Computational Study –Thrust Vector Control Supersonic Secondary Flow Injection

B.Ragul[1], S.Anand kumar[2], M.Dinesh kumar[3]
[1,2,3] *Excel engineering college*

Abstract- **This paper presents the Computational study of flow through the Rocket Nozzle in order to manoeuvre the Rocket by means of Supersonic Secondary Injection of Hot gases. Well-designed nozzles can accelerate these gases to speeds of thousands of meters per second. In addition to accelerating the gases, nozzles are often responsible for steering (or "vectoring") the flow in order to control the rocket's direction of flight in supersonic speed.**

OBJECTIVE

The primary objective is to determine the Optimum angle of Injection of Hot gases and it's Injecting Pressure. Comparative study has been carried out on Secondary injection of hot gases into the core of Nozzle to appraise the effectiveness against each other.

INTRODUCTION

The supersonic nozzle is responsible for converting the thermal energy of the hot combustion gases into kinetic energy that can be used to propel the rocket. Understanding of supersonic nozzle starting is important for the design and optimization of new nozzles and analysis of their performance under different operational conditions. The nozzle starting process is complex and highly non-stationary. Behind the primary transmitting shock wave, the transient flow field is formed, which contains, in particular, a back-facing, so-called secondary shock – one of the main features of the process. The secondary shock interacts with the wall boundary layers, resulting in the shock bifurcation and formation of separation bubbles. Choking of flow is an important consideration in internal compressible flows. A converging duct decelerates a supersonic flow until the choking point; any further reduction in area would be accompanied by the formation of a shock within the duct. In case of frictional choking, because of the growing boundary layers, which simulate a variable area duct for the inviscid core, the length of the duct is the limiting variable and for cases of heat addition; amount of heat that can be added is limited by choking. Thus design and operating regimes of Nozzles are constrained by choking considerations.

NOZZLE

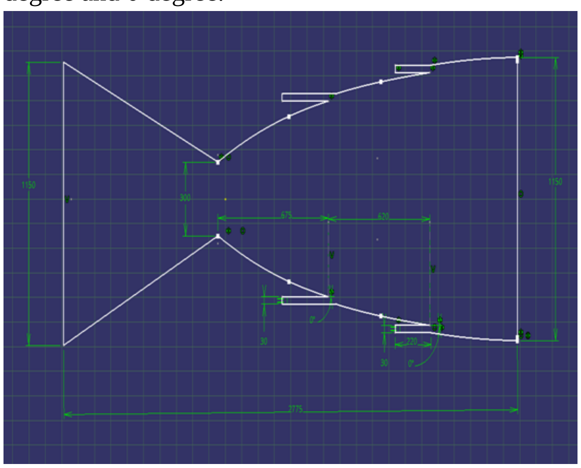
Nozzles are a uniformly varying area duct through which the flow is accelerated by the reduction of pressure. The function of the nozzle is of expand the hot gases down to ambient pressure, transferring the thermal energy to directed kinetic energy to produce thrust. A nozzle is a tube of varying cross-sectional area (usually axisymmetric) aiming at increasing the speed of an outflow, and controlling its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum, as we can feel by hand- holding a hose and opening the tap. In the simplest case of a rocket nozzle, relative motion is created by ejecting mass from a chamber backwards through the nozzle, with the reaction forces acting mainly on the opposite chamber wall, with a small contribution from nozzle walls.

THRUST VECTOR CONTROL

In addition to providing a propulsive force to a flying vehicle, a rocket propulsion system can provide moments to rotate the flying vehicle and thus provide control of the vehicle's attitude and flight path. By controlling the direction of the thrust vectors through the mechanisms described later in the chapter, it is possible to control a vehicle's pitch, yaw, and roll motions. All chemical propulsion systems can be provided with one of several types of thrust vector control (TVC) mechanisms. Some of these apply either to solid, hybrid, or to liquid propellant rocket propulsion systems, but most are specific to only one of these propulsion categories.

CAD DEVELOPMENT

The CAD Model is developed in CATIA V5 and the angle of Injection is varied from 40 degree to 0 degree while the Area ratio is kept identical as 8. The Schematic diagram of Convergent – Divergent Nozzle is furnished below. The geometry of Nozzle is remain unaltered for every Computational Flow Analysis and so the angle of injection is varying entity as like 40 degree, 30 degree, 20 degree, 10 degree and 0 degree.



CAD development

METHODOLOGY

This shows the systematic procedure of numerical study carried out for every nozzle configuration.

1. Cad Model Development
2. Fluid/Boundary Condition
3. Solving
4. Is The Numerical Results Are Grid Independent?
5. Utilization of Results For Performance Evaluation

GRID INDEPENDENCE STUDY

Grids or nodes are the points at which the transport equations of fluid flow are solved. These transport equations represent the spatial and time variation of different flow variables such as pressure, velocities in X, Y, Z directions, temperature, and turbulence properties etc. The partial differential quotients approximated as differential quotient for first order approximation with forward difference scheme as follows.

$\partial f(x) \partial x = \lim \nabla x \rightarrow 0 (f(x+\Delta x) - f(x) / \nabla x)$

According to Taylor expansion, which numerically approximates property $x+\Delta x$ is given by,

$f(x+\nabla x) = f(x) + \partial f(x) \partial x \Delta x / 1! + \partial 2 f(x) \partial x 2 (\Delta x) 2/2! + \partial 3 f(x) \partial x 3 (\Delta x) 3/3! + \ldots\ldots\ldots$

Boundary condition

- Pressure Inlet : 800 KPa
- Temperature Inlet – Primary : 500 K
- Temperature Inlet – Secondary : 1000 K
- Hot Gas Used : Hydrogen Fluoride
- Operating Pressure : 38 KPa
- Pressure Ratio : 1.05 (Sonic at Throat)
- Outlet : Pressure outlet with zero
- static pressure
- Walls : Non Slippery

SOLVER SETUP

ANSYS–FLUENT 16.0 is used as the solver. Assumptions made for this problem are follows,

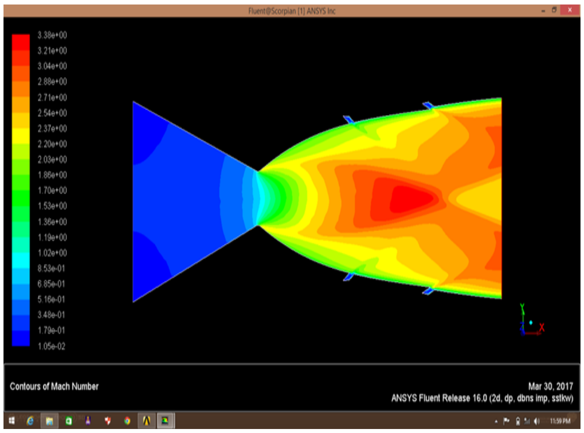
- K - ω Turbulence – SST – Viscous Heating is selected to meticulous features of Fluid flow over the body
- Density based Solver is selected to capture the Density – Pressure – Temperature Interaction
- Second Order Discretization is used to steer the solution towards Supersonic Flow
- FMG Initialization Coding is hooked up to avoid divergence of Solution

RESULT

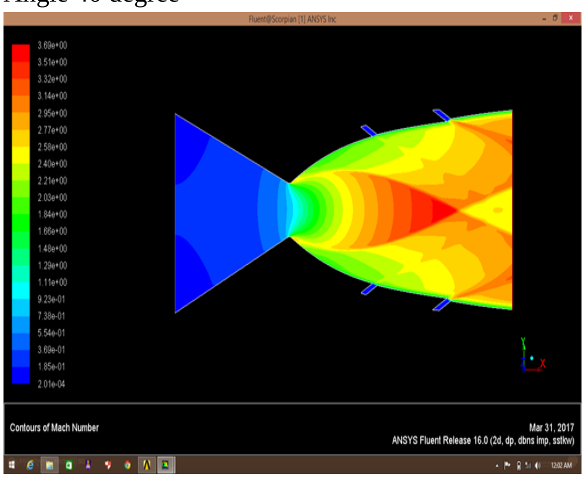
Grid Independence Study and Turbulent Models study has been carried out on Convergent – Divergent Nozzle with Inlet Pressure as 800 KPa to find the optimum Numerical Procedure, in order to capture physics appropriatelyProcedure, which captures the physics exactly is considered to optimum Numerical procedure forth coming cases. Numerical simulation for 40 degree Angled Convergent Divergent Nozzle has carried out by increasing the mesh count to some extent, until the Thrust Force remains same. From the graph, it is evident that no further variation of Thrust Force has been found after mesh count reaches about 122 thousands elements.

Hence, 122 thousands + elements is chosen to be mesh count required for capturing meticulous behaviour of fluid for all cases.

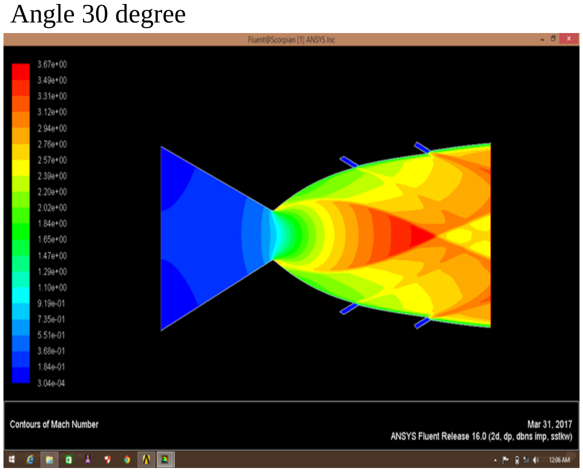
CONTOUR OF MACH NUMBER– SINGLE SIDED INJECTION



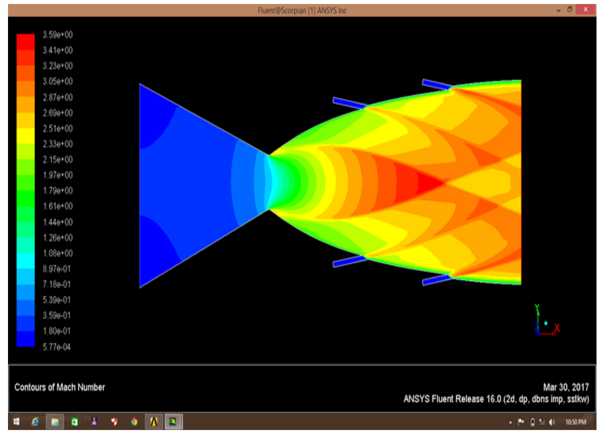
Angle 40 degree



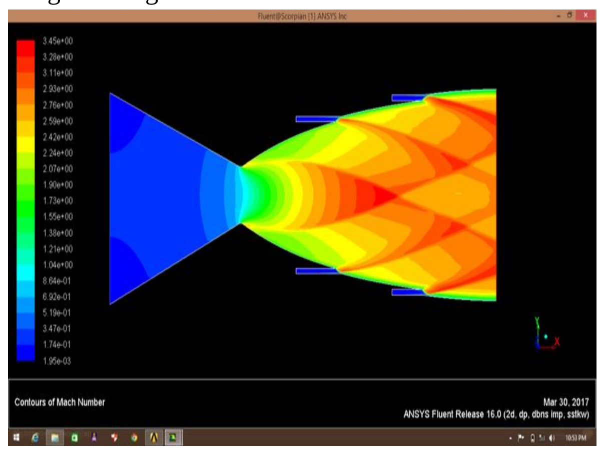
Angle 30 degree



Angle 20 degree



Angle 10 degree



Angle 0 degree

The above furnished Contours of Mach Number dictates some of the physicals facts, which are enlightened in both graphical and data interpolation manner.

Angle of Injection & Moment generated	Thrust Available
A40-36830	549199
A30-8427	567400
A20-21439.67	589242
A10-24618.84	596270
A0-27223.185	598537.14

Angle of injection in single side

CONCLUSION

Numerical Technique / Computational Analysis is carried out to appraise the effectiveness of Secondary injection of Hot gases into the Primary flow at different angle of injection. Tangential slotted injection of Hot gas is found to have better steering ability and better response in controllability of Rocket than the partial tangential slotted injection.

But, Tangential slotted injection deteriorates the performance of Nozzle when it shuts off. Hence, it must be addressed in order to be effective during shut off condition, by using it as tail pipe of Turbo Pump exhaust.

Injecting Pressure plays a significant role in Manoeuvring of Rocket, in the way of increasing the Injection Pressure appreciates the responding ability of rocket to steer its direction vector by means of reducing the magnitude of Thrust partially in order to appreciate the magnitude of Moment generated effectively.

REFERENCE

[1] Aravind kumar.T.R.,Jerin john., subanesh shyam.R., Naveen.N., Vignesh.R "Numerical studies on thrust vectoring using shock induced supersonic secondary jet", internal journal of mechanical aerospace, industrial, mechatronic and manufacturing engineering vol:7, No:8,2013.

[2] Edward walters.A and skaggs.V., Stephanie.R., "Progress Toward Replacing helon 2402 for Thrust vector control", Journal of aerospace solid propulsion vol:6, Bldg 2019,

[3] Michele Ferlauto.D., Roberto Marsilio.S., "Numerical simulation of fluidic thrust-vectoring", 23rd conference of the Italian association of aeronautics and astronautics politcenicodi torino 17-19 vol:4, November 2015.