Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

A New Modified k-ε Turbulence Model for Predicting Compressible Flow in Non- Symmetrical Planar-Curvature Converging-Diverging Supersonic Nozzle

Open Access

Hariyotejo Pujowidodor[1,*], Ahmad Indra Siswantara[2], Budiarso[2], Gun Gun Ramdlan Gunadi[3], Candra Damis Widiawaty[3]

[1] Center for Thermodynamics Engine and Propulsion (BT2MP-BPP Teknologi), Puspiptek Area, Setu, South Tangerang, banten, Indonesia
[2] Department of Mechanical Engineering, Faculty of Engineering, University of Indonesia, Depok, West Java, Indonesia
[3] Department of Mechanical Engineering, State Polytechnic of Jakarta, Depok,West Java, Indonesia

ARTICLE INFO

Article history:
Received 21 May 2020
Received in revised form 20 July 2020
Accepted 25 July 2020
Available online 31 July 2020

ABSTRACT

Convergent-Divergent (CD) nozzle of a compressible fluid is a common device to accelerate fluid flow to a higher supersonic speed and to direct or modify the fluid flow. CD nozzle has been applied in wide range of fluid equipment such as turbine power, chemical mixing equipment, turbo-jet engine, and rocket. The performance of CD nozzle is strongly affected by its geometry at certain pressure ratio and the flow characteristic. In the case of complicated flow phenomena within a supersonic flow, especially in the turbulence flow, many studies use the computational simulation to obtain the detailed behavior and properties of flow. However, the k-epsilon turbulence model has limitations in predicting the effect of dissipation due to the viscous friction. This study aims to propose the new modified *k-ε* turbulence model in planar- curvature Convergent-Divergent (CD) nozzle of a compressible fluid. Two equations model of modified Standard k-ε for predicting the compressible flow within planar- curvature CD nozzle was discussed. The simulation model was run in 2D and steady, while fluid was assumed as an ideal gas with domain size was 0.65 m length, 0.071 m width. In addition, it has been discretized in 3510 structured independent grid cells. The results depicted that in the divergent section of the nozzle (supersonic region), the fluid expansion caused the change in fluid parameters such as time-average of pressure, temperature, density, and velocity. This study found that the expanded cross-sectional area with non-symmetrical planar curvature affected the turbulence behavior and properties. Furthermore, the new modified constants of c_2 in dissipation equation and c_μ of eddy viscosity model could give a better prediction than the original constant of the k-ε turbulence model.

Keywords:
Dissipation rate: k-ε RNG; k-ε standard; RSM; turbulence kinetic energy

---

[*] *Corresponding author.*
E-mail address: hariyotejo.pujowidodo@bppt.go.id (Hariyotejo Pujowidodo)

https://doi.org/10.37934/cfdl.12.7.5769

1. Introduction

Nozzle is a common device to convert potential energy to kinetic energy in many engineering applications such as in power generation and mixing process. This device has a gradual contour of cross-sectional area that will change the fluid properties, particularly for the compressible fluid. The convergent and divergent geometry causes the fluid to be contracted and expanded. The change of fluid properties is the result of converting its potential energy (heat and flow pressure) to kinetic energy. This energy change also affects the flow behaviour and characteristic within the nozzle, especially for compressible fluid because of its compressibility. In a supersonic nozzle, there is a gradual change in pressure, temperature, density, and velocity along the cross area and consequently causes the higher Reynolds number. The pressure ratio between stagnation at the upstream side and downstream side is the significant parameter to determine the fluid speed at given geometry and dimension. The numerical simulation could predict complex phenomena concerning fluid behaviour and its properties alteration. The fluid mean and turbulence are the most influenced parameters in analysing supersonic Convergent-Divergent (CD) nozzle. Turbulence modelling is one of methods to obtain detailed characteristics of high Reynolds flow where the inertial force is more dominant than the viscous force. In addition, some models are available to solve closure terms in the Reynolds Averaging Navier-Stokes (RANS). The appropriate turbulence model is the key to obtain a good prediction.

Numerical turbulence modelling solves the mathematical expression in differential equation of the physical problems based on the governing equation of mass, momentum, and energy. Irregular, random, fluctuating, diffusive, dissipative, high vortices are the main features of turbulent flow, which allow describing it computationally. Turbulent flow is very difficult to solve because of unsteady, thus it should be resolved into its mean and fluctuating velocity components and substituting into the Navier-Stokes. Followed by time averaging, and results the Reynolds Stress which defined as the closure term for turbulence modelling approach with one or two equations. Turbulent kinetic energy will be transferred from a large scale (eddies) to be a smaller scale. When at smallest scale then it will be dissipated into thermal energy by molecular activity (viscosity) which named as dissipation rate. Therefore, these turbulence parameters are the noteworthy factors in modelling analysis.

Dealing with supersonic nozzle analysis, many studies have been done by CFD modelling to improve the design. Many turbulence models are used to describe the detailed characteristic and properties of fluid flow within the nozzle. The Standard k-ε model is useful in wide range of applications such as predicting the compressible flow of the supersonic nozzle, also investigating the nozzle performance. The operating parameters such as pressure, temperature, velocity, and fluid properties play a significant role for affecting the Mach number (Ma). The Standard k-ε model was studied and compared to analytical study on the effect of pressure, temperature, and velocity on Ma, in supersonic air ejector and CD nozzle by several authors [1-3]. The k-ϵ Standard turbulence model was employed by Koten *et al.,* [4] to predict turbulent levels effects in cold flow process simulation of engine performance study.

The geometry factor of supersonic nozzle, particularly on the expansion or divergent section has a significant effect on the nozzle performance and the downstream flow behaviour. The ratio of length to diameter was investigated with Standard k-ε turbulence modelling for various nozzle pressure ratios on supersonic nozzle by Chu and Luckring [5]. The optimum thrust of the supersonic nozzle was also analysed by Swaroopini *et al.,* [6] using the Standard k-ε turbulence model with varying divergent angle for some considerations of Nozzle Pressure Ratio and Nozzle Area Ratio. The Standard k-ε turbulence model also was used by Ayeleso and Khan [7], to study the performance of supersonic nozzle for various exhaust gas in Magneto Hydrodynamics (MHD) system by analysing the contour distribution of pressure, temperature, and Mach number along the nozzle wall position. The investigation of efficient turbulence model in simulation of supersonic cruise nozzle has been completed by Mummidisetti *et al.,* [8] and the Standard k-ε turbulence model was suitable for capturing the plume shock diamonds accurately. The study of the effect of pressure ratio in CD nozzle also was performed with the Standard k-ε turbulence model by Abid and Khan [9], to analyse the Ma on the suddenly expanded flow and the shock occurrence. The CFD analysis using the Standard k-ε turbulence model was simulated by Prafulla *et al.,* [10], to investigate the impact of rotor nozzle geometry and its axial clearance on the turbine performance. The simulation study was conducted by Kostić *et al.,* [11] to investigate the influence of the obstacle at the exit area of supersonic nozzle on the rocket thrust. The effect of divergent region on supersonic nozzle was investigated by Hossain *et al.,* [12] and Kumar *et al.,* [13] to describe the contour of pressure, temperature, and Ma. The heat transfer analysis using CFD modelling on the supersonic nozzle and the aspect of expansion ratio of conical and contour of geometry was analysed by Koli *et al.,* [14] and Venkantesh and Reddy [15] to obtain the flow characteristic within the CD nozzle.

The analysis of turbulence flow in supersonic nozzle was most widely investigated using CFD simulation to figure out detail properties of the flow. The turbulent kinetic energy (k) and dissipation rate (ε) are the fundamental parameters of nozzle flow modelling where the flow has higher Reynolds number. The combination of turbulence model Spalart-Allmaras and Standard k-ε were applied by Ramji and Hasan [16], to analyse the minimum length of the divergent section for various Ma in supersonic and hypersonic nozzle. In addition, they were used to plot the graph of static pressure and Ma on the centreline and near the wall as was conducted by Kiran *et al.,* [17]. This combined models simulation was also conducted by Najar *et al.,* [18] to figure out the suitable model for flow separation and shock structures in supersonic nozzle standard and was used to find the appropriate mixing supersonic combustion standard.

More detail information about the behaviour and flow properties within supersonic CD nozzle had been done using k-ω to analyse the effect of shear stress from turbulent flow that resulted from high Reynolds number and the divergent region. The thrust performance in turbojets and turbofan using k-ω turbulence model was investigated as the two equation model likewise Standard k-ε by Akhil [19], to analyse the coaxial flow with fix and variable geometry in CD nozzle [20]. Furthermore, the k-ω Shear Stress Transport (SST) model was used in order to investigate the effect of placing an obstacle at the exit of supersonic CD nozzle.

The investigation of associated physical flow phenomena in CD rocket nozzle also was performed using the k- ω SST for various operating conditions nozzle by Balabel *et al.,* [1]. Numerical investigation of over-expanded nozzle has been studied computationally in 2D and 3D by Sellam *et al.,* [3], to analyse detailed behaviour and shock structures by using turbulence models of k-ω SST, Spalart- Allmaras, and Baldwin-Lomax. RNG k-ε simulation model was completed by Yang *et al.,* [21] and gave a good agreement for analysing the three working conditions of divergent factor and predicting the thrust coefficient of 2-D CD nozzle. The comparative study of the Standard k-ε with RNG k-ε and RSM was conducted to analyse the turbulence properties such as k, ε, and eddy viscosity. This study found that the Standard k-ε gave an effective prediction about the pressure and velocity of air in the wind tunnel as studied by Ramdlan *et al.,* [22]. Whereas in case of cross flow turbine nozzle, Pujowidodo *et al.,* [23] found that the RSM model has a good agreement than standard k-ε and RNG k-ε. The RNG k-ε turbulence model had been used by Koten *et al.,* [24] to improve the ability k-ε Standard for predicting recirculating flows and complex shear layers in ultralow emission study of compressed bio gas-diesel dual fuel engine. The modified k-ε by adding additional term in dissipation rate transport equation was conducted by Saqr *et al.,* [25], and has given the better agreement with experimental results of confined vortex flow simulation and better capturing of the occurred core vortex flow inside.

One of the significant keys in turbulence modelling is the consideration of the closure scheme. In terms of Standard k-ε and RNG k-ε models, their accuracy is determined by empirical constants as found by Aziz and Khan [26]. It has been reviewed by Mishra and Aharwal [27] that for better estimating *coanda* effect of flow separation and attachment effect from ceiling, it was taken the constant of c_μ= 0.12 or 0.15 above default values 0.09 but there was no evidence from experiment. Investigation nozzle type, size, and pressure was conducted by Salim and Sidik [28] to analyse the effects on spray distribution and found that pressure variations will cause significant difference in width and spray distribution.

Many studies have found that the turbulence model has limitations in providing the good acceptance results of compressible flow supersonic nozzle, compared to other model results. This is because the constants of diffusion and dissipation in the equation of k-epsilon model are obtained empirically.

The present work deals with the study of the turbulence models, which comparing the Standard k-ε model, Renormalization Group (RNG) k-ε model, and Reynolds Stress Model (RSM) for predicting a hot air flow in the supersonic nozzle with non-symmetrical planar-curvature geometry.

This study will figure out the flow parameters such as time –average of pressure, temperature, density, and velocity. In addition, the turbulence properties like turbulent kinetic energy (k) and energy dissipation rate (ε) will be analysed. The objective of this study is to propose the new modified k-ε turbulence model for giving a better prediction.

2. Methodology

By analysing statistical data, the turbulent flow shows the fluctuating values which always vary in space and time. In flow field, its values adhere to the rule of conservation of mass, momentum, and energy. The conservation of flow in the finite volume gives the change rate of properties Φ equals to net flux J by convective, diffusive, and net rate of production.

$$\frac{\partial \rho \varphi}{\partial t} = div.J \tag{1}$$

Therefore, in general by using tensor notation the balance of mass for continuity and force for momentum conservation could be stated in differential form as shown in Eq. (2) and (3), respectively.

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \tag{2}$$

For 2-dimensional model u_i is vector component of u, v, whilst in direction x_i, there are x, y, and i for index 1, 2.

Basically, the balance of force is derived in Navier-Stokes equation from the conservation of momentum, which denotes that the rate change of force and by convective within the control volume equals to the sum of all forces applying on the surface and body force F_i. The forces which acting on the surface comprises the components of normal stress by pressure p_i and shear stress by the tangential gradient of velocity.

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_i u_j) = F_i - \frac{1}{\rho}\frac{\partial p_i}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial}{\partial x_j}(\frac{\partial}{\partial x_j}u_i) \tag{3}$$

where ρ is the density and μ is the dynamic viscosity of fluid, F_i is the buoyancy force.

Using the Reynolds averaging principles, the decomposition of average and fluctuation components of the turbulence flow velocity, has resulted in a closure term to solve the momentum equation. This equation is known as the Reynolds Averaging Navier-Stokes (RANS) and the closure term called as Reynolds Stress (R_{ij}).

$$U_j \frac{\partial}{\partial x_j}(U_i) = \quad -\frac{1}{\rho}\frac{\partial P_i}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\frac{\mu}{\rho}\frac{\partial}{\partial x_j}U_i\right) + \frac{\partial(R_{ij})}{\partial x_j} \tag{4}$$

Where $R_{ij} = -\overline{u'_i u'_j} = 2\frac{\mu_t}{\rho}S_{ij}$, with $S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$; and μ_t is eddy viscosity calculated by $\mu_t = c_\mu \frac{k^2}{\varepsilon}$; with $c_\mu = 0.09$. While $k = \frac{1}{2}\overline{u'_i u'_i}$ and the buoyancy force is ignored in this equation.

Several turbulence modelling for resolving RANS equation are Standard k-ε (STD k-ε), Renormalization Group k-ε (RNG k-ε), and Reynolds Stress Model (RSM). The difference of these models is depending on the number of equation to close the momentum equation for turbulent flow.

Turbulence kinetic Energy (k) and energy dissipation rate (ε) applying two equations model while the RSM using 6 equations. For STD k-ε model

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = \quad \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right] + 2\mu_t S_{ij}S_{ij} - \rho\varepsilon \tag{5}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon\rightleftarrows u_j)}{\partial x_j} = \quad \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + c_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t S_{ij}S_{ij} - c_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

here the constant values in above equation are: $\sigma_k = 1.00$; $C_{1\varepsilon} = 1.44$; $C_{2\varepsilon} = 1.92$. Meanwhile the RNG k-ε model is stated as following equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon \tag{7}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + c_{1\varepsilon}\frac{\varepsilon}{k}P_k - c*_{2\varepsilon}\,\rho\frac{\varepsilon^2}{k} \tag{8}$$

Noted that P_k is production term of turbulent kinetic energy, and calculated by the following correlation.

$$P_k = 0.5\frac{\mu_t}{\rho}\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)^2 \tag{9}$$

where

$$c*_{2\varepsilon} = c_{2\varepsilon} + \frac{c_\mu \eta^3(1-\eta_0)}{1+\beta\eta^3} \qquad and \quad \eta = \frac{Sk}{\varepsilon}\ ;\ S = \sqrt{2S_{ij}S_{ij}} \tag{10}$$

$c_\mu = 0.0845$; $\sigma_k = \sigma_\varepsilon = 0.7194$; $c_{1\varepsilon} = 1.42$; $c_{2\varepsilon} = 1.68$; $\eta_0 = 4.38$ and $\beta = 0.012$ are defined as constants for closure coefficients. In RSM model, the closure problem of Reynolds Stress is solved directly and is not using eddy viscosity.

$$\frac{\partial \tau_{ij}}{\partial t} + u_k \frac{\partial \tau_{ij}}{\partial x_k} = -\tau_{ik} \frac{\partial u_i}{\partial x_{k_{ij}}} + 2\frac{\mu}{\rho}\overline{\frac{\partial u'_i}{\partial x_k}\frac{\partial u'_j}{\partial x_k}} + \left(\overline{\frac{u'_i}{\rho}\frac{\partial p'}{\partial x_j}} + \overline{\frac{u'_j}{\rho}\frac{\partial p'}{\partial x_i}}\right) + \frac{\partial}{\partial x_k}\left(\frac{\mu}{\rho}\frac{\partial \tau_{ij}}{\partial x_k} + \overline{u'_i u'_j u'_k}\right) \tag{11}$$

and in this equation:

$\tau_{ij} = -\rho\overline{u'_i u'_j}$ (that has 6 components of $\overline{u'_1}^2$, $\overline{u'_2}^2$, $\overline{u'_3}^2$, $\overline{u'_1 u'_2}$, $\overline{u'_1 u'_3}$, $\overline{u'_2 u'_3}$).

Thermodynamic concepts used to analyze the energy conversion from potential energy into kinetic energy. The characteristic of supersonic converging-diverging nozzle is determined by fluid compressibility which depends on the flow operational parameters which are cross-sectional area (geometry), stagnation pressure, and temperature. In addition, other important parameters such as specific heat ratio (k) and Mach number (Ma) would affect the properties ratios for pressure, temperature, and density.

Figure 1 shows the model of 2-D non-symmetrical planar-curvature convergent-divergent nozzle. The fluid flows from the inlet at left side to the outlet at right side while the top side is in form of planar contour and the bottom side is in form of curvature contour. Some assumptions are considered that are steady flow, adiabatic wall and ideal gas.

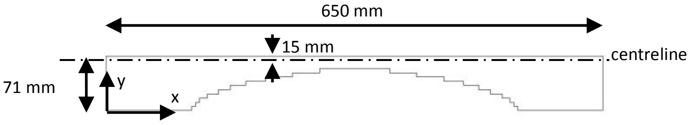


Fig. 1. Planar-curvature CD nozzle

The hot-air with temperature of 971 ^{o}K and pressure of 213.795 kPa.G flows through the planar-curvature CD nozzle. It becomes a supersonic at the expansion section if the speed is in sonic regime at throat section. Furthermore, the flow properties like pressure, temperature, and density will be changed along the nozzle.

Discretization of finite volume for numerical computation had been done for 3 different sizes of mesh in Cartesian coordinates afterwards; a grid independency test was carried out. The model was verified by comparing the absolute static pressure and dissipation rate of those three different sizes of mesh.

This study analyzed the Standard k-ε model which focused on the properties of turbulent kinetic energy and dissipation rate. The results were compared with RNG k-ε and RSM models which were used as the benchmarks. Figure 2 presents the methodology of turbulence modeling analysis for a planar–curvature CD nozzle.

3. Results

3.1 Grid Independency

The model was discretized using structured Cartesian from CFD-SOF® for 3 different sizes of mesh, which are 130x27; 258x27; and 258x52. Figure 3 shows a 130x27 grid dimension of non-symmetrical planar –curvature CD nozzle.

Figure 4 (a) and (b) plotted the absolute static pressure and dissipation rate, respectively, on a centreline inside the planar-curvature nozzle for three different sizes of mesh which depicted similar distribution. Furthermore, 130x27 cells is used for simulation analysis of turbulence modelling.

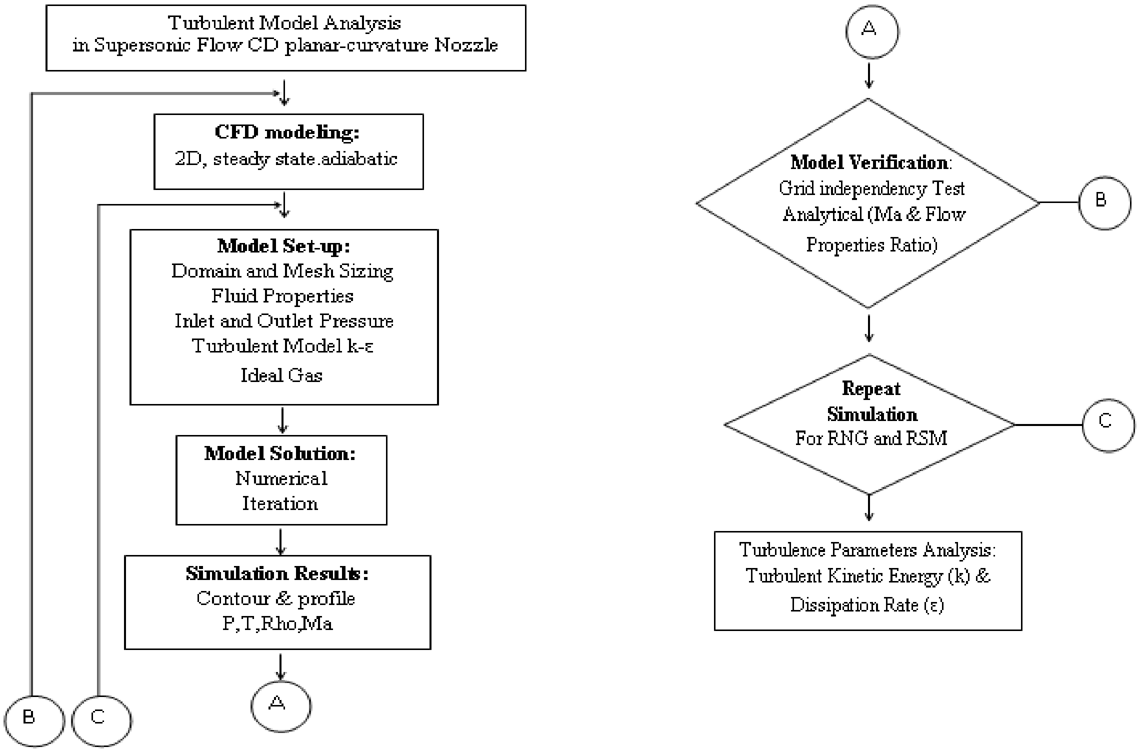


Fig. 2. Methodology of turbulent modeling analysis

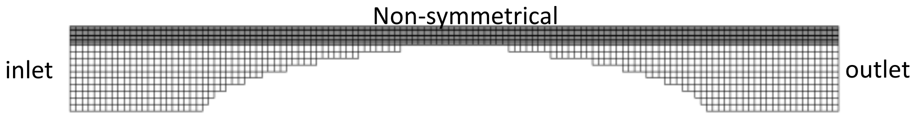
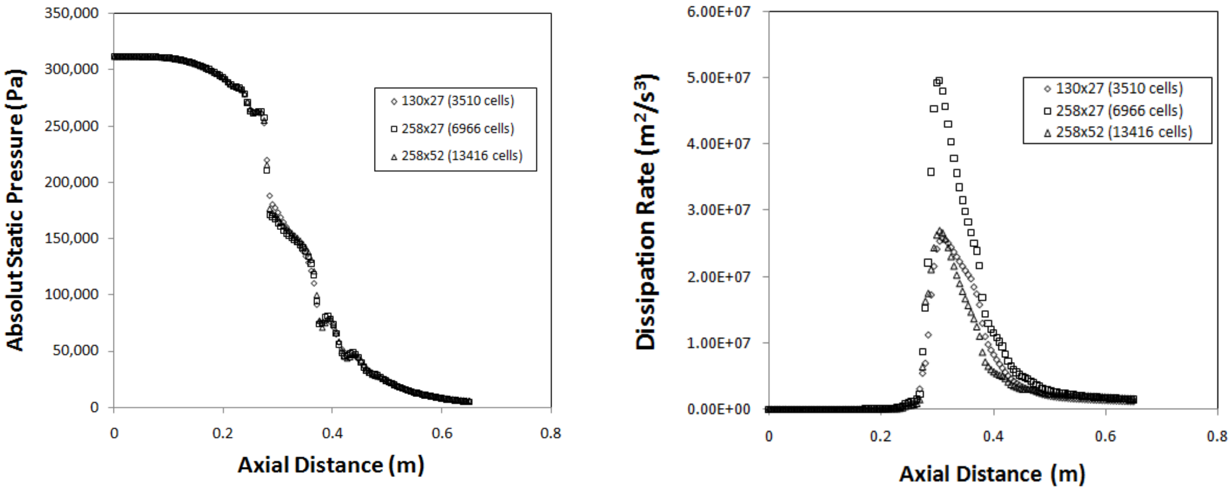


Fig. 3. Independent grid for 130x27 cells



(a) (b)

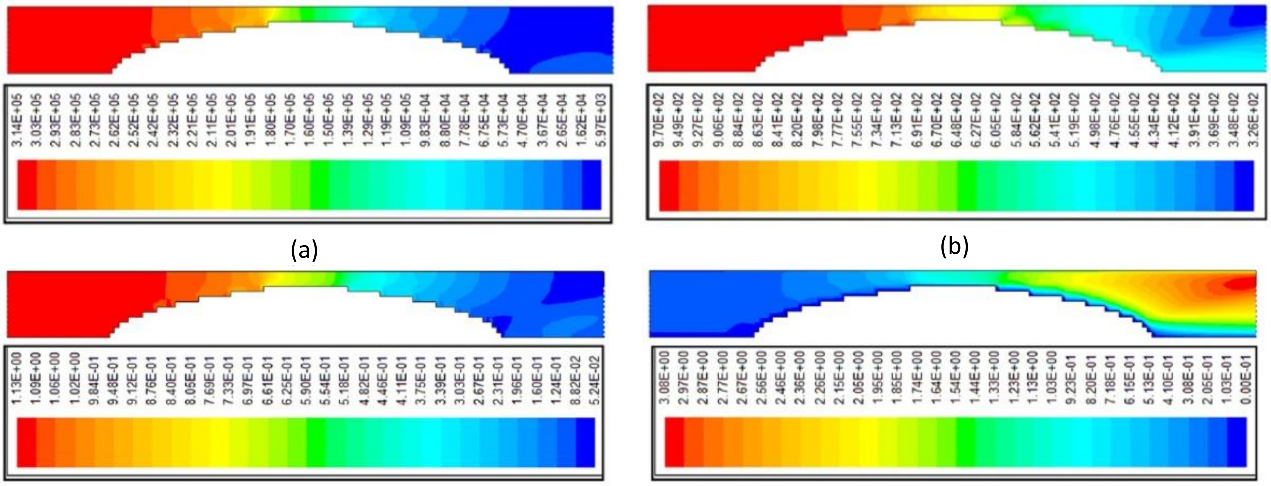
Fig. 4. Grid independency test for parameters of static pressure (a) and dissipation rate (b)

3.2 Compressible Flow Analysis

Analysis of the fluid flow on the planar -curved supersonic nozzle applied the contour distribution to obtain more detailed information about the characteristics of fluid. The analysis deeply concerns to the parameters of mean and turbulence

3.2.1 Contour of pressure, temperature, density, and Mach number

Pressure, temperature, density, and velocity (in Ma) would be analysed for flow field within the model. These parameters are the key variables in considering of nozzle design. Principally the thermodynamic parameters such as pressure and temperature would influence the density and enthalpy. When the enthalpy is converted to the kinetic energy the velocity will increase. While the change of density even affects the supersonic velocity. All of these considerations mainly rely on the geometry factor. These changes could be seen in the Figure 5.



(a) (b)

(c) (d)

Fig. 5. Contour Distribution of flow parameters of modified k-ε standard (a) absolute static pressure (Pa) (b) static temperature (°K) (c) density (kg/m^3) (d) Ma number (solved by CFD-SOF™)

The narrowing and expanding of the cross-sectional area of flow will cause the pressure and temperature decrease. Consequently, the density is also decreasing and the flow is growing to be supersonic as the flow at throat section gets in sonic condition.

Refers to Table 1 below, it was found that the flow parameters resulting from 3 different turbulent models showed the small difference. Two turbulent models of Standard k-ε and RNG k-ε give the closer result for absolute static pressure, density, and velocity. For downstream temperature, the RNG and RSM have a slight difference, comparing to the Standard k-ε.

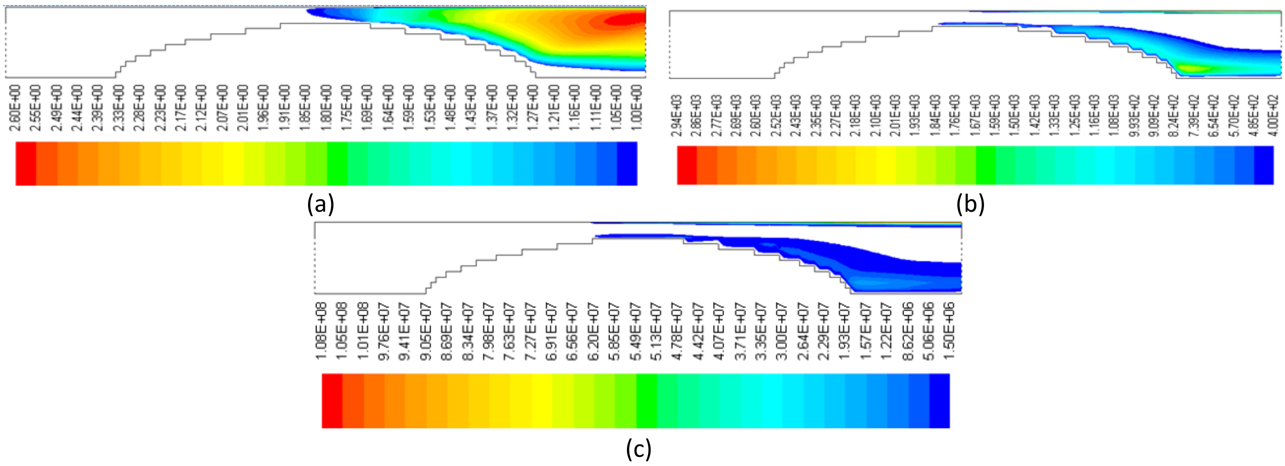
Table 1
Flow Parameters at Outlet Area for Hot Air inlet 213.795 kPa.G and 971 oK

Flow Parameters	k-ε STD	RNG	RSM
Absolute Static Pressure (Pa)	9.2126E+03	9.2284E+03	9.2075E+03
Downstream temperature (°K)	3.702+02	3.7081E+02	3.7087E+02
Density (kg/m^3)	8.2689E-02	8.2646E-02	8.2412E-02
Velocity (Ma)	2.3647E+00	2.3647E+00	2.3647E+00

3.2.2 Turbulence behaviour and k-є standard modification

Turbulent flow usually occurs at high Reynolds number, above 2000, where the inertia force more dominates than the viscous force. In supersonic flow within a CD nozzle, the closure term in turbulent modelling those are the parameters of turbulent kinetic energy (k) and energy dissipation rate (ε) would be found in the simulation result. Turbulent energy kinetic is the kinetic energy per mass unit, resulting from the fluctuating velocity components. Meanwhile, the dissipation rate is the rate of turbulent kinetic energy into thermal internal energy.

Figure 6 below show that the turbulence behaviour due to the existence of shearing action (flow disturbance) along the wall boundary in the region of sonic and supersonic flow. The adverse velocity gradient leads to the high dissipation of turbulent kinetic energy. The highest turbulent kinetic energy (above 5.74E+03 m^2/s^2) occurs on the wall at the end of curvature from a divergent part. Meanwhile, the highest dissipation rate lies on the upper side wall (above 1E+08 m^2/s^3). This is due to the highest Mach number near the upper exit side, which causes higher dissipation. Whilst on the bottom wall after divergent part as a result of pressure gradient, will create the alteration of surface stresses on the fluid elements. This flow field of turbulent would be taken into account to analyse the modification of Standard k-ε.



(a) (b)

(c)

Fig. 6. Contour distribution of turbulence behavior (a) sonic and supersonic region (b) turbulent kinetic energy k-ε (m^2/s^2) (c) dissipation rate (m^2/s^3) (solved by CFDSOF™)

Figure 7 and Figure 8 (a) and (b) show that modification of turbulent diffusivity of the Standard k-ε model does not show a significant change for turbulent kinetic energy (k) and dissipation rate (ε). While the comparison of Standard k-ε with RNG k-ε and Reynolds Stress Model (RSM) in Figure 7 and Figure 8, depict that the turbulent model of Standard k-ε producing the higher kinetic energy and dissipation rate cause the flow field more diffusive. Therefore, it is needed to change other constants of turbulence model in the Standard k-ε for lowering k and ε.

By modifying the additional constants in the turbulence model of Standard k-ε could obtain a good agreement with the benchmarked model of RNG k-ε. The Standard k-ε model with the default constants $\sigma_k = 1$ and $\sigma_\varepsilon = 1.3$ and other default values as shown in Figure 7 and Figure 8, also give the great discrepancies for turbulent kinetic energy and dissipation rate. Therefore, these results could be minimized by making the turbulent model of k and ε with the modified additional constant of $c_{2\varepsilon}$ in Eq. (6) and c_μ for calculation turbulent viscosity μ_t in Eq. (4).

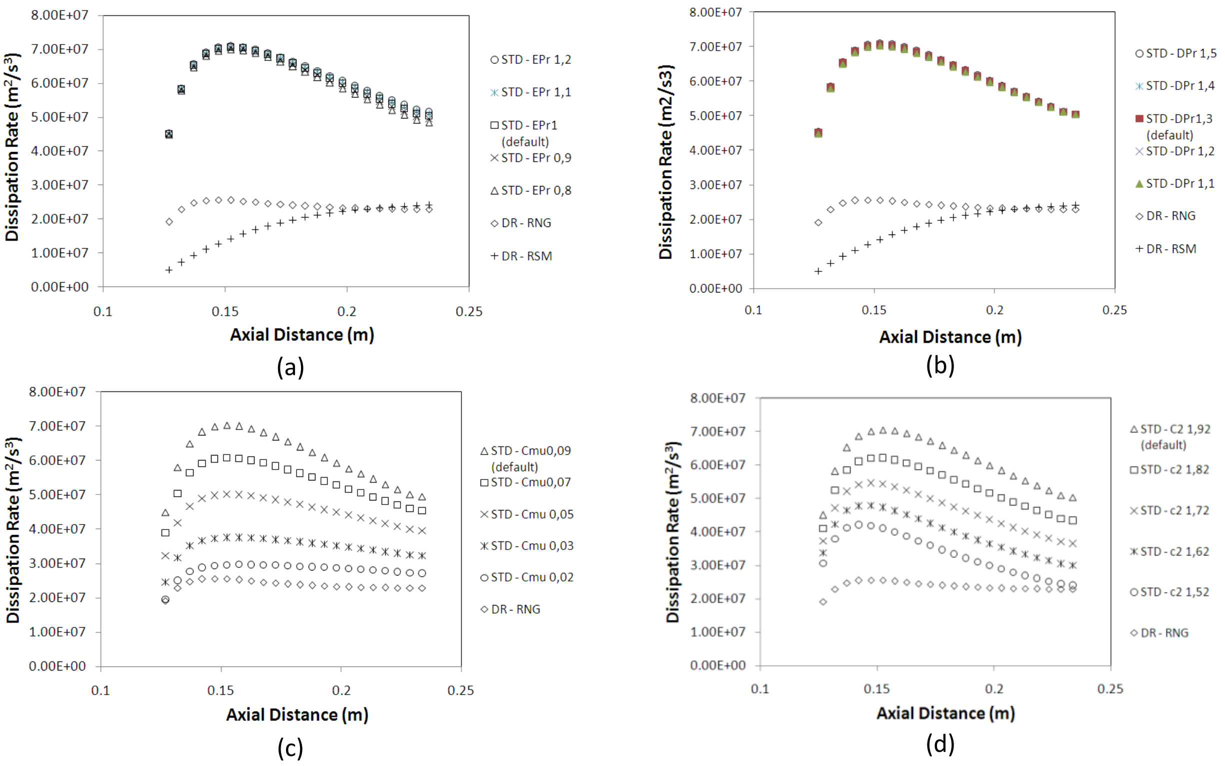


Fig. 7. Analysis of dissipation rate for some modified constants (a) kinetic energy, σ_k ; (b) for dissipation rate, σ_ε ; (c) c_2 ; (d) c_μ

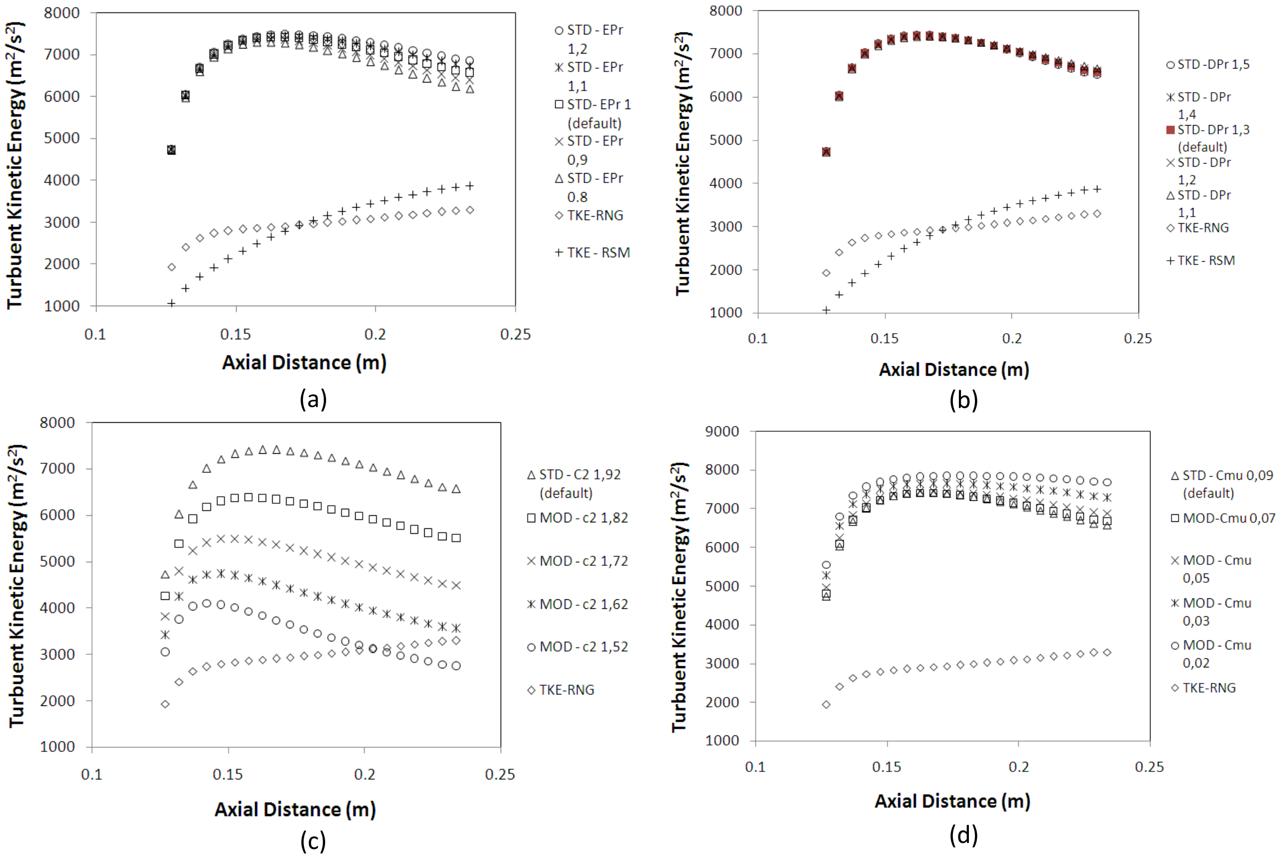
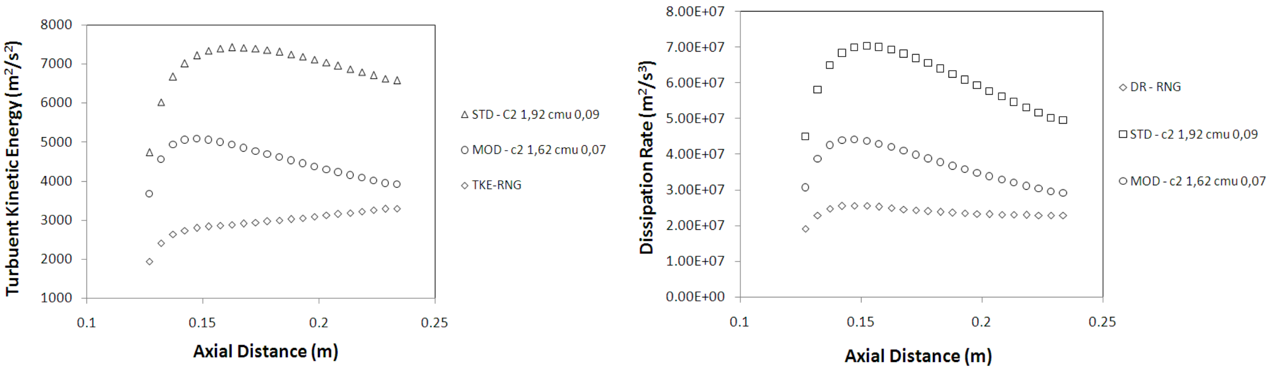


Fig. 8. Analysis of turbulent kinetic energy for some modified turbulent constants (a) kinetic energy, σ_k ; (b) for dissipation rate, σ_ε ; (c) c_2 ; and (d) c_μ

By applying the modified constants c_2 and c_μ by the values of 1.62 and 0.07 successively into the turbulence parameters in CFD modelling and without changing values for the default parameter of σ_k and σ_ε , the result of turbulent kinetic energy and dissipation rate tends to the better prediction with the RNG k-ε. It was shown in Figure 9 that the turbulent kinetic energy decrease to approximately 5E03 m^2/s^2 while the dissipation rate to 4E07 m^2/s^3, where RNG k-ε predicts about 3E03 m^2/s^2 and 2.5E07 m^2/s^3, respectively.

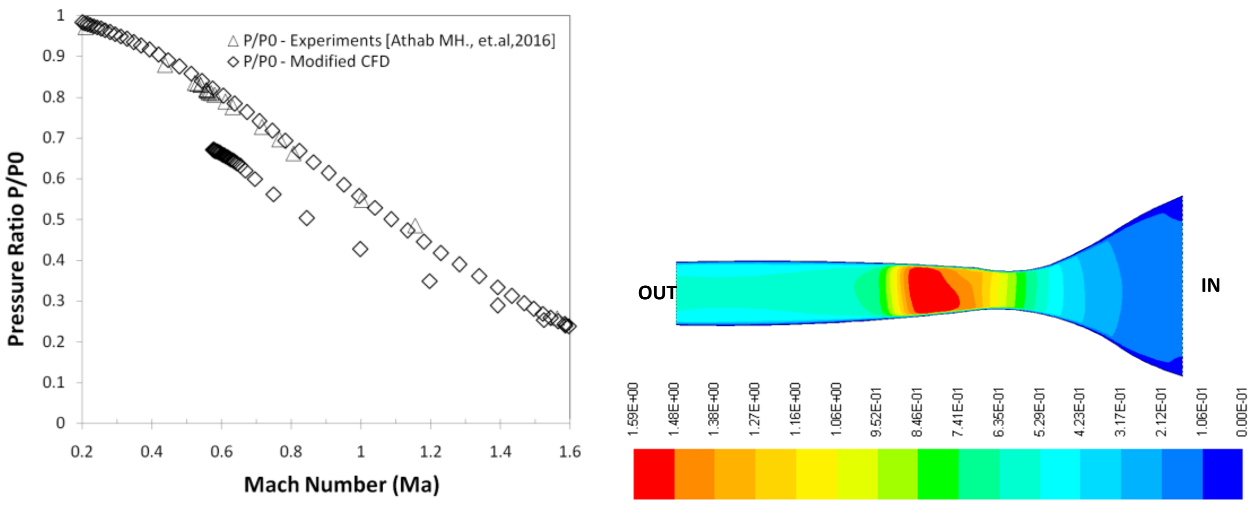


(a) (b)

Fig. 9. Result of modified turbulent model constant c_2 and c_μ (a) for turbulent kinetic energy (b) for dissipation rate

3.3 Validation

The modified turbulence model was validated with the experimental results of steam CD nozzle as had been reported in literature. The 17 static pressure taps were placed along the nozzle and were used to calculate the Mach number [29]. The results of measurement were normalized by pressure and length of nozzle. These experimental results were used to validate the modified turbulence model by comparing the pressure ratio with the Ma from simulation (Figure 10).



(a) (b)

Fig. 10. (a) Validation of pressure ratio at various mach number and (b) velocity contour of cd nozzle model

Figure 10 (a) shows that the pressure ratio of modified Standard k-ε turbulence model has the quite similarity with the experimental results from literature [29]. The modified turbulence model of

Standard k-ε gives a good prediction for pressure ratio at various Mach number. At some regions within the CD nozzle, the geometry is not precise symmetrical shape. Consequently, at certain position, the fluid velocities are different between upper and lower sides from centreline, as is shown in Figure 10 (b).

4. Conclusions

This simulation was conducted in two-dimensional model, steady flow. In addition, impermeable-adiabatic wall and an ideal fluid were assumed in this study. The result of this model was compared with the prediction using the RNG k-ε and RSM turbulence models. All the turbulence models showed the similar prediction of time-average of pressure, velocity, temperature, and density. However, it was found that the standard k-ε model over predicted for the turbulence parameters namely turbulent kinetic energy and dissipation rate. With new modified constants of c_2 for dissipation equation and c_μ for eddy viscosity model, the model could give a better prediction than the original constant of the k-ε turbulence model.

Acknowledgement

The authors would like to thanks, DRPM Universitas Indonesia for funding this research through “Hibah Riset Q1Q2 UI 2019” [NKB-0284/UN2.R3.1/HKP.05.00/2019] and to PT. CCIT Group Indonesia for CFD Package

References

[1] Balabel, A., A. M. Hegab, M. Nasr, and Samy M. El-Behery. "Assessment of turbulence modeling for gas flow in two-dimensional convergent–divergent rocket nozzle." *Applied Mathematical Modelling* 35, no. 7 (2011): 3408-3422. https://doi.org/10.1016/j.apm.2011.01.013.

[2] Rao, P. Srinivasa, C. Moorthy, and V. Srinivas. "Turbulence Modeling and Numerical Analysis for the Configuration of a Supersonic Air Ejector." *International Journal of Mechanical Engineering and Technology* 8, no. 10 (2017): 130-139.

[3] Sellam, Mohamed, Guillaume Fournier, Amer Chpoun, and Ph Reijasse. "Numerical investigation of overexpanded nozzle flows." *Shock Waves* 24, no. 1 (2014): 33-39. https://doi.org/10.1007/s00193-013-0458-3.

[4] Koten, Hasan, Mustafa Yilmaz, and M. Zafer Gul. "Compressed biogas-diesel dual-fuel engine optimization study for ultralow emission." *Advances in Mechanical Engineering* 6 (2014): 571063. https://doi.org/10.1155/2014/571063.

[5] Chu, Julio, and James M. Luckring. *Experimental surface pressure data obtained on 65 delta wing across Reynolds number and Mach number ranges*. National Aeronautics and Space Administration, Langley Research Center, 1996.

[6] Swaroopini, A. Shanthi, M. Ganesh Kumar, and T. Naveen Kumar. "Numerical Simulation and Optimization of High Performance Supersonic Nozzle at Different Conical Angles." *International Journal of Research in Engineering and Technology (IJRET)* 4, no. 09 (2015): 268–273. https://doi.org/10.15623/ijret.2015.0409049

[7] Ayeleso, Ayokunle O., and Mohamed TE Kahn. "Modelling of a combustible ionised gas in thermal power plants using MHD conversion system in South Africa." *Journal of King Saud University-Science* 30, no. 3 (2018): 367-374. https://doi.org/10.1016/j.jksus.2017.01.007.

[8] Mummidisetti, Karthik, Sampath Emani, Gurunadh Velidi, and Ugur Guven. "Investigation of Efficient Turbulence Model for Two-Dimensional Nozzle Designed for Supersonic Cruise Using STAR-CCM." (2013).

[9] Fharukh, Ahmed GM, Abdulrehman A. Alrobaian, Abdul Aabid, and Sher Afghan Khan. "Numerical analysis of convergent-divergent nozzle using finite element method." *International Journal of Mechanical and Production Engineering Research and Development* 8, no. 6 (2018): 373-382. https://doi.org/10.24247/ijmperddec201842

[10] Prafulla, Ms B. Krishna, Dr V. Chitti Babu, and Sri P. Govinda Rao. "CFD analysis of convergent-divergent supersonic nozzle." *International Journal of computational engineering research* 3, no. 5 (2013): 5-16.

[11] Kostić, Olivera, Zoran Stefanović, and Ivan Kostić. "CFD modeling of supersonic airflow generated by 2D nozzle with

and without an obstacle at the exit section." *FME Transactions* 43, no. 2 (2015): 107-113. https://doi.org/10.5937/fmet1502107k

[12] Hossain, M. S., Muhammad Ferdous Raiyan, and Nahed Hassan Jony. "Comparative study of supersonic nozzles." *International Journal of Research in Engineering and Technology* 3, no. 10 (2014): 351-357. https://doi.org/10.15623/ijret.2014.0310056

[13] Kumar, Uttam, Sudhir Singh Rajput, and Praveen Borkar. "CFD analysis and parameter optimization of Divergent Convergent Nozzle." *International Journal for Advance Research and Development* 3, no. 10 (2018): 9-13.

[14] Koli, Jyoti Kumari1 CS. "Modelling and Simulation of Supersonic Nozzle using ANSYS Workbench with New Measuring Data." *International Journal for Scientific Research & Development 4*, no. 11 (2017): 553–556.

[15] Venkatesh, V., and C. Jayapal Reddy. "Modelling and simulation of supersonic nozzle using computational fluid dynamics." *International Journal of Novel Research in Interdisciplinary Studies* 2, no. 6 (2015): 16-27..

[16] Ramji, V., Raju Mukesh, and Inamul Hasan. "Design and numerical simulation of convergent divergent nozzle." In *Applied Mechanics and Materials*, vol. 852, pp. 617-624. Trans Tech Publications Ltd, 2016. https://doi.org/10.4028/www.scientific.net/AMM.852.617.

[17] KiranKopelli, John Raja Ravi, M. Sudhakar, and G. Babu Rao. "Comparative cfd analysis of three flow viscous models for air flowing at high speed through a convergent-divergent nozzle." *Journal of Technological Advances and Scientific Research* 1, no. 3 (2015): 147-153.

[18] Najar, Nadeem Akbar, D. Dandotiya, and Farooq Ahmad Najar. "Comparative analysis of k-ε and spalart-allmaras turbulence models for compressible flow through a convergent-divergent nozzle." *The International Journal of Engineering and Science* 2, no. 8 (2013): 8-17.

[19] Kostić, Olivera, Zoran Stefanović, and Ivan Kostić. "CFD modeling of supersonic airflow generated by 2D nozzle with and without an obstacle at the exit section." *FME Transactions* 43, no. 2 (2015): 107-113. https://doi.org/10.5937/fmet1502107k

[20] Akhil, R., and Vishnu CA Ashik Shah. "Comparison of nozzle pressure ratios." *International Research Journal of Engineering and Technology* 4, no. 7 (2017): 641–644.

[21] Yang, Yaxiong, Eri Qitai, Qiang Wang, and Xijuan Zhu. "Development of Two-Dimensional Convergent-Divergent Nozzle Performance Rapidly Analysis Project." In *2015 International Forum on Energy, Environment Science and Materials*. Atlantis Press, 2015. https://doi.org/10.2991/ifeesm-15.2015.162.

[22] Ramdlan, G. Gun Gun, Ahmad Indra Siswantara, Asyari Daryus, and Hariyotejo Pujowidodo. "Turbulence model and validation of air flow in wind tunnel." *International Journal of Technology* 7, no. 8 (2016): 1362-1371. https://doi.org/10.14716/ijtech.v7i8.6891

[23] Pujowidodo, Hariyotejo, Ahmad Indra Siswantara, Budiarso, Gun Gun R. Gunadi, and Asyari Daryus. "Turbulence model and validation of air flow in crossflow turbine nozzle." In *AIP Conference Proceedings*, vol. 2001, no. 1, p. 060006. AIP Publishing LLC, 2018. https://doi.org/10.1063/1.5050007.

[24] Koten, Hasan, Mustafa Yilmaz, and M. Zafer Gul. "Compressed biogas-diesel dual-fuel engine optimization study for ultralow emission." *Advances in Mechanical Engineering* 6 (2014): 571063. https://doi.org/10.1155/2014/571063.

[25] Saqr, Khalid M., Hossam S. Aly, Mazlan A. Wahid, and Mohsin M. Sies. "Numerical simulation of confined vortex flow using a modified k-epsilon turbulence model." *CFD letters* 1, no. 2 (2009): 87-94.

[26] Aziz, T. N., J. P. Raiford, and A. A. Khan. "Numerical simulation of turbulent jets." *Engineering applications of computational fluid mechanics* 2, no. 2 (2008): 234-243. https://doi.org/10.1080/19942060.2008.11015224.

[27] Mishra, Pankaj, and K. R. Aharwal. "A review on selection of turbulence model for CFD analysis of air flow within a cold storage." In *IOP Conf. Ser. Mater. Sci. Eng*, vol. 402, pp. 012145-012153. 2018. https://doi:10.1088/1757-899X/402/1/012145.

[28] Hassen, Nasir Salim, and Nor Azwadi Che Sidik. "Laboratory investigation of nozzle type, size and pressure effects on spray distribution." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 61, no.1 (2019): 140-146.

[29] Athab M.H., Al-Taie A., Mashi H.W. "Experimental Study of Steam Flowing a Convergent-Divergent Nozzle." *International Journal of Novel Reserach in Electrical and Mechanical Engineering* 3, no.3 (2016): 7-13.