Investigation of Efficient Turbulence Model for Two-Dimensional Nozzle Designed for Supersonic Cruise Using STAR-CCM+

Karthik Mummidisetti, Sampath Emani, Gurunadh Velidi & Ugur Guven

Department of Aerospace Engineering, University of Petroleum and Energy Studies
E-mail : Karttik.m@gmail.com, sampath.evs@gmail.com

Abstract – **In the present work, investigation of various turbulence models has been carried out for predicting the efficient turbulence model for a two-dimensional nozzle designed for a supersonic cruise nozzle. Initially, a computational domain was created for a two-dimensional nozzle for a supersonic cruise, then, with an appropriate mesh size, various turbulence models has been used for simulations. The main objective of the present work is to determine the efficient turbulence model for nozzle designs. As till date, commercial software's are implementing many advanced technique, the test of turbulence model is very much needed for today's research. The results obtained from the computational approach were compared with experimental approach which was conducted in the Langley 16-Foot Transonic Tunnel at Mach numbers from 0.8 to 1.2 by NASA Langley Research Centre, Virginia. These supersonic cruise nozzles have a wide range of applications in designing Fighter jets and supersonic cruise aircraft's.**

The present work was conducted by using the commercial Computational Fluid Dynamics Software, STAR-CCM+. Initially, Nozzle at a free stream Mach number 0.9 was designed and all the initial and boundary conditions were calculated. From the results obtained in the present investigation, we can conclude that there was an excellent correlation between the experimental and computational data for K-Epsilon turbulence model.

***Keywords* – CFD, Nozzle, Turbulence Models, STAR-CCM+, boattail flap.**

I. INTRODUCTION

In today's simulation world, one of the most important fields of research can be said as Turbulence Modeling. As the advances in today's computational techniques has been increased rapidly, it's very important to choose the appropriate turbulence model for a specific kind of problem. Today simulation research in trying to solve many complex many in realistic life scenario's, therefore predicting an efficient turbulence model plays a very vital role in today's computational fluid dynamic research.

In today's commercial CFD software's, there are many turbulence models are into existence for simulations. For each specific kind of problem, people started to choose a turbulence model base on the research which was held many years ago. Due to advanced techniques in CFD, the effect of turbulence model may or may not change. Therefore in our work, we selected a specific problem which created an excellent plume of shock diamonds from the nozzle due to the free stream supersonic mach. Therefore, to simulate the flow behind this nozzle which was designed for an supersonic cruise, we simulated the domain using various K-Epsilon turbulence models. In current research K-Epsilon, K-Omega, Spallart Almaras and RST turbulence models are used for analyzing the nozzle domain.

Till date, K-omega is used extensively for external aerodynamics problems; Spallart Almaras is used for finding the aerodynamic properties. In the same way, for each and every turbulence model, there are specific applications. The results of this work show the way K-Omega, Spallart Almaras, K-Epsilon and RST turbulence models affected on the specific problem.

II. VARIOUS TURBULENCE MODELS

In this work we are using four turbulence models namely, k-epsilon, k-omega, Spalart Allmaras and RST. Therefore in this section, a brief description about these models is given below.

A. K-Epsilon

K-Epsilon is one of the turbulence models which are much into interest in today's simulation world. This turbulence model is a 2-equation model for which there is no specific dissipation for simulating the problem therefore this kind of model is mainly useful for fully turbulent flows. Till date, many research papers had chosen K-epsilon turbulence model for capturing shock diamonds. In e-epsilon, again three kind of models are developed for specific problems. In this work, we selected standard k-epsilon turbulence model.

B. K-Omega

K-Omega is one of the turbulence models which are highly used for external aerodynamics models. This turbulence model is a 2-equation model for which there is a specific dissipation for simulating the problem. Therefore it is mainly useful for external aerodynamic cases.

C. Spalart Allmaras

Spalart Allmaras is a one equation turbulence model. Therefore it is mainly useful for predicting the aerodynamic properties of a body,

D. Reynolds Stress Turbulence

RST is a higher level, elaborate turbulence model in which eddy viscosities are discarded and Reynolds stresses are directly computed.

III. PROBLEM SPECIFICATION

In this work, we are designing a two-dimensional nozzle for supersonic cruise and discretizing the domain by refining the mesh near the nozzle with 12 prism layers on the nozzle and boattail flap walls. After discretizing successfully, analysis is performed by calculating the initial and boundary conditions needed for the simulations. All the initial and boundary conditions are mentioned in below tables.

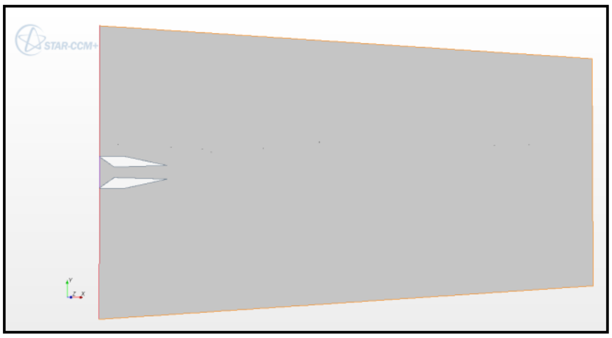

Fig. 1: 3D-CAD Model of the Nozzle domain

A. Nozzle Design:

The above figure represents the 3D-CAD model designed in STAR-CCM+. This is the whole nozzle domain with boattail walls. In the pressure study, we are validating the pressure distributions on the boattail top wall. All the design parameters of the nozzle domain can be found from the below table. As there is no radius of curvature for the boattail walls, we can observe a sharp corner at the beginning of the boattail.

Nozzle throat height	2.02 in
Nozzle exit height	2.70 in
Length of the Nozzle	13.14 in
Radius of curvature of boattail walls	0 degrees
Length of flap	8.42 in
Height of the Nozzle	6.2 in

Table 1: Design parameters of the Nozzle

The below figure represents the various nozzle parameters used for designing the nozzle. The nomenclature of these parameters is clearly described in the initial pages of this report.

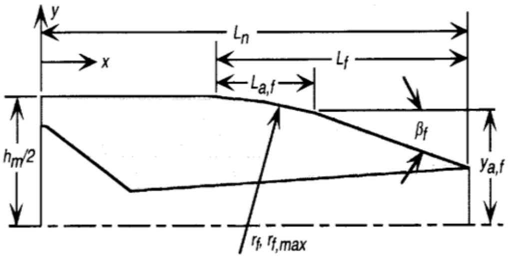

Fig. 2: Basic geometry of the nozzle

B. Mesh Continua:

As the discretization of the geometry is a very important task for predicting the results, the present nozzle geometry was discretized with a very fine mesh near the nozzle walls and boattail walls. In order to capture the shock diamonds, a volumetric cone shape was created from the nozzle exit to the end of the domain and this volumetric shape was discretized with a fine mesh and a course mesh has been applied to the other nozzle domain. Trimmer mesh is applied for the whole nozzle domain with 12 prism layers on the nozzle and boattail walls. These prism layers are stretched gradually with an initial thickness of 3 mm. Finally, this decent overall mesh and prism layers helped in converging the solutions with a plume of shock diamonds.

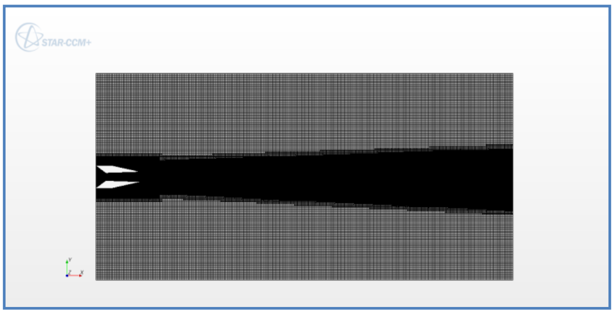

Fig. 3: Discretized Nozzle Domain

From the above figure, we can say that, the mesh near the nozzle walls and boattail walls is very fine, mesh for the volumetric cone shaped from the nozzle exit to the end of the domain is fine and the other domain has a course mesh.

C. Initial and Boundary Conditions:

After a fine mesh was generated, all the initial and boundary conditions were calculated for the transonic investigations of the designed nozzle.

In the below table, a brief mesh generation data has been provided for the further reference. As per the experimental data given, temperature units are taken in rankines and dimensions are taken in inches.

Free Stream Mach	0.9
Reference Pressure	35710 Pa
Static temperature	455 R
Total Temperature	528.71
Velocity Magnitude	277 m/s
Total Pressure	142480 Pa
NPR	5

Table 2: Initial and Boundary Conditions

The above table gives us detailed information's about all the initial and boundary conditions which are calculated according to the isentropic relations for the present validations.

IV. RESULTS AND DISCUSSIONS

The below images show us the simulation results for each turbulence model after 7000 iteration steps. We have successfully plotted the Mach number contours and the variation plot for pressure co-efficient w.r.t. its position in inches.

A. K-Epsilon

Below, Mach number contours are shown for k-Epsilon turbulence model

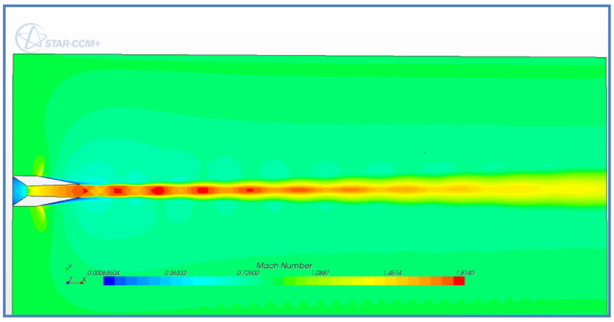

Fig. 4: Mach number Contour

The above figure represents the scalar scenes of the whole nozzle domain with k-epsilon turbulence model. From this scalar scene, we can say that as the flow is supersonic and as there is no radius of curvature on the boattail top wall, expansion is achieved through an expansion fan with infinite number of Mach waves on the boattail top wall. Also, we can clearly observe the plume of the shock diamonds formed. As the pressure of the gas exiting the nozzle is different from the ambient pressure, these shock diamonds can be observed.

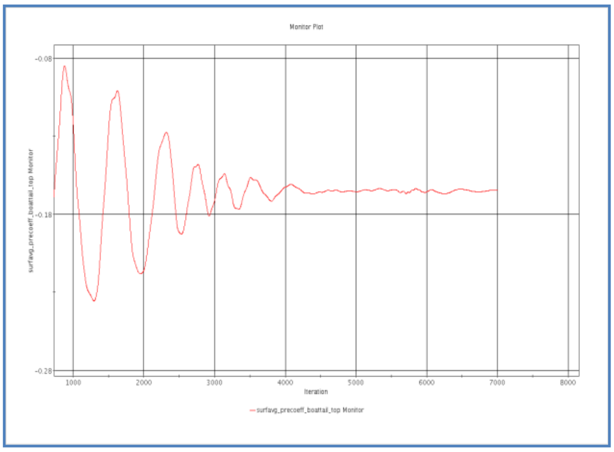

Fig. 5: Pressure vs. Position Plot

The above figure represents the pressure coefficient variations along the boattail top wall. We can clearly observe that with respect to the number of iterations, pressure coefficient follows a straight path. This figure indicates that the solution has reached convergence criteria.

B. K-Omega

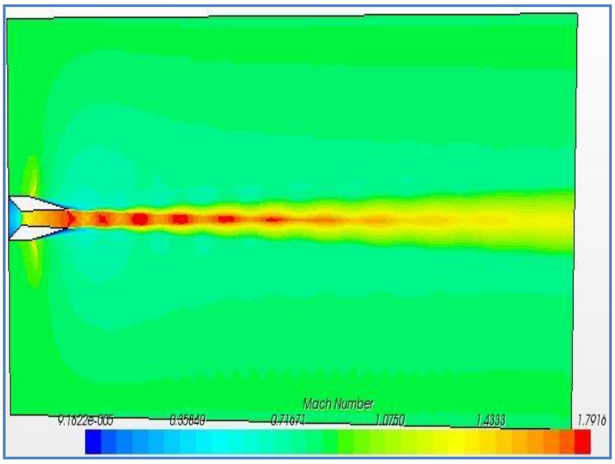

Fig. 6: Mach number Contour

The above figure represents the scalar scenes of the whole nozzle domain with K-Omega turbulence model. From this scalar scene, we can say that as the flow is supersonic and as there is no radius of curvature on the boattail top wall, expansion is achieved through an expansion fan with infinite number of Mach waves on the boattail top wall. Also, we can clearly observe the plume of the shock diamonds formed. As the pressure of the gas exiting the nozzle is different from the ambient pressure, these shock diamonds can be observed.

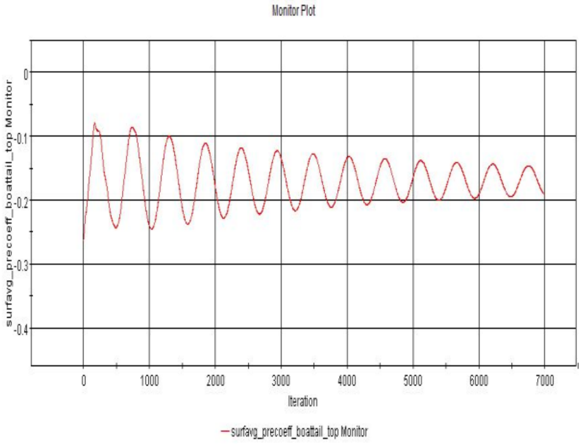

Fig. 7: Pressure vs. Position Plot

The above figure represents the pressure coefficient variations along the boattail top wall. We can clearly observe that with respect to the number of iterations, pressure coefficient did not reach the convergence criteria. Therefore, as compared with k-epsilon, this turbulence model is not efficient for simulating nozzle flows.

C. Spalart Allmaras

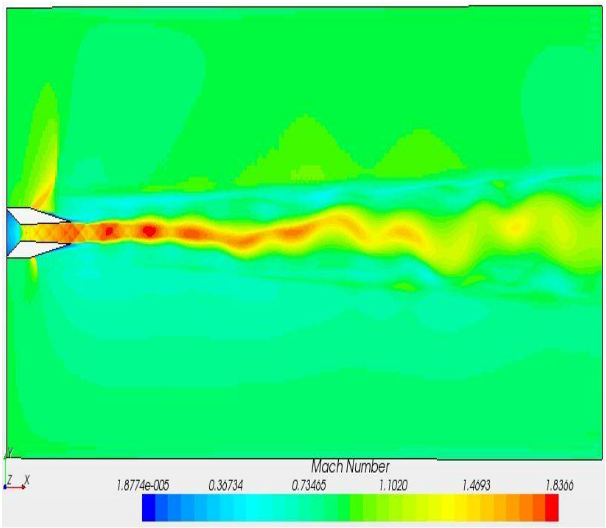

Fig. 8: Mach number Contour

The above figure represents the scalar scenes of the whole nozzle domain with Spalart Allmaras turbulence model. From this scalar scene, we can say that as the flow is supersonic and as there is no radius of curvature on the boattail top wall, expansion is achieved through an expansion fan with infinite number of Mach waves on the boattail top wall. Also, there are no plumes of the shock diamonds in this domain.

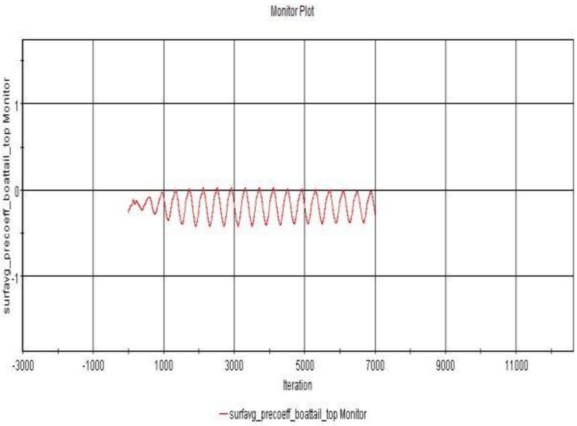

Fig. 9: Pressure vs. Position Plot

The above figure represents the pressure coefficient variations along the boattail top wall. We can clearly observe that with respect to the number of iterations, pressure coefficient did not reach the convergence criteria. Therefore, as compared with k-epsilon, this turbulence model is not efficient for simulating nozzle flows.

D. RST

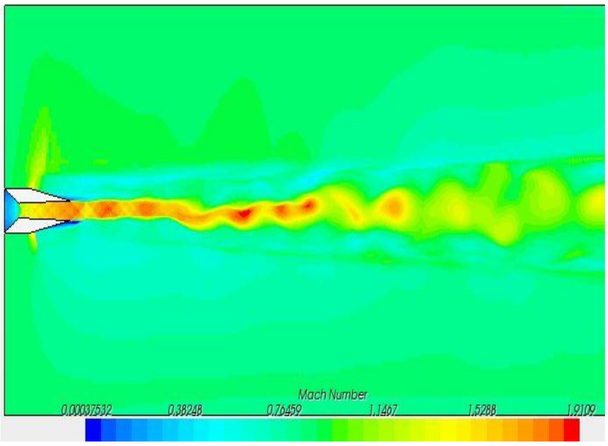

Fig. 10 : Mach number Contour

The above figure represents the scalar scenes of the whole nozzle domain with RST turbulence model. From this scalar scene, we can say that as the flow is supersonic and as there is no radius of curvature on the boattail top wall, expansion is achieved through an expansion fan with infinite number of Mach waves on the boattail top wall. Also, there are no plumes of the shock diamonds in this domain.

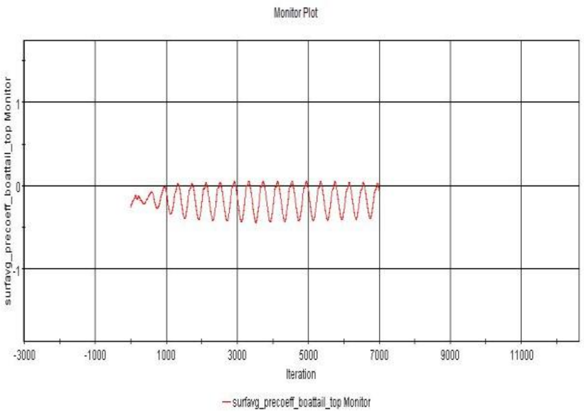

Fig. 11: Pressure vs. Position Plot

The above figure represents the pressure coefficient variations along the boattail top wall. We can clearly observe that with respect to the number of iterations, pressure coefficient did not reach the convergence criteria. Therefore, as compared with k-epsilon, this turbulence model is not efficient for simulating nozzle flows.

V. CONCLUSIONS

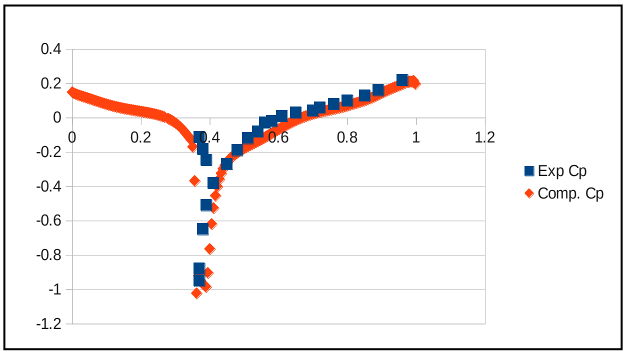

Fig. 12: Pressure Co-efficient on boattail top wall vs. Position Plot

After reaching the convergence criteria, pressure coefficient on the boattail top wall w.r.t position was plotted and compared with the experimental work.

The above pictorial representation is the experimental and computational pressure distribution comparisons of the nozzle designed in the present work. Red coloured data represents the computationally predicted values and blue coloured data represents the experimental value. From this plot, we can conclude that STAR-CCM+ successfully validated the transonic investigations of a two-dimensional nozzle designed for a supersonic cruise and the investigations indicate that excellent correlation between the experimental and computational results was obtained. And also, as there is no radius of curvature for the boattail wall, a very low pressure coefficient can be observed in that area. This is due to the sharp corner of the boattail.

From the above CFD simulations performed, we can clearly say that, K-Epsilon turbulence model was efficient for simulating the nozzle domains which can capture the plume shock diamonds accurately. As seen, other turbulent models did not capture the plume of shock diamonds when compared to k-Epsilon turbulent model. Also, when the results are obtained by computational domain with various turbulence models are compared with existing experimental data, an excellent correlation was obtained with k-Epsilon turbulence model. Therefore from these results and discussion we can say that usage of k-Epsilon is efficient for these kinds of problems.

VI. ACKNOWLEDGEMENTS

I would like to thank University of Petroleum and Energy Studies for carrying this work. I would also like to that our head of the department Dr. Om Prakash for his support in this work. Also, I would like to thank the parents of Karthik M, Mr. Seshagiri Rao M and Mrs. Patris Lumumba for their support in his higher education.

VII. REFERENCES

[1] F.J. Capone, K.A. Deere, “ Transonic Investigation of Two-Dimensional Nozzles Designed for Supersonic Cruise”, 37th AIAA joint Propulsion Conference and Exhibit, 8-11 July 2001, Salt Lake City, Utah

[2] Karen A. Deere, Mohagna J. Pandya, “ Propulsion Simulations With The Unstructured-Grid CFD Tool TetrUSS”, 37th AIAA joint Propulsion Conference and Exhibit, 8-11 July 2001, Salt Lake City, Utah

[3] Karen A. Deere, “ Summary of Fluidic Thrust Vectoring Research Conducted at NASA Langley Research Centre", the 21st AIAA Applied Aerodynamics Conference, June 23-26, 2003, Orlando, Florida

◆◆◆