

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

CFD Analysis of a Thrust Vectoring Nozzle

Baggio yesudas¹, Dr. S.Sankar², Sreejith K.V.³

M.Tech Scholar, Dept. of ME, NCERC, Pampady, Kerala, India¹

H.O.D, Dept. of ME, NCERC, Pampady, Kerala, India²

Asst. Professor, Dept. of ME, NCERC, Pampady, Kerala, India³

ABSTRACT:Thrust Vectoring is a moderately new innovation which has been discussed for quite a while, and it can give present day military flying machine with various points of interest in regards to performance such as enhanced manoeuvrability, shorter take-off , broadened flight envelope and survivability. Furthermore, as a result of Thrust Vectoring, there is additionally the ability to independently control the exit section of the nozzle, which permits to have dependably an "adjusted" nozzle to each flight condition and motor power requirements.

The work aims at "prediction of temperature profile and speed at the exit of nozzle of rocket in space utilizing fluent tool in Ansys workbench." The simulation includes the forecast of temperature profile and velocity profile inside the rocket nozzle. The simulation done is utilized as a part of external air conditions by varying of pressure in vacuum thought in external air condition. creo ptc is used for design of models and ansys fluent is used for cfd analysis. By taking a standard arrangement of the nozzle the shapes and charts are acquired. Further comparisons are done for modified configuration by varying the exit section shape from circle to decagon, keeping length and diameter constant.

KEYWORDS:ANSYS CFD, creo ptc, converging-diverging nozzle.

I. INTRODUCTION

Thrust Vectoring is a moderately new innovation which has been discussed for quite a while, and it can give present day military flying machine with various points of interest in regards to performance such as enhanced manoeuvrability, shorter take-off , broadened flight envelope and survivability. Furthermore, as a result of Thrust Vectoring, there is additionally the ability to independently control the exit section of the nozzle, which permits to have dependably an "adjusted" nozzle to each flight condition and motor power requirements.

A Nozzle is one of the real conduits which is utilized to quicken flow out of a progression of mach regimes. These mach regimes constitute to be Subsonic, Sonic, Supersonic and Hypersonic regions. The kind of nozzle that is decided for a specific application mostly relies upon the exit velocity of the course through nozzle and the design and working conditions. The efficiency of a nozzle incredibly impacts the general execution of the framework where it is utilized. The main applications of the nozzle mainly covers greater part of the designing, compound and restorative fields. In any case we are for the most part bothered about the outline and improvement of the channel in this way bringing about enhancing the general efficiency and limiting the work input to the framework. nozzles are mostly used to control the speed and rate of the stream alongside the direction and the pressure of the stream that rises up out of them.

II. RELATED WORK

Olivera Kostic, Zoran Stefanovic^[1] studied, a CFD (Computational Fluid Dynamics) computational model, with the expect to appropriately re-enact complex supersonic stream produced by a 2D united disparate spout. With the end goal of rocket motor push vector control (TVC) recreations, at first a few stream tab positions were utilized to create up to 30% shadowing of the spout exit, without and with a hole amongst tab and exit. Numerical outcomes were contrasted and the current test information, both quantitatively and subjectively, and reasonable understandings have been acquired in the two detects. The same CFD settings have then been connected for the computational examination of

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

another TVC compose – the stream vane, which was broke down in four diverse embraced arrangements as for the spout. Stable joining of arrangements has been accomplished, up to 40° of vane avoidances. Performed figurings have empowered correlations of the two considered TVC composes, in the feeling of the accomplished push constrain diversions, and push misfortunes.

Rahul Rai, Danish Khan^[2] led CFD to examine stream example of supersonic rocket nozzle at different level of disparate edge, mach numbers and so on to think about the conduct of stream in concurrent dissimilar spout by breaking down different parameters like static weight, temperature and speed in Mach No utilizing computational liquid progression programming (C.F.D). These comes about were additionally plotted contrasting them and investigative qualities.

Pathan Khizar Ahmed^[3] conducted investigation to think about the impact of zone proportion and spout weight proportion on the stream field of axi-symmetric sudden development from focalized disparate spouts to a roundabout channel of bigger cross-sectional zone than that of spout leave territory. The examination is concentrating on the weight and speed dispersion along pivotal length. The examination is finished by differing the region proportions and spout weight proportions. The territory proportions for the investigation are 1, 2, 4, 6, 8 and 10. The estimations of spout weight proportions considered are 2, 4, 6, 8 and 10. The outcomes are plotted with the assistance to charts. By watching every one of the outcomes it can be presumed that the stream field in developed conduit is highly influenced by territory proportion.

Venkatesh .V, C Jaya buddy Reddy^[4] directed an exchange about the outline strategy of supersonic merged disparate (C-D nozzle). The C-D spouts both funnel shaped and form are planned on a suspicion of the isentropic stream of the ideal gas. The PC code which utilizes the technique for attributes and the stream capacity to characterize high effectiveness spout profile for isentropic, inviscid, ir-rotational supersonic streams of any working liquid for any userdefined leave Mach number. The planned spout zone proportion is contrasted with hypothetical zone proportions for the chose liquid and wanted leave Mach number. The spout geometry got from the code is freely checked with the business Computational Fluid Dynamics (CFD) code. ANSYS-FLUENT has been utilized to reenact stream on spout to confirm the isentropic stream.

Shaik Khaja Hussain^[5] directed cfd to reproduce and comprehend Supersonic streams with single fly Flow at different Mach numbers. The reason for existing is to accurately comprehend the liquid elements and variety of stream properties, for example, speed, weight and turbulence in supersonic stream administration for different Mach numbers weight proportion and dimensionless dispersing single fly spout. The fly spout can take three distinct materials like Titanium, Tungsten and Nickel. It can watch the material is to be in the supersonic stream in the spout. The Mach number at the nozzle exit is seen to be less in examination with composed esteem. This is because of the consistency and turbulence in liquid close to the mass of the pipe. The Mach number declines because of stun wave and reversible flow. contrasted with indirect planes of a comparable corresponding separation over.

III.METHODOLOGY

PROJECT STATEMENT

The work aims at "prediction of temperature profile and speed at the exit of nozzle of rocket in space utilizing fluent tool in Ansys workbench." The simulation includes the forecast of temperature profile and velocity profile inside the rocket nozzle. The simulation done is utilized as a part of external air conditions by varying of pressure in vacuum thought in external air condition. creo ptc is used for design of models and ansys fluent is used for cfd analysis. By taking a standard arrangement of the nozzle the shapes and charts are acquired. Further comparisons are done for different configuration by varying the exit section shape to decagon, keeping length and diameter constant.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

SOLID MODELING

As we discussed earlier for aerodynamics analysis we require either a scaled prototype of the car for wind tunnel testing or a computer model to carry out simulation in ANSYS. As wind tunnel testing is expensive, requires large set-up and time consume therefore this project relies on CFD analysis technique for carrying out analysis. For analysis in CFD we require a 3D model which can be modelled using softwares like Creo, CATIA V 5, and Solidworks etc. any of these softwares can be used depending on one's skill. For this project work creo solid modeller has been used to develop a 3D-Model of the nozzle body. Various modelling tools have been used like Multi-section, Fill,blend, line, circles, Sweep section, Point etc. A nozzle 3D solid model has been created and then it is analysed using fluent tool in workbench.

MESHING

The process of discretization model into number of elements is known as meshing. It is the most important step in a FEA analysis. The quality of mesh defines the accuracy of the result. We must require refining the meshing to get crisp and clear elements with uniformity. The meshing is done in ANSYS Mesh Generator.

CFD SOLVER (SET UP)

This is the main part of CFD where our model solved and analysed according to the given boundary. Special care should be taken while defining the steps in the solver. The setup of the solver varies with the type of analysis we are performing. It takes up the most part of time of CFD analysis. It is here where the size of generated comes into play. Finer the mesh more the time will it consume to solve our problem because the equations are solve for each and every element, smaller the element size more will be the number of elements for a given model. The solver setup includes following steps:-

Type of analysis: For our flow analysis we carry out 3D model analysis which density based with transient conditions. We do transient state analysis.

Models: here the type of numerical model used for the analysis is defined. For fluid flow analysis Viscous based numerical model. The type of viscous based model used is realizable k-epsilon and scalable wall function model

Materials: here the material for the model is defined. In this work the analysis is carried out on Air which is the flowing fluid. The wall material is selected as titanium. Here properties of the material can be retrieved from ANSYS Library.

Boundary conditions: this is the core part of ANSYS Solver here the initial boundary conditions are defined. Without defining these variables we cannot initialize the solution that is why they are called initial boundary values. The initial boundary values for our analysis are :-

- Inlet guage total pressure(pa): - 20Mpa.
- Supersonic/initial guage pressure(pa):-21Mpa
- Direction specification method:-direction vector
- Inlet temperature condition:-1500K
- Outlet gauge pressure:-0pa
- Backflow total temperature:-343K
- Wall: - Stationary

Solution methods :-

- a. gradient:-least square cell based
- b. flow:-second order upwind
- c. Turbulent kinetic energy:-second order upwind

After carrying out all these initial set-ups run the calculation with required number of iterations. More the number of iterations the obtained result will be more close to the actual values.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

IV. EXPERIMENTAL RESULTS

The existing reference model was analysed in ANSYS and the following data were found:-

1. The temperature coefficient was found to be 1300K at outlet section.
2. The thrust force was found to be 527293N.

Table 1: comparison of results obtained from cfd analysis of existing and modified model

PARAMETERS	ACTUAL MODEL	MODIFIED MODEL
Inlet pressure	20Mpa	20Mpa
Inlet temperature	1500K	1500K
Thrust force	527293N	545773N
Inlet diameter	300mm	300mm
Throat diameter	122mm	122mm
Exit diameter	510mm	510mm
Length of converging section	400mm	400mm
Length of diverging section	1200mm	1200mm

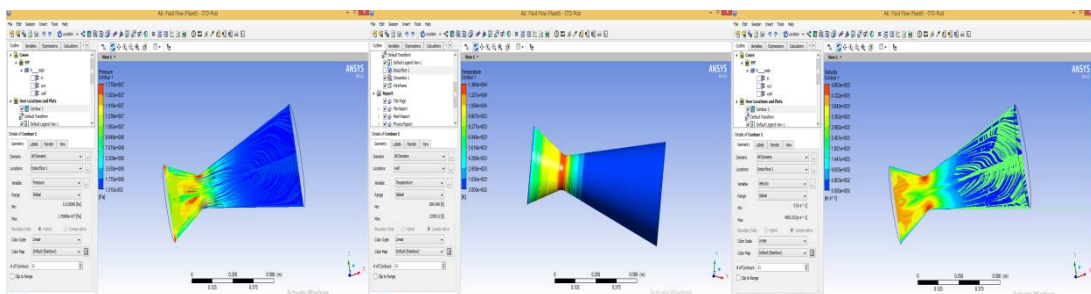


Fig.1(a).Pressure Contour Plot (b)temperature profile (c) velocity profile of reference model

PRESSURE CONTOUR PLOT

In the first place result to be confirmed is the pressure gradient of the liquid to be gone through the nozzle. The condition for acquiring high power at the nozzle exit is that the pressure of the liquid should diminish as it goes from inlet to the outlet. As the pressure diminishes, the speed of the liquid increments. Thus pressure should be inversely

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

proportional to the velocity obtained. Seeing the reference model shape, the pressure of the liquid is high at the inlet and it diminishes at the uniting segment of the nozzle and increments as it grows. The above condition is checked for the modified model and the contours tends for optimisation.

However the condition as said above is fulfilled yet because of turbulence there was an adjustment in values.

TEMPERATURE CONTOUR PLOT

The temperature is maximum at the inlet and goes on decreasing till the outlet In this setup the temperature at the throat all of a sudden increments as the throat zone is little, hence there is an entire transfer of temperature because of friction to the air at the inside at the throat region.

VELOCITY CONTOUR PLOT

The velocity is minimum at the inlet and goes on increasing till the nozzle exit.

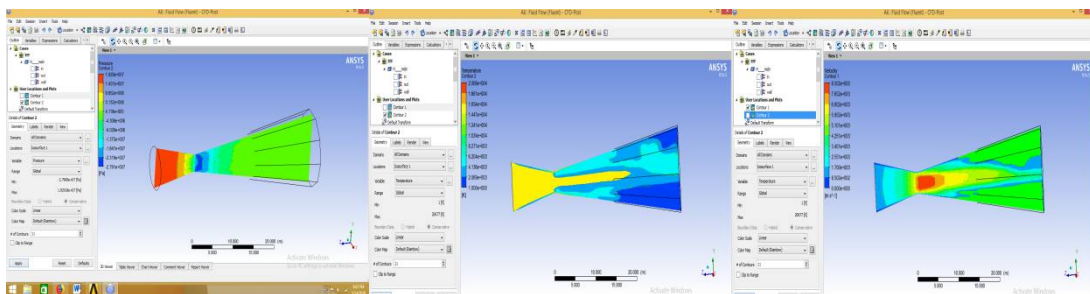


Fig. 2. (a) Pressure Contour Plot (b) temperature profile (c) velocity profile of modified model

The existing reference model was analysed in ANSYS and the following data were found:-

1. The temperature coefficient was found to be 1639K at outlet section.
2. The thrust force was found to be 545773N.

V. CONCLUSION

From the analysis we compared the results of the existing and the modified model which as shown in tabular column given above:-

It can be seen clearly that the modified model has improved temperature value at the exit section as compared to the reference model. Also the net thrust force shows an increment of value by 18480N when compared with reference model. Therefore the modified model is the optimized model.

REFERENCES

- [1]. Olivera Kostić, Zoran Stefanović, Ivan Kostić “COMPARATIVE CFD ANALYSES OF A 2D SUPERSONIC NOZZLE FLOW WITH JET TAB AND JET VANE” ISSN 1330-3651 , ISSN 1848-6339.
- [2]. “ FLOW ANALYSIS IN A CONVERGENT-DIVERGENT NOZZLE USING CFD “ Rahul Rai Danish Khan ISSN (O): 2393-8609, Volume 4 – No.2, April 2017.
- [3]. “CFD Analysis of The Supersonic Nozzle Flow with Sudden Expansion” Pathan Khizar Ahmed, P. S. Dabeer, S. A. Khan IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN : 2320–334X PP 05-07.
- [4]. “Modelling and Simulation of Supersonic Nozzle Using Computational Fluid Dynamics” Venkatesh .V, C Jaya pal Reddy ISSN 2394-9716 International Journal of Novel Research in Interdisciplinary Studies Vol. 2, Issue 6, pp: (16-27), Month: November-December 2015.
- [5]. Modelling and Computational Fluid Dynamic Analysis on Jet Nozzle Shaik Khaja Hussain, B V Amarnath Reddy, A V Hari Babu ISSN: 2455-2631 © September 2016 IJSDR | Volume 1, Issue 9 IJSDR1609066 International.



ISSN(Online): 2319-8753
ISSN (Print): 2347-6710

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 5, May 2018

- [6]. NUMERICAL SIMULATION AND OPTIMIZATION OF HIGH PERFORMANCE SUPERSONIC NOZZLE AT DIFFERENT CONICAL ANGLES A.Shanthi Swaroopini, M.Ganesh Kumar, T.Naveen Kumar”, Budapest University of Technology and Economics. IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163| pISSN: 2321-7308.