Turbulent modelling analysis of saturated steam flow in curved convergent divergent nozzle

H Pujowidodo1*, A I Siswantara2, A Daryus3 and G G R Gunadi4

1 Center for Thermodynamics, Engine, and Propulsion, BPP Teknologi, Serpong 15314, Indonesia
2 Department of Mechanical Engineering, Universitas Indonesia, Depok 16424, Indonesia
3 Mechanical Engineering Department, University of Darma Persada, Jakarta, Indonesia
4 Mechanical Engineering Department, State University of Jakarta, Depok 16424, Indonesia

*hariyotejo.pujowidodo@bppt.go.id

Abstract. The objective of this study is to analyze turbulent flow characteristics of turbulence modeling in Steam Curved Converging Diverging Nozzle. The three turbulence models have been conducted to simulate flow characteristics using k-ε standard, k-ε Renormalization Group (RNG), and Reynolds Stress Model (RSM). Fluid dynamics parameters such as pressure, velocity, and/or density have been analyzed graphically to know the more suitable model. The k- k- ε STD and RNG show give the similar profile of the turbulent kinetic energy (k) and energy dissipation rate (ε) that is differ with the RSM model. After throat position (position 0.67 m) the k-k-ε STD and RNG give the magnitude more than 1200 (m²/s²) for its turbulent kinetic energy and more than 3.2E06 (m²/s³) for its energy dissipation rate. The RSM model give the lowest values which far below 200 (m²/s²) for turbulent kinetic energy and below 2E05 (m²/s³) for energy dissipation rate. The turbulence modeling analysis of steam saturated flow in plane curved nozzle present the increasing steam velocity about 3 Ma for 200 kPa steam inlet.

1. Introduction
Convergent Divergent nozzle has the geometry shape which is the converging section joined to the diverging section, to increase the velocity to supersonic flow. This nozzle many applied for compressible fluid, which has the potential energy of pressure and heat, for momentum improvement such as in turbine power as well in vacuum pumping to eject other evacuated substances. The fundamental principle of this device is converting the potential flow and thermal energies into kinetic energy. The compressibility of the fluid has the important role for this energy conversion and throat section (joint area between converging and diverging sections) as well. The throat section must attain sonic condition to increase the flow to higher Mach number. Geometry and operating parameters (pressure and temperature) are the two important factors that influence the nozzle performance.

Turbulence is the flow regime which has the high Reynolds number. It is shown by the correlation of inertial force to viscous force. Thus, for turbulence condition, it could be understood that the inertial
force (quantity of flow) has high gradient velocity. The flow in turbulence is irregular, fluctuating, and high vortices.

Study of supersonic nozzle many carried out by researchers to improve and investigate the internal flow specially to know the deep impact of geometry and flow parameters on nozzle performance. The design of geometry properly had been studied for improving thrust of rocket nozzle from combustion gas enthalpy conversion [1]. The study of simulation by CFD also had been done by analyzing meshing upon the contours of pressure, velocity, Mach number, cell Reynolds number and cell skew in rocket nozzle modeling [2]. The fluid properties such as pressure, temperature, and velocity are strongly dependence on cross sectional area and expansion condition in convergent-divergent nozzle [3]. The optimization of conical supersonic nozzle for 3 Ma, was modeled by CFD in 2D by varying the difference of divergence angle [4]. The back pressure and area ratio also studied by CFD modeling to optimize the Mach [6]. Investigation steam jet ejector dealing with geometry and inlet condition had been modeled by CFD to analyze the mixing process of evacuated substance [7,8].

In this paper would be presented the study of turbulence modeling on the saturated steam supersonic nozzle with the plane-curved geometry. The aim of this study is to obtain the comparative modeling using $k-\varepsilon$ standard, $k-\varepsilon$ Renormalization Group (RNG), and Reynolds Stress Model (RSM). The flow properties like pressure, velocity, and density, and turbulence parameters such as turbulent kinetic energy ($k$) and energy dissipation rate ($\varepsilon$) would be analyzed to get the result.

2. Materials and methods

By statistic data, the turbulent flow has the fluctuating values which always change with space and time. In flow field, there obeys the principle of conservation of mass, momentum, and energy. The analyze of flow upon the finite volume gives the conservation that the volume change rate (of properties $\Phi$) is equal to net flux $J$ by convective, by diffusive, and net rate of production.

\[ \frac{\partial \rho \Phi}{\partial t} = \text{div} \cdot J \] (1)

Generally using tensor notation, the mass balance and force balance for continuity and momentum conservation also could be written in equation form as follows.

\[ \frac{\partial u_i}{\partial x_i} = 0 \] (2)

Where $u_i$ is component vectors $u, v, w$ in direction $x_i$, that is $x, y, z$, and $i$ for index 1, 2, 3. The force balance could be derived from the momentum conservation in Navier-Stokes equation, which states that
the rate change of force and by convection inside the control volume is the sum of forces acting on the surface and body force \( F_i \).

\[
\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_i u_j) = \frac{1}{\rho} \frac{\partial P_i}{\partial x_j} + \frac{\partial}{\partial x_j} (\rho \frac{\partial}{\partial x_j} u_i) - \frac{1}{\rho} \frac{\partial}{\partial x_j} (\frac{\partial}{\partial x_j} u_i)
\]

(3)

The velocity in turbulence flow is the decomposition of average and fluctuation components, which using the averaging Reynolds it is obtained that there is a closure term in order to solve the momentum equation. This equation is known as the Reynolds Averaging Navier Stokes (RANS) and the closure term named by Reynolds Stress (Rij).

\[
U_j \frac{\partial}{\partial x_j} (U_i) = - \frac{1}{\rho} \frac{\partial P_i}{\partial x_j} + \frac{\partial}{\partial x_j} (\mu \frac{\partial}{\partial x_j} U_i) + \frac{\partial (R_{ij})}{\partial x_j}
\]

(4)

Where \( R_{ij} = -u'_i u'_j \) with \( u'_i = 2 \frac{\mu}{\rho} S_{ij} \) and \( \mu, \rho \) is dynamic viscosity and density of fluid, and \( \mu_i \) is eddy viscosity determined by \( 0.09 \rho \kappa / \epsilon \). While \( k = \frac{1}{2} u'_i u'_j \) and in this equation the buoyancy neglected.

Some of turbulence modeling for solving RANS equation are standard \( k-\epsilon \) (STD \( k-\epsilon \)), Renormalization Group \( k-\epsilon \) (RNG \( k-\epsilon \)), and Reynolds Stress Model (RSM). The distinction of these models is the number of equations to close the momentum equation for turbulent flow.

Energy kinetic turbulence \( (k) \) and energy dissipation rate \( \epsilon \) use two equations model while the RSM using 6 equations.

For STD \( k-\epsilon \) model:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + 2 \mu_i S_{ij} S_{ij} - \rho \epsilon
\]

(5a)

\[
\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + c_{1\epsilon} \frac{\epsilon}{k} 2 \mu_i S_{ij} S_{ij} - c_{2\epsilon} \rho \frac{\epsilon^2}{k}
\]

(5b)

\( \sigma_k = 1.00; \ C_{1\epsilon} = 1.44; \ C_{1\epsilon} = 1.92 \) are model constants.

While for RNG \( k-\epsilon \) model is written as equation below:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_i}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + P_k - \rho \epsilon
\]

(6a)

\[
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_j} (\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_i}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} + c_{1\epsilon} \frac{\epsilon}{k} P_k - c_{2\epsilon} \rho \frac{\epsilon^2}{k}
\]

(6b)

Where \( P_k \) is production of turbulent kinetic energy, calculated by:
In this equation: \( \tau_{ij} = -\rho u_i u_j \) (which has 6 components of \( u_i^2, u_i^2, u_i^2, u_i u_j, u_i u_j, u_i u_j \)).

Supersonic converging-diverging nozzle thermodynamically determined by fluid incompressibility where depending on cross sectional area (geometry) and stagnation pressure, temperature as well (flow operational parameters). The specific heat ratio (k) and Mach number (Ma) would drive the properties ratios for pressure, temperature, and density.

\[
Ma = \frac{v}{c} = \frac{v}{\sqrt{kRT}}
\]  

where: \( v = \) supersonic speed (m/s); \( c = \) speed of sound (m/s); \( k = c_p/c_v \) (\( c_p, c_v = \) specific heat capacity at constant pressure and constant volume (kJ/kg. K)), \( R = \) universal gas constant= 0.4615 (kJ/kg. K), \( T = \) temperature (°K)

\[
\frac{P_0}{P} = \left[ 1 + \left( \frac{k - 1}{2} \right) Ma^2 \right]^{\frac{1}{k-1}}
\]

\[
\frac{T_0}{T} = \left[ 1 + \left( \frac{k - 1}{2} \right) Ma^2 \right]
\]

\[
\frac{\rho_0}{\rho} = \left[ 1 + \left( \frac{k - 1}{2} \right) Ma^2 \right]^{\frac{1}{k-1}}
\]

The convergent-divergent nozzle, which is modeled, has the inlet and outlet from the left and right sides in 2-dimensional domain, and planar and curved forms on the top and bottom sides. The fluid flowing supersonic nozzle is the saturated steam with temperature 120 °C(200 kPa.G).
Figure 2. Steam saturated convergent-divergent plane-curved nozzle.

The steam in saturated condition flows through the convergent-divergent plane-curved nozzle will get the supersonic speed as flowing pass expansion and sonic at throat areas. The flow properties such as pressure, temperature, and density will change along nozzle length.

Figure 3. Methodology of turbulent modeling analysis.

Discretization in finite volume for numerical computation is done for 3 mesh sizing and will be tested for grid independency test. The verification for model carried out by comparing the supersonic velocity and flow properties.

The following picture presents the methodology of turbulent modeling analysis for steam saturated convergent-divergent plane-curved nozzle.
3. Results and discussion

3.1. Grid and analytical verification

The model has been discretized using Body Fitted Coordinates from CFD-Pre® for 3 meshing sizes, which are 34x8; 76x 16; and 131x28. Meanwhile for thermodynamic calculation of compressible fluid supersonic flow, has been plotted in the profiles of pressure, temperature, and density ratios from EES™. The contour distribution for vacuum pressure is chosen to verify the grid independence.

![Figure 4. BFC’s grid for 34x8 cells.](image)

![Figure 5. Distribution of pressure below 1 bar for grid size (a) 34x8; (b) 76x16; and (c) 131x28.](image)

From figure 4 the contour of vacuum pressure inside the plane-curved nozzle depicts similar distribution, thus for simulation analysis of turbulent modelling is used the grid dimension 34x8 cells. The pressure distribution leaving from the throat section gradually changes into more vacuum where flow energy converted to kinetic energy.
Figure 6. Properties ratio of pressure, temperature, and density by analytical calculation (solved by EES™).

Figure 7. Profile Distribution of pressure along axial distance (Pa) (plotted by CFD-SOF™).

Table 1. Comparison between analysis and simulation.

| Verified Parameters          | Analysis | Simulation | % Bias  |
|-----------------------------|----------|------------|---------|
| Back Pressure (kPa)         | 3        | 3.1977     | 0.0659  |
| Downstream temperature (°K) | 149.2    | 154        | 0.032   |
| Ma                          | 3.302    | 3.267      | -0.0106 |

In Figure 6 and 7, the distribution of pressure along axial distance gives the similar trend. As the consequence of descending and ascending the cross-sectional area, when fluid choked at throat thus the density will change and the velocity also will increase. This profile also has the same profile with the trend obtained by other cited study [6].

The result of verification given in table 1, the difference between simulation and analysis is not higher than 1% and the simulation ideally could present the good prediction.

3.2. Flow contour distribution

For analysis the flow passing through plane-curved supersonic nozzle, it is used the contour distribution to get more detail about the information its characteristics. It 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) are going to be analyzed for flow throughout the model. These parameters are the main variables which determining nozzle design considerations. Principally the thermodynamics parameters those are pressure and temperature would affect the density, and then flow and thermal energy, driven by pressure and temperature, converting to kinetic energy will increase the velocity. Its density change also affects the supersonic velocity. All of these considerations mainly depend on the geometry factor.

![Contour Distribution of flow parameters](image)

Figure 8. Contour Distribution of flow parameters (a) relative Static Pressure (Pa) (b) Static Temperature (°K) (c) Density (kg/m³) (d) Ma number (solved by CFD-SOF™).

The narrowing and expanding the cross sectional area of flow will cause the pressure and temperature decreasing, resulting the density also minimizing and the flow going to be supersonic as the flow is in sonic condition at throat section [3][4].

| Flow Parameters                  | k-ε STD        | RNG           | RSM           |
|----------------------------------|----------------|---------------|---------------|
| Relative Static Pressure (Pa)    | -9.7052E+04    | -9.7171E+04   | -9.7161E+04   |
| Downstream temperature (°K)      | 7.891E+01      | 7.253E+01     | 7.7339E+01    |
| Density (kg/m³)                  | 1.1758E-01     | 1.1701E-01    | 1.1714E-01    |
| Velocity (Ma)                    | 2.9452E+00     | 2.9790E+00    | 2.9811E+00    |

Table 2. Flow parameters at outlet area for steam saturated inlet 200 kPa (120 °C).

Refers to Table 2 above, it could be found that the flow parameters resulting from 3 turbulent model show the small difference. The two turbulent models of RNG and RSM give the closer result for relative
static pressure, density, and velocity. For downstream temperature, the k-ε STD and RSM have the slight difference, comparing to the RNG.

3.2.2. Turbulence parameters. The closure term in turbulent modeling would be found in the simulation result in the parameters of turbulent kinetic energy (k) and energy dissipation rate (ε). 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 9](image_url)

**Figure 9.** Contour distribution of turbulence parameters (a) Turb. KE k-ε (m²/s²) (b) Turb. KE RNG (m²/s²) (c) Turb. KE RSM (m²/s²) (d) Dissip. Rate k-ε (m²/s³) (e) Dissip. Rate RNG (m²/s³) (f) Dissip. Rate RSM (m²/s³) (solved by CFD-SOF™).
Figure 9 above; give the turbulence contour of 3 models, and shows that k-ε STD and RNG presenting the distribution inside downstream area the spread pattern while for RSM there is the contrast area lying on the bottom side. Turbulent intensity given by the its kinetic energy would be higher after passing through the throat because there is a sharp gradient of the diverging section [4].

The dissipation rate distribution of RNG and RSM show the similar location, which on the bottom side of downstream area there is higher dissipation into thermal heat energy, while from k-ε STD model, the dissipation more spreader at position near the outlet region.

![Graph](image1.png)

**Figure 10.** Turbulence parameters at center of throat along nozzle length (a) Turbulent KE (k) and (b) Dissipation Rate (ε).

As could be seen in figure 8, there are the curve of turbulence parameters along nozzle length at throat center. The k-ε STD and RNG show the same curve form, and differ with the RSM model. In this figure, it also could be observed that after throat position (position 0.67 m) the k-k-ε STD and RNG give the magnitude more than 1200 (m²/s²) for its turbulent kinetic energy and more than 3.2E06 (m²/s³) for its energy dissipation rate. The RSM model give the lowest values which far below 200 (m²/s²) for turbulent kinetic energy and below 2E05 (m²/s³) for energy dissipation rate.
4. Conclusion
The turbulence modeling analysis of steam saturated flow in plane-curved nozzle present the increasing steam velocity about 3 Ma for 200 kPa steam inlet. This supersonic flow occurred by the converting flow and thermal energies into kinetic energy in gradual cross section area and sonic condition at throat section area. The k- k-ε STD and RNG show give the similar profile of the turbulent kinetic energy (k) and energy dissipation rate (ε) that is differ with the RSM model.

Acknowledgements
The authors would like to thanks, DRPM Universitas Indonesia for funding this research through “Hibah Publikasi Internasional Terindeks untuk Tugas Akhir Mahasiswa UI 2018” with contract number 2367/UN2.R31/HKP.05.00/2018, and to PT. CCIT Group Indonesia for CFD Package and EES software license.

References
[1] Belega B A and Nguyen T D editors 2015 Proceedings of International Conference of Scientific Paper AFASES Brasov, Romania
[2] Narayana K P S S and Reddy K S 2016 Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis 58-65
[3] Satyanarayana G 2013 Acta Technica Corviniensis-Bulletin of Engineering 6 3 139
[4] Pandey K and Singh A 2010 International Journal of Chemical Engineering and Applications 1 2 179
[5] Patankar S 1980 Numerical heat transfer and fluid flow (CRC press)
[6] Shariatzadeh O J 2015 J Clean Energy Technol 3 3 pp 220-225
[7] Singhal A, Chitkara T K and Ameenuddin M 2013 National Convention of Aerospace Engineers (27th NCAE) (Dehradun, India)
[8] Versteeg H and Malalasekera W 1995 Computational fluid dynamics The finite volume method