Figure 2.5 Input conditions for density and specific energy from the BUCKY model.

Results

The results of the analytical calculations outlined above for the shock/blast wave propagation are summarized in Figure 2.6. From the BUCKY model, the Mach number of the shock at the inner wall of the target chamber is 3.4. A Mach 3.4 shock wave then enters the short section of beam tube, and ignoring any entrance effects and friction for simplicity, exits the diverging short beam tube section at Mach 6.3. The shock then propagates spherically into the gap losing momentum to reach the entrance of the long beam tube section at Mach 1. It then speeds up again as it passes through the diverging long beam tube section to exit at Mach 3.2, again ignoring entrance and frictional effects.
Figure 2.6 Results of the analytical model in terms of Mach number.

Figure 2.7 shows the results of the numerical model in terms of overpressure and Mach number contours of the propagating blast wave through the system at 0, 4, 15, 30, and 60 ms. The 4 micro-second image shows the spherical nature of the shock wave propagation across the gap (gray-scale overpressure), while the colored contour lines show a maximum Mach number of over 5 (red contour) exiting the short beam tube section with an entrance Mach number to the long beam tube section around 1 (light blue contour). These conditions persist throughout the simulation until about 60 ms, at which time we start to see the effects of the input source decay. At 60 ms there is still a Mach 2 shock propagating out the exit of the long beam tube section. These results are similar to those of the analytical calculations discussed previously.
Figure 2.7  Overpressure and Mach number contours at time = 0, 4, 15, 30, and 60 ms.
Figure 2.8 shows the time history of the pressure experienced at the center of the final optic. While the magnitude of the pressures experienced by the final optic is not relatively large, it is thought that the successive impulses (pressure peaks times their respective very short time durations) are the damage mechanism to the optic.

![Figure 2.8 Pressure history at the would-be center of the final optic.](image)

**Conclusions**

Both the analytical and numerical models show that a supersonic blast/shock wave caused by fusion ignition at the center of the target chamber propagates from the outer wall of the target chamber through the short beam tube section. The advancing blast/shock is spherically dispersed across the gap, but still reaches the opening of the long beam tube section at sonic speeds. Since the blast/shock wave enters the long beam tube section at sonic velocities, the blast/shock wave then picks up speed through the diverging section and exits at the location of the final optic with high impulsive strength.
In answer to the first posed question, the blast/shock wave appears to propagate to the final optic given the input conditions within the target chamber. We next turn our attention to answering the second posed question, “Is the propagated blast/shock wave strong enough to damage the final optic?”
CHAPTER 3

DOES THE PROPAGATED BLAST/SHOCK WAVE DAMAGE THE FINAL OPTIC?

Conceptual Model

As described in Chapter 1, critical to ICF is the protection and survival of the final optic that resides approximately 20 to 25 meters from the target, but sits in direct line of sight of target fusion emissions which consist of high-energy (14.1 MeV) neutrons, X-rays, charged burn product and debris ions, as well as an advancing blast/shock wave of “dirty gas.” As established in Chapter 2, the blast/shock does propagate to the final optic. Due to the near vacuum environment within the system the blast/shock wave will be relatively low in pressure (see Figure 2.8), and in and of itself, not very harmful to the optic. However, the relatively low force(s) applied over very short periods of time (micro to milliseconds) will transfer an impulsive momentum to the thin lens (0.5 millimeters) of the final optic that could result in significant displacement and stress. If that displacement is large enough, and/or if the stress exceeds the modulus of rupture, than the lens will be irreparably damaged.

Conceptually, the final optic is modeled as a very flexible circular membrane that is free to pivot at it outer edge and deflect inward and outward at its center, similar to the membrane of drum. It should be noted that this is an over-simplification of the final optic and the way that it will be held in place, but since the design is still under development, the “drum membrane” model will need to suffice. The metric used to determine irreparable damage to the final optic is an allowable deflection angle of \(4 \times 10^{-6}\) radians. Additionally, the maximum deflection distance at the optic center is calculated for a qualitative damage assessment.
Modeling Approach

The “drum membrane” model was implemented analytically using a methodology informally developed by Ralph Moir of LLNL using the results of both the analytical and numerical wave propagation models, as well as numerically using a structural/mechanical code called LS-DYNA.

Analytical Model for Deflection Angle and Displacement of the Final Optic

In the Moir model for the deflection of a thin optic such as a Fresnel lens, it is assumed that an impulsive “puff” of gas pushes on the optic, causing the optic to expand under constant pretension until it comes to rest in an approximate spherical deflection (see Figure 3.1).

![Analytical model for deflection angle and displacement of the final optic.](image)

For small values of deflection:

\[ r = \frac{d^2}{8x} \]  
(3-1)

The mass of the optic is:

\[ m = \rho \frac{\pi d^2}{4} \delta \]  
(3-2)

The volume of the spherical sector is:

\[ V = \frac{\pi}{8} d^2 x \]  
(3-3)

The pressure experienced by the membrane is:
The velocity of the deflection is:

\[ v = \frac{pt}{\rho \delta} \]  (3-5)

The deflection can be derived by calculating the work to inflate a membrane under constant pretension and equating it to the kinetic energy of the membrane just after it has been impulsively loaded:

\[ Work = \int p \, dV = \frac{1}{2} m v^2 \]  (3-6)

The work term is given by:

\[ \int p \, dV = \int \frac{16 \sigma \delta}{d^2} x \frac{\pi d^2}{8} \, dx = \pi \sigma \delta x^2 \]  (3-7)

The kinetic energy term is given by:

\[ \frac{1}{2} m v^2 = \frac{1}{2} \rho \pi d^2 \delta \, v^2 = \frac{\pi d^2 \rho^2 t^2}{8 \rho \delta} \]  (3-8)

Equating the work and kinetic energy:

\[ \frac{\pi d^2 \rho^2 t^2}{8 \rho \delta} = \pi \sigma \delta x^2 \]  (3-9)

The deflection is then given by:

\[ x = \sqrt{\frac{1}{8 \sigma \rho} \frac{dpt}{\delta}} \]  (3-10)

The deflection angle is given by:

\[ \phi = \frac{4x}{d} \]  (3-11)

The allowable optic deflection is given by:

\[ \phi_{allow} = \frac{e_{allow}}{L} \]  (3-12)

Where

\[ p = \frac{2 \sigma \delta}{r} \]  (3-4)
x = optic displacement
\sigma = optic pre-stress
\rho = optic density
d = optic diameter
p = pressure
t = time
\delta = optic thickness
\phi = optic deflection
\phi_{allow} = allowable optic deflection
e_{allow} = allowable pointing error
L = optic standoff

The \( pt \) term in the equation for the deflection is the impulsive load experienced by the optic. The impulsive load can be obtained from the results of the GEODYN pressure history (see Figure 2.8) by doing a base-lined step-wise integration of the peak pressure response. Figure 3.2 shows a zoomed-in view of the pressure history over the 30 to 40 ms time period. The impulsive load is calculated from the area under the curve highlighted in yellow.
Figure 3.2  Zoomed-in view of the pressure history showing the impulsive load as the area under the curve highlighted in yellow that was used as for the $pt$ term in the Moir analytical model.

Given the following ductile material properties for a Fresnel lens:

\[ \sigma = 100 \text{ MPa} \]
\[ \rho = 2.2 \text{ g/cm}^3 \]
\[ d = 200 \text{ cm} \]
\[ \delta = 0.05 \text{ cm} \]

The allowable deflection angle is calculated to be $4.0 \times 10^{-6}$ radians. The optic displacement and deflection angle for the peak impulsive load between 34.3 and 35.6 ms (as shown above in Figure 3.2) can be then calculated from the Moir equations. The results are summarized later in this chapter.
Numerical Model for Deflection Angle and Distance of the Final Optic

An LLNL proprietary code called LS-DYNA was used to numerically model the response of the final optic to the load imposed upon it by the propagated shock wave discussed in Chapter 2. LS-DYNA is a general purpose, multidimensional, Lagrangian, explicit finite element code for analyzing the static and dynamic response of structures coupled to fluids.

Governing Equations

Considering a structural body undergoing time-dependent deformation from a reference point in a fixed rectangular Cartesian coordinate system to a new position in the same coordinate system, the governing equation for continuity, momentum, and energy are, respectively (Hallquist, 2007):

\[
\rho \dot{V} = \rho_0 \quad (3-13)
\]

\[
\sigma_{ij,j} + \rho f_i = \rho \ddot{x}_i \quad (3-14)
\]

\[
\dot{E} = V s_{ij} \dot{e}_{ij} - (p + q) \dot{V} \quad (3-15)
\]

with

\[
s_{ij} = \sigma_{ij} + (p + q) \delta_{ij} \quad (3-16)
\]

where

\[
\rho = \text{density}
\]

\[
V = \text{volume}
\]

\[
\rho_0 = \text{reference density}
\]

\[
\sigma_{ij} = \text{Cauchy stress}
\]

\[
f = \text{body force}
\]

\[
\ddot{x} = \text{acceleration}
\]

\[
s_{ij} = \text{deviatoric stresses}
\]

\[
p = \text{pressure}
\]
Methodology

Conceptually, the optic is structurally modeled as the membrane of a drum pinned at its outer edge. Taking advantage of symmetry, the domain of the optic was modeled as a one-quarter-circle shell in the x- and y-directions with a diameter of 100 cm and a lens thickness of 0.05 cm using quadrilateral elements. The material properties of the Fresnel (SiO$_2$) optic are shown in Table 3.1.

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Fresnel (SiO$_2$) Optic (at 25 °C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Modulus</td>
<td>73 GPa</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>31 GPa</td>
</tr>
<tr>
<td>Rupture Modulus</td>
<td>50 MPa</td>
</tr>
<tr>
<td>Bulk Modulus</td>
<td>36.9 GPa</td>
</tr>
<tr>
<td>Apparent Elastic Limit</td>
<td>55 MPa</td>
</tr>
<tr>
<td>Compressive Strength</td>
<td>1.1 GPa</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>50 MPa</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.17</td>
</tr>
</tbody>
</table>

The pressure load from the shock wave applied orthogonally in the z-direction to the optic was initially at rest. The optic was free to deflect and deform in all three Cartesian directions.

Results

The results of the Moir analytical model show that displacement and deflection angle for the peak impulsive load between 34.3 and 35.7 ms are 0.4762 cm and 0.0095 radians. Given that the thickness of the optic is 0.05 cm, the displacement is almost 10-
times as much, which for a somewhat brittle material would appear to exceed its structural capacity. Additionally, the deflection angle is over 200-times larger than the allowable deflection angle of $4.0 \times 10^{-6}$ radians. Clearly, the impulse imposed on the lens due to the blast/shock is strong enough to compromise the structural integrity of the optic.

Figures 3.3 and 3.4 show the results of the numerical model in terms of contours of optic displacement and maximum principle stress, respectively, at 42 ms. The maximum displacement naturally occurs at the center of the optic and is about 0.45 cm, similar to the value predicted by Moir analytical model. The maximum principle stress naturally occurs at the pinned edge of the optic. Figure 3.5 shows the time-history of maximum principle stress, and is plotted against the rupture modulus for the Fresnel lens and the input pressure. The successive loading and unloading caused by the input pressure generates cyclic maximum principle stresses that far exceed the allowable rupture modulus of the optic.
Figure 3.3  Contours of displacement (mm) at 42 ms for the optic as calculated by the numerical.
Figure 3.4 Contours of maximum principle stress (MBar) at 42 ms for the optic as calculated by the numerical LS-DYNA model.
Figure 3.5 Time-history of maximum principle stress plotted against the rupture modulus and the input pressure.

Summary

Both the analytical and numerical models show that impulsive loading of the optic caused by the propagating blast wave is indeed strong enough to irreparably damage the final optic. Damage occurs in the form of large displacements at the optic center that are nearly 10-times the thickness of the lens, as well as an extremely large deflection angle that far exceeds the structural integrity of the optic. Additionally, the maximum principle stresses at the pinned edge of the optic cyclically exceed the rupture modulus for a thin Fresnel lens. In answer to the second posed question, the propagated blast wave is capable of irreparably damaging the final optic. We next turn our attention to answering the third and final posed question, “What types of mitigation strategies can be deployed to lessen or eliminate the impacts of the blast wave on the final optic?”
CHAPTER 4

CAN THE EFFECTS OF THE PROPAGATED BLAST BE MITIGATED?

Outline

Determining what types of mitigation strategies can be deployed, and their effectiveness in lessening or eliminating the impacts of the propagating blast wave on the final optic comprise the remainder and bulk of the original research presented herein. While the two previous posed questions were investigated and answered entirely within their own separate chapters, the investigation into answering whether or not the propagation of the blast wave can be mitigated in defense of the final optic will require the next three chapters.

Chapter 5 will present the theory of normal shock and moving shock behavior.

Chapter 6 will present the results of a literature search into shock wave propagation for diverging channels and attenuation of shock waves in channels using and mechanical methods and shock wave/boundary layer interactions.

Chapter 7 will investigate the numerical implementation of various strategies, including those found to be promising from the literature search, to attenuate the blast wave. These will include: (1) parametrically changing the diameters and lengths of the beam tubes of the system; (2) disrupting the planar flow of the shock wave by introducing surface “features” to the beam tube inner walls that cause the flow to be turbulent and unsteady.

Chapter 8 will summarize the key numerical and experimental findings, offer conclusions, and ultimately make recommendations for possible future work.
CHAPTER 5
THEORY OF NORMAL AND MOVING SHOCK BEHAVIOR

The Governing Equations

Consider the rectangular control volume for 1-D flow, shown in Figure 5.1, where properties of the fluid are uniform but abruptly change when crossing from the left side (1) of the control volume to the right side (2). This change could represent a hydraulic jump in an incompressible flow or a shock in a compressible flow.

Figure 5.1 Rectangular control volume for 1-D fluid flow (Anderson, 2003).

Continuity

By applying the integral equations of conservation to the control volume, the continuity equation is given as:

$$- \iiint_S \rho \vec{V} \cdot d\vec{s} = \frac{\partial}{\partial t} \iiint_V \rho dV$$  \hspace{1cm} (5-1)

Since the flow is assumed to be steady:

$$\frac{\partial}{\partial t} = 0$$  \hspace{1cm} (5-2)

The continuity equation can be rewritten as:

$$\iiint_S \rho \vec{V} \cdot d\vec{s} = 0$$  \hspace{1cm} (5-3)
Expanding the continuity equation using the variables defined for the control volume yields:

\[
\iiint_{A_1} \rho_1 \bar{u}_1 \cdot \bar{n}_1 \, dA + \iiint_{A_2} \rho_2 \bar{u}_2 \cdot \bar{n}_2 \, dA = 0
\] (5-4)

The previous equation says that the net flux is zero (net flux because a closed surface integral is being evaluated). Since the velocity of the first term is normal to the area but in the opposite direction of the unit normal, and the velocity of the second term is also normal to the area but in the same direction of the unit normal:

\[
\bar{u}_1 \cdot \bar{n}_1 = -u_1
\] (5-5)

\[
\bar{u}_2 \cdot \bar{n}_2 = u_2
\] (5-6)

The continuity equation can be rewritten again as:

\[- \iiint_{A_1} \rho_1 u_1 \, dA + \iiint_{A_2} \rho_2 u_2 \, dA = 0\] (5-7)

Evaluating the integrals yields:

\[\rho_1 u_1 A_1 = \rho_2 u_2 A_2\] (5-8)

Since we’ve defined the control volume as rectangular:

\[A_1 = A_2\] (5-9)

Therefore, the continuity equation, which holds for both compressible and incompressible flows, becomes:

\[\rho_1 u_1 = \rho_2 u_2\] (5-10)

**Momentum**

By applying the integral equations of conservation to the control volume, the momentum equation is given as:

\[
\iiint_S \left( \bar{\rho} \bar{V} \cdot d\bar{S} \right) \bar{V} + \iiint_V \frac{\partial (\bar{\rho} \bar{V})}{\partial t} \, dV = \iiint_V \bar{\rho} \bar{f} \, dV - \iiint_S \bar{p} \, d\bar{S}
\] (5-11)

Since the flow is assumed to be steady and there are no body forces:
\[ \frac{\partial}{\partial t} = 0 \]  

(5-12)

\[ \dot{f} = 0 \]  

(5-13)

So the momentum equation can be rewritten as:

\[ \iiint_S (\rho \dot{V} \cdot d\vec{S}) \dot{V} = -\iiint_S Pd\vec{S} \]  

(5-14)

Since the flow is 1-D, we need only concern ourselves about the scalar components in the x direction. Expanding the momentum equation using the variables defined for the control volume yields:

\[ \iiint_{A_1} (\rho_1 \vec{u}_1 \cdot \vec{n}_1 dA) u_1 + \iiint_{A_2} (\rho_2 \vec{u}_2 \cdot \vec{n}_2 dA) u_2 \]

\[ = -\iiint_{A_1} P_1 dA - \iiint_{A_2} P_2 dA \]

(5-15)

Using the dot products from Eqs. 5.5 and 5.6 and evaluating the integrals, the momentum equation simplifies to:

\[ \rho_1 (-u_1) A_1 u_1 + \rho_2 (u_2) A_2 u_2 = -(P_1 A_1 + P_2 A_2) \]

(5-16)

Again, since we’ve defined a rectangular control volume, the momentum equation, which also holds for both compressible and incompressible flows, becomes:

\[ P_1 + \rho_1 u_1^2 = P_2 + \rho_2 u_2^2 \]  

(5-17)

**Energy**

By applying the integral equations of conservation to the control volume, the energy equation is given as:

\[ \iiint_v \dot{q} dV = \iiint_s \rho \dot{V} \cdot d\vec{S} \]

\[ + \iiint_v \rho (\dot{f} \cdot \dot{V}) dV \]

\[ = \iiint_v \frac{\partial}{\partial t} \left[ \rho \left( e + \frac{v^2}{2} \right) \right] dV \]

\[ + \iiint_s \rho \left( e + \frac{v^2}{2} \right) \vec{V} \cdot d\vec{S} \]  

(5-18)

Since the flow is assumed to be steady with no body forces nor heat generation:
\[
\frac{\partial}{\partial t} = 0
\]  
(5-19)

\[
\dot{f} = 0
\]  
(5-20)

\[
q = 0
\]  
(5-21)

The energy equation can be rewritten as:

\[
- \iint_S p\vec{V} \cdot d\vec{S} = \iiint_S \rho \left( e + \frac{V^2}{2} \right) \vec{V} \cdot d\vec{S}
\]  
(5-22)

Expanding the energy equation using the variables defined for the control volume yields:

\[
- \left( \iiint_{A_1} P_1 \vec{u}_1 \cdot \vec{n}_1 dA + \iiint_{A_2} P_2 \vec{u}_2 \cdot \vec{n}_2 dA \right)
\]

\[
= \iiint_{A_1} \rho_1 \left( e_1 + \frac{u_1^2}{2} \right) \vec{u}_1 \cdot \vec{n}_1 dA + \iiint_{A_2} \rho_2 \left( e_2 + \frac{u_2^2}{2} \right) \vec{u}_2 \cdot \vec{n}_2 dA
\]  
(5-23)

Again using the dot products and evaluating the integrals, the energy equation simplifies to:

\[
-(P_1 (-u_1) A_1 + P_2 u_2 A_2)
\]

\[
= \rho_1 \left( e_1 + \frac{u_1^2}{2} \right) (-u_1) A_1 + \rho_2 \left( e_2 + \frac{u_2^2}{2} \right) (-u_2) A_2
\]  
(5-24)

Again, since we’ve defined a rectangular control volume, the energy equation becomes:

\[
P_1 u_1 - P_2 u_2 = -\rho_1 \left( e_1 + \frac{u_1^2}{2} \right) u_1 + \rho_2 \left( e_2 + \frac{u_2^2}{2} \right) u_2
\]  
(5-25)

Rearranging:

\[
P_1 u_1 + \rho_1 \left( e_1 + \frac{u_1^2}{2} \right) u_1 = P_2 u_2 + \rho_2 \left( e_2 + \frac{u_2^2}{2} \right) u_2
\]  
(5-26)

Noting the final form of the continuity equation in Eq. 5-10 and dividing the left side of Eq. 5-26 by \( \rho_1 u_1 \) and the right side by \( \rho_2 u_2 \), the final form of the energy equation becomes:
\[
\frac{p_1}{\rho_1} + e_1 + \frac{u_1^2}{2} = \frac{p_2}{\rho_2} + e_2 + \frac{u_2^2}{2}
\]

(5-27)

The definition of enthalpy is:

\[
h = e + Pu
\]

(5-28)

Substituting for the energy term using the enthalpy equation into the Eq. 5-27 yields:

\[
\frac{p_1}{\rho_1} + h_1 - p_1 u_1 + \frac{u_1^2}{2} = \frac{p_2}{\rho_2} + h_2 - p_2 u_2 + \frac{u_2^2}{2}
\]

(5-29)

Simplifying yields:

\[
h_1 + \frac{u_1^2}{2} = h_2 + \frac{u_2^2}{2}
\]

(5-30)

Normal Shocks

Consider a flat-faced cylinder placed in subsonic and supersonic flows, as shown in Figure 5.2. Since the flow is composed of individual molecules, some of which impact the front face of the cylinder, there is a change in molecular energy and momentum due to those impacts with the obstruction. In subsonic flow, that change in molecular energy and momentum can be communicated through the random motion of the molecules and propagated upstream. Molecules upstream are warned of the presence of the obstruction and begin to adjust their flow paths to go around it. In supersonic flow, that communication upstream cannot propagate upstream, and a coalescence occurs a short distance ahead of the obstruction. A thin shock wave forms. Ahead of that shock wave the flow is uninformed of the presence of the obstruction, while behind the flow is subsonic and adjusts its streamlines to go around. Quantitatively, there is a discontinuity in the flow properties across the shock. If we assume there is no heat added or taken away from the flow, then the flow across the shock is adiabatic and the governing equations developed previously for continuity, momentum, and energy may be applied.
Applying those general governing equations to a calorically perfect compressible gas where:

\[ p = \rho RT \quad (5-31) \]
\[ h = c_p T \quad (5-32) \]

and dividing the momentum equation by the continuity equation gives:

\[ \frac{p_1}{\rho_1 u_1} - \frac{p_2}{\rho_2 u_2} = u_2 - u_1 \quad (5-33) \]

with:

\[ a = \sqrt{\gamma RT} = \frac{\sqrt{\gamma p}}{\rho} \quad (5-34) \]

Eq. 5-34 can be rewritten as:
\[
\frac{a_1^2}{\gamma u_1} - \frac{a_2^2}{\gamma u_2} = u_2 - u_1 \quad (5-35)
\]

The energy equation can be rewritten as:

\[
c_p T_1 + \frac{u_1^2}{2} = c_p T_2 + \frac{u_2^2}{2} \quad (5-36)
\]

with:

\[
c_p = \frac{\gamma R}{\gamma - 1} \quad (5-37)
\]

If point 1 corresponds to a point where \(u_1 = a\), and point 2 corresponds to some hypothetical location where the fluid element is adiabatically brought to Mach 1 and \(u_2 = a^*\), the energy equation can be rewritten again as:

\[
\frac{\gamma R T_1}{\gamma - 1} + \frac{u_1^2}{2} = \frac{\gamma R T_2}{\gamma - 1} + \frac{u_2^2}{2} \quad (5-38)
\]

\[
\frac{a_1^2}{\gamma - 1} + \frac{u_1^2}{2} = \frac{a_2^2}{\gamma - 1} + \frac{u_2^2}{2} \quad (5-39)
\]

\[
\frac{a^2}{\gamma - 1} + \frac{u^2}{2} = \frac{a^{*2}}{\gamma - 1} + \frac{a^{*2}}{2} \quad (5-40)
\]

\[
\frac{a^2}{\gamma - 1} + \frac{u^2}{2} = \frac{\gamma + 1}{2(\gamma - 1)} a^{*2} \quad (5-41)
\]

or:

\[
a_1^2 = \frac{\gamma + 1}{2} a^{*2} - \frac{\gamma - 1}{2} u_1^2 \quad (5-42)
\]

\[
a_2^2 = \frac{\gamma + 1}{2} a^{*2} - \frac{\gamma - 1}{2} u_2^2 \quad (5-43)
\]

Substituting into the combination of the continuity and momentum equation gives:

\[
\frac{\gamma + 1}{2} \frac{a^{*2}}{\gamma u_1} - \frac{\gamma - 1}{2} u_1 - \frac{\gamma + 1}{2} \frac{a^{*2}}{\gamma u_2} + \frac{\gamma - 1}{2} u_2 = u_2 - u_1 \quad (5-44)
\]

\[
\frac{\gamma + 1}{2\gamma u_1 u_2} (u_2 - u_1) a^{*2} + \frac{\gamma - 1}{2\gamma} (u_2 - u_1) = u_2 - u_1 \quad (5-45)
\]
\[
\frac{\gamma + 1}{2\gamma u_1u_2} a^* + \frac{\gamma - 1}{2\gamma} = 1
\]  
(5-46)

Solving for \(a^*\) yields the Prandtl relation:

\[
a^* = u_1 u_2
\]  
(5-47)

**The Normal Shock**

Consider the case where the normal shock is stationary, as shown in Figure 5.3a. The Prandtl relation is a very useful intermediate equation for normal shock behavior.

From it:

\[
1 = \frac{u_1 u_2}{a^* a^*} = M_1^* M_2^*
\]  
(5-48)

or:

\[
M_2^* = \frac{1}{M_1^*}
\]  
(5-49)
Referring back to Figure 5.2, the flow ahead of the shock wave is supersonic, thus \( M_1 > 1 \), which implies that \( M_1^* > 1 \). From Eq. 5-49 it can then be seen that \( M_2 < 1 \) and \( M_2 < 1 \), thus proving that the Mach number behind the normal shock is always subsonic.

This relation holds for non-calorically perfect gases as well. Dividing Eq. 5-41 by \( u^2 \) gives:

\[
\frac{\left(\frac{a}{u}\right)^2}{\gamma - 1} + \frac{1}{2} = \frac{\gamma + 1}{2(\gamma - 1)} \left(\frac{a^*}{u}\right)^2
\]  

(5-50)

From Eqs. 5-50 and 5-48 and a relationship between the Mach number and characteristic Mach number can be obtained:
\[
\left(\frac{1}{M}\right)^2 + \frac{1}{2} = \frac{\gamma + 1}{2(\gamma - 1)} \left(\frac{1}{M^*}\right)^2
\]

or:

\[
M^2 = \frac{2}{\frac{\gamma + 1}{M^{*2}} - (\gamma - 1)}
\]

Solving for \(M^*\) gives:

\[
M^{*2} = \frac{(\gamma + 1)M^2}{2 + (\gamma - 1)M^2}
\]

Substituting Eq. 5-53 into Eq. 5-49 gives:

\[
\frac{(\gamma + 1)M_2^2}{2 + (\gamma - 1)M_2^2} = \left[ \frac{(\gamma + 1)M_1^2}{2 + (\gamma - 1)M_1^2} \right]^{-1}
\]

Solving for \(M_2^2\) gives:

\[
M_2^2 = \frac{1 + \left(\frac{\gamma - 1}{2}\right)M_1^2}{\gamma M_1^2 - \left(\frac{\gamma - 1}{2}\right)}
\]

Eq. 5-55 shows that for a calorically perfect gas with a constant specific heat ratio, the Mach number behind the shock is only a function of the Mach number ahead of the shock. Ratios of other properties can also be obtained. For example, combining the equation for continuity, the Prandtl relation, and Eq. 5-53 yields:

\[
\frac{\rho_2}{\rho_1} = \frac{u_1}{u_2} = \frac{(\gamma + 1)M_1^2}{2 + (\gamma - 1)M_1^2}
\]

Using the equation for momentum yields:

\[
\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1)
\]

and using the equation of state yields:

\[
\frac{T_2}{T_1} = \frac{h_2}{h_1} = \left[ 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 - 1) \right] \left[ \frac{2 + (\gamma - 1)M_1^2}{(\gamma + 1)M_1^2} \right]
\]
The Moving Shock

Now consider the case of the moving shock in Figure 5.3b. The previously derived equations for continuity, momentum, and energy still apply (Eq. 5-10, 27, and 30). Since $W$ is the velocity of the gas ahead of the shock relative to the wave, and $W - u_p$ is the velocity of the gas behind the shock, again relative to the wave, the governing equations can be rewritten as:

\[ \rho_1 W = \rho_2 (W - u_p) \]  \hspace{1cm} (5-59)

\[ p_1 + \rho_1 W^2 = p_2 + \rho_2 (W - u_p)^2 \]  \hspace{1cm} (5-60)

\[ h_1 + \frac{W^2}{2} = h_2 + \frac{(W - u_p)^2}{2} \]  \hspace{1cm} (5-61)

Substituting Eq. 5-59 into Eq. 5-60 are rearranging gives:

\[ W^2 = \frac{p_2 - p_1}{\rho_2 - \rho_1} \left( \frac{\rho_2}{\rho_1} \right) \]  \hspace{1cm} (5-62)

or

\[ (W - u_p)^2 = \frac{p_2 - p_1}{\rho_2 - \rho_1} \left( \frac{\rho_1}{\rho_2} \right) \]  \hspace{1cm} (5-63)

Substituting Eq. 5-62 and Eq. 5-63 into Eq. 5-61 and simplifying gives:

\[ e_2 - e_1 = \frac{p_1 + p_2}{2} (v_1 - v_2) \]  \hspace{1cm} (5-64)

Eq. 5-64 is known as the Hugoniot equation that relates the changes in the thermodynamic variables across a normal shock. For a calorically perfect gas, the Hugoniot equation can be rewritten as:

\[ \frac{T_2}{T_1} = \frac{p_2}{p_1} \left( \frac{\gamma + 1}{\gamma - 1} + \frac{p_2}{p_1} \right) \]  \hspace{1cm} (5-65)

\[ \rho_2 = \frac{1 + \frac{\gamma + 1}{\gamma - 1} \left( \frac{p_2}{p_1} \right)}{\frac{\gamma + 1}{\gamma - 1} + \frac{p_2}{p_1}} \]  \hspace{1cm} (5-66)
If the Mach number of the moving shock is defined as:

$$M_s = \frac{W}{a_1}$$  \hspace{1cm} (5-67)

Then substituting this definition into Eqs. 5.59, 60, and 61 and using the calorically perfect gas relations gives:

$$M_s = \sqrt{\frac{\gamma + 1}{2\gamma} \left( \frac{p_2}{p_1} - 1 \right) + 1}$$  \hspace{1cm} (5-68)

Solving $M_s$ and then substituting for its definition, Eq. 5-67 gives:

$$W = a_1 \sqrt{\frac{\gamma + 1}{2\gamma} \left( \frac{p_2}{p_1} - 1 \right) + 1}$$  \hspace{1cm} (5-69)

and:

$$u_p = W \left( 1 - \frac{\rho_1}{\rho_2} \right)$$  \hspace{1cm} (5-70)

Substituting Eq. 5-65 and 5-69 into 5.70 gives:

$$u_p = \frac{a_1}{\gamma} \left( \frac{p_2}{p_1} - 1 \right) \sqrt{\frac{2\gamma}{\gamma + 1} \left( \frac{\gamma - 1}{\gamma + 1} + \frac{p_2}{p_1} \right)}$$  \hspace{1cm} (5-71)

Eqs. 5-64, 65, 66, 69, and 71 are commonly used and referred to as the moving shock relationships.

**Area-Velocity Relation**

In subsonic fluid flow, it is intuitive that the velocity of the flow will increase through a converging channel or nozzle, and slow through a diverging channel or nozzle. Most have practical experience with these phenomena when using an ordinary garden hose and nozzle. However, the supersonic case is less intuitive.

Consider an incremental volume as depicted in Figure 5.4.
The continuity equation derived in Chapter 5 for steady quasi-one-dimensional flow:

\[ \rho_1 u_1 A_1 = \rho_2 u_2 A_2 \]  

or, in the general case:

\[ \rho u A = \text{const} \]  

The differential form of the general case is given by:

\[ d(\rho u A) = 0 \]  

If expanded out, the differential form of the continuity equation of the general case becomes:

\[ \frac{dp}{\rho} + \frac{du}{u} + \frac{dA}{A} = 0 \]  

Similarly, the differential form of the momentum equation derived in Chapter 5 for the general case is:

\[ pA + \rho u^2 A + pdA \]

\[ = (p + dp)(A + dA) + (\rho + d\rho)(u + du)^2 (A + dA) \]

Dropping out all the 2nd order terms involving products of differentials gives:

\[ Adp + Au^2 d\rho + \rho u^2 dA + 2\rho u Adu = 0 \]

Multiplying the expanded form of the continuity equation by \( u \) gives:
Equating Eqs. 5-77 and 5-78 and simplifying yields Euler’s equation:

\[ dp = -\rho u du \]  

(5-79)

Using Euler’s equation to eliminate the differential pressure term in the expanded form of the continuity equation gives:

\[ \frac{dp}{\rho} = \frac{dp}{d\rho} = -u du \]  

(5-80)

Since the flow is adiabatic and inviscid, or isentropic, any change in pressure, \( dp \), is accompanied by a corresponding isentropic change in density, \( d\rho \), expressed as:

\[ \frac{dp}{\rho} = \frac{\partial p}{\partial \rho} = a^2 \]  

(5-81)

Combining these last equations gives:

\[ a^2 \frac{d\rho}{\rho} = -u du \]  

(5-82)

or:

\[ \frac{d\rho}{\rho} = -\frac{udu}{a^2} = -\frac{u^2 du}{a^2} = -M^2 \frac{du}{u} \]  

(5-83)

Substituting this result into the expanded form of the differential equation for continuity for the general case yields the area-velocity relationship:

\[ \frac{dA}{A} = (M^2 - 1) \frac{du}{u} \]  

(5-84)

Application of the area-velocity relationship for subsonic flow (\( 0 \leq M < 1 \)) mathematically shows the intuitive case where flow velocity increases where the flow area decreases, and the flow velocity decreases when the flow area increases. It also mathematically proves the non-intuitive case for supersonic flow (\( M > 1 \)) where the flow velocity of increases when the flow area increases (e.g. supersonic flow in a diverging channel or tube), and the flow velocity decreases when the flow area decreases (e.g. supersonic flow in a converging channel or tube).
CHAPTER 6

LITERATURE SEARCH

Attenuation of Propagating Shock Waves

For Laser Inertial Fusion-Fission Energy (LIFE), the ideal configuration for the laser-beam-line is a converging channel or tube from the final focusing optic to the cryogenically frozen deuterium fusion target. However, the ideal configuration for the laser-beam-line is diametrically opposed to the ideal configuration to naturally attenuate a moving shock originating at the target and propagating in the opposite direction of the laser-beam. The moving blast/shock wave sees a diverging channel or tube.

Much research exists for the latter case, attenuating a moving or propagating shock in a converging channel or nozzle, but very little research exists for the former case, attenuating a moving shock in a diverging channel where the flow naturally wants to perpetuate. Some research that was found and is pertinent follows.

**Attenuation the Propagating Shock Waves Using Mechanical Means**

*The Behavior of Shock Waves in Ducts and When Entering Entrance Structures – Schardin and Reichenbach, 1965*

Schardin and Reichenbach in 1965 investigated the attenuation of shock waves in ducts of various diameters and smoothness using shock tubes and Schlieren photography. They surmised that for ducts of varying diameters, the speed of the shock wave, and thus the peak pressure of the shock, was reduced as the duct diameter decreased. This conclusion is intuitive as the boundary layer plays a more significant role in the attenuation of the shock as the duct diameter decreases to be on the same order of the boundary layer thickness. Schardin and Reichenbach also observed that the attenuation of the shock wave was increased by increasing the roughness of the duct.
walls, the reasons for which are again related to the thickness of the boundary layer. Figures 6-1 And 6-2 show their observations.

Figure 6.1 Schlieren photography of a shock wave propagating through pipes of increasing diameters (top to bottom) and time (right to left) (Schardin and Reichenbach, 1965, Figure 2a-2c)

Figure 6.2 Schlieren photography of a shock wave propagating through pipes of increasing roughness (top to bottom) and time (left to right) (Schardin and Reichenbach, 1965, Figure 4a-4b)

*Attenuation of Shock Waves Propagating Over Arrayed Baffle Plates – Ohtomo et. al., 2005*
Ohtomo et. al. investigated the attenuation of shock waves propagating over arrayed baffle plates in 2005. Their application to a synchrotron radiation factory is somewhat similar to that of a LIFE reactor. They performed shock tube experiments using vertically symmetric and oblique and staggered baffle plate arrangements for Mach flows ranging from 1.2 to 3.0 in air, as shown in Figure 6.3. Pressures were measured along the shock tube sidewall. They also performed numerical simulations of the experiments, and compared the results. They found that indeed they could attenuate the shock wave using baffled plates, and that the oblique arrangement provided for a more pronounced effect.

![Figure 6.3 Test section showing the vertically symmetric and oblique and staggered baffle plate arrangements (Ohtomo et. al., 2005, Figure 3)](image)

Figures 6.4 and 6.5 show double exposure holographic interferometry of the supersonic air flow through the experimental arrangements for Mach flows of 1.5 at various times. For the vertically symmetric baffle plates, the flow remains fairly
symmetrical through the first three sets of baffle plates, but becomes quite asymmetric and turbulent through the fourth and fifth sets of baffle plates. The flow through the oblique and staggered baffle plates is never symmetrical. Similar results were obtained for Mach flows of 3.0 as shown in Figures 6.6 and 6.7, though the asymmetry is much more pronounced for both baffle plate arrangements.

Figure 6.4 Interferogram for $M_s = 1.5$ at 2.40 ms for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 8e)

Figure 6.5 Interferogram for $M_s = 1.5$ at 2.51 ms for the oblique and staggered baffle plates (Ohtomo et. al., 2005, Figure 12d)

Figure 6.6 Interferogram for $M_s = 3.0$ at 2.34 ms for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 10f)
Figures 6.8 through 6.11 show pressure histories as a function of position along the shock tube walls (“a” to “h” in the direction of the flow) and the corresponding instances in time to the interferograms (“A” to “F”). These pressure histories clearly show reductions in pressures as a function of position (e.g. as the flows moving through the baffle plate arrays) and over time, with the oblique staggered plate arrangement showing the most significant attenuation of the moving shock.

Figure 6.8 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 1.5$ for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 9)
Figure 6.9 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 1.5$ for the oblique staggered baffle plates (Ohtomo et. al., 2005, Figure 13)

Figure 6.10 Pressure variation along the shock tube wall from “a” to “h” for $M_s = 3.0$ for the vertically symmetric baffle plates (Ohtomo et. al., 2005, Figure 11)
waves and rigid obstacles or orifice plates, modifies considerably the flow field by types of large geometrical obstacles. They note the center of the end—Expe consequence of attenuating the propagation. The flow to impede the laser be blocked by large obstacles in the flow. This research presented herein and baffle plates that extended well into the flow, In 2009, Berger et. al. investigated the effects (pressures and loads) induced on though this research was conducted using a constant cross-sectional area shock tube and baffle plates that extended well into the flow, it still has application to the research presented herein for a diverging channel and where the laser line-of-sight cannot be blocked by large obstacles in the flow. Potentially, surface features that would not impede the laser-beam could be employed on the walls of the beam tube that would cause the flow to behave similarly by becoming asymmetrical and turbulent with the natural consequence of attenuating the propagation.

*Experimental Investigation on the Shock Wave Load Attenuation by Geometrical Means*

— Berger et. al., 2009

In 2009, Berger et. al. investigated the effects (pressures and loads) induced on the center of the end-wall of a shock tube by a shock wave passing through different types of large geometrical obstacles. They noted that, “The interaction between shock waves and rigid obstacles or orifice plates, modifies considerably the flow field by
introducing new waves (shocks, compressions and rarefaction), vortices, and regions of intense turbulence. These new waves can reduce the energy traveled with the transmitted shock wave and the load imposed by it.”

Their experimental set-up is shown in Figure 6.12, and included a 5.5 m long horizontal shock tube with an 8 cm square internal cross-section. Moderately low Mach number (\(M_s \approx 1.2\)) shock waves were generated by rupturing a Mylar diaphragm that initially separated the driven section from the driver section. Transparent plexi-glass sidewall windows were installed in a test-section of the shock tube for visualization via Schlieren photography, a pulsed frequency light source, and a shutterless high-speed camera.
A test-section was fabricated with aligned grooves on the top and bottom walls to allow for the insertion of the various geometrical obstacles. Each grooved section was spaced 4 cm center-to-center from the next. The geometrical obstacles included thin plates oriented at inclination angles of 45°, 90°, and 135°. The heights of the obstacles also varied to explore the effects of differing relative opening fractions (ROF), which was...
calculated as the distance between the obstacles mounted to the top and bottom walls, and the height or width of the shock tube ($d/w$). Multiple obstacles could be inserted and varying distances apart from each other. Figure 6.13 is a schematic of a test-section using (a) a single obstacle, and (b) multiple obstacles at inclination angles of $45^\circ$.

![Diagram](image)

Figure 6.13 Schematic of a typical test-section (Berger et al., 2009, Figure 3).

Results from a single obstacle inclined at $45^\circ$ with an ROF=0.375 at M=1.2 are shown in Figures 6.14 and 6.15. The time between consecutive images is 56 $\mu$s. With the shock entering from the left, a reflected curved shock propagates upstream (Figure

<table>
<thead>
<tr>
<th>Relative opening fraction (ROF)</th>
<th>Obstacle inclination ($\alpha$)</th>
<th>Number of obstacles</th>
<th>Obstacles separation distance (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.375</td>
<td>45$^\circ$</td>
<td>1</td>
<td>49</td>
</tr>
<tr>
<td>0.5</td>
<td>90$^\circ$</td>
<td>2</td>
<td>80</td>
</tr>
<tr>
<td>0.625</td>
<td>135$^\circ$</td>
<td>3</td>
<td>18 cm</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4</td>
<td>18 cm</td>
</tr>
<tr>
<td></td>
<td></td>
<td>5</td>
<td>18 cm</td>
</tr>
</tbody>
</table>

Table 1 The values of the various obstacle-geometry parameters that were investigated

Most of the experiments with each obstacle geometry were repeated at least three times in order to reduce the uncertainty. For finding the effect of the obstacles separation distance on the load that is developed on the end-wall, only the three obstacles configurations were used. To implement this experimental setup, every other groove in the test section (see Fig. 3) was used. This created an obstacles separation of 80 mm.
6.14b) then expands into the opening downstream of the obstacle. Vortices develop at the tips of the plates (Figure 6.14c) and then detach to propagate in a curved path towards the top and bottom of the test-section walls. The main shock propagates downstream to the right inducing a complex series of reflected shocks and rarefacted shocks off the end-wall. The authors note that the 45° obstacle tends to act as a converging nozzle.

Figure 6.14 Schlieren images from a single obstacle test at 45° with an ROF=0.375 at M=1.2 (Berger et. al., 2005, Figure 4).
The authors repeated the single obstacle testing for inclination angles of 90° and 135° and for different ROFs of 0.625, 0.5, 0.375, and 0.25. They conclude that the larger the ROF, the higher the pressure jump across the transmitted and reflected shock waves, though very little difference was observed for each obstacle configuration having ROFs of 0.5 and 0.625. Keeping the ROF constant and varying the obstacle inclination angle also resulted in different pressure jumps, with a more pronounced effect for smaller ROF values.

Berger et al. performed a similar suite of tests using multiple obstacles of the same kind (inclination angle) from 2 to 5 along the top and bottom walls of the test-chamber. Figure 6.16 shows three sets of Schlieren photographs with 0.112 ms in between consecutive images for three obstacles per test-section for inclination angles of 45°, 90°, and 135° for a M=1.2. For the multi-obstacle configuration inclined at 45°
(Figure 6.16, column I), the shock propagates both downstream and upstream as it passes each set of plates. Strong vortices are induced, detach, and spin off towards the top and bottom walls in a curved path, though secondary reflected shocks hit these vortices and slow their motions. All this motion creates a very complex and turbulent flow field. For the multi-obstacle configuration inclined at 90° (Figure 6.16, column II), similar flow patterns occur but are less intense because the transmitted shock does not propagate upstream as strong as it did for the 45° inclination angle case. Vortex flow is much less intense for the 135° inclination angle case (column III). This diverging nozzle type of obstacle crops the shock wave with the central part expanding through the center of the test-section past the other obstacles and the non-central parts of the flow becoming trapped in the space between the obstacle plate and the top and bottom walls.
The authors used a single parameter that represented the load at the center of the end-wall of the shock tube to make comparisons between the various geometric configurations. They termed this parameter the impulse linear slope, which they calculated as:

\[
Impulse \ Linear \ Slope = \frac{I(t_c)}{t_c - t_0} \tag{6-1}
\]

\[
I(t_c) = \int_{t_0}^{t_c} P_{ew}(t') \, dt' \tag{6-2}
\]

Where

\[
I(t_c) = \text{Time integral of the pressure measured at the center of the end-wall}
\]
\( P_{ew} = \) Pressure at the center of the end wall

\( t_0 = \) Time when the incident shock reached the end-wall

\( t_c = \) Integration time

Figure 6.17 shows calculated impulse linear slopes for ROFs of 0.375 and 0.625, inclination angles of 45°, 90°, and 135°, number of obstacles from 1 to 5, and integration times of 0.25 ms, 1 ms, 2 ms, and 5 ms. For each combination of ROF and inclination, increasing the number of obstacles reduced the load at the center of the end-wall, with the obstacles with the inclination angle of 135° (diverging nozzle configuration) showing the best ability to attenuate shock wave load. They also noted that increasing the distance between obstacles also decreased the impulse load, with the inclination angle of 135° attenuating the load most effectively.
Berger et. al. summarized that for the case of a single geometrical obstacle, ROF played the most significant role on shock wave load attenuation, the larger the ROF the greater the attenuation. For multi-obstacle configurations, the shock wave load was attenuated with increasing the number of obstacles at early times, or before the reflected shock waves off the end-wall were reflected back to the end wall, with the converging configuration (inclination angle of 45°) having the most prominent effect on shock wave reflection off and back to the end-wall because the geometrical shape of the obstacle traps the reverberations. They concluded that divergent-nozzle type geometric obstacles were the best for attenuating the shock wave load on the end-wall.

Turbulent Shock Wave Boundary Layer

Shock wave/boundary layer interactions (SWBLI) is an increasingly popular field of study, with a number of recent papers and text books published that include:

- **Turbulent Shear Layers in Supersonic Flow** by Smits and Dussauge, 1996
- **Numerical Simulation of Viscous Shock Layer Flow** by Golovachov, 1995
- **Turbulent Shear Layer/Shock Wave Interactions** by Delery, 1985
- **Some Physical Aspects of Shock Wave/Boundary Layer Interactions** by Delery and Dussauge, 2009
- **Shock Wave Boundary Layer Interaction** by Hadjadj and Dussauge, 2009

*Some Physical Aspects of Shock Wave/Boundary Layer Interactions – Delery and Dussauge, 2009*

Delery and Dussauge discussed some physical aspects of the interactions of shock waves with boundary layers in a paper published in Shock Waves in 2009. The shock wave/boundary layer interactions (SWBLI) produce complex phenomena because of what they term, “the rapid retardation of the boundary layer flow and the propagation of the shock in a multilayered structure.” In essence, the boundary layer experiences an
adverse pressure gradient caused by the shock that greatly distorts the boundary layer velocity profile. As well, when the flow is turbulent, that turbulence is enhanced and the coupled effect leads to viscous dissipation and large unsteadiness in the flow. Figure 6.18 shows how Delery and Dussauge graphically show the complexity of the SWBLI on the flow.

![Diagram of shock wave/boundary layer interactions](image.png)

**Figure 6.18** Complex flow of shock wave/boundary layer interactions.

The inviscid shock \((C_1)\) penetrates into the rotational inviscid part of the boundary layer where its Mach number decreases causing it to progressively bend to the point that it weakens and disappears when it reaches the boundary layer sonic line. At the same time, incident shocks form upstream because of the pressure rise creating complex interaction of shock and boundary layer velocity profile disturbances. While these interactions are mechanically undesirable for machine or vehicle performance, such as in jet engines, it’s precisely this mechanism that will be investigated in the next chapter to mitigate moving shocks in an expanding duct.
CHAPTER 7

NUMERICAL STUDY

Numerical Modeling and Mitigation of a Moving Shock in an Expanding Duct

A comprehensive numerical study was performed to assess the effectiveness of various mitigation strategies that might be deployed and/or engineered to lessen or eliminate the impacts of the propagating blast/shock wave on the final optic. First, a parametric computational analysis was performed to determine what effects changes in the geometrical configuration of the target chamber and beam tube might have on the blast/shock wave propagation, and included altering the short and long beam tube lengths and openings, and also introducing simple wall treatments designed to promote turbulence and flow structure detachment (e.g. boundary layer separation). The results of the parametric computational study prompted the development of a more rigorous numerical model that was used to study the effects of turbulence and flow structure detachment on the flow regime and shock propagation for mitigation.

The Miranda Hydrodynamics Code

The Miranda hydrodynamics code, developed by Lawrence Livermore National Laboratory (LLNL) was used to perform the parametric studies and advanced numerical modeling on LLNL’s supercomputing platforms and environment. Miranda is a proprietary, multi-disciplinary, multi-dimensional, multi-physics, parallel, adaptive mesh refinement (AMR) code capable of multi-fluid hydrodynamics with real equation-of-state behavior.

Miranda Governing Equations

Taking the fluid to be inviscid and compressible, the index notation of the governing equations for continuity, momentum, and energy implemented in the code are, respectively:
\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \]  
(7-1)

\[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j + p \delta_{ij}) = 0 \]  
(7-2)

\[ \frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_j} ((\rho E + p) u_j) = 0 \]  
(7-3)

with

\[ E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho |u|^2 \]  
(7-4)

The governing equations are closed using real equation-of-state behavior, which for an ideal gas is given by:

\[ p = \rho R T \]  
(7-5)

where

\( \rho \) = fluid density

\( u \) = fluid velocity

\( p \) = pressure

\( \delta \) = Kronecker delta (1 if \( i = j \), 0 if otherwise)

\( E \) = total energy per unit mass of the fluid

\( \gamma \) = ratio of specific heats of the fluid

\( R \) = gas constant

\( T \) = fluid temperature

\( t \) = time

\( i, j \) = indices ranging from 1 to 3 for the three component directions, \( x, y \) and \( z \)

**Turbulence Modeling**

Miranda uses a modified form of large eddy simulation (LES), which is a numerical technique, used to solve the partial differential equations (PDEs) governing turbulent fluid flow. Meteorologists first formulated LES in the 1960’s as a way to
computationally capture very high Reynolds number flows using coarse gridding schemes.

In terms of computational effort, LES stands between direct numerical simulation (DNS) and Reynolds-averaged Navier-Stokes (RANS) approaches. DNS is very often too computationally expensive even for modern day supercomputers, while RANS methods lack the ability to capture the detailed flow structures of turbulent flow.

LES is based on the theory that large eddies in the flow are dependent on the flow geometry, while smaller eddies are self-similar and have a more universal nature. For the bulk flow, that is the flow not affected by walls in the domain, LES seeks to explicitly solve for the larger eddies, while modeling the effects of the smaller eddies on the larger ones using sub-grid scale (SGS) models. Near the walls where boundary layers are apt to develop, LES is often coupled to zonal approaches with RANS or other empirically based models capable of resolving the boundary layer.

SGS models typically solve the unresolved flow structures (unresolved because the eddies are smaller than the grid scale) by applying a filter to the Navier-Stokes equations. A common filtering approach to compensate for the unresolved turbulent scales is to add an eddy viscosity term to the governing equations.

Another approach, named implicit large eddy simulation (ILES), integrates the filtered equations between grid points to generate a set of second-order finite difference equations. Those equations are solved using a numerical reconstruction scheme, but such a scheme is often subject to large dissipation and dispersion errors, while also being highly susceptible to grid imprinting.

Miranda employs an artificial fluid large eddy simulation (AFLES) for turbulence modeling. This technique is described by a paper by Cook in 2007, and attempts to model the large-scale behavior of a fluid using artificial properties that simulate the characteristics of the real fluid in lieu of filtering the governing equations.
The artificial properties consist of modifications to the transport coefficients of shear viscosity, bulk viscosity, thermal conductivity, and species diffusivity. AFLES allows the freedom to choose a high-fidelity numerical scheme that works directly with the governing equations, rather than having to employ numerical schemes on the filtered equations like ILES. The SGS employed by AFLES is a numerical damping scheme designed to provide the correct energy transfer rate through the cutoff wavenumber. The AFLES damping scheme allows the artificial properties (viscosity and diffusivity) to impart a high-wavenumber bias to the dissipation, and therefore approximates the cusp in the Heisenberg-Kraichnan spectral viscosity for isotropic turbulence.

AFLES adds grid dependent components to the transport coefficients (dynamic viscosity, bulk viscosity, thermal conductivity, species-i diffusion coefficient) in the governing equations:

\[
\begin{align*}
\mu &= \mu_f + \mu^* \\
\beta &= \beta_f + \beta^* \\
\kappa &= \kappa_f + \kappa^* \\
D &= D_{f,i} + D_i^* 
\end{align*}
\]

where \(f\) denotes the fluid property and \(^*\) denotes the artificial property. The artificial properties are required to be positive definite, frame invariant, and carry over to the incompressible limit (i.e. where the viscosity is not dependent on the speed of sound in the medium). However, unlike the real fluid properties, the artificial properties are designed to vanish in smooth regions while providing strong damping near discontinuities.

The models for the artificial properties act like switches, turning on only where fields are insufficiently smooth with respect to grid scale. The artificial bulk viscosity term in the governing equations allows the scheme to capture shocks without excessive
damping of vorticity. The artificial thermal conductivity terms helps to remove ringing at the heat fronts. The artificial diffusivity term helps keep mass fractions between zero and one.

A tenth-order compact finite difference scheme is used to solve the first and second derivatives of the governing equations. A five-step fourth-order explicit Runge-Kutta method is used to advance the solution of the governing equations in time. The stability criterion of the numerical solution is determined by the inviscid Courant-Friedrichs-Lewy (CFL) condition, as well as the maximum viscosity, conductivity, and diffusivity existing within the domain. A description of the numerical methods and Navier Stokes solver for compressible flow is given in Appendix 1.

**Summary of the Miranda AFLES Scheme**

In summary, the given governing equations with the addition of the viscous/Reynolds stress tensor term solve compressible fluid flow with large (i.e. grid-scale) turbulent structures. When those turbulent structures become smaller than the grid-scale, or when sharp discontinuities are present (i.e. shocks, heat fronts, fluid mixing fronts), those features are not resolvable. However, the AFLES scheme includes “switches” to turn on artificial properties that modify the transport coefficients (dynamic viscosity, bulk viscosity, thermal conductivity, and diffusivity) in order to numerically resolve the smaller than grid-scale turbulent structures or the flow discontinuities.

**Parametric Modeling Methodology**

A parametric computational analysis using Miranda was performed to determine what effects changes in the geometrical configuration of the target chamber and beam tube had on the blast wave propagation, and included altering the primary and secondary beam tube lengths and openings, and also introduced simple wall treatments designed to promote turbulence. For computational efficiencies, each model domain was simulated in 2-D rather than 3-D. Due to the possibility of turbulent flow conditions, the 2-D
domains were fully simulated, as an axisymmetric boundary condition down the length of
the domain would have been inappropriate. For cases 1 through 6, slip flow conditions
were used at the walls. Table 7.1 summarizes the cases run and compared.

**Table 7.1** Parametric modeling case descriptions.

<table>
<thead>
<tr>
<th>Case</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depiction</td>
<td><img src="image1" alt="Depiction" /></td>
<td><img src="image2" alt="Depiction" /></td>
<td><img src="image3" alt="Depiction" /></td>
<td><img src="image4" alt="Depiction" /></td>
<td><img src="image5" alt="Depiction" /></td>
<td><img src="image6" alt="Depiction" /></td>
<td><img src="image7" alt="Depiction" /></td>
<td><img src="image8" alt="Depiction" /></td>
<td><img src="image9" alt="Depiction" /></td>
</tr>
<tr>
<td><strong>Target Chamber Radius (cm)</strong></td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td>250</td>
</tr>
<tr>
<td><strong>Short Beam tube Length (cm)</strong></td>
<td>282</td>
<td>100</td>
<td>347</td>
<td>282</td>
<td>100</td>
<td>347</td>
<td>282</td>
<td>100</td>
<td>347</td>
</tr>
<tr>
<td><strong>Short Beam tube Exit Diameter (cm)</strong></td>
<td>20.78</td>
<td>13.67</td>
<td>23.32</td>
<td>83.28</td>
<td>54.79</td>
<td>93.46</td>
<td>20.78</td>
<td>13.67</td>
<td>23.32</td>
</tr>
<tr>
<td><strong>Short Beam tube Wall Function</strong></td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
</tbody>
</table>
The basecase and Common Initial Conditions

The basecase is the same as that presented in Chapter 2, albeit fully simulated with Miranda instead of GEODYN and without an axisymmetric boundary condition. The initial conditions within the target chamber for each case are the same, and are obtained from the results of 1-D hydrodynamic BUCKY simulations. Given an initial input energy of 8.3 MJ at the center of the target chamber, it takes approximately 1 ms for the front of the plasma shock to reach the inner wall of the target chamber. At that point, the specific energy and density as a function of radial distance (see Figure 7.1) are extracted from the BUCKY output and used as the initial conditions for within the target chamber for all the other cases. Everywhere else within the flow regime, the xenon gas density and temperature are initially set to 2 \( \mu \text{g/cm}^3 \) and 1000 K. Each case was simulated for up to 100 ms. In all cases, the distance from the center of the target chamber to the entrance of the second section of beam tube remained the same. As well, the entrance diameter and half-angle of the second section of beam tube remained the same for every case.
Results and Comparisons of Geometrical Changes to the Basecase (Cases 1 through 6)

Case 2 differed from the basecase in that the first section of beam tube was shortened from 282 cm to 100 cm, thus decreasing the short beam tube exit diameter from 20.78 cm to 13.67 and enlarging the plenum or gap between the exit of the first section of beam tube and the entrance to the second section of beam tube. The half-angle of the first section of beam tube was the same for both cases. Computationally, Case 2 does not converge on a solution after about 65 ms and the simulation crashes. However, enough of a solution is reached to draw conclusions. A comparison of the basecase (Case 1) to Case 2, shown in Figure 7.2, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison clearly shows that a shorter section of beam tube slows the arrival of the propagating blast/shock wave, but does little to alter its magnitude.
Figure 7.2 Comparison of the basecase (Case 1) to Case 2.

Case 3 differed from the basecase in that the first section of beam tube was lengthened from 282 cm to 347 cm, thus increasing the short beam tube exit diameter from 20.78 cm to 23.32 cm and shortening the gap between the exit of the first section of beam tube and the entrance to the second section of beam tube. The half-angle of the first section of beam tube was the same for both cases. A comparison of the basecase (Case 1) to Case 3, shown in Figure 7.3, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison clearly shows that a longer section of beam tube has little affect on the arrival of neither the propagating blast/shock wave nor its magnitude.
Figure 7.3 Comparison of the basecase (Case 1) to Case 3.

Case 4 differed from the basecase by increasing the half-angle of the first section of beam tube, which changed the short beam tube entrance diameter from 9.77 cm to 39.14 cm and the short beam tube exit diameter from 20.78 cm to 83.28 cm. The gap space between the exit of the first section of beam tube and the entrance to the second section of beam tube remained the same. A comparison of the basecase (Case 1) to Case 4, shown in Figure 7.4, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison clearly shows that a wider first section of beam tube quickens the arrival of the propagating blast/shock wave and increases its magnitude.
Figure 7.4 Comparison of the basecase (Case 1) to Case 4.

Case 5 is a combination of Case 2 and Case 4. Case 5 differs from the basecase by shortening the first section of beam tube was from 282 cm to 100 cm, and widening the half-angle of the short beam tube resulting an entrance diameter change from 9.77 cm to 39.14 cm and an exit diameter from 20.78 cm to 54.79 cm. The resulting gap space between the exit of the first section of beam tube and the entrance to the second section of beam tube is longer. A comparison of the basecase (Case 1) to Case 5, shown in Figure 7.5, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison shows that a wider and shorter first section of beam tube slows the arrival of the propagating blast/shock wave and decreases its magnitude.
Case 6 is a combination of Case 3 and Case 4. Case 6 differs from the basecase by lengthening the first section of beam tube was from 282 cm to 347 cm, and widening the half-angle of the short beam tube resulting an entrance diameter change from 9.77 cm to 39.14 cm and an exit diameter from 20.78 cm to 93.46 cm. The resulting gap space between the exit of the first section of beam tube and the entrance to the second section of beam tube is shorter. A comparison of the basecase (Case 1) to Case 6, shown in Figure 7.6, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison shows that a wider and longer first section of beam tube hastens the arrival of the propagating blast/shock wave and increases its magnitude.
Given these first five comparisons to the basecase, an obvious way to mitigate the effects of the propagating blast wave on the final optic is to widen and shorten the first section of beam tube. However, other factors, including the ease of design and engineering may not allow for the widening of the short section of beam tube. This prompted an investigation into using surface treatments on the walls of the beam tubes to promote turbulence as a means of slowing the arrival of the propagating blast wave and decreasing its magnitude. Cases 7, 8, and 9 explore this option.

*Conceptual Implementation of Wall Treatments to Promote Turbulence*

Inviscid flow theory governs the bulk motion of a gas at some distance away from the walls (i.e. outside the boundary layer). That is to say, there are no viscous forces in the bulk flow, which is modeled using the Euler equations. The Euler equations are simply the Navier-Stokes equations with zero viscosity terms. Since there are no viscous forces in the bulk flow, and since Re number is the ratio of inertial forces to
viscous forces, Re number no longer becomes a measurable parameter with respect to inviscid flow. Essentially the Re goes to infinity for an "ideal" inviscid fluid. For real inviscid fluids, the flows are simply characterized as having very, very high Re numbers.

An ideal inviscid fluid, when brought into contact with a surface or wall, would naturally slip right past it since the fluid has no viscosity. However, for a real fluid a thin boundary layer forms between the wall or surface and the bulk flow. Classical boundary layer theory governs this flow. That boundary layer naturally starts as laminar, but grows in thickness over the length of the boundary until it either separates (due to an adverse pressure gradient) or becomes turbulent. Generally, a turbulent boundary layer is thicker (or reaches further into the bulk flow) than a laminar boundary layer.

For the situation of interest herein, the beam tube blast/shock wave propagation model is bulk inviscid flow coupled with boundary layer theory, complicated by the fact that the inviscid flow is supersonic and shock. Typically, the boundary layer over the length of the short section of beam tube (between the target chamber and the gap) stays laminar because that section of beam tube is not long enough for the boundary layer to naturally trip to turbulent. The bulk flow is supersonic with a very high implied Re number, but retains a parallel streamlined profile.

Upon encountering the abrupt change in area at the interface between the short beam tube section of the gap, the flow separates and eddies/vortices appear, but the flow stays symmetrical as it propagates across the gap because the boundary layer upstream stayed laminar and the bulk flow lacked eddies/vortices. The bulk flow undergoes another abrupt change as it enters the long section of beam tube. This section is long enough for the boundary layer to trip to turbulent and the bulk flow to develop eddies/vortices.

By purposely tripping the boundary layer in the short section of beam tube, the turbulent boundary layer may grow in thickness to point where it reaches far enough into
the bulk flow to cause the bulk to flow to lose its parallel streamlined looking profile. The turbulent boundary layer may also reach far enough into the bulk flow that it "sees" the turbulent boundary layer from the opposite side of the wall, thus really knocking the bulk flow out of its streamlined pattern. Upon exiting the short section of beam tube, this turbulent flow is not axisymmetric like it was for the basecase model, isn't directly in-line with the opening to the longer beam tube section, and therefore loses much of its "punch" upon propagating down the longer beam tube section.

Results and Comparisons of Wall Treatment Changes to the Basecase (Case 1 and Cases 7 through 9)

Case 7 only differs from the basecase (Case 1) in that the walls of the short section of beam tube are treated as non-slip and numerically given a surface treatment. This is done numerically in Miranda using a technique called “blocking” which essentially forces the wall boundary conditions to be diffusely enforced. A comparison of the basecase (Case 1) to Case 7, shown in Figure 7.7, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. The comparison clearly shows that tripping the flow into turbulent conditions slows the arrival of the propagating blast/shock wave and decreases its magnitude.
Figure 7.7 Comparison of the basecase (Case 1) to Case 7.

Case 8 is compared to Case 2 (short first section of beam tube), with the only difference being that for Case 8 the walls of the short section of beam tube are numerically given a surface treatment. A comparison of the Case 2 to Case 7, shown in Figure 7.8, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. While both simulations don’t converge on a solution causing them to crash, the pressure time histories are similar. This would indicate that this shorter section of beam tube is not long enough to promote significant turbulence.
Figure 7.8 Comparison of the Case 2 to Case 7.

Case 9 is compared to Case 3 (long first section of beam tube), with the only difference being that for Case 9 the walls of the short section of beam tube are numerically given a surface treatment. A comparison of the Case 3 to Case 9, shown in Figure 7.9, is made by plotting the pressure time histories at the center of the exit of the second section of beam tube. These results are similar to what was observed between the basecase (Case 1) and Case 7, indicating that once turbulence is tripped and the boundary layer extends far enough into the bulk flow, increasing the length even more does little else to slow the arrival or the decrease the magnitude of the propagating blast/shock wave.
The Rigorous Numerical Modeling

Based on the promising results of Case 7 presented in the previous section, a more rigorous numerical modeling effort was performed using Miranda within LLNL’s supercomputing environment. Runs were made on multi-node processors and took anywhere from hours to days to run, depending on the desired level of fidelity of the results. For the sake of computational efficiency, and the restrictions and hierarchy of the LLNL high-performance computing environment (e.g. computing resources given to higher profile projects), the rigorous numerical modeling was performed in two-dimensions rather than in three-dimensions. It should be noted here that two-dimensional calculations of turbulence might be unrealistic, even if they are perfectly resolved. This is because when solving the Navier Stokes equation in two-dimensions, one of the velocity components is implicitly set to zero and does not allow for variation of the fluid's
properties in that direction. Thus, the vorticity (which usually has 3 components) will only have one. The result of these conditions often create long “paint-like mixing” structures because the vorticity has no place to go. If the Reynolds number was fairly low, and the flow was laminar and didn't shed vorticity, then a two-dimensional result would be equivalent to a three-dimensional result. As will be seen in the results presented later in this section, the flow is higher Reynolds number and the results could be fairly different in three-dimensions. As such, a recommendation in Chapter 8 is given that future modeling be performed in three-dimensions.

**Computational Mesh**

The two-dimensional computational mesh is generated within Miranda itself by mapping a structured mesh to the flow regime bounded by the target chamber and short and long sections of beam tube. For the models presented in this section, the mesh is not axisymmetric so as to allow the full formation of turbulent flow structures and detachment/re-attachment. Though the mesh size was refined and the results visually compared before settling on an appropriate cell height, width, and density, a rigorous grid independence study was not performed because of the issues outlined previously for two-dimensional versus three-dimensional flow modeling.

**Initial and Boundary Condition**

The initial conditions within the target chamber are obtained from the results of BUCKY simulations as noted in Chapter 2, and presented again in this chapter (see Figure 7.1). BUCKY is a 1-D radiation hydrodynamic code used to simulate the behavior of high energy density plasmas typical in inertial confinement fusion and target chambers. Given an initial input energy of 8.3 MJ at the center of the target chamber, it takes approximately 1 ms for the front of the plasma shock to reach the inner wall of the target chamber. At that point, the specific energy and density as a function of radial distance (see Figure 2.5) are extracted from the BUCKY output and used as the initial
conditions for within the target chamber for the GEODYN shock propagation model. Everywhere else within the flow regime, the xenon gas density and temperature are initially set to $2 \mu g/cm^3$ and 1000 K. The simulations are run out to between 100 ms to 200 ms.

Figure 7.10 shows the model domain and boundary conditions. Neumann, or zero flux boundary conditions are used at the walls of the target chamber and beam tubes (shown as red lines in Figure 7.10). At the far right end of the domain, the final optic is represented used an open or out-flow boundary condition to eliminate upstream rarefaction of the shock waves. This type of boundary condition (shown as a blue lines in Figure 7.10) allows for a simple continuation of the solution outside the domain, essentially setting all gradients to zero. This same type of out-flow boundary condition is also used at the plenum interfaces between the short and long beam tube sections, and for the target chamber (honoring the circular nature of that part of the domain). As noted earlier, parametric cases 1 through 6 allowed flow slip at the beam tube walls for computational efficiency. For more accurate results, no-slip flow conditions at the beam tube walls are used to allow for boundary layer development. Additionally, the mesh was refined adaptively near the beam tube walls to improve the flow modeling.

![Figure 7.10 Model domain showing the boundary conditions used for the Miranda modeling.](image)
Real equations of state for the fluid regime are used to close the governing equations.

Results

Figure 7.11 shows a time-lapse of the flow, color contoured by Mach number, from its origination point at the target, to the interface between the target chamber and the short section of beam tube, to the final optic (far right boundary of the domain). Results are shown for times starting at 0.00 ms, 4.12 ms to 13.12 ms in 1 ms increments, 21.42 ms, 24.12 ms, 29.12 ms, 40.12 ms, and 44.12 ms. At 4.12 ms, the blast initiated at the center of the target chamber has propagated to the outer wall of the target chamber, easily overcome any entrance effects at the opening of the short section of beam tube, and supersonically propagated to the entrance of the plenum used to clear the target chamber of gas and debris in between successive shots (e.g. gap between the short and long beam tube sections). At 5.12 ms, driven by the divergent nature of the short beam tube section, the flow gains velocity to greater than Mach 3 and expands spherically into plenum. By 6.12 ms, the shock front has propagated across the plenum to the entrance of the long beam tube section, and by 7.12 ms has overcome entrance effects and propagated into the long beam tube section. Times 8.12 ms to 13.12 ms show the propagation of the flow, still supersonic, down the long beam tube section. By 24.12 ms, the flow in the long beam tube section has become quite asymmetric and turbulent, though the upstream flow in the short beam tube section and plenum remain laminar and symmetrical. By 29.12 ms, the flow across the plenum has become asymmetric and turbulent caused by the depletion in the energy of the source from the target chamber, though the flow in the short beam tube section remains symmetric and laminar for about the first 20 ms of the simulation. By 40.12 ms the shock front has reached the final optic, and though the flow behind the shock front is turbulent as desired, it is still impulsively strong enough to destroy the final optic. Though the optic is not physically modeled, the last time-lapse
image at 44.12 ms shows how the flow would continue to propagate given the less the futile resistance of the final optic to the impulsive shock.
Figure 7.11 Time-lapse of the simulated flow from the target chamber to the final optic.
Figures 7.12 Through 7.15 are zoomed in views of the model domain, clipping the length of the long beam tube section by 15 m from the final optic toward the target chamber, and respectively show the bulk viscosity, temperature, pressure and bulk velocity of the flow at 6 ms into the simulation. Along side each color contoured plot of the respective variable (e.g. bulk viscosity, temperature, pressure, and velocity), is an x-y plot that shows the variations of that same variable as a function of position along the 10 m long length of the domain for from the target chamber for (1) a half-angle of zero degrees or along the lengthwise axis, and (2) a small half-angle or very near the wall of the beam tube sections. Note that for each of these sets of figures, there is an offset of 0.5 m down the length of the domain for the color contoured image and the x-y plots. Note also that units are in CGS (centimeters, grams, and seconds).

As expected, the bulk viscosity along the lengthwise axis changes sharply at the shock fronts at the entrance of the short beam tube section (2.5 m in the color contoured plot, and 2.0 m in the x-y plot), and at the leading front of the shock in the long section of beam tube (~7.5 m in the color contoured plot, and ~7.0 in the x-y plot). Note also the drop of the bulk viscosity at the entrance of the plenum as the flow expands spherically into the open space.

Of note in the plots of the temperature (reported in units of eV), is the gradient within the target chamber, the rapid cooling through the beam tube sections, and the sharp rise in temperature just before the entrance of the long beam tube section as the flow piles up on itself trying to propagate into the entrance. This sharp rise in temperature near the long beam tube entrance correlates well to the rise in pressure at the same location as seen in Figure 7.13. The x-y plots of the bulk viscosity, temperature, and pressure along the zero half-angle and the small half-angle are fairly close in magnitude and track each other. Not so, however, for the bulk velocity shown in Figure 7.14. The flow along the lengthwise axis is markedly faster than the flow near the wall.
boundary of the short beam tube while the spherical expansion into the open space of the plenum allows the off-axis velocity to catch up.

Figure 7.12 Bulk viscosity of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in red) and ~2 degrees (shown in gray).
Figure 7.13 Temperature of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in red) and ~2 degrees (shown in blue).

Figure 7.14 Pressure of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in green) and ~2 degrees (shown in blue).
Figure 7.15  Velocity of the fluid regime at 6 ms (left) and an x-y plot of bulk viscosity down the length of the flow domain (shifted 0.5 m from the target chamber center) for half-angles of zero degrees (shown in orange) and ~2 degrees (shown in pink).

Referring back to Figure 7.11 for times later than 13.12 ms, the flow down the long beam tube section exhibits an observed phenomenon relative to diverging nozzle diffusers of rocket engines. The diverging nature of the long beam tube section causes the supersonic flow to accelerate and generates shock waves that impinge on the beam tube walls. This interaction between the shock and the turbulent fluid near the wall causes vortices to shed and leads to unsteadiness in the flow itself. In the field of rocket engine design, this phenomenon is categorized as either free shock separation (FSS) or restricted shock separation (RSS). Figure 7.16 generalizes these two types of shock separations. As summarized by Olson in 2012, “RSS is characterized by a small separation region or ‘bubble’ which exists immediately downstream of the shock wave. In this region, the mean flow circulates (moving upstream in some regions) before the flow reattaches to the wall and continues down the length of the nozzle as an attached boundary layer…In FSS, the separation region downstream of the shock fails to reattach
for the remaining length of the nozzle. A shear layer forms and the region of separation grows as it is convected down the length of the nozzle.” This unsteadiness causes lateral forces or side loads on the walls confining the flow, and in the case of the rocket engine adversely affects the engine’s stability. For the case of the beam tubes, however, this unsteadiness is exactly what is sought after, and in fact, the objective is to enhance it as it helps to dissipate the impulsive load on the final optic.

Figure 7.16 Generalized depiction of free flow separation (FSS - Left) and restricted shock separation (RSS – Right) for a rocket engine (Oslund and Muhammad-Klingmann, 2005).

Figure 7.17 shows a time-lapse of density and Mach number for an even more zoomed in view of the flow regime highlighting the short beam tube section from its entrance to about 4.75 m from the target chamber center (or about 0.5 m from the interface of the exit of the short beam tube section and the plenum). Results for times shown range from 0.00 ms to 2.50 ms in 0.50 ms increments, then for 5 ms, 10 ms, and 20 ms. The density is shown as a gray-scaled color contour range, and the Mach number as discrete rainbow colored line contours. The velocity of the flow, as seen in the Mach number line contours, clearly increases with time as it propagates down the diverging channel, with the Mach number going from 1 to 1.5 at 1 ms to greater than 3 at 10 ms.

For up to 1.00 ms, the flow resembles that of a normal moving shock, with shock “diamonds” forming behind the shock front from 1.0 ms to 2.5 ms. At 5 ms, the formation of vortices at the outer edges of the bulk flow is seen, with some of those
vortices detaching near the beam tube walls. At 10 ms some of the detached vortices have reattached and the increase in velocity as seen in the Mach number line contours produces even more pronounced shock diamonds in the bulk flow. At 20 ms, the symmetric flow begins to breakdown and become unsteady near the exit of the short beam tube, but the propagation of the shocked flow up to that point in time is still impulsively strong enough to damage the final optic. Unsteadiness of the flow in the short beam tube causes the flow into the plenum to be “flappy” and not jet straight across to the long beam tube section as shown in Figure 7.11 at 29.12 ms, 40.12 ms, and 44.12 ms. Enhancing this effect could cause the flow to enter the long beam tube section at a velocity below Mach 1 or subsonically, and in turn the diverging nature of the beam tube would decelerate the flow and reduce the impulsive forces on the final optic. One way to accomplish this, as suggested by Ohtomo and Berger in the documented literature searches of Chapter 6, is to introduce physical obstructions in flow field, though in this case those obstructions need to be located along the walls of the beam tubes and small enough so as to not interfere with the propagation of the laser to the target.

Figures 7.18 and 7.19 show the results of adding short notches 180 degrees opposed on the walls of the short beam tube. In three dimensions, this would be a short triangular ring around the internal beam tube wall. For comparison purposes to Figure 7.17, Figures 7.18 and 7.19 show the same time-lapse of density and Mach number for the short beam tube section from its entrance to about 4.75 m from the target chamber center (or about 0.5 m from the interface of the exit of the short beam tube section and the plenum). The density is again shown as a gray-scaled color contour range, and the Mach number as discrete rainbow colored line contours. The addition of the obstructions cause the formation of vortices that both detach and reattach much earlier in the simulation, though the notches also act as a sort of diffuser and increase the Mach number of the flow directly downstream when compared to the case without any notches.
In fact, multiple notches seem to “pump” the flow as seen in Figure 7.19 for up to 2.5 ms. The addition of these notches causes the flow to become unsteady around 5 ms, some 15 ms earlier than the case without any notches. The unsteadiness both slows the velocity of the flow and the propagation further downstream, and by 20 ms the flow in the short beam tube section has nearly dissipated.

Figure 7.20 shows a comparison at 3.2 ms for the three cases. For the smooth walled beam tube case, the Mach number of the flow at 4.75 m is between Mach 2 and 2.5. Adding a notch at 2.75 m to the otherwise smooth walled beam tube, the Mach number of the flow at 4.75 m has slowed to just above 1, and around 4.50 m the flow begins to be asymmetrical and unsteady. By adding multiple notches at 2.75 m, 3.25 m, and 3.75 m to the otherwise smooth walled beam tube, the Mach number of the flow at 4.75 m has slowed to around 0.5 and has gone subsonic. Around 3.35 m the flow begins to be asymmetrical and unsteady and compounds with distance traveled down the beam tube. Figure 7.20 shows the pressure that the final optic experiences as a function of time for the three cases. For the single notch case, the pressure is both later and reduced when compared to the case without notches. For the case with multiple notches, the peak pressure response on the final optic is greatly reduced and experienced more than 10 ms later than the case without notches.
Figure 7.17 Timelapse of the flow through the small beam tube with no notches.
Figure 7.18 Timelapse of the flow through the small beam tube with notches.
Figure 7.19 Timelapse of the flow through the small beam tube with multiple notches.
Figure 7.20. Density and Mach number contours at 3.2 ms for the short beam tube section without and with notches.

Figure 7.21 Plots of pressure versus time at the final optic for the case without and with notches.
CHAPTER 8

CONCLUSIONS

As conceptually demonstrated by authors investigating similar phenomena and through a course of parametric and rigorous numerical modeling presented in the previous chapters, it is possible to slow the propagation of a strong shock wave in a diverging channel or duct by introducing small wall obstructions that cause the shock waves to interact with the boundary layer, forcing turbulence and unsteadiness, thereby mitigating the impulsive force experienced at the end of the channel of duct, or in this case by the final focusing optic of a LIFE reactor. The primary mechanism for slowing the propagation of the moving shock is boundary layer separation. For wall-bounded flow, at high Reynolds number a laminar boundary layer will begin to become unstable, and small perturbations will grow causing the flow to transition to turbulent. Adverse pressure gradients within the boundary layer cause the flow to detach or separate, sometimes reattaching and sometimes not. If the flow reattaches, the effects of the separation that occurred upstream persist. Instabilities in the separated flow regime will drive the transition to turbulence to be faster causing large unsteadiness in the flow. Additionally, asymmetries arise due to instability of the boundary layer. Very small disturbances in the boundary layer grow exponentially. Small wall obstructions in the flow, like notches, trigger faster transition to turbulence, unsteadiness, and asymmetry.

The physics of the simulations presented herein are qualitatively accurate. Obstructions to wall geometry can trigger separation and enhance mixing and turbulence as noted. However, depending on the quantities one desires to capture, higher fidelity calculations may be needed. For example, higher fidelity could be achieved by modeling the phenomenology in three-dimensions rather than two. While computationally very expensive and beyond the reach of the efforts presented herein, the turbulence modeling
boundary layer interactions would be more realistic and resolved. Additionally, there are some caveats to the results of the simulation presented herein with respect to the way that the Miranda code models the boundary layer. Miranda uses a blocking methodology that forces the wall boundary conditions to be diffusely enforced. This blocking methodology is computationally efficient and has application where the physical boundary layer thickness is much larger than the blocking thickness, which may be the case for flow in small diameter ducts like in the beam tubes modeled herein. However, it may not have proper application for certain flows that rely heavily on the physics near the wall, causing numerical representation of the flow to dominate the physics.

That said, there is certainly future work that could be done, namely modeling the flow regime in three dimensions, using an adaptive mesh refinement, a more physical treatment of the shock wave boundary layer, and experimental work to validate the results and conclusions presented herein.
APPENDIX 1

NUMERICAL METHODS – THE MIRANDA CODE

A description of the numerical methods employed by the Miranda code is reproduced here from Olson, 2012, Appendix A.

Governing Equations for the Compressible Navier-Stokes Solver

Miranda solves the compressible Navier-Stokes equations in a Cartesian coordinate system given as:

\[
\frac{\partial U}{\partial t} + \nabla_x \cdot F_x = \phi
\]

(A-1)

Where

\[
\nabla_x \equiv \left( \frac{\partial \rho}{\partial x}, \frac{\partial \rho u}{\partial y}, \frac{\partial \rho w}{\partial z} \right)
\]

(A-2)

\[x \equiv (x, y, z)\]  

(A-3)

The equations for conservation of mass, momentum, and total energy are:

\[U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E \end{pmatrix} \]

(A-4)

\[F_x = \begin{pmatrix} \rho u \\ \rho u^2 + p - \tau_{xx} \\ \rho uv - \tau_{xy} \\ \rho uw - \tau_{xz} \\ u(E + p - \tau_{xx}) - v\tau_{xy} - w\tau_{xz} - q_x \end{pmatrix} \]

(A-5)

\[F_y = \begin{pmatrix} \rho v \\ \rho uv - \tau_{xy} \\ \rho v^2 + p - \tau_{yy} \\ \rho vw - \tau_{yz} \\ v(E + p - \tau_{yy}) - u\tau_{xy} - w\tau_{yz} - q_y \end{pmatrix} \]

(A-6)
The source term is given as:

$$\phi = \begin{pmatrix} 0 \\ \rho g_x \\ \rho g_y \\ \rho g_z \\ \rho (g_xu + g_yv + g_zw) \end{pmatrix} \quad (A-7)$$

The viscous stress tensor for Newtonian fluids is given as:

$$\tau = 2\mu S + \left(\beta - \frac{2}{3}\mu\right)(\nabla_x \cdot \mathbf{u})\delta \quad (A-8)$$

Where the symmetric strain rate tensor is given as:

$$S = \frac{1}{2} (\nabla_x \mathbf{u} + (\nabla_x \mathbf{u})^T) \quad (A-9)$$

Fourier’s law for the conductive heat flux is:

$$\mathbf{q} = -k\nabla_x T \quad (A-10)$$

Real equations of state are used to close the governing equations and are given as:

$$p = (\gamma - 1)\rho e \quad (A-11)$$

$$T = (\gamma - 1)\frac{e}{R} \quad (A-12)$$

Spatial Finite Differencing

The first derivative of the tenth-order compact finite difference scheme is numerically approximated as:

$$\beta f'_{j-2} + \alpha f'_{j-1} + f' + \alpha f'_{j+1} + \beta f'_{j+2} = a \frac{f_{j+1} - f_{j-1}}{2\Delta} + b \frac{f_{j+2} - f_{j-2}}{4\Delta} + c \frac{f_{j+3} - f_{j-3}}{6\Delta} \quad (A-13)$$

Where $f'_j$ is the derivative of the continuous variable $f$ at node $j$ and $\Delta$ is the grid spacing between nodes. The coefficients for the derivative are:
This derivative operator is applied along grid directions and yields a penta-diagonal matrix of the form:

\[ A\vec{f}' = \vec{b}(f) \]  

(\text{A-14})

Whose solution is a vector of the derivatives of \( f \).

For non-periodic boundary conditions, telescoping geometric arrangements of nodal groups, or stencils, are constructed to maintain conservation, such that only the boundary nodes contribute to the boundary fluxes.

Similarly, in the tenth-order finite difference scheme, the second derivatives comprising the Laplacian operators in the artificial fluid properties are computed as:

\[
\beta f_{j-2}'' + \alpha f_{j-1}'' + f''' + \alpha f_{j+1}'' + \beta f_{j+2}'' \\
= a \frac{f_{j+1} - 2f_j + f_{j-1}}{\Delta^2} + b \frac{f_{j+2} - 2f_j + f_{j-2}}{4\Delta^2} + c \frac{f_{j+3} - 2f_j + f_{j-3}}{9\Delta^2}
\]  

(\text{A-15})

The coefficients for the second derivative are:

\[
\alpha = \frac{334}{899} \\
\beta = \frac{43}{1798} \\
a = \frac{1065}{1798}
\]
\[ b = \frac{1038}{899} \]

\[ c = \frac{79}{1798} \]

**Temporal Integration**

The governing equations are advanced in time by casting them and integrating using a five-step fourth-order Runge-Kutta. This scheme is used for its broad stability for both the convective and viscous terms. The scheme is given as:

\[ Q^\eta = \Delta t F^{\eta-1} + A^\eta Q^{\eta-1} \quad \text{(A-16)} \]

\[ \Phi^\eta = \Phi^{\eta-1} + B^\eta Q^\eta \quad \text{(A-17)} \]

For \( \eta = 1, \ldots, 5 \) and \( A^\eta \) and \( B^\eta \) are:

\[ A^1 = 0 \]

\[ A^2 = -6234157559845/12983515589748 \]

\[ A^3 = -6194124222391/4410992767914 \]

\[ A^4 = -31623096876824/15682348800105 \]

\[ A^5 = -122511854476824/11596622555746 \]

\[ B^1 = 49439346753/4806282396855 \]

\[ B^2 = 4047970641027/5463924506627 \]

\[ B^3 = 9795748752853/13190207949281 \]

\[ B^4 = 4009051133189/8539092990294 \]

\[ B^5 = 1348533437543/7166442652324 \]

The fraction of \( \Delta t \) for which the solution advances after each subset is:

\[ \eta = 1 \rightarrow 494393426753/4806282396855 \]

\[ \eta = 2 \rightarrow 4702696611523/9636871101405 \]

\[ \eta = 3 \rightarrow 3614488396635/5249666457482 \]

\[ \eta = 4 \rightarrow 9766892798963/10823461281321 \]
\[ \eta = 5 \rightarrow 1 \]

Compact Filter

Partial de-aliasing is accomplished by applying an eight-order compact filter to the conserved variables after each Runge-Kutta substep. The compact filter is used to remove 10\% of the wavenumbers in as sharp a manner as possible so that the results remain independent of the frequency of the filter. This helps prevent the artificial fluid properties from becoming too large. The filter is:

\[
\begin{align*}
\beta f_{j-2} + \alpha f_{j-1} + f_j + \alpha f_{j+1} + \beta f_{j+2} \\
= a f_j + \frac{b}{2} (f_{j-1} + f_{j+1}) + \frac{c}{2} (f_{j-2} + f_{j+2}) + \frac{d}{2} (f_{j-3} + f_{j+3}) \\
+ \frac{e}{2} (f_{j-4} + f_{j+4})
\end{align*}
\] (A-18)

Where

\[ \alpha = 0.66624 \]
\[ \beta = 0.16688 \]
\[ a = 0.99965 \]
\[ \frac{b}{2} = 0.66652 \]
\[ \frac{c}{2} = 0.16674 \]
\[ \frac{d}{2} = 0.00004 \]
\[ \frac{e}{2} = -0.00005 \]

Gaussian Filter

The formation of artificial fluid properties requires the application of a truncated Gaussian filter. This filter eliminates cusps introduced by the absolute value operator, which in turn, ensures that the artificial transport properties are positive definite. The truncated Gaussian filter is given as:
\[ f(x) = \int_{-L}^{L} G(|x - \xi|; L)f(\xi)d\xi \quad \text{(A-19)} \]

Where:

\[ G(\xi; L) = \frac{e^{-\frac{6\xi^2}{L^2}}}{\int_{-L}^{L} e^{-\frac{6\xi^2}{L^2}} d\xi}, \quad L = 4\Delta \quad \text{(A-20)} \]
APPENDIX 2

NUMERICAL METHODS – THE BUCKY CODE

A description of the numerical methods employed by the BUCKY code is reproduced here from MacFarlane, et. al., 1995. BUCKY is a one-dimensional Lagrangian radiation-hydrodynamics code developed at the University of Wisconsin Fusion Technology Institute to model Inertial Confinement Fusion (ICF) high energy density plasmas. It solves a single fluid equation of motion, where electrons and ions are assumed to move together, with pressure contributions from electrons, ions, radiation, and fast charged particles. Shocks are handled using a von Neumann artificial viscosity. BUCKY uses high-quality equations of state and multi-group opacity tables which provide data for both low-Z and high-Z plasmas over densities ranging from the dilute ideal gas to highly compressed matter. In addition to radiation, the following physical processes are included in the electron and ion energy equations as source terms:

- Fast ion (beam or target debris) energy deposition
- Heating due to the deposition of fast charged particles and neutrons during the fusion burn phase
- Laser energy deposition
- X-ray heating of a cold buffer gas

Fusion burn equations from deuterium-tritium, deuterium-deuterium, and deuterium-helium 3 reactions are solved, and charged particle reaction products are transported and slowed using a time-dependent particle tracking algorithm.

Governing Equations for the Mass, Momentum, and Energy Conservation

The conservation of mass and momentum in Lagrangian coordinates are given as:
\[
\frac{\partial V}{\partial t} = V \frac{\partial u}{\partial r} = \frac{\partial}{\partial m_o} (r^{\delta-1} u)
\] (B-1)

\[
\frac{\partial u}{\partial t} = - \frac{1}{\rho} \frac{\partial}{\partial r} (P + q) = - r^{\delta-1} \frac{\partial}{\partial m_o} (P + q) + u_{TN}
\] (B-2)

Where \( V = 1/\rho \) is the specific volume, \( u \) is the fluid velocity, and \( m_o \) is the Lagrangian mass variable, \( P = P_e + P_i + P_r \) is the total fluid pressure, \( q \) is the von Neumann artificial viscosity, and \( u_{TN} \) is the velocity change due to momentum exchange from the slowing down of fast non-thermal particles. The artificial viscosity is introduced into the inviscid equation of motion to deal with shocks, as its function is to smooth the shock fronts by adding a small amount of dissipation into the equation. The density and specific volume in Eq. B-1 are actually computed after the time-dependent radii are computed from the updated velocities. Once the velocities of the boundaries at \( t^{n+1/2} \) are known, the new boundary positions at \( t^{n+1} \) can be calculated. For Eq. B-2, the explicit difference equation used to solve the partial differential equations and is given as:

\[
\frac{u_j^{n+1/2} - u_j^{n-1/2}}{\Delta t^n} = -(-r^{\delta-1})^n_j \left( \frac{\Delta P_j^n + \Delta q_j^{n-1/2}}{\Delta m_o} \right) + u_{TNj} \] (B-3)

The conservation of energy is represented by temperature diffusion equations for the electrons (e) and ions (i). The Lagrangian forms are give as:

\[
C_{ve} \frac{\partial T_e}{\partial t} = \frac{\partial}{\partial m_o} \left( r^{\delta-1} K_e \frac{\partial T_e}{\partial r} \right) - \omega_e (T_e - T_i) - [(E_e)_\nu + P_e] \frac{\partial V}{\partial t} T_e + A - J
\] (B-4)

\[
C_{vi} \frac{\partial T_i}{\partial t} = \frac{\partial}{\partial m_o} \left( r^{\delta-1} K_i \frac{\partial T_i}{\partial r} \right) - \omega_e (T_e - T_i) - [(E_i)_\nu + P_i] \frac{\partial V}{\partial t} T_i + A - J
\] (B-5)

\[
-e
\]
Where $c_{ve}$ and $c_{vi}$ are the electron and ion specific heats, $K_e$ and $K_i$ are the electron and ion thermal conductivities, $\omega_c(T_e - T_i)$ is the electron-ion collisional coupling term, $A$ and $J$ are the radiative heating and cooling terms, and $S_e$ and $S_i$ are source inputs to the electrons and ions. These equations are posed in a convenient matrix form for the purposes of the numerical solution.

**Radiation Transport Model**

The multi-group radiation transport equation is given as:

$$
V \frac{\partial E_R^g}{\partial t} = \frac{\partial}{\partial m}(\tau^{\delta-1} \kappa_R^g \frac{\partial E_R^g}{\partial r}) - \frac{4}{3} E_R^g \dot{\nu} - c \sigma_{p,A}^g E_R^g + J^g,
$$

(B-6)

$g = 1, \ldots, G$

Where $E_R^g$ is the radiation energy density, $\kappa_R^g$ is the radiation conductivity for the frequency group $g$, $J^g$ is the rate of radiation emitted by the plasma into group $g$, $\sigma_{p,A}^g$ is the Planck absorption opacity for group $g$. The multi-group radiation equations are written in finite difference forms and computed using the finite difference scheme described a bit later.

**Fusion Burn Energy Deposition**

For the fusion burn reaction and energy deposition in BUCKY, the thermonuclear reaction for deuterium-tritium is given as:

$$
1D^2 + 1T^3 \rightarrow 2He^4 (3.5 \text{ MeV}) + 0n^1 (14.1 \text{ MeV})
$$

(B-7)

The deuterium-deuterium reactions are:

$$
1D^2 + 1D^2 \rightarrow 2He^3 (0.82 \text{ MeV}) + 0n^1 (2.45 \text{ MeV})
$$

(B-8)

$$
1D^2 + 1D^2 \rightarrow 1T^3 (1.01 \text{ MeV}) + 1p^1 (3.02 \text{ MeV})
$$

(B-9)

The deuterium-helium 3 reaction is:

$$
1D^2 + 2He^3 \rightarrow 2He^4 (3.6 \text{ MeV}) + 1p^1 (14.7 \text{ MeV})
$$

(B-10)
BUCKY includes the reaction rates for these reactions and solves the rate equations describing the depletion of the individual species. These are solved using simple Euler difference equations.

**Time Step Control**

The finite difference scheme used in BUCKY is a backward substitution solution to the implicit Crank-Nicholson difference scheme. All values are evaluated at both \( t^n \) and \( t^{n+1} \). This implicit numerical scheme solves two coupled equations using matrices of the scalar coefficients that are inverted to block tridiagonals. For linear equations, the Crank-Nicholson scheme is unconditionally stable and accurate to order \((\Delta t)^2\) and \((\Delta x)^2\). This generally allows for a much larger time step than the explicit scheme. For the non-linear equations, stability issues arise unless the time step is restricted. This time step restriction is given as:

\[
\Delta t^{n+3/2} = \max \left[ \Delta t_{min}, \min \left( \Delta t_{max}, \frac{K_1}{R_1^{n+1}}, \frac{K_2 \Delta t^{n+1/2}}{R_2^{n+1}}, \ldots \frac{K_5 \Delta t^{n+1/2}}{R_5^{n+1}} \right) \right] \tag{B-11}
\]

Where:

\[
R_1^{n+1} = \max \left( \frac{V_j^{n+1} p_j^{n+1}}{\Delta x_j^{n+1/2}} \right) \tag{B-12}
\]

\[
R_2^{n+1} = \max \left( \frac{V_j^{n+1} - V_j^n}{V_j^{n+1/2}} \right) \tag{B-13}
\]

\[
R_3^{n+1} = \max \left( \frac{E_{j-1/2}^{n+1} - E_{j-1/2}^n}{E_{j-1/2}^{n+1/2}} \right) \tag{B-14}
\]

\[
R_4^{n+1} = \max \left( \frac{T_{i-1/2}^{n+1} - T_{i-1/2}^n}{T_{i-1/2}^{n+1/2}} \right) \tag{B-15}
\]
\[ R_{5}^{n+1} = \text{Max} \left( \frac{T_{e_j}^{n+1} - T_{e_j}^n}{T_{e_j}^{n+1/2} - T_{e_j}^{n/2}} \right) \] (B-16)
BIBLIOGRAPHY


Moir, R., 2001, “Consider a thin silica optics hit by a puff of gas. How much deflection can be expected and tolerated?” Informal communication.


VITA
Graduate College
University of Nevada, Las Vegas

Veraun Chipman

Address:
3023 Lenoir St., Las Vegas, NV, 89135

Degrees:
Bachelor of Science, Civil and Environmental Engineering, 1995
Brigham Young University, Provo, UT

Master of Science, Civil and Environmental Engineering, 1995
Brigham Young University, Provo, UT

Employment:
National Security Technologies, Senior Scientist, 2011 to Present
Las Vegas, NV

Lawrence Livermore National Laboratory, Physicist/Engineer, 2001 – 2011
Livermore, CA

Santa Fe, NM

Selected Publications:

Chipman, V., Culbreth, W., “Modeling the Effects of Using Small Wall Obstructions on the Propagation of a Shock Wave in a Diverging Duct,” In draft, Intended to be submitted to the journal Shock Waves.


Dissertation Title:
Mitigation of Moving Shocks in an Expanding Duct

Ph.D. Committee:
Dr. William Culbreth
Dr. Darrell Pepper
Dr. Brendan O”Toole
Dr. Evangelos Yfantis
Dr. Sahendra Singh