International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.3., Issue.2, 2015

RESEARCH ARTICLE

ISSN: 2321-7758

ANALYSIS OF SCRAMJET NOZZLE IN CONNECTED PIPE TEST FACILITY USING 'FLUENT'

M.VIJAYAN[1], S.OSWIN ERIC[2], M.MANI BHARATHI[3]

[1,2,3]PG-Scholar, Department of Mechanical Engineering, Government College of Engineering Salem, Tamilnadu, India

Article Received: 07/04/2015 | Article Revised on:14/04/2015 | Article Accepted on:16/04/2015

M.VIJAYAN

S.OSWIN ERIC

M.MANI BHARATHI

ABSTRACT

Scramjet, supersonic combustion ramjet engine makes a drastic change in the world of aerospace by overtaking the speed of sound approximately 25 times. According to the reference from NASA standards that scramjet has reached the mach number of slightly above 7. Scramjet attains the supersonic speed in the combustion chamber by funneling the shocks to its proper shape. Scramjet test facility is the ground test facility set up of scramjet engine with the various altitudes and mach numbers ranging from 4 to 8. This engine needs to undergo development tests, qualification tests and acceptance tests in the ground test facilities. Scram jet connected pipe test facility is considered here for simulation purpose. To simulate the proper mach numbers and static pressures at the combustion chamber entry, a convergent divergent nozzle is provided at the combustor inlet. This work describes the configuration design development scramjet facility nozzle by theoretical calculations using area Mach number relation, validation of result using CFD software namely fluent and compared both results.

Key words: Scramjet Nozzle; CFD; Connected Pipe Test Facility

I.INTRODUCTION

Scramjets, Supersonic Combustion Ramjets are the simplest air breathing engines whereas piston engines and turbine engines use complex mechanical components to compress the air then extract the energy for useful work. The scramjet compresses the air and expands the exhaust simply by means of its shape and its movement through the air. It requires no moving parts. Atmospheric air enters in to the scramjet finds a widened area called the diffuser and spread out. When it does it slows down and increases in pressure. After the air enters the engine, the fuel is added through an injector to provide a combustible mixture. This fuel/air mixture is ignited to raise its temperature so that it exists out the nozzle at a much higher velocity than it came in with.

As the fuel/air mixture burns to become exhaust products it heats up and accelerates towards the exhaust nozzle. The nozzle throat provides a constriction to help maintain pressure inside the combustion chamber. The hot exhaust products pass through the throat and expand in the nozzle exit to approximately atmospheric pressure. The scramjet net thrust is the difference between two opposing forces, drag and thrust.

The scramjet differs from the ramjet in that combustion takes place at supersonic air velocities though the engine. To accomplish this, scramjet engineers rely heavily on the shape of the entire aircraft. The front section of the aircraft is shaped such that the shock waves produced at the aircraft's leading edges are funneled into the engine. This concept, along with the geometry of the engine inlets, compresses the air for combustion with the fuels. Scramjets can use hydrocarbon fuels or liquid hydrogen.

There are two main advantages of scramjet propulsion over rocket propulsion for lifting vehicles into orbit. The first major advantage is weight savings. By avoiding the requirement to
Carry one's own oxidizer; weight of orbital vehicles can be drastically reduced. This in turn requires drastically less fuel, the end result being a dramatic reduction in overall cost.

The second major advantage to the scramjet is the maneuverability added to any mission. Once ignited, solid fuel rockets will burn until their fuel is exhausted. A malfunction on a launch vehicle with solid propellant in almost all cases results in the loss of the vehicle and its payload. A launch vehicle propelled by a scramjet engine could, at any time during ascent, abort its mission and maneuver for return to a suitable landing site.

SCRAMJET TEST FACILITY

Scramjet Test Facility is the ground test facility setup of Scramjet engine with the various altitudes from 20Km to 40Km and Mach numbers from 4 to 8. This engine need to undergo development tests, qualifications tests and acceptance tests in the ground test facilities. There are various types of test facilities.

The purpose of the Direct-Connect Supersonic Combustion Test Facility is to test scramjet combustors in flows with stagnation enthalpies duplicating that of flight at Mach numbers from 4 to 8 in direct-connect, or connected-pipe, fashion so that the entire facility test gas mass flow passes through the combustor. The flow at the exit of the facility nozzle simulates the flow exiting an inlet and entering the combustor of a scramjet in flight.

Figure 1.1 shows the actual design of Scramjet engine, the in let is acting as a diffuser before the combustion chamber. But in Figure 1.2, in the test facility a convergent divergent nozzle is placed as a direct connect pipe to combustion chamber to get the mass flow rate and simulating conditions of original engine.

Vitiated gas is supplied from air storage and is passed through heater and facility nozzle and enters into the combustion chamber where the fuel is injected. Unlike the real flying condition where the air **has only** oxygen and nitrogen, this vitiate air has equivalent oxygen and the remaining part is mostly steam vapor with they little quantity of nitrogen.

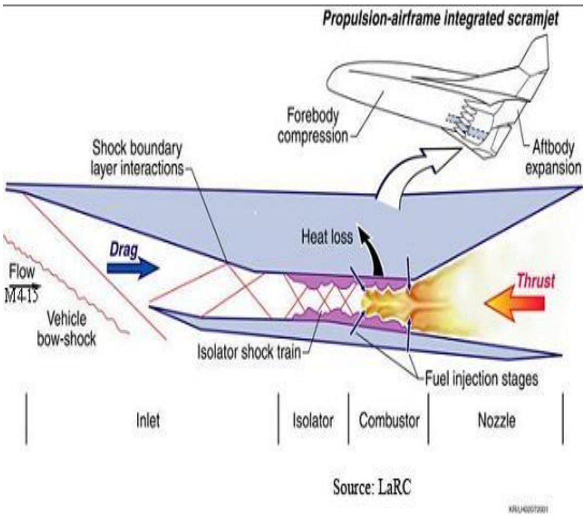


Figure 1.1 Scramjet Engine

CALCULATIONS OF NOZZLE CONFIGURATION FOR SCRAMJET TEST FACILITY

SCRAMJET-CONNECT PIPE TESTING

Test Article Specification

1) Scramjet Engine is flying at varying attitude from 20km to 40km.

2) Typical Combustor Conditions

TABLE I. CALCULATIONS OF OUTLET CONDITIONS

3) Engine air flow rate - 10 Kg/Sec

4) Engine fuel flow rate - 1 Kg/Sec

Hence the Engine Outlet mast flow rate should be (10+1) = 11 Kg/Sec

Model Calculation for Mach No.2 & Altitude 20 Km.

From gas tables for Z = 20Km

P2 = 0.05529bar T2 = 217K

so Stagnation Pressure for Mach No: 2

$$\frac{P_2}{P_o} = 0.128 \Rightarrow P_o = 0.431\text{bar}$$

$$\frac{T_2}{T_o} = 0.555 \Rightarrow T_o = 390\text{K}$$

S. No.	Mach No.	Attitude (Km)	STG PR (bar)	STG Temp (K)	Static Pr. (bar)	Static Temp (k)
1.	2	20	0.431	390	0.05529	217
2.	2.5	25	0.435	500	0.02549	222
3.	3	30	0.478	533	0.01197	227
4.	3.5	35	0.521	673	0.00574	237

FACILITY CONFIGURATION DESIGN –INLET SIDE

The basic arrangement of the engine in the facility is shown in the following sketch.

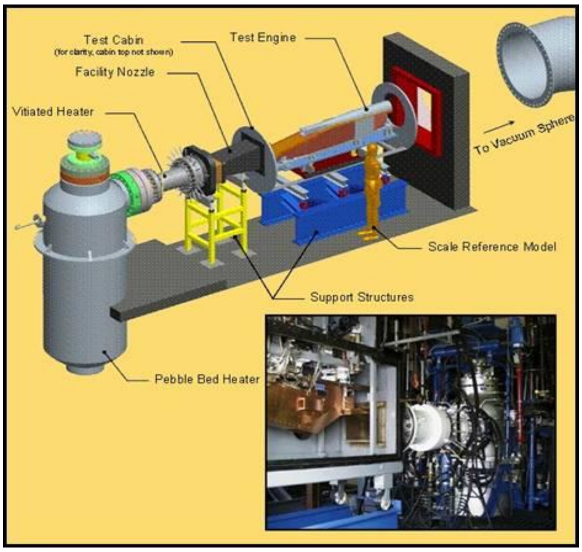


Figure 1.2 Scramjet Connect Test Facility

The inspection of the inlet data gives the view that we should have a minimum of 4.28 bar and 2800 K at the inlet to the nozzle. Since, the flow system involves huge pressure losses, generally a pressure of the order of around 2.5 times need to be maintained at the outlet of the supplying device (i.e) we should get a pressure of around 10 bar.

FACILITY REQUIREMENTS

To supply 10 kg/sec inside the engine, it is proposed to supply around 30 kg/sec of (vitiated) air so that the engine will face the free steam condition without any disturbance.

Considering 150 sec of testing, the total air required is equivalent to 30 x 150 = 4500 kg. The data of the inlet nozzle, which will be supplying the (vitiated) air at M = 4, 5, 6, 6, 5, 7 and 8 are given in the following table.

TABLE II. CALCULATIONS OF NOZZLE DIMENSIONS

M	2	2.5	3	3.5
γ	1.4	1.36	1.350	1.34
A/A*	1.687	2.637	4.235	6.789
A*(m2) Based on $\left[\frac{m\sqrt{T_o}}{A^*P_o} = 0.0404\right]$	0.124	0.139	0.1315	0.1355
A(m2)	0.209	0.366	0.536	0.919
d*	0.39	0.42	0.409	0.415
d2	0.52	0.682	0.841	1.081

Model calculation

for Mach No:2

from has table

$$A_2 / A^* = 1.687$$

And also

$$\frac{m\sqrt{T_o}}{A^*P_o} = 0.0404 \Rightarrow A^* = \frac{11\sqrt{390}}{0.0400\text{x}0.431\text{x}105} = 0.124\text{m}^2$$

So A2 = 1.687 x 0.124 = 0.209m2

So
$$\frac{\pi}{4}d^{*2} = 0.124 \Rightarrow d^* = 0.39\text{m}$$

$$\frac{\pi}{4}d_2{}^2 = 0.209 \Rightarrow d_2 = 0.52\text{m}$$

for Inlet condition

M1 = 0.15 P01 = 1 bar, To1 = 300k

$$A^* = \frac{11\sqrt{300}}{0.0404\text{x}1\text{x}10^5} = 0.047\text{m}^2$$

For M1 = 0.15

$$\frac{A_1}{A^*} = 3.9 \Rightarrow A_1 = 0.183\text{m}^2$$

d* = 0.244m

d1 = 0.482m

from gas tables

$$\frac{T_1}{T_{01}} = 0.995 \Rightarrow T_1 = 298.5\text{K}$$

$$\frac{P_1}{P_{01}} = 0.984 \Rightarrow P_1 = 0.984\,\text{bar}$$

Simulation Parameters – Facility Nozzle

The inspection of the inlet data gives the view that we should have a minimum of 48.41 bar and 2831 k at the inlet to the nozzle. Since, the flow system involves huge pressure losses, generally a pressure of the order of around 2.5 times need to be maintained at the outlet of the supplying device (i.e.) we should get a pressure of 120 bar.

FLUENT ANALYSIS

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative case. Supported mesh types include 2D triangular / quadrilateral, 3D tetra-hedral / hexa hedral / wedge and mixed meshes. FLUENT also allows refining or coarsening the grid based on the flow solution. Fluent is based on Finite volume method. The finite volume method is a numerical method for solving partial differential equations that calculates the values of the conserved variables averaged across the volume. Advantage of finite volume method is, it does not require a structured mesh as in the case of Finite element method. It is preferable comparing with other methods as a result of the fact that boundary conditions can be applied non-invasively. This is true because the values of the conserved variable are located within the volume elements, and not at nodes or surfaces. Finite volume methods are specially powerful on coarse non-uniform grids and in calculations where the mesh moves to track interfaces or shocks.

A. Introduction

CFD is a method for solving complex fluid flow and heat transfer problems. This method is used to simulate the flow problem. For simulation following procedure is used.

I. CFD provides a flow area into a large number of cells or control, volumes, collectively referred to as the “mesh” or “grid”.

II. In each of the cells, the Navier Stokes Equation, i.e. the partial differential equations that describe fluid flow are rewritten algebraically; tolerate such variables as pressure, velocity and temperature in neighboring cells.

III. The equations are then solved numerically yielding a picture of the flow corresponding to the level of resolutions of the mesh.

The basic steps in simulation:

I. Pre-processing
II. Solving
III. Post processing

PRE-PROCESSING

Gambit is used for pre-processing which involves the following geometry modeling.

a) Vertices are defined according to the dimensions of nozzle.
b) Edges are defined (using predefined vertices)
c) Faces are defined (using predefined edges)

MESHING

Line meshing

By selecting all the edges, the appropriate interval count is given

Face meshing:

By selecting faces, the face meshing is done

DEFINING THE BOUNDARY CONDITIONS

Boundary conditions are defined by selecting the available boundary types. For continuum boundary condition is taken as combustion gases which are default. After defining the boundary conditions the file is converted into the “mesh files”. Then it is exported into “FLUENT” by using the export option.

Solver – segregated

Model – K-epsilon

SOLUTION (PROCESSOR)

Processing is done in FLUENT. The steps are

I. Read-case and data. Enter the file name (to open the file)
II. Grid – check (To ensure the absence of negative volume)
III. Scale is set as m (meter)
IV. Models are defined

Conditions are for viscous model

Solver – segregated, implicit, steady state and 2D

Model K-epsilon

By using material properties option the properties of fluid are defined. (Density, viscosity and molecular weight)

OPERATING CONDITIONS

I. Operating pressure – 101325 Pascal
II. By using boundary conditions option, the values at boundaries are given (i.e) velocity at inlet is
III. In the solve-controls option second order upwind scheme is selected for solution of discredited equation.
IV. In the solve-monitor option, residual values are given for continuity and momentum equation.
V. In the solve, Intialize option, the value of gauge pressure as Pascal and the initial velocity as Zero.

VI. Finally by using iterates option, the number is given and the solution is obtained by convergence.

POST PROCESSING:

The solution obtain is post processed to obtain the required restless.

Graphical display:

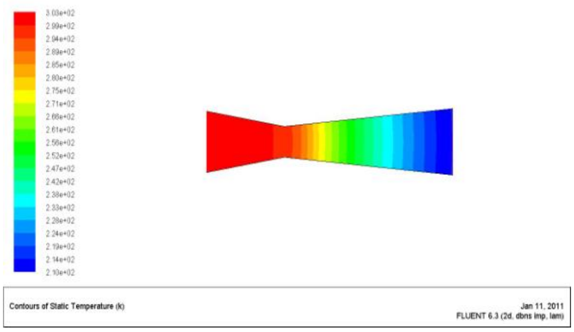
Main menu – XY plot-selection of required parameters

X-axis:

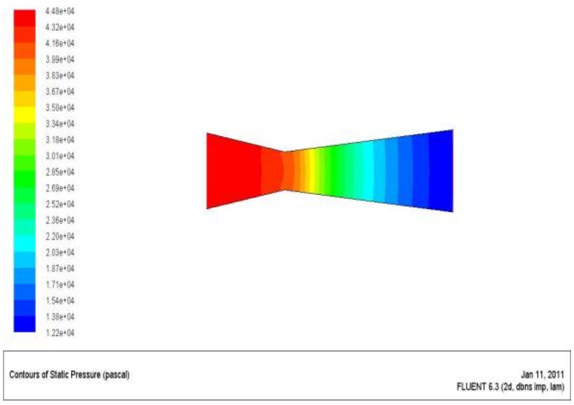
a) Static pressure
b) Velocity magnitude
c) Static temperature

RESULTS

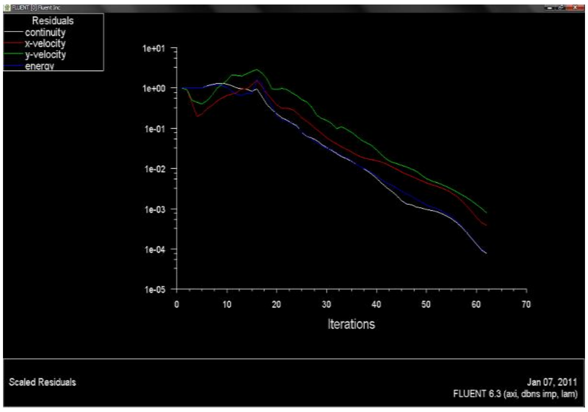
A. Fluent analysis results for pressure contour



B. Temperature contour



C. Scaled residuals iterations



CONCLUSION

The configuration design development of scramjet facility nozzle is done by theoretical calculations using area Mach number relation, these results are compared with the results of CFD software namely Fluent. Excellent validation of results is found. CFD code is too generated for numerical solutions of quasi-one-dimensional and two-dimensional CD nozzle flow (compressible) using finite difference method by McCormack's Technique using equations of motion. The flow variables over the entire length of the nozzle are also plotted. This work is to be done for three dimensional CD nozzle flow as a future work.

REFERENCES

[1]. Fundamentals of CFD - Harvard Lomax, Thomas H. Pulliam & David W. Zing
[2]. Modern Compressible Flow - John D. Anderson
[3]. Prandtl's Essentials of Fluid Mechanics - Herbert Oertel
[4]. Standard Handbook For Aeronautical And Astronautical Engineers - Scott Eberhardt
[5]. Airplane Aerodynamics and Performance - Roskam, Jan & Lan C.E.
[6]. Computational Fluid Dynamics - Anderson J.D.
[7]. Theoretical Aerodynamics Second edition - L.M. Milne –Thomson
[8]. W. H. Howe, L. D. Dinapoli, J. B. Arant -Venturi Tubes, Flow Tubes, and Flow Nozzles-, Béla Lipták(2003)
[9]. Stephen N. Rudnick et al-Particle Collection Efficiency in a Venturi Scrubber, Environ. Sci. Technol., Vol. 20, No. 3.
[10]. Edward J. Efsic III- Supersonic Combustion Ramjet Propulsion Engines. Embry-Riddle Aeronautical University(2002).