



CFD Simulation Analysis of Two-Dimensional Convergent-Divergent Nozzle

R Ramesh Kumar & Yuvarajan Devarajan

To cite this article: R Ramesh Kumar & Yuvarajan Devarajan (2018): CFD Simulation Analysis of Two-Dimensional Convergent-Divergent Nozzle, International Journal of Ambient Energy, DOI: [10.1080/01430750.2018.1517683](https://doi.org/10.1080/01430750.2018.1517683)

To link to this article: <https://doi.org/10.1080/01430750.2018.1517683>



Accepted author version posted online: 31 Aug 2018.



Submit your article to this journal [↗](#)



View Crossmark data [↗](#)

Publisher: Taylor & Francis & Informa UK Limited, trading as Taylor & Francis Group

Journal: *International Journal of Ambient Energy*

DOI: 10.1080/01430750.2018.1517683



CFD SIMULATION ANALYSIS OF TWO-DIMENSIONAL CONVERGENT-DIVERGENT NOZZLE

Ramesh Kumar R^{1,*} Yuvarajan Devarajan¹

1 Department of Mechanical Engineering, Vel Tech Rangarajan Dr.Sagunthala R&D

Institute of Science and Technology, Chennai, India.

Corresponding author: ramesh.mech37@gmail.com

Abstract

Computational fluid dynamics is one of the parts of fluid mechanics and calculating numerical methods are solving the various flows of fluid. Two-dimensional models, which is to solve the flow rate of supersonic flow and analyzed the mathematical operations, which has solved the energy equations. This iteration process has analyzed and shown better results of temperature and static pressure, velocity, static temperature. The investigating results showing better temperature and velocity flows. The 2D nozzle line diagram is done in Ansys 14.5 geometry modular and calculated iteration process can be done in fluid flow. The nozzle is done and meshes using automatic method and sizing of different value of the meshing process. In the order to analyzed the Ansys fluent software and solved the flow process of the convergent-divergent iteration nozzle. Standard nozzle equation manually calculated and compared with analyzing the results.

keywords: Nozzle; CFD; Ansys 14.5; Temperature; Statistic pressure; Velocity.

Introduction:

Kinetic energy is created in the combustion chamber by chemical-thermal energy generation using in the conversion of the nozzle. The nozzle converts the high-temperature gas to lower temperature gas, higher pressure gas to lower pressure gas and lower velocity of higher velocity gases (L. Sushma et al (2017); Seongmo Koo et al (2017)). The French engineer who is trying to develop the more efficiency of the steam engine and turbine jet blades were designed successfully. The turbine jet is generating the warm power from the hot steam boiler. The nozzle project report results are shown to control the sound of speed and good efficiency. When the jet engine speed is increasing, it is affecting the sound levels (Hyungyu Lee et al (2017); Y.W. Son et al (2017)). This effect causes to heat energy dissipation. A nozzle is transforming the energy from pressure into kinetic depends on nozzle shapes. A C-D nozzle which is used to achieve supersonic flow speeds.

Easily we can solve the experimental work to use computational fluid dynamics. Fluid dynamics are not a limited iteration process solving method. It is solving the unlimited iterations among the other iteration processes. CFD is

appeared to change any iteration process could be the transport of velocity to pressure process JVS Praveen et al (2017); Nikita Bavishi et al (2017); Mazzelli Federico et al (2017)). We can solve any engineering problems like experimental methods, analytical methods and numerical methods. Among these methods are compared to analytical is very complicated and difficult. Other than the experimental method cost is very high. If any errors showing in design, prototype is to be changed all the setup of experimental prototypes and reassemble again. It may need more time and more cost (A. Eswara Kumar et al (2017); Hairong Yuan and Yue H (2009)). These methods are very useful to save time and cost consuming process. This process is converged a gas which is produced by highest pressure compression of liquid or solid process, containing oxidizer and fuel parts, inside the combustion chamber (Naoki Tsuge (2015); B. S. Karpinos and V. M. Kulish (2017)) CD nozzle indicates to the unstable fluid process of supersonic flow converged to an exit section. M. Rasidi Pairan et al (2017) revealed to be increased of fluid mixing process can get better efficiency of mixing of fluid and air process. Fluid applications are used to getting more ability of turbofans, turbojets, ramjet and rocket engines.

D. Daljit Majil (2016) investigated one-dimensional classical flow analysis of laminar and turbulent, complex structural flow analysis based on the CFD process. Computer-based analyzed values are different as compared to theoretical values. Theoretical values are most accurate value as compared to analyze values of engineering problems. They used the fluent RANS flow of 2-D nozzle convergent and diverging of NPR (nozzle pressure ratio) generates as C-codes.

Pardhasaradhi Natta et al. (2012) established different divergent angles of nozzles and changing the pressure values of initial velocity conditions are calculated and analyzed successfully. He was found the divergence angle of 18° And Mach number is 1.12 on the stage of the throat. When the increasing of Mach numbers at running contains are 2.917 at the exit portion of the nozzle. Most of the degrees are formed at exit portion are 20° . An exit portion of the throat velocity is 260 m/s for all divergence results. Mach number was decreasing to near the wall of all nozzles. This fluid is maintained constant viscosity and turbulence flow. C-D nozzle divergent rate and Mach numbers as compared to other nozzles it could be efficient rates as very low. 30° of divergence angle and 3.06 exits of Mach number results are compared to 40° of divergence angle and 3.19 exits of Mach number values are very high. But 20° of divergence and Mach number values are very low. 20° of the divergence angle is very high-intensity turbulence having in exit conditions. The conical nozzle provides as maximum velocity is 30° and 40° .

J. Z. Zhou et al. (2013) have investigated a paper which is defined as a one-dimensional gas dynamics of the exact solution for applied engineering, modeling, and simulation of compressible fluids working. Analytical solutions are well-known matches exact results. The last century has formed a one-dimensional gas dynamics theory show limited of isothermal, isentropic or adiabatic flow process. This flow process takes sample types of analytical solutions. The fluid properties changes have brought out an exact solution for the single factor in the gas dynamic process. The author published work as ODE (ordinary differential equation) for a nonlinearity process. It shows the results of ideal gases and compressible flow of nozzle divergences. It shows separate system variables for ODE (ordinary differential equation) and elementary functions to be identical in the exit region. The detailed derivations are presented in this paper and shown the most important results are calculated in analytical methods.

Zoltan Fuszko and Robert Olsiak (2016) published the paper to analyze and derived flow rate has been successful. It shows the flow rate of temperature is more able to provide in fluid properties. They took chemical types are N, NO, N_2O , O_2 . The Euler equations are solved by the flow rate of computational fluid dynamics in explicit conditions. A Euler equation solves the finite volume method into a kinetic model of the Zeldovich process. They did a work of CFD process could be completed of 150 mesh nodes with the X-axis. It appears a very good result of the process mentioned above.

Ami A. Patel (2017) proved the experimental setup of nozzle positions are calculated in inlet and exit conditions. Whereas Lewis and Carlson [7] have done, an experimental work is calculated supersonic nozzle distance is under the distance from the exit plane.

Kaviya Sundar and Thanikaivel Murugan, D (2017) investigated the separation flow mechanisms from nozzle partition. Back. et.al [9] proved the expected flows are measured through pointed supersonic nozzles. They calculated conical nozzle static pressure wasn't applicable in the one-dimensional process. They found the flow process through the transonic region depends on the configurations.

Aikaterini Katsandri (2017) presented optimized based on transient flows, expanding in the numerical process. The nozzle wall is having separation of side-loads in start-up operations and shutdown operations. The current results are showing that justification of numerical solutions of axisymmetric based on 2D problems. It has been maintained and connected to optimize outcomes. They presented the initial process is started from characteristics of the flowering process. Finite element methods are easily solved to difficult numerical calculations of Navier-stokes and Reynolds numbers.

Coupled field implicit methods are quickly solved by governing equations. Turbulence models have analyzed the flow rate of the equations incompressible flow process. In the experimental values are narrowly matched with analytical pressure values. They observed experimental values of inlet and outlet walls varying and shows more complex structures. When the maximum pressure shows at the nozzle, the start-up wall is changing from the FSS to the RSS portion. FSS and RSS phases are changing in the in the nozzles it is showing more pressure and shear stresses. By checking analytical method nozzle wall pressure results are very high in the transition process. High frequencies results are variable in different oscillatory approaches are produced at the maximum magnitude of the velocity.

Petra Tisovska et al (2017) submitted test gas is converted from high temperature to low temperature is during the period of axis-symmetric nozzle elements. They used Mac – Cormack implicit method based the governing equation is solved flow energy equations. Kinetic energy ($k-\omega$) was suggested for the flow of turbulence. This work has been classified into two categories: first categories are used to leading nozzle and collar removed from the nozzle. Second categories are used to without fielding comprises the wind and jet nozzles. All types of meshes have been studied in the field of computational fluid dynamics. Both categories are applied to the suitable mesh element of automatic method and sizing. The test gas is calculated as the separation of an ideal gas. Mac-Cormack methods are solved in the predictor and corrector flows of the finite volume method. The same author presented turbulent flow numerical calculations of nitrogen-based axisymmetric revolutions in single state simulations. They are done three fields of studies like as nozzle, jet and lower field based on independent mesh element.

Gaurab Kumar Khanra et al. (2017) done a work of numerical calculation is calculated based on a physical analysis of turbulent blades. They have approached the investigation of the ideal contour of the natural coordinating system. The structurally analyzed formulation is based on the finite volume method of governing equations. The nozzle dimensions and boundary conditions are shown in table 1&2.

Srinivas M (2017) investigated turbulent compressible flow behavior of numerical simulation is converging of Navier Stokes of statistical method. They preferred time investigation of numerical predictor correction was slowly diverging.

3. MATHEMATICAL CALCULATIONS:

Below shown Standard Dimensions are taken from the spray nozzle manufacturing standards

Table 1 standard dimensions of the nozzle

S.No	Size	Dimensions
1	Inlet Φ (D)	25 mm
2	Throat Φ (D)	10 mm
3	Outlet Φ (D)	35 mm
4	Convergent (θ)	28°
5	Divergent (θ)	14°
6	The total length (L)of the nozzle	75 mm

Table 2 Boundary Conditions

S. No	Size	Dimensions
1	Inlet pressure (P)	140 bar
2	Inlet temperature (K)	120k

C-D Nozzle Formulation:

Conservation of Mass:

$$m = \rho VA = \text{constant}$$

$$\frac{dp}{\rho} + \frac{dV}{V} + \frac{dA}{A} = 0$$

Conservation of Momentum

$$\rho V dV = -dp$$

Isentropic Flow :

$$\frac{dp}{p} = \gamma \frac{d\rho}{\rho}$$

$$dp = a^2 d\rho$$

Combine with Momentum :

$$-M^2 \frac{dV}{V} = \frac{dp}{\rho}$$

Combine with Mass

$$(1 - M^2) \frac{dV}{V} = -\frac{dA}{A}$$

For subsonic flow ($M < 1$) increased the flow of area ($dA > 0$)Causes of flow velocity is to be decreased ($dV < 0$)For supersonic flow ($M > 1$) increased the flow of area ($dA > 0$)Causes flow velocity is to be increased ($dV > 0$)(M) = Mass flow rate; (A) = area; (V) = velocity; (P) = pressure; (ρ) = density; (M) = Mach; (γ) = specific heat ratio**General procedure for CFD analysis**

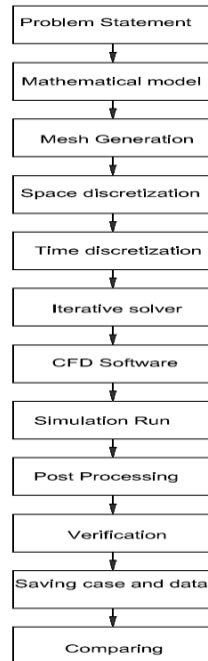


Figure 1. The methodology of CFD Process

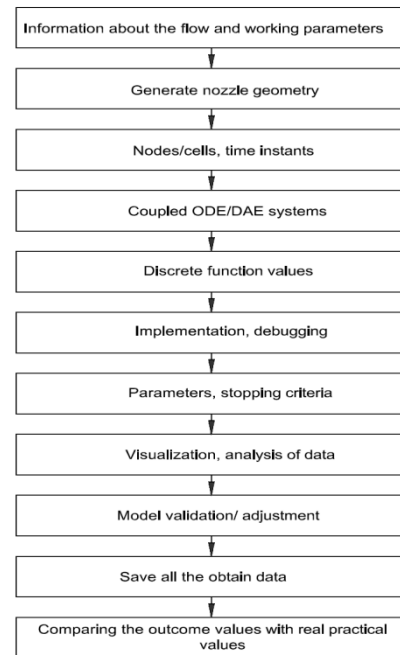


Figure 2. Steps involved in CFD process

4. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics are to be the hope of solving the engineering, mathematical calculation from derived experimental work. This aim is not limited iteration process. It may consider the thousands of thousand calculations to be solved easily. The same method we have been calculating manually is not possible. It needs much more time and calculating is to be very difficult. CFD methodology has to be shown in fig 1. Any transporting process can be solved very easily. In other ways, we have solved in engineering problems like experimental work, analytical work, and testing prototypes. The CFD mathematical calculations are very difficult to solve to manually calculate. Sometimes it's going to very large iteration mathematical process which is solved in manually. But we are compared to the experimental setup is required more cost of money needed. Prototype testing results are describing any error detection is to be dismantled and clear the errors and reassemble and again testing. This process is required more time and money. So this much of the complicated process is to be solved are not possible. CFD steps are shown in fig 2. Ansys Company solved this type of difficult are very easily. It is introduced to computational fluid dynamic systems are in the field of engineering, is to be a Renaissance process to solve the difficult iteration. The computers are helped to be solved in the field of CFD problems like simulations and flow equations. We checked before to calculate the predicted values of experimental work. This project scope is to be calculated for the nozzle which is presenting the highest exit velocity and accumulate the hustle requirements. The engine thrust speed might be calculated from the inside of the nozzle which is reducing the flow creation of instabilities of exit Mach numbers. This divergent angle might be omitted from the flow parameters. This result is showing the angles are 5° , 10° , and 15° . The prototype testing experimental works to be calculated is needed much more time and cost. CFD operations are shown the capable results to be defeated in previous limitations. In this work proved different parameters are analyzed successfully. Turbulent model flowcharts are shown in figure 3 as given below.

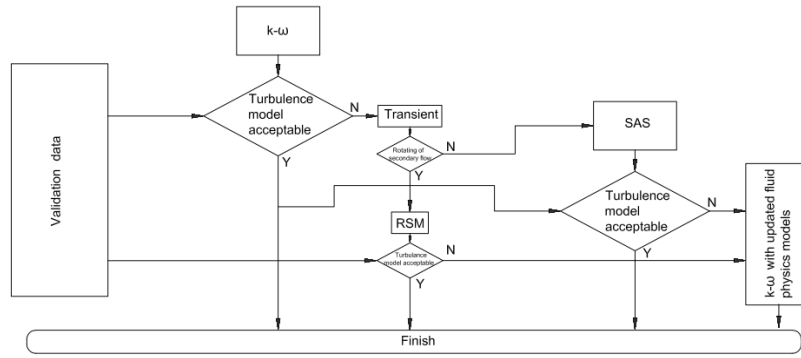


Figure 3. Turbulence model

CFD APPLICATIONS

Applications are used in CFD

- I. Combination of flow and transfer of heat dissipated in industrial applications likes heat exchanger, blowers, piping, boilers, etc.
- II. Aerodynamic structures, aerospace vehicles, aircraft machines, etc.
- III. Thermal coating, nano-particles, film factors in material testing applications.
- IV. Flow process to be used in power generations.
- V. Integrated circuit manufacturing decades to produce in the field of CVD (chemical vapor deposition)

2. Design of the Nozzle

A nozzle is a device which is to control the behavior of liquid and flow properties. To change is the direction or modification of the fluid flow, a device which has a varying cross-section of the pipe or tube is the called nozzle. To control the speed, direction, flow, pressure, fluids, nozzles are often used. The velocity of fluid increases with the decrease of pressure of the fluid in the nozzle. The nozzle line diagram is shown in fig 4. The nozzle can be classified into two categories. First categories are convergent which is reduced from a larger diameter to the smaller diameter. Second categories are divergent, which is enlarged from a smaller diameter to larger diameters.

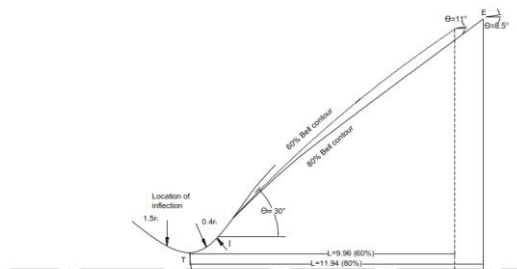


Figure 4. Contour nozzle.

5. COMPUTER ASSIST TO SIMULATION

CFD helps to solve the engineer methods like experimental, analytical, and numerical. Computational Fluid Dynamic nozzle performance is given below and steps are shown in fig 5:

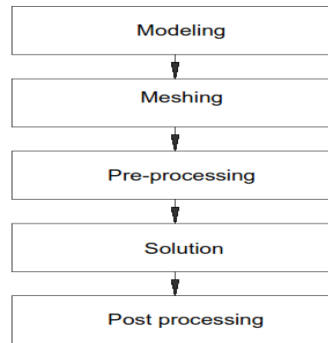


Figure5. CFD processing

Modeling: The two-dimensional nozzle model was done in ANSYS modeling parameters. The nozzle model dimensions are the listed table as given below:

Meshing

ANSYS ICEM solved the meshing process is done in after modeling process. These meshing elements are to be sized into two elements. First one of the elements is "sizing method" which is divided into 0.5 mm near the wall. The second one, of the meshed elements, is used to "contact method" which is split into 0.75 mm of convergent and divergent areas.

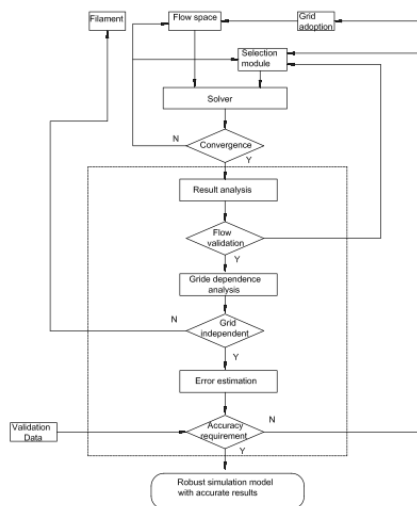


Figure 6. Meshing and post-processing model

The residual criteria are used to converge and CFD methods are shown in Fig. 6. The steady state process I (iteration), C (convergence) and D (Divergence) will be modified in the full simulation iterations. Irrespective of simulation, convergence status, and solver setups are noted. Post-processing results are matched with high expected accuracy results if the solution is converged. If not matching the converged results, the adoption will be changing the exits simulation results.

The physical model flow chart is shown in fig 7. Simulation-based on the highest value of Mach number and Reynolds number flow systems are checked twice and analyzed the simulation model to be modified or not. Suppose the flow calculations diverge in the nozzle. The CFD setup should be modified to convergence angles. Solver configuration is allowed to be starting a new iteration process which is to simulate during every time simulation scheme to be noted in different parameters. All simulation intervention is to be solved many simulation iterations and simulated the convergent iteration of diagnostic.

Pre-Processing

ANSYS FLUENT helps to solve the Pre-processing elements. Ansys element's settings are solved in double precision with two-dimensional modeling. All the dimension units are specified in mm. This meshing was checked while entering the setup; it is not showing any errors.

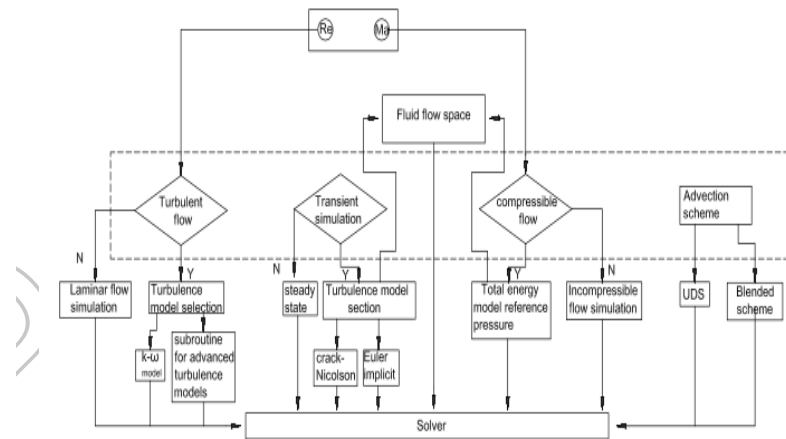


Figure 7. Turbulence physical model

Design and data processing of the CFD model depicted in the figure. 8. The fluid intractable problems can be initialized from the CFD database. The initial data, calculations could be obtained from the flow parameters of Mach numbers and Reynolds numbers are successfully verified.

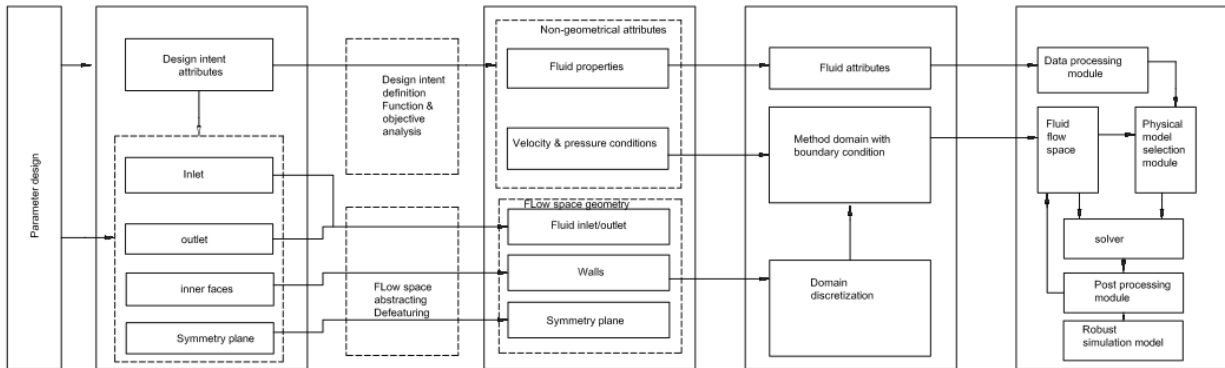


Figure 8. Design and CDF flow process

Solution

CFD setups are iterated with 500 numbers of iterations and which was converged in 218th iterations. After completion of the calculation, it is showing accurate iteration values. The iteration process could be fined to the CFD values are replaced by the next iteration input as previous output. After completion of the iteration process could be find converged values and calculated the Mach numbers. The pressure gradient is showing the boundary layers where is raising the pressure gradient. These stages the pressure levels are intensively increased and flows are varying from the nozzle walls. The flow efficiency is to be affected by walls, by following directions.

6. RESULTS AND DISCUSSIONS

The Following contour plots were showed as given below:

- 1) Velocity Contours:** The initial stages of inlet nozzle velocity flows are Minimum. Exit nozzle stage velocity flows are Maximum. The throat section velocity of Mach numbers is one in the nozzle stage. This process is called as flow stations. The nozzle flow rate is 2920.00 m/sec, which is Mach number of 2.82.
- 2) Temperature Contours:** The maximum inlet temperature is 298.195 K shows in the nozzle inlet and quickly decreasing the temperature is 129.838 K in the nozzle outlet.
- 3) Pressure Contours:** The maximum inlet pressure value is 291899 [pa] in the nozzle inlet and quickly decreasing static pressures is -87094.9 [pa] in the nozzle outlet.

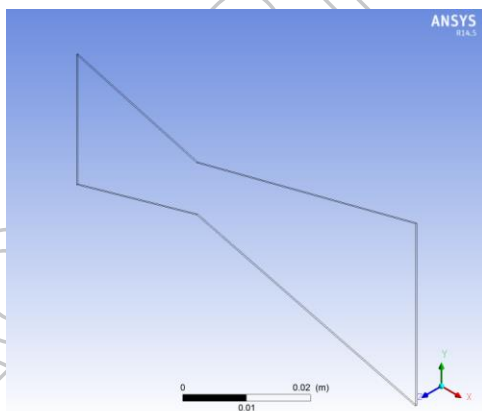


Figure 9. 2-D model of nozzle

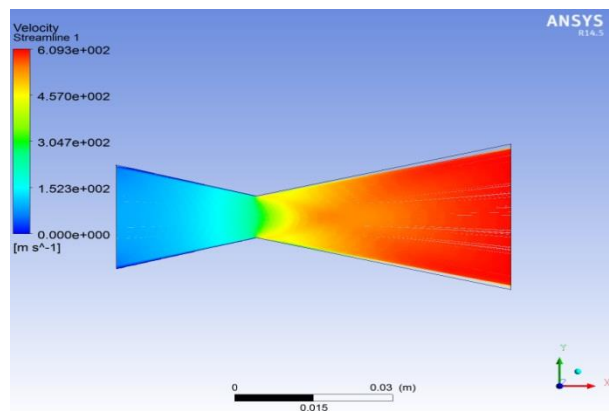


Figure 10. velocity flow constraints

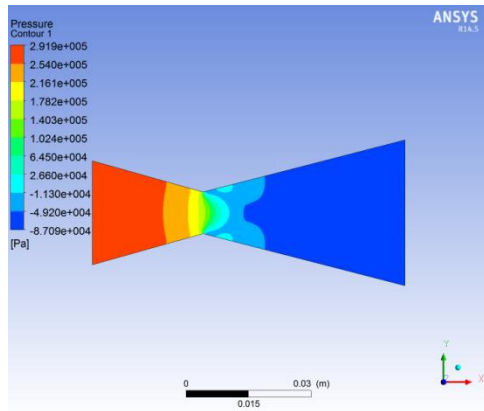


Figure 11. Pressure constraints

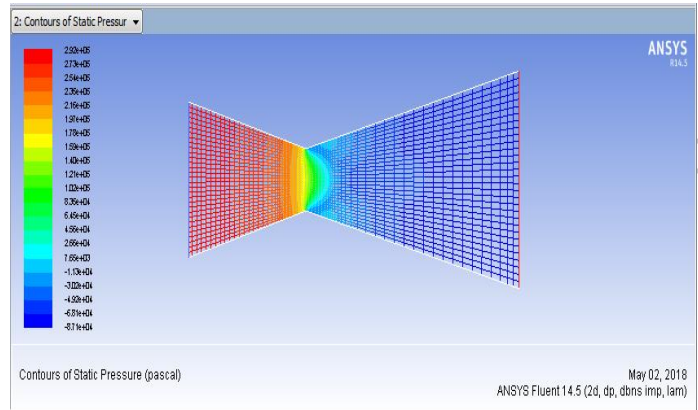


Figure 12. Static pressure contours

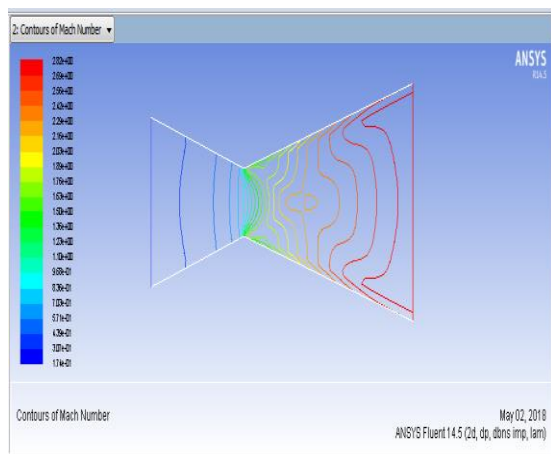


Figure 13. Contour Mach number

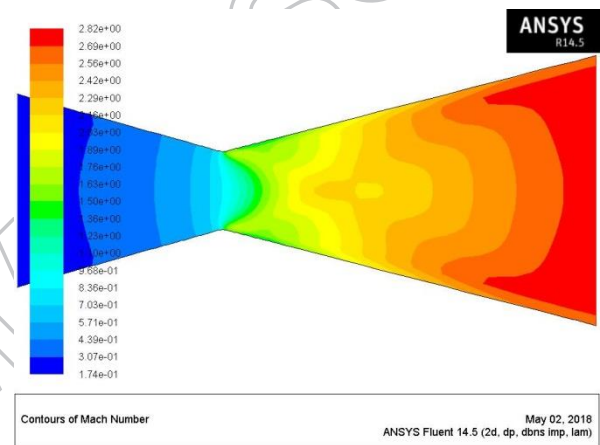


Figure 15. Static pressure contours

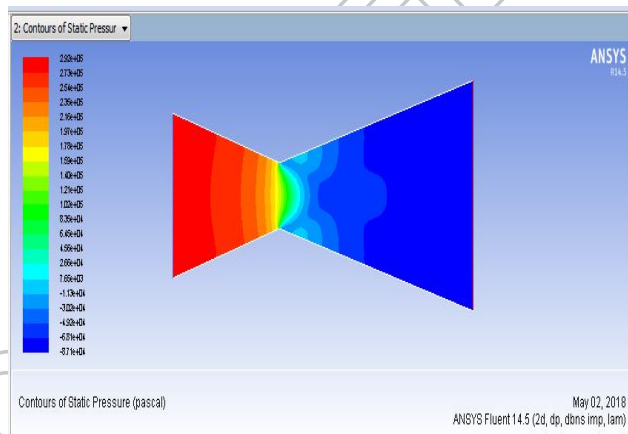


Figure 16. Contour static pressure

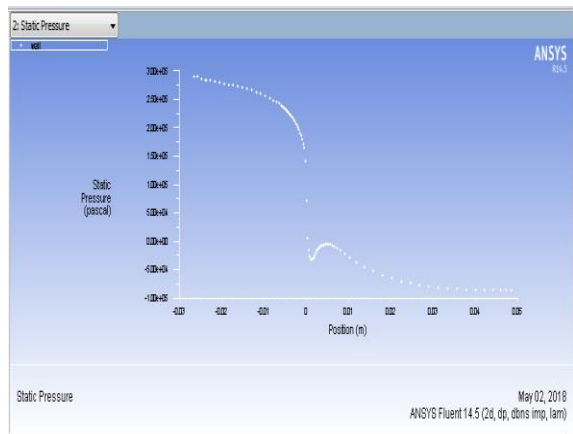


Figure 17. Graph of static pressure flow

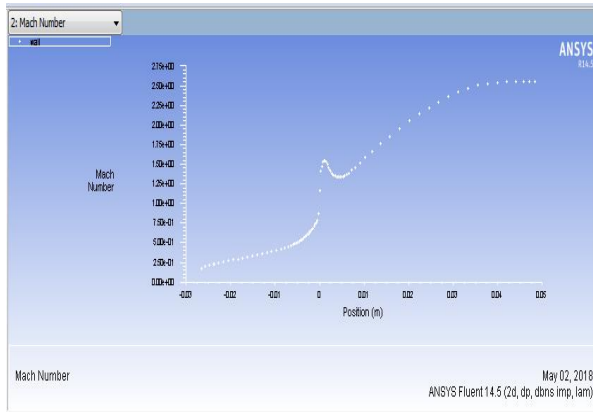


Figure 18. Mach number flow process

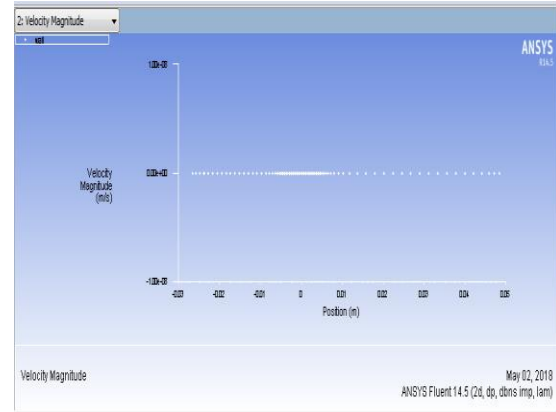


Figure 19. Velocity magnitude flow process

Velocity Magnitude: The rectangular nozzle is permitted to allow inlet and outlet flow through the nozzle is incurred maximum velocity flows in inlet valves are 610 m/s. We have checked the same flow rate in square shape incurred velocity of the maximum is 540 m/s. We have checked the same flow rate of the straight, circular nozzle incurred velocity of the maximum is 480 m/s, so we finally decided rectangular shape nozzle was to be better off compared.

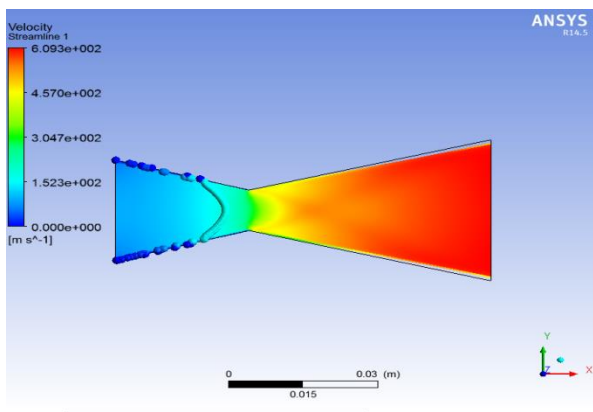


Figure 20. Velocity streamline Entering throat

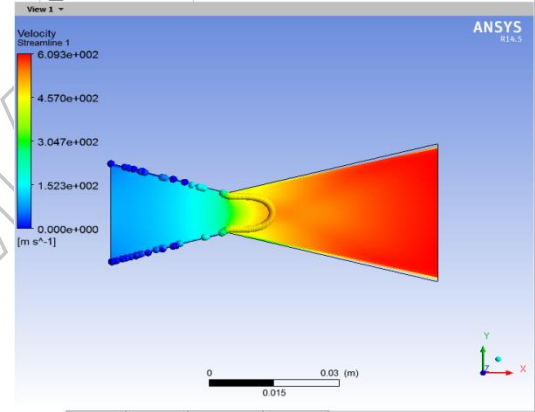


Figure 21. velocity streamline passing throat

We can easily identify the viscous flow at a lower velocity in the nozzle wall. It is got the throat section values are 0.22 m in the exit nozzle. The velocity of magnitudes is increasing from the inlet to the outlet. The inlet stage of velocity is 609.346 m/s. The throat section of the velocity results vary from $3.28e^2$ m/s to $1.45e^2$ m/s. Inlet to outlet the nozzle velocity flows is smoothly increasing.

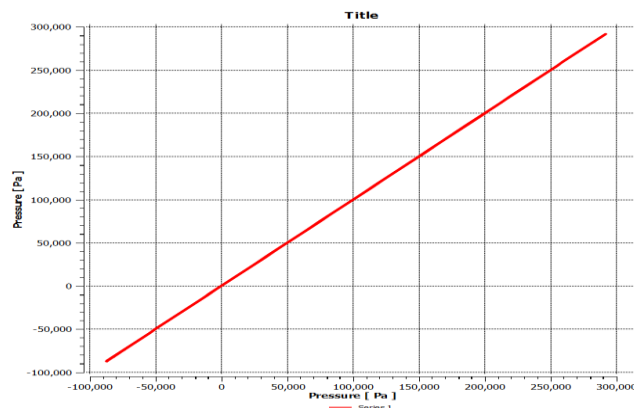
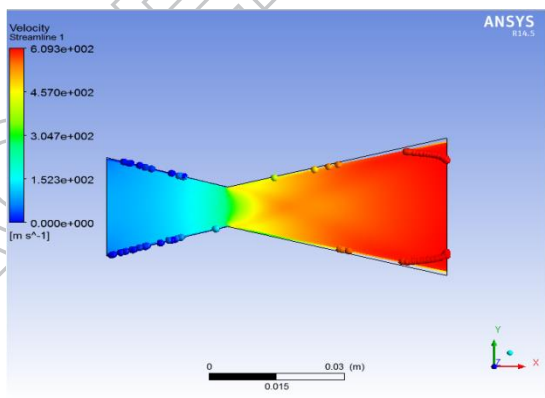


Figure 22. velocity streamline Exiting throat

Figure 23. Wall and throat pressure flow

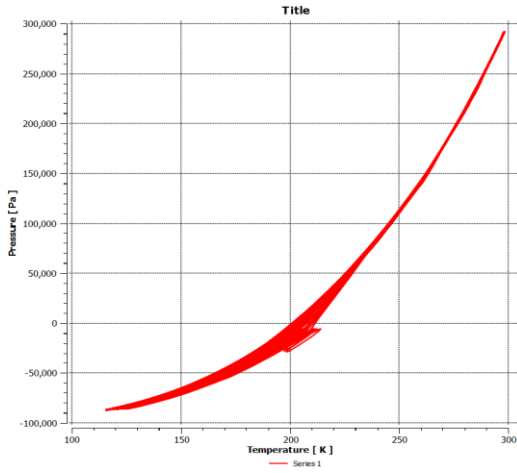


Figure 24. Pressure vs. Temperature Region

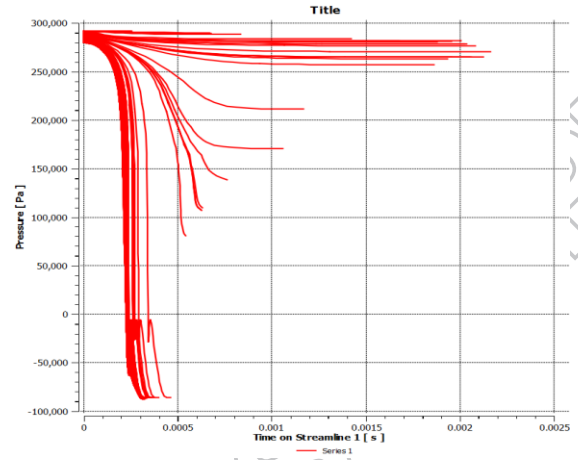


Figure 25. Pressure vs. Time on stream line

The inlet static pressures are calculated 7.56×10^6 Pa in the flow region controls towards moving from the inlet to throat flows are found to be 4.78×10^6 Pa. Next to the throat area flows rapidly move from the inlet to the throat. It could be reduced the values are 1.28×10^4 Pa for the exit section of the nozzle. We found the static pressure results were Maximum at combustion port.

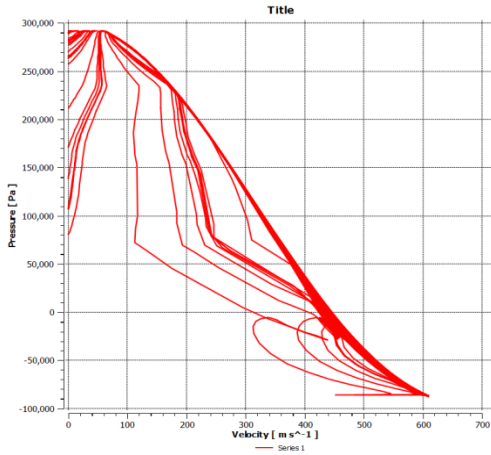


Figure 25. Pressure vs. velocity

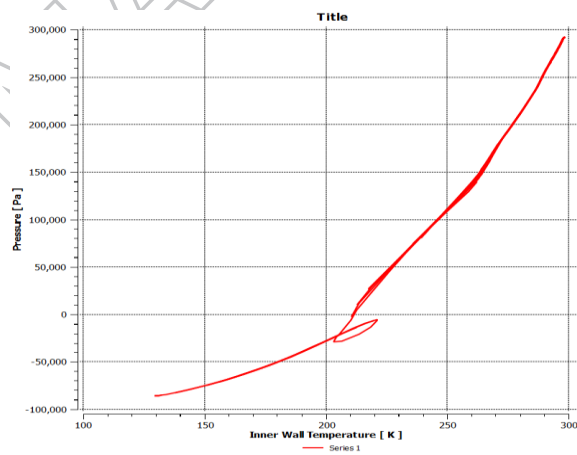


Figure 26. Pressure vs. inner wall Temperature

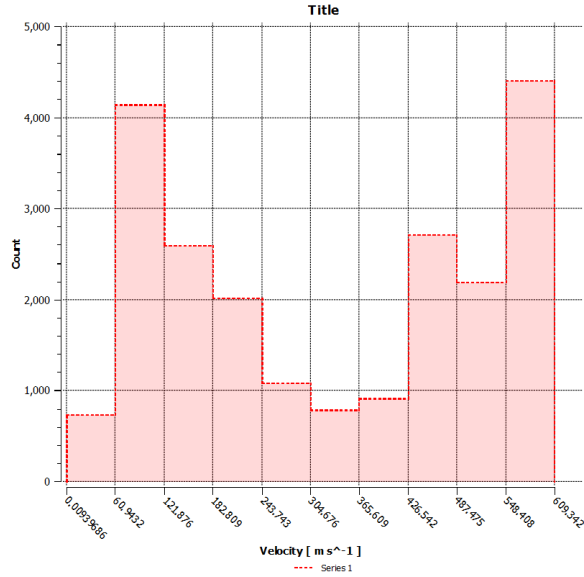


Figure 27. Flow count vs. Velocity distribution

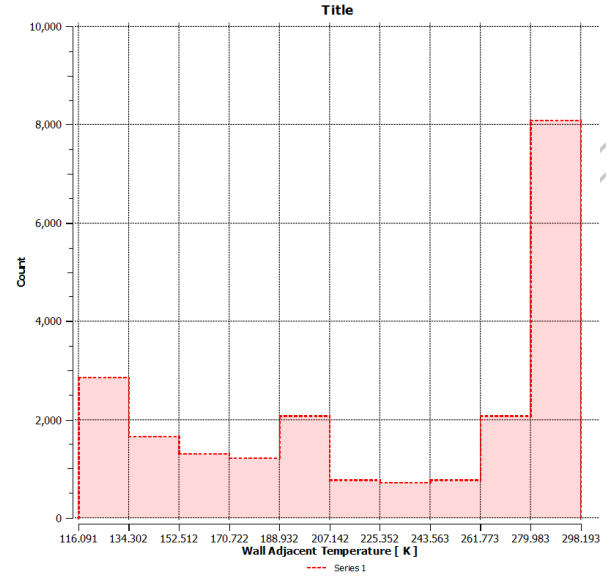


Figure 28. Flow count vs. adjacent wall temperature

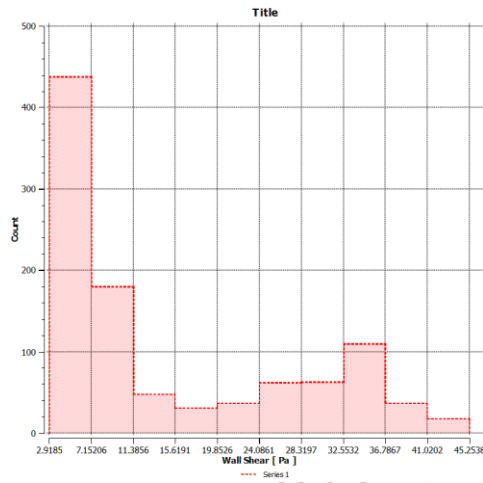


Figure 29. Flow count vs. Wall shear

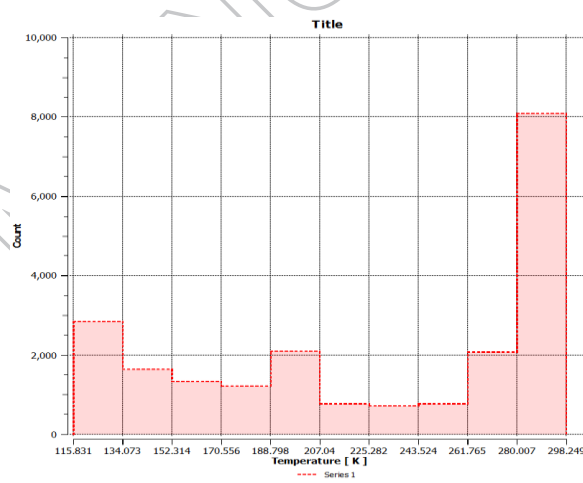


Figure 30. Flow count vs. Temperature

We have calculated the fluid static pressure values are directly proportional to static temperature. The static pressure decreases while the static temperature also decreasing correspondingly. The static temperature increases while Mach number values are decreasing correspondingly. We have checked the static temperature; it could be higher in the throat and divergent.

CONCLUSION

After completing the investigation to be observed the fluid flow nozzle speed is increasing and the divergent angle is also increasing. But static pressure is normally decreasing among the nozzle in case of the divergence angle is increasing. The fluent computer codes are indicated, the dynamical numerical solutions of the turbulence temperatures are grandly increased through the velocity. The operating pressure ratio could be increased ahead of Mach numbers. The entire nozzle wall Mach number values are randomly decreasing for the rear wall. Turbulence and velocity in the field calculated through the analysis. Finally, we have calculated, when the velocity increased at the moving speed the pressure of the fluid is reduced in the nozzle. We have found the results of rectangular nozzle drop of pressure is 610 m/s, square shape incurred velocity of the maximum is 540 m/s and straight, circular nozzle incurred velocity of the maximum is 480 m/s. Finally, as we compared to the square nozzle, circular nozzle, and the rectangular nozzle, the rectangular nozzle provides more drop of

pressure is 17. 23% below compared to other shapes. The rectangular nozzle provides a drop of temperature is 298.248 k. This temperature value is more as compared to other shapes.

References

- [1] M. Rasidi Pairan, Norzelawati Asmuin, Nurasikin Mat Isa, and Farid Sies (2017). "Characteristic study of flat spray nozzle by using particle image velocimetry (PIV) and ANSYS simulation method " AIP Conference Proceedings, Volume 1831, Issue 1. <https://doi.org/10.1063/1.4981150>
- [2] D. Daljit Majil (2016). "Design and Analysis of Jet Vane Thrust Vectoring Nozzle using CFD and Optimization of Nozzle Parameters " Indian journal of science and technology, Volume 9, Issue 39. [10.17485/ijst/2016/v9i39/100778](https://doi.org/10.17485/ijst/2016/v9i39/100778)
- [3] Pardhasaradhi Natta, V.Ranjith Kumar, Dr.Y.V.Hanumantha Rao (2012). "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD)" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, Issue 5, pp.1226-1235 1226
- [4] J. Z. Zhou et al. (2013). "Analysis and Simulation of the Fluid Field in Thermal Water-Jet Nozzle Based on ANSYS FLUENT & ICEM CFD", Applied Mechanics and Materials, Vols. 423-426, pp. 1677-1684, 2013. <https://doi.org/10.4028/www.scientific.net/AMM.423-426.1677>
- [5] Zoltan Fuszko and Robert Olsiak (2016). "Design and CFD analysis of an axisymmetric supersonic plug nozzle for an air-air ejector ", AIP Conference Proceedings, Volume 1768, Issue 1. <https://doi.org/10.1063/1.4963055>
- [6] Ami A. Patel (2017). "CFD Analysis Of Simple Carburetor For Different Positions Of Modified Aerodynamic Shape Throttle Valve And Fuel Nozzle Valve" International Journal of Advanced Engineering and Research Development, Volume 4, Issue 4. <https://doi.org/10.21090/ijaerd.43094>
- [7] Kaviya Sundar and Thanikaivel Murugan. D (2017). "CFD Analysis of Supersonic Nozzle with Varying Divergent Profile " International Journal of Engineering and Technology, Vol. 9, No.3 Page(s):2457-2467. <https://doi.org/10.21817/ijet/2017/v9i3/1709030336>
- [8] Aikaterini Katsandri (2017). "A theoretical analysis of a spacer filled flat plate membrane distillation modules using CFD: Part I: velocity and shear stress analysis " Desalination, volume 408 on pages 145 to 165. <https://doi.org/10.1016/j.desal.2015.09.001>
- [9] Petra Tisovska, Pavel Peukert, Jan Kolar (2017). "Verification of ANSYS Fluent and Open FOAM CFD platforms for prediction of impact flow" EPJ Web of Conferences, Volume 143, page(s):5. <https://doi.org/10.1051/epjconf/201714302130>
- [10] Gaurab Kumar Khanra, Sree Ganesh K, Ajith M, Dr. T. Jayachandran (2017). "Axi-Symmetric Thermal Analysis of Regenerative Cooled Cryogenic Engine Nozzle Using Fem ", International Journal of Science and Research (IJSR), volume 6 issue 7 on pages 400 to 405. <https://doi.org/10.21275/art20171281>
- [11] Srinivas M (2017). " Thrust Enhancement of a Convergent-Divergent Nozzle by Using CFD, " International Journal for Research in Applied Science and Engineering Technology, volume 5 issue 11 on pages 2687 to 2695. <https://doi.org/10.22214/ijraset.2017.11370>
- [12] L. Sushma, A. Udaya Deepik, Sathish Kumar Sunnam, M. Madhavi (2017). "CFD Investigation for different nozzle jets,"Materials Today: Proceedings, volume 4 issue 8 on pages 9087 to 9094. <https://doi.org/10.1016/j.matpr.2017.07.263>
- [13] Seongmo Koo and Hyuksang Chang (2017). "CFD Analysis on the Effect of the Nozzle Arrays and Spray Types in the Hydrogen Peroxide Mixing Quencher to Improve the Mixing Efficiency," Clean Technology, volume 23 issue 1 on page 42 to 53. <https://doi.org/10.7464/ksct.2017.23.1.042>
- [14] Hyungyu Lee, Jungsoo Lee, Donghwa Kim, Jinsoo Cho(2017). "Pre-swirl Nozzle Geometry Optimization to Increase Discharge Coefficient Using CFD Analysis " The KSFM Journal of Fluid Machinery, volume 20 issue 1 on

pages 21 to 28. <https://doi.org/10.5293/kfma.2017.20.1.021>

- [15] Y.W. Son, J.H. Lee and S.M. Chang(2017). "CFD ANALYSIS ON THE NOZZLE OF HIGH VISCID ROW MATERIAL FOR URETHANE FOAM, " Journal of Computational Fluids Engineering, volume 22 issue 3 on pages 79 to 85. <https://doi.org/10.6112/kscfe.2017.22.3.079>
- [16] JVS Praveen and Mehboob Pathan(2017). "Flow Coefficient Analysis for a Globe Valve by using CFD," International Journal of Science and Research (IJSR), volume 6 issue 12 on pages 756 to 763. <https://doi.org/10.21275/art20178896>
- [17] Nikita Bavishi and Hitesh Raiyani (2017). "Design and CFD Analysis of Liquid Ring Vacuum Pump," International Journal of Advanced Engineering and Research Development, Volume 4, Issue 4. <https://doi.org/10.21090/ijaerd.61414>
- [18] Mazzelli Federico, Giacomelli Francesco, Milazzo Adriano (2017). "CFD modeling of the condensation inside a Supersonic Nozzle: implementing customized wet-steam model in commercial codes, " Energy Procedia, volume 126, pages 34 to 41. <https://doi.org/10.1016/j.egypro.2017.08.053>
- [19] A. Eswara Kumar, K. Somanadha Sastry, K. Manideep, M. Priyanka (2017). "Dynamic Analysis of Flex Seal of Solid Rocket Motor Nozzle," Materials Today: Proceedings, volume 4 issue 2 on pages 1590 to 1597. <https://doi.org/10.1016/j.matpr.2017.01.182>
- [20] Hairong Yuan and Yue H (2009). "Transonic potential flows in a convergent-divergent approximate nozzle," Journal of Mathematical Analysis and Applications, volume 353 issue 2 on pages 614 to 626. <https://doi.org/10.1016/j.jmaa.2008.12.005>
- [21] Naoki Tsuge (2015). "Existence of global solutions for isentropic gas flow in a divergent nozzle with friction," Journal of Mathematical Analysis and Applications, volume 426 issue 2 on pages 971 to 977. <https://doi.org/10.1016/j.jmaa.2015.01.031>
- [22] B. S. Karpinos and V. M. Kulish (2017). "Effect of Cooling Parameters on the Thermo stressed State of Gas Turbine Nozzle Blades," Strength of Materials, volume 49, issue 3 on pages 412 to 421. <https://doi.org/10.1007/s11223-017-9881-5>