

# Computational Analysis of Ramjet

<sup>1</sup>Prof. Omkar Pandey, Assistant Professor,  
<sup>2</sup>Anuj Karthik S, <sup>3</sup>Gagan.P, <sup>4</sup>Gaganesh.L, <sup>5</sup>Charan Simha.SU, <sup>6</sup>Hruthvik Sagar.S  
<sup>2,3,4,5,6</sup>Dr.ACS College of Engineering, Bengaluru, India

## ABSTRACT

A miniature ramjet engine designed to perform at Mach 4.0 was tested in a supersonic wind tunnel. Cryogenic strain gauges were used to measure drag and Schlieren imaging techniques were used to observe the inlet Mach cone profile at Mach numbers of 4.0. Three different nozzle configurations were tested to confirm computational models used to predict back pressure and normal shock locations at the inlet.

Using ANSYS-CFX, a cold flow, computational fluid dynamics model of the ramjet in the wind tunnel was evaluated to compare with the experimental results. This model was then used as a base for an eddy dissipation combustion model. Hydrogen was modeled as being injected into the combustion chamber of the ramjet through inlet struts and then reacting with atmospheric oxygen to produce combustion. Drag predictions were inconclusive, however, the computational model remained stable during combustion

Calculations

## I.INTRODUCTION

The idea of rotorless, air-breathing engine dates back to a French inventor named René Lorin. His idea was well before his time, however, as even he recognized that the velocities required to make the compression cycle of his engine work were in excess of the speed of sound in air. This was just not possible of aircraft before the First World War The use of ramjets in military aircraft and weapons was intermittent after the second World War, with the most well-known example being Lockheed Martin's SR-71 Blackbird using the Pratt & Whitney J58 turbojet-assisted ramjets at unclassified speeds reaching Mach 3.2 at altitude. Ramjet application in both the military and civilian sectors waned after the retirement of the SR-71 platform in favor of more efficient (turbofan) and faster (SCRamjet) engines. However, a recent resurgence has been seen in the interest of the use of ramjets in military technology. November of 2006 saw the implementation of the BrahMos, the world's fastest cruise missile, into the arsenals of the Indian and Russian Federation's militaries. Interest in ramjets is currently spiking due to the design's potential to be scaled down because of the design's absence

of moving integral parts. As the ranges demanded of certain weapons increase and the size of unmanned aerial systems decrease, ramjets are being looked at as the propulsion design of choice.

This thesis continues the work of Kevin M. Ferguson [1], Wee T. Khoo [2], and Bingqiang Chen [3] to analyze the performance envelope of a ramjet small enough to be ballistically launched from within a 35 mm diameter barrel.

In [1], a ramjet was designed for operation in a free stream velocity of Mach 4.0 and was then tested at operating speed in a supersonic wind tunnel (SSWT). Analyses were also done using the GRIDGEN and OVERFLOW computer programs for computational fluid dynamics (CFD). A follow-up was conducted in [2] with an attempt at using the CFDFASTRAN code to model the combustion process in the ramjet. Unfortunately, the computing limitations and the two-dimensional model used prevented an optimal analysis of the operating conditions of the ramjet.

In [3], combustion CFD analysis was done using ANSYS-CFX with better but still sub-optimal results. An attempt to measure drag was also done in the SSWT using cryogenic strain gauges. For reasons that will be explained in Chapter III, the results of those drag measurements were not satisfactory. The ANSYS software did allow detailed, accurate coldflow analyses of Mach 4.0 flow over the ramjet. These analyses revealed that the inlet conditions were not optimal. Thus, CFD analysis was done altering the nozzle by reducing the throat area of the converging-diverging section.

The present study seeks to refine the tests and data gathered from these preceding studies by continuing the CFD combustion analyses done in [3] and completely redesigning the drag measurement flexure arms, which hold the ramjet in the SSWT. The changes made were done in order to improve the quality of the data collected in the SSWT experimental drag measurement and the cold flow CFD drag prediction. To validate the CFD predictions done in [3] of the different nozzles, two of the nozzle designs with reduced throat areas were also tested in the SSWT with drag measurements recorded.

The data acquisition for the strain measurements was also upgraded to a National Instruments unit that was more user friendly. Additionally, the calibration procedure

before and after testing of the ramjet in the SSWT was updated, which resulted in reduced uncertainty of the drag measurements.

## II. COLD F CFD ANALYSIS

### A Background and Methodology

In [3], a three-dimensional simulation of the ramjet in the SSWT was performed in order to predict the drag experienced. Symmetry was employed at two planes so that only one quarter of the ramjet and one of the winglets would have to be modeled, reducing computational costs. The drag calculated for the quarter ramjet and the winglet were then multiplied by 4 and 2, respectively. This introduced an inaccuracy due to the two planes of symmetry essentially accounting for four winglets in the computation, while in the experiment there were only two winglets used. In these simulations, only one plane of symmetry was employed to increase accuracy, making it necessary to only model half of the ramjet and half of the winglet. Also in [3], analyses were done for different configurations of the nozzle. Analyses were done with a ramjet using the base nozzle design, and then with throat area reduced by 10%, 20%, 30%, and 40% from the base area. To analyze the accuracy of those computations, this thesis had two new nozzles (20% and 40% reductions in the nozzle throat area) manufactured for testing in the SSWT. Computational models of these new nozzles were also performed simulating the conditions in the SSWT.

These CFD analyses utilized SolidWorks modeling software to create the models of the ramjet with the winglet attachment as well as the SSWT section. These models were then imported into ANSYS-CFX Workbench 14.0, where a mesh was created, flow conditions were set, and the computation was run using typically 12 processors.

### B. COMPUTATIONAL MODEL SETUP

#### 1. Geometry

To exploit the axis of symmetry present on the model and winglet in the SSWT, half of the ramjet, winglet, and SSWT section were modeled utilizing the symmetry plane shown in Figure 12. The winglet was also simplified to reduce computational costs. A comparison of the physical winglet and the simplified winglet model used in the CFD analysis is shown in Figure 13.

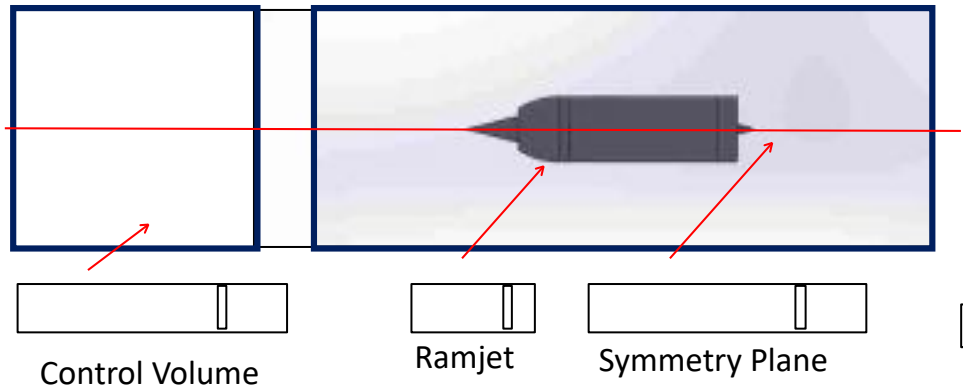


Figure 12 Ramjet model in computational domain.

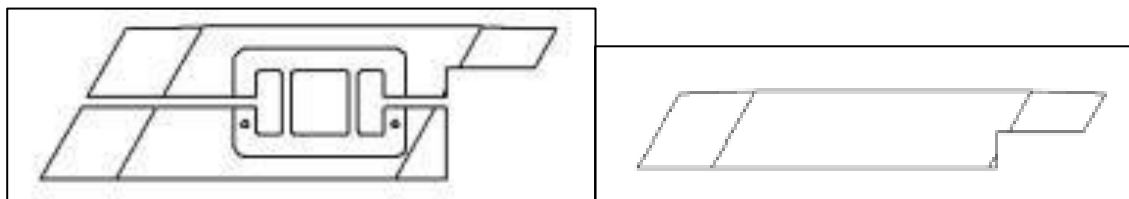


Figure 13 Comparison of physical flexure model and equivalent CFD flexure model

The ramjet, winglet, and SSWT section were modeled in SolidWorks before they were transferred into the geometry program within ANSYS. ANSYS CFX was then used to subtract the ramjet model from the SSWT test section model to create the computational domain. The computational domain is shown below in Figure 14.

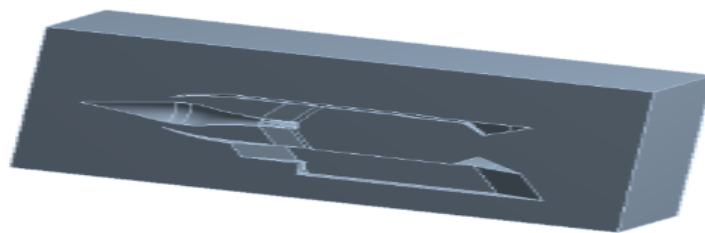


Figure 14 Computational domain of the CFD analysis.

## 2. Mesh

The meshing module within ANSYS was then utilized to create the computational mesh. Inflation layers were created near each wall of the ramjet and winglet. The parameters of the mesh can be found in Appendix D. For the model with the base nozzle design (nozzle A), the utility created 3.83 million nodes and 20.03 million elements. Figure 15 shows an

example section of the mesh, with the smaller inflation layers visible at the nozzle/domain interface.

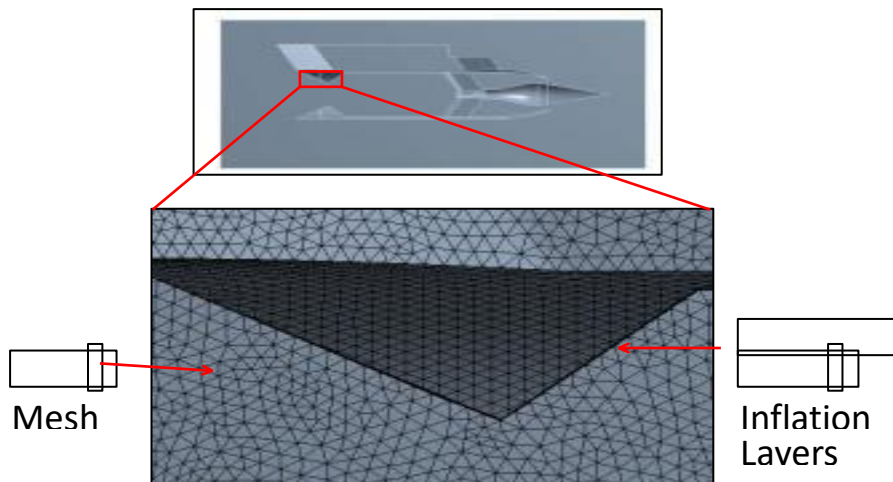


Figure 15 Close up of ANSYS generated mesh.

### 3. Boundary Conditions

All of the computational parameters were set up in ANSYS CFX-PRE. All of the parameters can be found in Appendix D while a snapshot can be seen in Table 2.

Table 2 Boundary conditions of the SSWT CFD analysis.

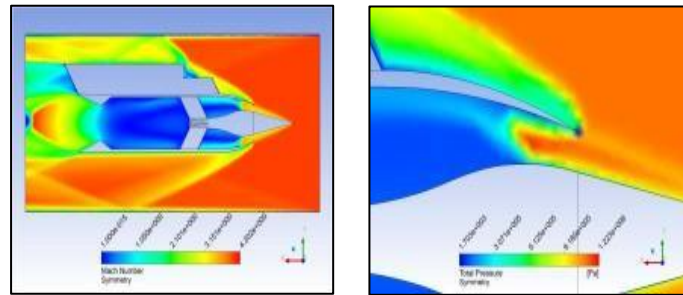
Boundary	Type	Boundary Condition
Inlet	Inlet	Supersonic, $V = 661 \text{ m/s}$ , $P = 7378 \text{ Pa}$ , $T = 68 \text{ K}$
Outlet	Outlet	Supersonic
Ramjet	Wall	No-Slip Wall
Winglet	Wall	No-Slip Wall
Symmetry Planes	Symmetry	-
SSWT Sides	Wall	No-Slip Wall

## C. RESULTS

### 1. Mach Profiles and Shock Locations

Figure 16 shows the Mach number profile for the whole symmetry plane and the total pressure profile at the area of the shock at the inlet cowl. This figure can be compared with

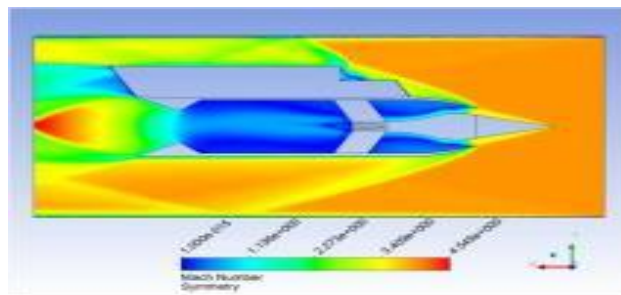
Figure 5 in [3] to see the similarities between the two computational models. however, the figures in [3] do not show the winglet, as this was not included in that analysis. Figures 17 and 18 show the same Mach number profile and total pressure profile for the models with a 20% and 40% reduction in nozzle throat area respectively. They can be compared with Figures 13 and 15 in [3]. Figure 19 exhibits the shock indicators at the inlet cowl for all three setups. Comparing this with Figure 16 in [3] shows very favorable comparison between the two sets of computational simulations.



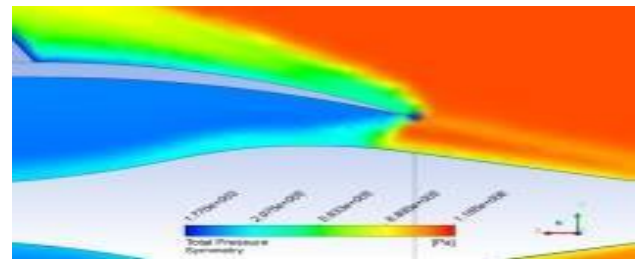
Mach Number

Total Pressure

Figure 16 Mach profile and total pressure at inlet for model with 100% nozzle throat area.

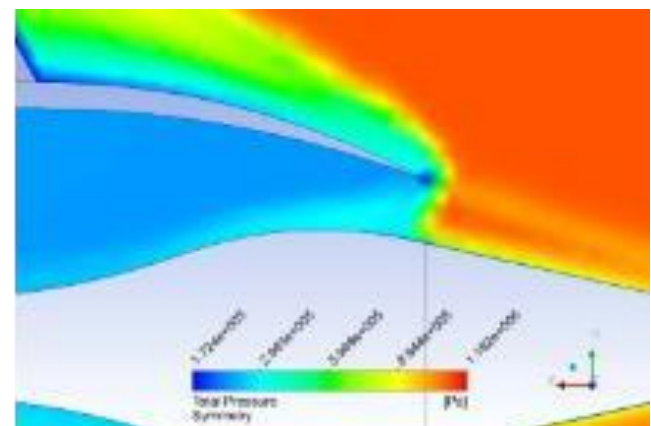
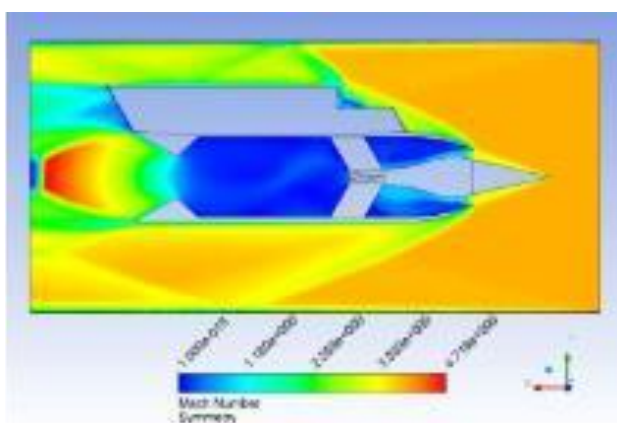


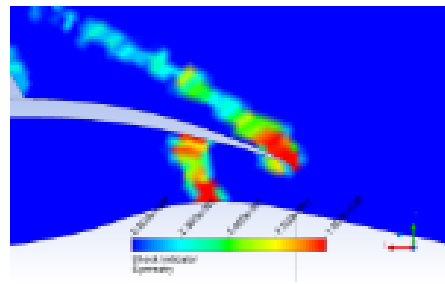
Mach Number



Total Pressure

Figure 17 Mach profile and total pressure at shock for model with 80% nozzle throat area.



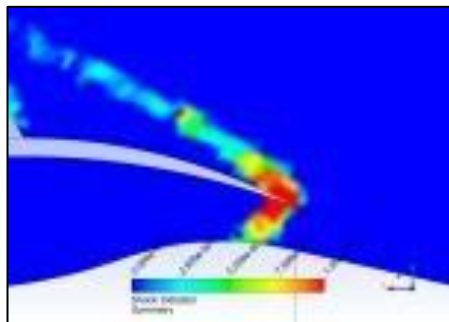


Mach Number

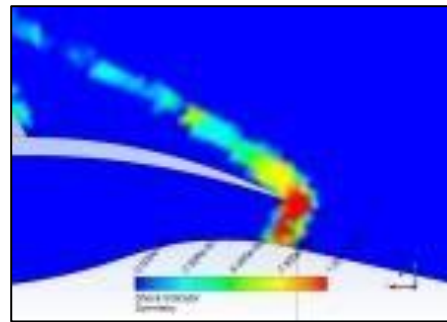
Total Pressure

Figure 18 Mach profile and total pressure at shock for model with 60% nozzle throat area.

100% Nozzle Throat Area



80% Nozzle Throat Area



60% Nozzle Throat Area

Figure 19 Shock indicator profiles at inlet cowling Drag Predictions

Drag calculations of each model were able to be completed using the ANSYS CFX-Post function calculator. The drags on each of the configurations are listed in Table 3. A property of interest when predicting drag on a surface is the dimensionless wall distance, or  $y^+$ , defined as  $y^+ = \tau_w y / \mu$ , where  $\tau_w$  is the wall shear stress,  $\rho$  is the fluid density,  $y$  is the distance from the wall, and  $\nu$  is the kinematic viscosity. A  $y^+$  value between 1.0 and 3.0 indicates that the calculated values close to the walls are accurate. The  $y^+$  values for these computational models, shown in Table 4, are small enough to assume accurate computational measurements at the walls.

Table 3 Computational drag measurements.

Model	Nozzle Throat Area Compared to Base Design	Drag on Half Ramjet	Drag on Half Winglet	Total Drag
A	100%	13.36 N	5.33 N	37.38 N
B	80%	12.63 N	5.42 N	36.10 N
C	60%	14.73 N	5.31 N	40.08 N

Table 4 Average  $y^+$  values for surfaces in computational models.

Model	Nozzle Throat Area Compared to Base Design	$y^+$ Ramjet	$y^+$ Winglet
A	100%	1.63	1.198
B	80%	1.869	1.31
C	60%	1.965	1.316

#### D. COMPARISON OF EXPERIMENTAL MEASUREMENTS AND COMPUTATIONAL PREDICTIONS

The results of the drag measurements for both the SSWT experiments and the ANSYS CFX simulations are tabulated in Table 5. These results show that the drag on the model with nozzles A and C were consistently under predicted by the computational models by around 14 percent. The one run that was able to be completed on the ramjet with nozzle B showed significantly more drag than the others. The comparison of the experimental drag measurements on the ramjet with Nozzle B with the drags experimentally measured on the other ramjet configurations as well as the computational models led to the conclusion that the drag measured on the ramjet with nozzle B included some source of error that led to such high drag on the ramjet. The disparity between the experimental and the computational should be more consistent between nozzle configurations.

Table 5 Comparison of experimental measurements and computational drag predictions.

Model	Nozzle Throat Area Compared to Base Design	Experimental Drag Measurement	Computational Drag Measurement	% Error
A	100%	42.9 - 45.2	37.38	13.0 - 17.4
B	80%	52.6	36.10	31.4
C	60%	46.5 - 47.3	40.08	13.8 - 15.2

#### REFERENCES

- [1] K. M. Ferguson, "Design and cold flow evaluation of a miniature Mach 4 ramjet," M.S. Thesis, Department of Aeronautical Engineering., Naval Postgraduate School, Monterey, CA, June 2003.
- [2] W. T. Khoo, "Cold flow drag measurement and numerical performance prediction of a miniature ramjet at Mach 4," M.S. Thesis, Department of Aeronautical Engineering, Naval Postgraduate School, Monterey, CA, December 2003.
- [3] B. Chen, "Numerical performance prediction of a miniature ramjet at Mach 4," M.S. Thesis, Department of Mechanical and Aerospace Engineering, Naval Postgraduate School, Monterey, CA, September 2012.
- [4] M.J. Foust et al., "Experimental and analytical characterization of a shear coaxial combustor GO<sub>2</sub>/GH<sub>2</sub> flowfield," presented at the American Institute of Aeronautics and Astronautics 34th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 15-18, 1996, Paper 96-0646.