CFD MODELING OF THRUST VECTOR CONTROL THROUGH JET VANE

Madiha Imdad[1], Mukkarum Hussain[2], Mirza Mehmood Baig[3], Tabinda Kanwal[4]
[1,3,4]Department of Mathematics, NEDUET, Karachi, Pakistan
[2]Institute of Space Technology (IST), Karachi, Pakistan
Corresponding Author: madiha.imdad@gmail.com

ABSTRACT: *Nozzle thrust vectoring is used for active controlling and maneuverability of the vehicle. Thrust vectoring can be achieved through flexible nozzle, secondary injection, and plume deflection through jet vanes. Experimental setup for design and analysis of thrust vectoring is very costly while numerical simulation of these cases is a challenging task. In present study numerical computations of a test case which is very much similar to thrust vectoring through jet vanes are carried out. The nozzle used for the test case has rectangular cross section and an obstacle is placed at lower wall of nozzle exit for thrust vectoring. 2D density based Roe's scheme with kw-SST turbulence model is used. Computed pressure profiles at upper and lower nozzle walls are compared with available experimental data. Present results reinforce the concept of numerical test rig and depicts that CFD would be very helpful for future development of thrust vector control system by minimizing the expensive and time consuming wind tunnel tests.*

Keywords: Convergent-divergent nozzle, Wall pressure distribution, Roe's method, RANS k-w SST, 2D flow

INTRODUCTION

In gas power systems combustion chamber is used to convert chemical energy of gases into thermal energy. After combustion chamber a nozzle is placed to expand gases where the temperature and pressure of gas decreases and velocity of gas increases drastically which generates required thrust force [1]. Due to their compactness gas power systems are used in flying vehicles like airplanes, rockets, missiles etc [2]. A flying vehicle is in mission, it is sometimes needed to control its flight path and speed due to outside disturbance or for a particular purpose. Thrust Vector Control (TVC) is a technique to adjust direction of flight path of the propulsion system [3] [4].

In addition to providing a thrust force to a flying vehicle, nozzle of gas power systems can be used to generate moments to rotate the flying vehicle and thus provide control of the vehicle's attitude and flight path [5]. Many different mechanisms have been used successfully for controlling the direction of the thrust vector which causes vehicle maneuvering. Commonly used mechanisms for controlling the direction of the thrust vector are mechanical deflection of the nozzle known as flexi-nozzle [6], injection of fluid into the side of the diverging nozzle section causing an asymmetrical distortion of the supersonic exhaust flow known as secondary injection [7] [8], and thrust vector control by deflection of exhaust gases through jet vanes [9] [10] [11].

For preliminary design and analysis numerical predictions are always recommended for quick estimation. The capability to numerically simulate these test conditions containing complex geometry and physical feature and generate precise results is a challenging task [12] [13]. Numerical predictions give researchers a chance to spread the investigation to many other possible geometry and flow conditions without performing expensive and time consuming wind tunnel tests [14].

Computational Fluid Dynamics (CFD) is a technology that enables to study the dynamics of things that flow [15] [16] [17]. CFD as a computational technology is eminently suited to develop the concept of numerical test rig or virtual wind tunnel [18]. In present study CFD computations are carried out to capture flow physics that take place in thrust vector controlling by deflection of exhaust gases through jet vanes. Test case nozzle cross-section is rectangular and an obstacle is placed at lower wall of nozzle exit for thrust vectoring [10]. Computed results are compared with available experimental data to investigate limitations and capability of numerical prediction. Results are very encouraging and depicts that CFD would be very helpful for future development of thrust vector control system by minimizing the expensive and time consuming wind tunnel tests.

TEST CASE

Rectangular cross-section nozzle with an obstacle placed at lower wall of nozzle exit for thrust vectoring is used in present study. Experimental data of this test case is available in literature [10] [19]. Wind tunnel tests for this test case were performed in VTI Žarkovo (Belgrade) by the joint team from the Faculty of Mechanical Engineering, University of Belgrade and Aeronautical Technical Institute Žarkovo. Geometrical description of test case used, grid generation and solver setting details are given in following sections.

Geometry Model and Grid Generation

Geometry modeling and grid generation are done through Gridgen software. Detailed dimension of nozzle model used in present study are shown in Fig. 1 and Fig. 2. Grid independence study is also carried out. Details of grids are given in Table 1 and Table 2 while shown from Fig. 4 to Fig. 7.

Table 1. Nozzle without obstacle mesh description

Grid	First Cell Height (mm)	Mesh size (Cells)
Coarse	0.1	13094

Table 2. Nozzle with obstacle mesh description

Grid	First Cell Height(mm)	Mesh size (Cells)
Coarse	0.1	16885
Medium	0.05	37695
Fine	0.01	128902

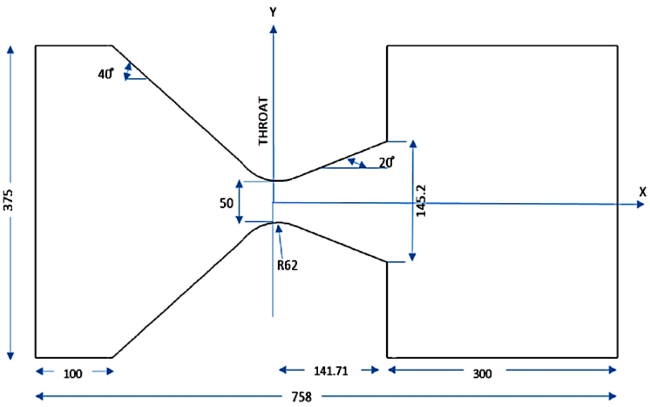

Fig. 1. Nozzle without Obstacle Geometry (dimensions are in millimeters)

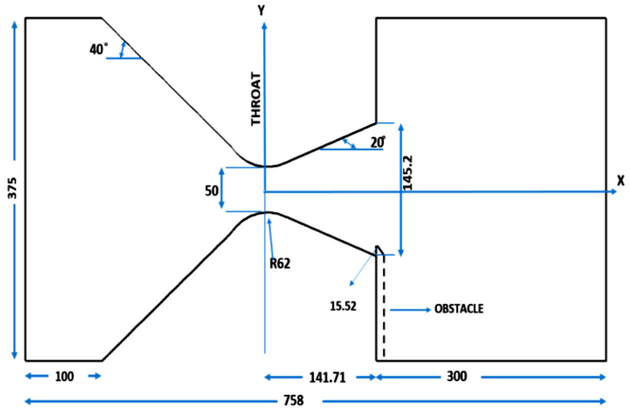

Fig. 2. Nozzle with Obstacle Geometry (dimensions are in millimeters)

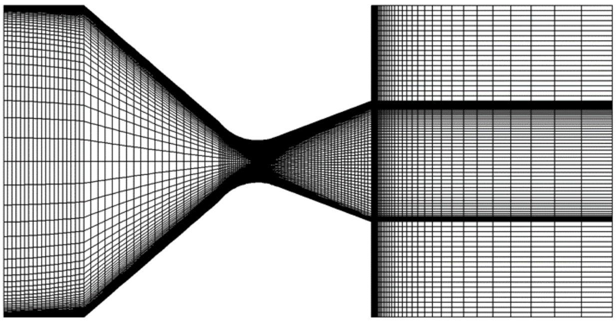

Fig. 3. Mesh used for Nozzle without

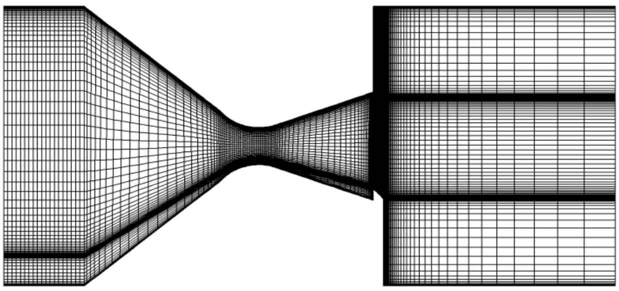

Fig. 4. Coarse Mesh for Nozzle with Obstacle

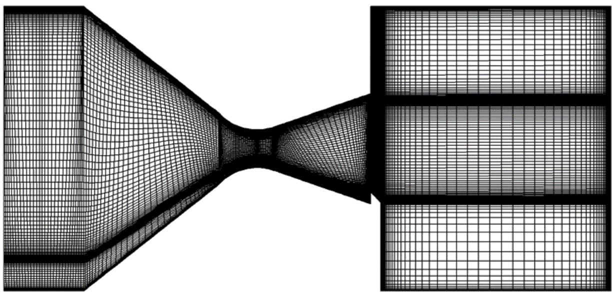

Fig. 5. Medium Mesh for Nozzle with Obstacle

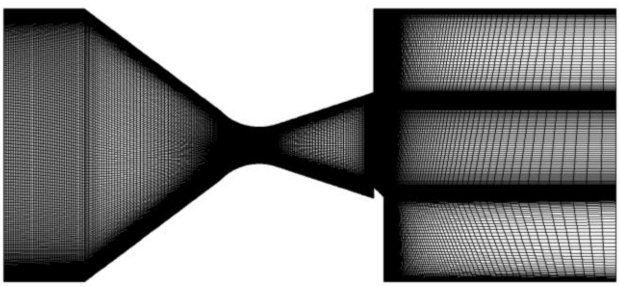

Fig. 6. Fine Mesh for Nozzle with Obstacle

Boundary Conditions

Boundary conditions applied in present study are shown in Fig. 7 and tabulated in Table 3. Solver setting used in Fluent solver is given in Table 4.

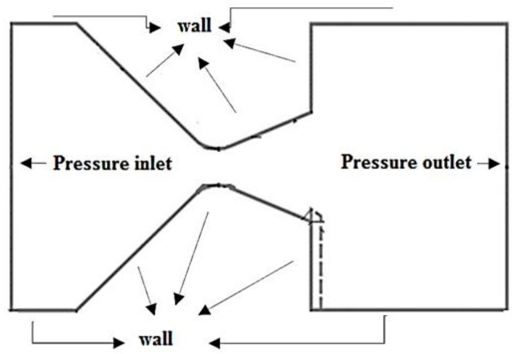

Fig. 7. Boundary Conditions used for Convergent-Divergent Nozzle

Table 3. Details of Boundary Conditions used in Present Study

M_{inlet}	0.086
T_{inlet} (K)	286.75
$P_{inlet(total)}$ (Pascal)	101054.2
$P_{inlet(static)}$(Pascal)	100532.3
P_{exit} (Pascal)	500
T_w (K)	Adiabatic Wall
R (J/kg-K)	287
Γ	1.4

Table 4. Ansys fluent solver setting

ANSYS FLUENT	
Numerical Method	
Algorithm	RANS
Method	FDS
Solver	Density Based
Linear Algebra and Accuracy	ILU
Multigrid	AMG
Spatial Discretization	
Convective terms	TVD (Second order upwind)
Pressure interpolation	First Order
Temporal Discretization	
Scheme	Implicit
Thermodynamics	
Compressibility	Ideal gas law
Dynamic Viscosity	Sutherland (three coefficients)
Turbulence Model	
Modified k-omega	k-omega SST
Boundary Conditions	
Inlet	Pressure Inlet
Outlet	Pressure Outlet
Walls	Stationary walls , No-slip

RESULTS AND DISCUSSION

Computations are carried out for rectangular cross-section nozzle without obstacle and with an obstacle placed at the end of lower wall of divergent section. Test case of nozzle with an obstacle placed at the end of lower wall of divergent section is very much similar to the thrust vector controlling through jet vanes. Although nozzles used for thrust vector controlling through jet vanes have mostly circular cross section area but the physical behavior of flow deflection through jet vanes and obstacle are almost similar.

In present study Roe's density-based solver is used to solve RANS system of equations. Turbulence effect is modeled through k-omega SST turbulence model. Three different grid spacing and sizes are used to investigate results dependency on grid quality. Experimental data of pressure profile at upper and lower wall of nozzle without obstacle and with an obstacle placed at the end of lower wall of divergent section are used for validation purpose [10][19]. Nozzle central line results for both cases are also compared with previous numerical studies.

Initially computations are run for smooth nozzle without any obstacle. Fig. 8 and Fig. 9 present comparison of pressure profile at upper and lower walls of nozzle. Results are in good agreement with experimental data. Flow is attached though out the nozzle. Separation and formation of shock and vortex are not found as shown in Fig. 14 to Fig. 16. Central line Mach number is also compared with CFD computations of Ivan A. Kostic [19] and presented in Fig. 10. Recently computed and previously predicted results are compared well. Mach number at the exit of nozzle plane is precisely predicted and is same as determined through experiment, i.e. M = 2.6. Overall results for smooth nozzle without obstacle are very satisfactory.

Furthermore, computations are carried out for nozzle with an obstacle placed at the end of the lower wall of divergent section. As compared to former problem, it is a challenging task. Flow separations, vortex and shock formation are basic characteristics of this type of flow which were not present in prior problem. Computed pressure profile at upper wall of nozzle is compared with experimental data in Fig. 11. Computed and experimental values are similar and are same as for smooth nozzle without obstacle. Flow at nozzle exit is supersonic, as shown in Fig. 17. Supersonic flow has certain regions namely, region of influence and region of dependence. Disturbance at a location transmit their effect within region of influence. Region of influence become shorter as Mach number is increased. Distance between upper and lower wall of nozzle divergent section is large enough and hence upper wall is outside from lower wall's region of influence. If same disturbance was produced at convergent section then it must be transmitted its effect on upper wall. Computed pressure profile at lower wall of nozzle is compared with experimental data in Fig. 12. Pressure profile is predicted well qualitatively as well as quantitatively. Sudden increase in pressure at lower wall near obstacle is computed precisely. Unsymmetrical flow pattern inside divergent section is also well predicted through numerical computations and presented in Fig. 17. A high speed flow decelerates when it reaches near solid surface. Its kinetic energy converts into potential energy and its pressure, temperature and density increases. Pressure, density and temperature raised due to stagnation are evident in Fig. 18 to Fig. 20, respectively.

Computed center line Mach number is also compared with previous CFD studies [19] and shown in Fig. 13. Results are in good agreement and depict that center line lies within influence region of disturbance generated by obstacle.

Overall results are not only satisfactory but also very encouraging. Results for nozzle without obstacle and an obstacle placed at the end of lower wall of divergent section are in good agreement with experimental data and previous CFD studies.

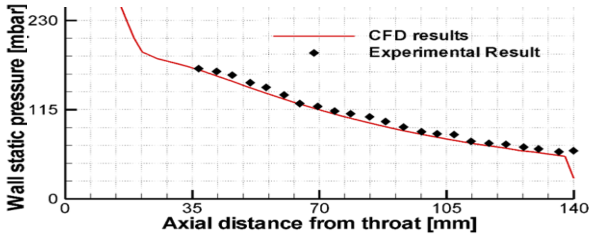

Fig. 8. Static pressure profile at upper divergent wall of nozzle (without obstacle)

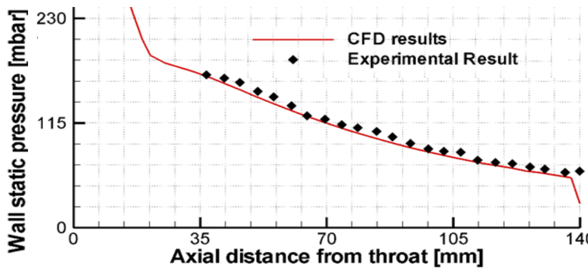

Fig. 9. Static pressure profile at lower divergent wall of nozzle (without obstacle)

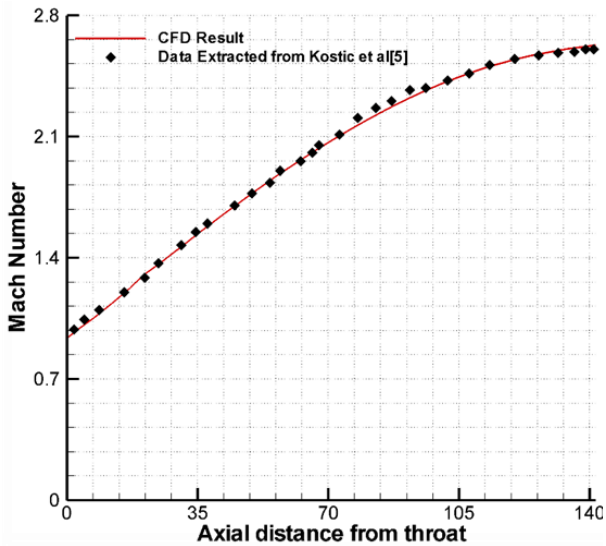

Fig. 10. Mach number along the nozzle axis, from throat to divergent section exit (without obstacle)

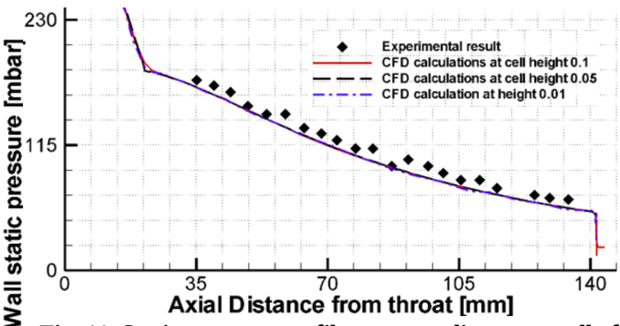

Fig. 11. Static pressure profile at upper divergent wall of nozzle (with 15mm obstacle)

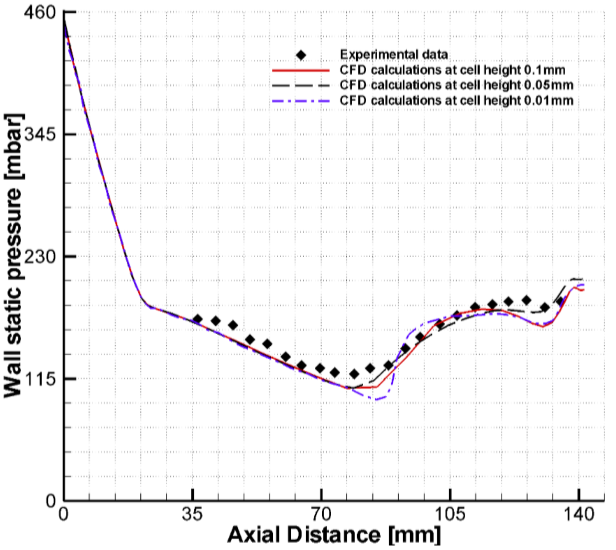

Fig. 12. Static pressure profile at lower divergent wall of nozzle (with 15mm obstacle)

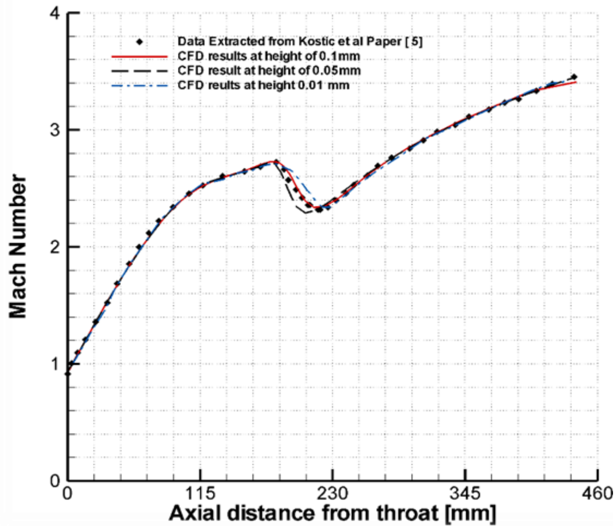

Fig. 13. Mach number along the Nozzle Axis, from Throat (with obstacle)

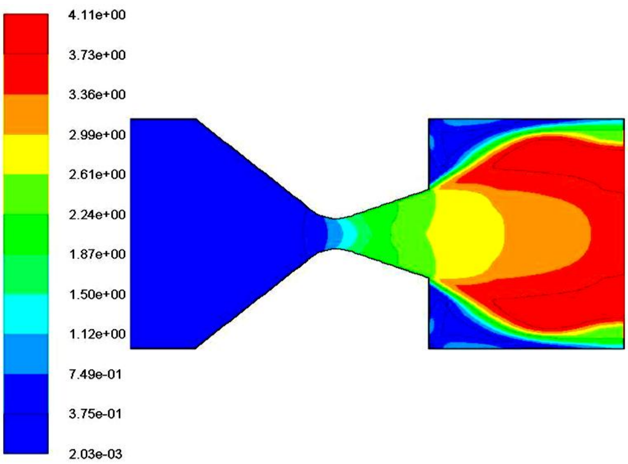

Fig. 14. Mach number Distribution Obtained through CFD Computation (without obstacle)

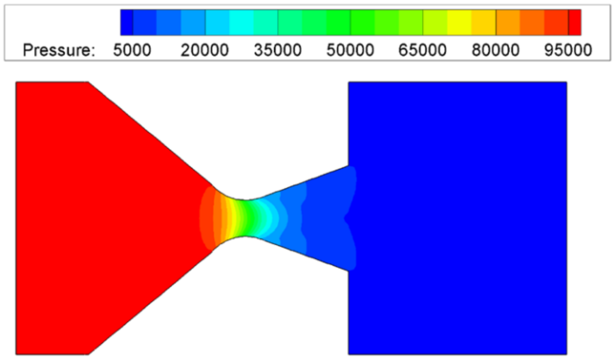

Fig. 15. Pressure Distribution Obtained through CFD Computation (without obstacle)

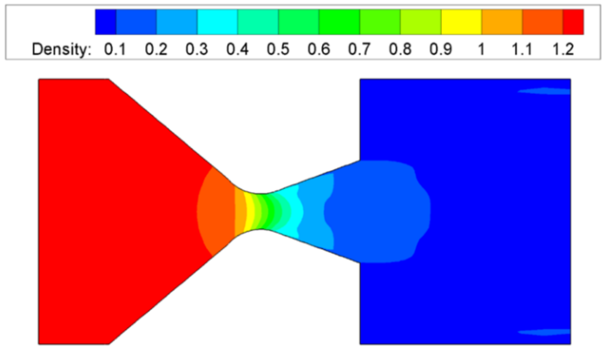

Fig. 16. Density Distribution obtained through CFD Computation (without obstacle)

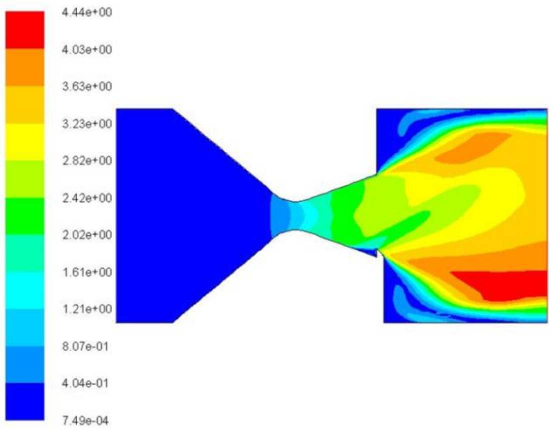

Fig. 17. Mach number Distribution Obtained through CFD Computation (with obstacle)

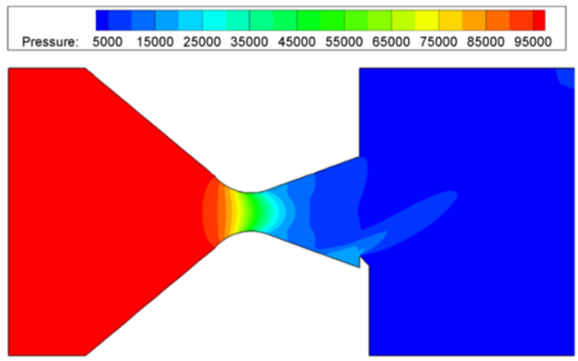

Fig. 18. Static Pressure Distribution Obtained through CFD Computation (with obstacle)

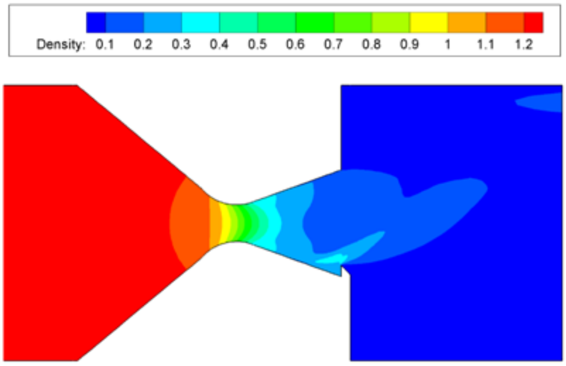

Fig. 19. Density Distribution Obtained through CFD Computation (with obstacle)

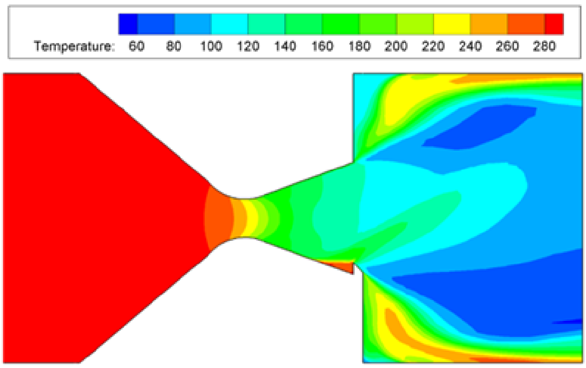

Fig. 20. Temperature Distribution Calculated through CFD Computation (with obstacle)

CONCLUSION

CFD is a numerical tool to predict flows. Its results before validation cannot be used with confidence. Validation of numerical computation of highly separated flow is presented in this study. Test case used in the present study is very much similar to thrust vector controlling through jet vanes. Results conclude that CFD is an appropriate tool to predict this type of complex flow. It will not only decrease design and analysis time but also reduce costs by minimizing costly wind tunnel tests.

ACKNOWLEDGMENT

Authors are thankful to Department of Mathematics, NEDUET, Karachi, Pakistan for their support throughout the work.

REFERENCES

[1] George. P. Sutton and Biblarz Oscar, *Rocket Propulsion Elements*, 7th ed.: JOHN WILEY & SONS, INC., 2001.

[2] M. Haluk Aksel and O.Cahit Eralp, *Gas Dynamics*.: Prentice Hall, 1994.

[3] B. Jojić and Z. Stefanović, "Theoretical Analysis and Design of 2D TVC Model for Wind Tunnel Testing, PhD thesis," Belgrade, 1986.

[19] Olivera P. Kostić, Zoran A. Stefanović , and Ivan A. K, "CFD Modeling of Supersonic Airflow Generated by 2D Nozzle With and Without an Obstacle at the Exit Section," 2015.

[4] M.Ryan Schaefermeyer, "Aerodynamics Thrust Vectoring for Attitude Control," 2011.

[5] Zeamer, Robert F.H. Woodberry , and Richard J., "Solid Rocket Thrust Vector Control," Cleveland, Ohio, December 1974.

[6] Nauparac, D.Prsić, D. Miloš, M. Samadžić, and Isaković, J , "Design Criterion to Select Adequate Control Algorithm for Electro-Hydraulic Actuator Applied to Rocket Engine Flexible Nozzle Thrust Vector Control Under Specific Load," vol. 41, No 1, pp. 33-40, 2013.

[7] Dan Miller, Pat Yagle, and Jeff Hamstra, "Fluidic Throat Skewing for Thrust Vectoring in Fixed Geometry Nozzle," 2015,.

[8] Erdem and Erinc, "Thrust Vector Control by Secondary Injection," 2006.

[9] Lloyd, R. and Thorp, G., "A review of Thrust Vector Control System for Tactical Missiles," in *14th Joint Propulsion Conference*.

[10] Zoran S. Marko M. Ivana T. and Milan P., "Ivestigation of Pressure Distribution in 2D Rocket Nozzle with a Mechanical System for Thrust Vector Control.," 2011.

[11] Chakraborty, M.S.R Chandra Murty, and Debasis, "Numerical Characteristics of Jet-vane based Thrust Vector Control System," *Defence Science Journal*, vol. 65, No.4, pp. 261-264, July 2015.

[12] Jhon C. Tannehill, Dale A. Anderson, and H. Pletcher, *Computational Fluid Mechanics and Heat Transfer, Second Edition*. United States of America: Taylor & Francis, 1984.

[13] Professor Eleuterio F. Toro, *Riemann Solvers and Numerical Methods for Fluid Dynamics, A Practical Introduction,Third Edition*. University of Trento, Italy: Springer Dordrecht Heidelberg London New York, 2009.

[14] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics,The Finite Volume Method, Second Edition*. England, United Kingdom: PEARSON Prentice Hall, 2007.

[15] Jr. John D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. University of Maryland ,United States of America.: McGraw-Hill, Inc., 1976.

[16] Klauss A. Hoffmann and Steve T. Chiang, *Computational Fluid Dynamics*. Wichita, Kansas: Engineering Education System, 2001.

[17] Klaus A. Hoffmann and Steve T. Chiang, *Computational Fluid Dynamics,Fourth Edition,Volume I*. Wichita,Kansas,USA.: Engineering Education System(EES), August 2000.

[18] Culbert B. Laney, *Computational Gasdynamics*. University ofColorado, New York, United States of America: Cambridge University Press, 1998.