

Technical Report ARWSB-TR-19009

Geometric Effect on Impulse in a Gas Redirected Shock Tube

David Marshall

Research, Development and Engineering Command
Armament Research, Development and Engineering Center
Weapons Software & Engineering Center
Benét Laboratories
Watervliet, NY 12189

October 2018



Distribution A. Approved for public release: distribution unlimited.

The views, opinions, and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy, or decision, unless so designated by other documentation.

The citation in this report of the names of commercial firms or commercially available products or services does not constitute official endorsement by or approval of the U.S. Government.

Destroy this report when no longer needed by any method that will prevent disclosure of its contents or reconstruction of the document. Do not return to the originator.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Washington Headquarters Service, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington, DC 20503.

PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.

| | | | | | |
|---|---------------------------|---|--|---|--|
| 1. REPORT DATE (DD-MM-YYYY) Oct 2018 | | 2. REPORT TYPE Technical Report | | 3. DATES COVERED (From - To) | |
| 4. TITLE AND SUBTITLE Geometric Effect on Impulse in a Gas Redirected Shock Tube | | | | 5a. CONTRACT NUMBER | |
| | | | | 5b. GRANT NUMBER | |
| | | | | 5c. PROGRAM ELEMENT NUMBER | |
| 6. AUTHOR(S) David Marshall | | | | 5d. PROJECT NUMBER | |
| | | | | 5e. TASK NUMBER | |
| | | | | 5f. WORK UNIT NUMBER | |
| 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) U.S. Army ARDEC-Benet Laboratories RDAR-WSB Watervliet, NY 12189-4000 | | | | 8. PERFORMING ORGANIZATION REPORT NUMBER ARWSB-TR-19009 | |
| 9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) | | | | 10. SPONSOR/MONITOR'S ACRONYM(S) | |
| | | | | 11. SPONSORING/MONITORING AGENCY REPORT NUMBER | |
| 12. DISTRIBUTION AVAILABILITY STATEMENT Distribution A: Approved for public release: distribution unlimited. | | | | | |
| 13. SUPPLEMENTARY NOTES | | | | | |
| 14. ABSTRACT The behavior of an expanding gas from a shock tube when redirected by external geometry can vary with changes implemented to the external geometry. Large force oscillations initially occurred, as modifications were made to the geometry the force-vs-time curve dampened. Changing the geometry impacts how the expanding flow reacts to the change in direction. The analysis conducted five numerical studies on different geometries at two different stability values. Force vs time curves were created and the impulse was computed from the force curves. The different geometries were compared to one another for impulse and force dampening. Results show that modifying the internal geometry does impact the impulse. Results also show that the explicit scheme was stable and that using a larger courant number improved computing resources while not affecting error in impulse values. By conducting numerical studies of how the impulse is impacted by geometry, a more stable, effective, efficient model was developed. | | | | | |
| 15. SUBJECT TERMS Gas; Shock Tube; Oscillations; Geometry; Force-vs-Time; Impulse | | | | | |
| 16. SECURITY CLASSIFICATION OF: | | | 17. LIMITATION OF ABSTRACT SAR | 18. NUMBER OF PAGES 7 | 19a. NAME OF RESPONSIBLE PERSON David Marshall |
| a. REPORT U | b. ABSTRACT SAR | c. THIS PAGE SAR | | | 19b. TELEPHONE NUMBER (Include area code) (518) 266-6232 |

INSTRUCTIONS FOR COMPLETING SF 298

1. REPORT DATE. Full publication date, including day, month, if available. Must cite at least the year and be Year 2000 compliant, e.g., 30-06-1998; xx-08-1998; xx-xx-1998.

2. REPORT TYPE. State the type of report, such as final, technical, interim, memorandum, master's thesis, progress, quarterly, research, special, group study, etc.

3. DATES COVERED. Indicate the time during which the work was performed and the report was written, e.g., Jun 1997 - Jun 1998; 1-10 Jun 1996; May - Nov 1998; Nov 1998.

4. TITLE. Enter title and subtitle with volume number and part number, if applicable. On classified documents, enter the title classification in parentheses.

5a. CONTRACT NUMBER. Enter all contract numbers as they appear in the report, e.g. F33615-86-C-5169.

5b. GRANT NUMBER. Enter all grant numbers as they appear in the report, e.g. 1F665702D1257.

5c. PROGRAM ELEMENT NUMBER. Enter all program element numbers as they appear in the report, e.g. AFOSR-82-1234.

5d. PROJECT NUMBER. Enter all project numbers as they appear in the report, e.g. 1F665702D1257; ILIR.

5e. TASK NUMBER. Enter all task numbers as they appear in the report, e.g. 05; RF0330201; T4112.

5f. WORK UNIT NUMBER. Enter all work unit numbers as they appear in the report, e.g. 001; AFAPL30480105.

6. AUTHOR(S). Enter name(s) of person(s) responsible for writing the report, performing the research, or credited with the content of the report. The form of entry is the last name, first name, middle initial, and additional qualifiers separated by commas, e.g. Smith, Richard, Jr.

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES). Self-explanatory.

8. PERFORMING ORGANIZATION REPORT NUMBER. Enter all unique alphanumeric report numbers assigned by the performing organization, e.g. BRL-1234; AFWL-TR-85-4017-Vol-21-PT-2.

9. SPONSORING/MONITORS AGENCY NAME(S) AND ADDRESS(ES). Enter the name and address of the organization(s) financially responsible for and monitoring the work.

10. SPONSOR/MONITOR'S ACRONYM(S). Enter, if available, e.g. BRL, ARDEC, NADC.

11. SPONSOR/MONITOR'S REPORT NUMBER(S). Enter report number as assigned by the sponsoring/ monitoring agency, if available, e.g. BRL-TR-829; -215.

12. DISTRIBUTION/AVAILABILITY STATEMENT. Use agency-mandated availability statements to indicate the public availability or distribution limitations of the report. If additional limitations/restrictions or special markings are indicated, follow agency authorization procedures, e.g. RD/FRD, PROPIN, ITAR, etc. Include copyright information.

13. SUPPLEMENTARY NOTES. Enter information not included elsewhere such as: prepared in cooperation with; translation of; report supersedes; old edition number, etc.

14. ABSTRACT. A brief (approximately 200 words) factual summary of the most significant information.

15. SUBJECT TERMS. Key words or phrases identifying major concepts in the report.

16. SECURITY CLASSIFICATION. Enter security classification in accordance with security classification regulations, e.g. U, C, S, etc. If this form contains classified information, stamp classification level on the top and bottom of this page.

17. LIMITATION OF ABSTRACT. This block must be completed to assign a distribution limitation to the abstract. Enter UU (Unclassified Unlimited) or SAR (Same as Report). An entry in this block is necessary if the abstract is to be limited.

Table of Contents

Table of Contents i
List of Figures ii
List of Tables ii
Abstract 12
Introduction 12
Background 12
Methods 12
Discussion/Results 12
Conclusions 12
References 12

List of Figures

Figure 1: Quarter Symmetry Representation 2
Figure 2: Fluid Volume and Boundary Conditions..... 3
Figure 3: Mesh Size Functions..... 4
Figure 4: Simulation 3-25% Front and Rear Radius Force-vs-Time Curve 5
Figure 5: Force vs Time Curves For Group 2 Simulations..... 7
Figure 6: Velocity Contours on Symmetry Plane 8
Figure 7: Velocity Vectors Simulation 1 9
Figure 8: Velocity Vectors Simulations 3..... 10

List of Tables

Table I: Simulation Radius Size..... 5
Table II: Simulation Impulse Results..... 6
Table III: CFL Computer Resource Usage 7

Geometric Effect on Impulse in a Gas Redirected Shock Tube

Abstract – The behavior of an expanding gas from a shock tube when redirected by external geometry can vary with changes implemented to the external geometry. Large force oscillations initially occurred, as modifications were made to the geometry the force-vs-time curve dampened. Changing the geometry impacts how the expanding flow reacts to the change in direction. The analysis conducted five numerical studies on different geometries at two different stability values. Force-vs-time curves were created and the impulse was computed from the force curves. The different geometries were compared to one another for impulse and force dampening. Results show that modifying the internal geometry does impact the impulse. Results also show that the explicit scheme was stable and that using a larger courant number improved computing resources while not affecting error in impulse values. By conducting numerical studies of how the impulse is impacted by geometry, a more stable, effective, efficient model was developed.

INTRODUCTION

In a closed end shock tube, the expanding gas exerts an axial force in the opposite direction of the expanding gas. If the expanding gas moving opposite of the closed end encounters a baffle or wall, a force is imparted on the wall which “turns” the flow in an angular direction to the axis of flow. In the case of a jet engine, reverse thrust acts against the forward momentum of an aircraft by directing the engines thrust forward, slowing the aircraft down. Specifically, the engine is not operated in reverse, rather deflectors are used to change the direction of the fast flowing air. As the aircraft touches down the engines use these deflectors to focus the rear thrust forward [1]. This change in momentum will affect the impulse on the system. Different geometries can improve or degrade the impulse. The research conducted will look at a simple geometry modification that will be used to improve functionality as well as numerical scheme stability.

BACKGROUND

A high pressure, high velocity gas travelling axially through a tube expands in three dimensions when released into the ambient air. When the expanding gas is redirected by a geometric interface a force is imparted on the surface of the geometry. The behavior of this force can determine the numerical stability. Altering the internal front and rear radius will impact the performance of the force when the gas reflects off of the surfaces. As the gas is expanding the force will oscillate as the surfaces redirect the flow. The force being applied by the gas will dampen as the internal radii decrease.

The focus of this study is to determine how the forces are impacted by altering the geometry. A quarter symmetry assembly of a tube with a geometric interface attached to the end of the tube will represent the numerical model, see figure 1.

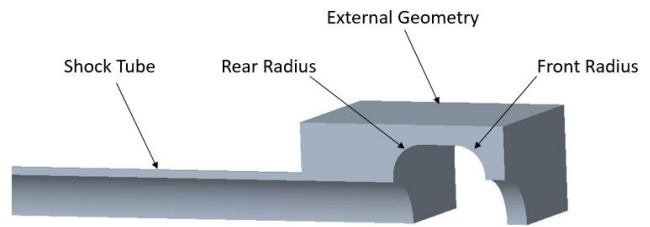


FIGURE 1
QUARTER SYMMETRY REPRESENTATION

$$I = \int_{t_1}^{t_2} F dt \quad (1)$$

Impulse is defined as an acting force during a specific change in time and can be determined by integrating the force over time curve, refer to (1). Optimizing this area under the curve will effectively optimize the total force on the system. For this study 5 separate cases with varying radius values will be analyzed:

- Initial radius of 1 for front and rear
- 50% of initial for front and rear
- 25% of initial for front and rear
- 1 for front and 50% of initial rear
- 50% of initial front and 1 for rear

Three dimensional models were created using Creo Parametric 3D Modeling Software. The CFD numerical simulations were set-up using the ANSYS workbench suite. ANSYS mesh was used to discretize the model, ANSYS Fluent was used to solve the conservation of mass, momentum and energy equations for velocity and pressure.

The paper will cover a methods section which will discuss the numerical set-up and simulation. A discussion/results section will explain the data compiled and discuss what the results represent.

METHODS

A baseline model was developed using a value of 1 for both the front and rear radius. The model was cut into quarter

symmetry in order to save on computation time. The fluid model was created by subtracting the geometry from an air-volume model. The air behind the turning geometry was neglected and the volume of external air was extended above and forward of the turning geometry, see figure 2. Six boundary conditions were determined; four symmetry surfaces, sixteen wall surfaces, one interior connection and three pressure outlet surfaces.

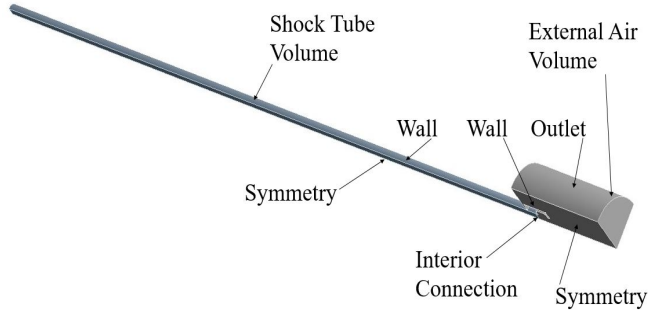


FIGURE 2
FLUID VOLUME AND BOUNDARY CONDITIONS

The meshing method used was tetrahedron with three size functions, see figure 3. Size Function1 was the surfaces related to the turning geometry wall surfaces, Size Function2 was the surface associated with the interior connection and Size Function3 was the symmetry surfaces relating to the tube.

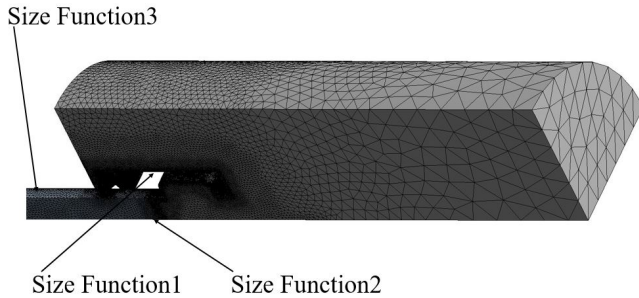


FIGURE 3
MESH SIZE FUNCTIONS

$$C_d = \frac{2F_d}{\rho u^2 A} \quad (2)$$

$$F_d = \frac{C_d \rho u^2 A}{2} \quad (3)$$

The problem was setup in Fluent as an explicit, first order upwind formulation with a density based, transient solver. The flow is considered compressible therefore the species model is treated as an ideal gas. Setting up a coefficient-of-drag monitor, the force was determined, refer to (2) and (3).

$$y = -2,288,235 * x + 99150000 \quad (4)$$

$$y = 117.46 * x \quad (5)$$

$$C \equiv \frac{c_x \Delta t}{\Delta x} + \frac{c_y \Delta t}{\Delta x} + \frac{c_z \Delta t}{\Delta x} \leq 1 \quad (6)$$

Solutions were patched in for the pressure and velocity profile of the shock tube. For the pressure profile, refer to (4). Equation (5) represents the velocity profile. For the solution controls, explicit stability and time step are a function of the 3-dimensional Courant Number (CFL), refer to (6). The CFL condition places a severe limitation on Δt_{max} [2]. As the courant number approaches 1 the explicit scheme can become unstable which can affect the results of the simulation. Two simulation groups were analyzed; group 1 set the courant number for the five simulations at 0.3, group 2 set the courant number for the five simulations at 1.

Table I lists the different radii values for their respective simulation run. The values are normalized, the largest radius has a value of 1 and the smallest radius is 25% of 1. Simulation 1 would be considered the baseline analysis for this study.

TABLE I
SIMULATION RADIUS SIZE

| Simulation | Front Radius | Rear Radius |
|------------|--------------|-------------|
| 1 | 1 | 1 |
| 2 | 0.5 | 0.5 |
| 3 | 0.25 | 0.25 |
| 4 | 1 | 0.5 |
| 5 | 0.5 | 1 |

DISCUSSION/ RESULTS

The force data compiled from the simulation has been normalized on a scale from 0 to 1. All the forces are a percentage of the peak force that was determined from the five simulations. The time scale is also on a scale from 0 to 1. The time value of 1 is the approximate time that the solutions converged to a zero force.

Impulse numbers for the group 1 and group 2 simulations are listed in Table II. Figure 4 illustrates the error that exists in the force-vs-time curve for simulation 3 when adjusting the Courant number. Comparing the Courant number (CFL) differences in Table II, the explicit scheme was stable at both the 0.3 and 1 value a ~1% variance. Very little difference in the force curve and subsequently the impulse existed by changing the courant number.

TABLE II
SIMULATION IMPULSE RESULTS

| Simulation | Group 1 Impulse (CFL=0.3) | Group 2 Impulse (CFL=1) | % Diff Impulse |
|------------|---------------------------|-------------------------|----------------|
| 1 | 0.2255 | 0.2253 | 0.07% |
| 2 | 0.2641 | 0.2649 | 0.30% |
| 3 | 0.2708 | 0.2700 | 0.30% |
| 4 | 0.2417 | 0.2449 | 1.33% |
| 5 | 0.2523 | 0.2525 | 0.08% |

TABLE III
CFL COMPUTER RESOURCE USAGE

| Simulation | Group 1 Solve time (hrs) | Group 2 Solve Time (hrs) | Solve Time Improvement |
|------------|--------------------------|--------------------------|------------------------|
| 1 | 36.5 | 7.5000 | 5 |
| 2 | 35.5 | 7.0000 | 5 |
| 3 | 29.0 | 8.5000 | 3 |
| 4 | 19.0 | 9.5000 | 2 |
| 5 | 26.0 | 8.0000 | 3 |

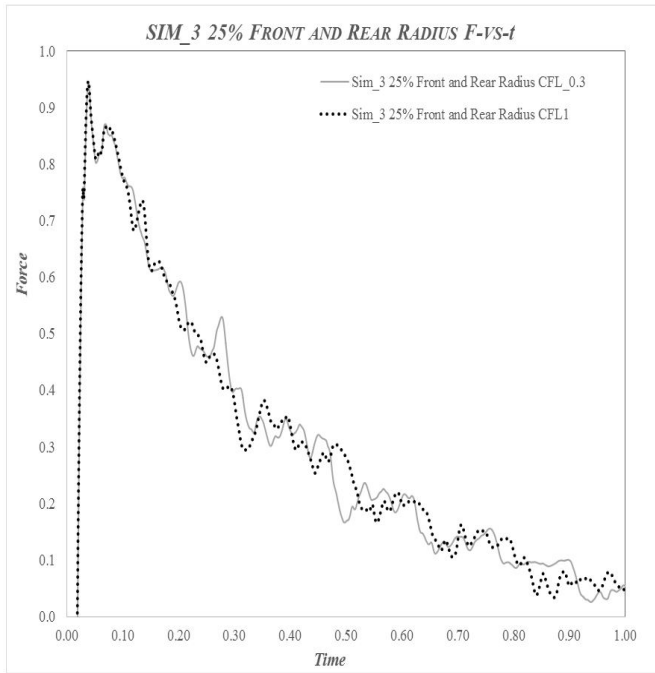


FIGURE 4

SIMULATION 3-25% FRONT AND REAR RADIUS FORCE -VS- TIME CURVE

The courant number had the largest impact on computer resources, see Table III. The simulation run times were three to five times faster with the CFL=1 value. Simulation 4 resource time was an outlier. It is difficult to determine what is causing the smaller difference in resource time at this specific simulation. Assumptions could be made that the time step needs to have a tighter control on this geometry. This could also explain why the CFL=0.3 had the more efficient run time compared to the other four CFL=0.3 simulations. Considering that the geometry is fairly simplistic has a real effect on the small differences in impulse and large difference in simulation run time. Complex geometry having complex mesh schemes will have a larger effect on the impulse numbers. These complexities will require a tighter control of the time step in an explicit analysis.

The remainder of the analysis will focus on the CFL=1 impulse numbers from Table II. Referring to figure 5, the geometry in simulation 3 has the largest impact on the impulse of the five simulations with a value of 0.27. There was noticeable force-vs-time curve dampening for simulation 2 through simulation 4 when compared to the force-vs-time curve of the baseline simulation 1, which had a value of 0.22. Simulation 4 appears to have dampened the curve the best but had lower peak forces when compared to simulation 3, affecting the overall impulse of the system. Simulation 5 had a similar force-vs-time curve to the simulation 1 curve.

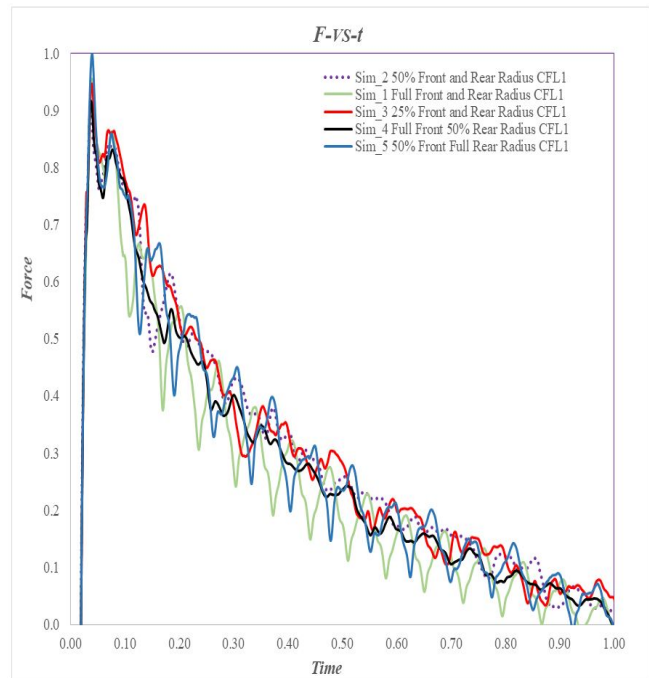


FIGURE 5

FORCE -VS- TIME CURVES FOR GROUP 2 SIMULATIONS

Inspecting the velocity vectors at time value 0.25 for the two most extreme simulations; simulation 1 and simulation 3, the effect of the radius can be seen. Figure 6 provides a view for orientation. Figures 7 and figure 8 show the velocity vector behavior. Stagnation areas in both simulations are noticeable forward of the shockwave as well as behind the shock wave. These stagnation areas affect the structure of the

shock wave. The fast moving gas compresses the air in front, pressure increases and fluid flow stops. These stagnation areas have a transient effect on the shock structure. When the radii is smaller as in simulation 3, the larger surface area of the front face creates a larger stagnation area. The stagnation area behind the shock barrel is created by the fast flowing gas exiting the shock tube and expanding into the larger volume. This stagnation region has much lower pressures than the stagnation point [3].

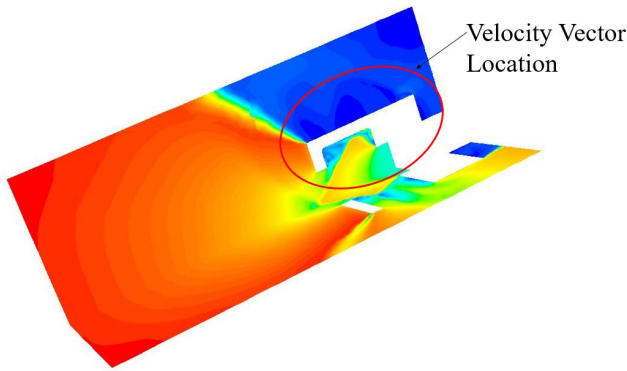


FIGURE 6
VELOCITY CONTOURS ON SYMMETRY PLANE

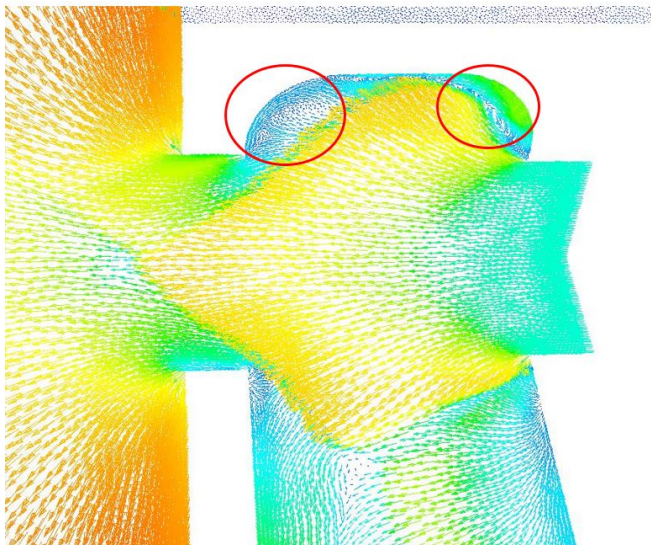


FIGURE 7
VELOCITY VECTORS SIMULATION 1

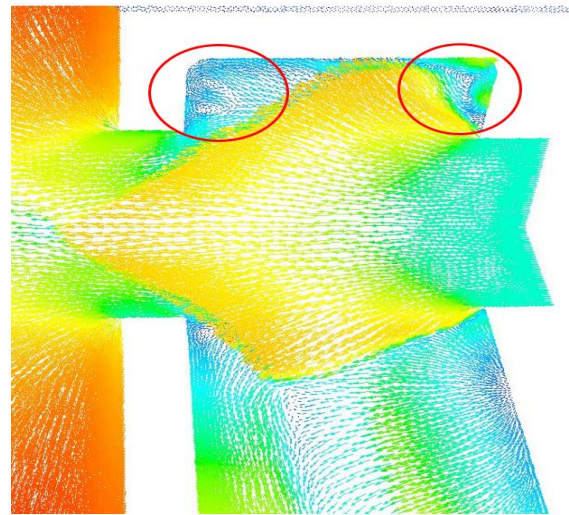


FIGURE 8
VELOCITY VECTORS SIMULATION 3

Decreasing the radius creates a larger surface for the gas to “impact” thus increasing the ability of the gas to “push” on the geometry. From just a fluids perspective the smaller the radius at both locations returns the best impulse with a significant curve damping. In order to make a well informed decision on what value to make the radii a structural study should be conducted to locate potential stress risers in the area of the radii.

CONCLUSIONS

The analysis of five different geometry configurations provided favorable results when focusing on the force-vs-time curve. Numerical stability of the explicit simulations was first determined in order to optimize resource time. Two groups of simulations were set-up and the computation time was analyzed. Computation solve time was 3-5 times faster when the stability control was set to allow the maximum time step. There was no major differences in impulse results between the two groups.

The analysis of five different geometry configurations provided favorable results when focusing on the force-vs-time curve. Dampening of the curve was accomplished by modifying the internal radii. The peak force also changed with the modification of the geometry. The impulse improved by 16% and 18% when changing both radii 50% and 25% respectively. Peak forces were also affected by the geometric change.

The emphasis of this study was solely on how the forces are affected by the flow of a fluid through geometry. Future work would consist of a finite element study to determine how the geometry changes affect the structural integrity of the system. Decreasing the radius until impulse numbers are the highest would be the fluid answer, stress risers and material yielding would ultimately decide on geometric shapes.

REFERENCES

- 1 P. U. Engineering, "Thrust Reversing," 1998. [Online]. Available:
<https://engineering.purdue.edu/~propulsi/propulsion/jets/basics/reverse.html>. [Accessed 28 09 2018].
- 2 L. C. Rajesh Bhaskaran, "Introduction to CFD Basics," [Online]. Available:
<http://dragonfly.tam.cornell.edu/teaching/mae5230-cfd-intro-notes.pdf>. [Accessed 18 09 2018].
- 3 J. M. C. Yunis A. Cengel, Fluid Mechanics: Fundamentals and Applications 3rd Edition, New York: McGraw-Hill, 2014.