RESEARCH ARTICLE | JANUARY 25 2019

The analytic and CFD modeling studies of saturated steam mass flow in curved convergent divergent nozzle FREE

Hariyotejo Pujowidodo; Ahmad Indra Siswantara; Asyari Daryus; Gun Gun Ramdlan Gunadi

AIP Conf. Proc. 2062, 020015 (2019)
https://doi.org/10.1063/1.5086562

The Analytic and CFD Modeling Studies of Saturated Steam Mass Flow in Curved Convergent Divergent Nozzle

Hariyotejo Pujowidodo[1,a)], Ahmad Indra Siswantara[2,b)],Asyari Daryus[3,c)], Gun Gun Ramdlan Gunadi[4,d)]

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

[a)]Corresponding author: hariyotejo.pujowidodo@bppt.go.id
[b)]a_indra@eng,ui,ac,id
[c)]asyari@yahoo.com
[d)]gungun.rg@mesin.pnj.ac.id

Abstract. The study of steam mass flow in a Curved Converging Diverging Nozzle has been analyzed using analytic and CFD modeling. The objective of this work is to get the comparison between the results of flow properties and characteristics from solution using thermodynamics analysis and turbulent model simulation. Using the post analysis of CFD simulation by integrating of the velocity at a cross flow section and the calculation supersonic flowing in the gradual area by *EES*® model, would be used to get the mass flow of steam, The simulation results give the good acceptance with the bias less than 7% compared to analytical. Saturated steam with stagnation pressure 200 kPa. and downstream area 71 cm^2 with throat area 16 cm^2, analytically and computationally could achieve the Ma about 3. At chocked condition, the maximum mass flow has the magnitude less than 0.016 kg/s, with the critical properties are 109.1 kPa; 341.9 ^{o}K; and 0.4092 kg.m^3. The calculated mass flow has the magnitude 1.55 kg/s, meanwhile from simulation gives the mass flux magnitude 1.22 kg/s.

INTRODUCTION

Converging diverging nozzle has the gradually geometry cross section area either in curved or planed shape. The main feature of this device is the capability to increase compressible fluid flow in supersonic velocity. Various utilization of this component for converting enthalpy energy into kinetic energy, hence many utilized in mechanic power turbine, thrust power for rocket, as well in vacuum mechanism using motive fluid as fluid ejector. The performance of this nozzle to give the suitable supersonic flow strongly depends upon the change flow properties and characteristic as passing through the throat section (interconnection region between convergent and divergent section). Compressible fluid has the characteristic of compressibility, that is the magnitude of specific volume change when the pressure changes. Geometry and flow parameters play significant role in supersonic fluid nozzle, like cross section areas of throat and exit, stagnation pressure at inlet side and back pressure at exit one. Ideally fluid process in nozzle is isentropic, where the friction is very small and usually negligible, and the analysis using the ideal gas and isentropically expansion equations.

Many studies on supersonic converging-diverging nozzle had been done by researchers to investigate the flow characteristics and properties to improve the performance. When the pressure ratio, referring to at throat section, is the same with critical ratio, thus the flow would be in sonic. If the ratio more lower than its critical, the flow velocity more supersonic. Conversely if the ratio is higher than critical ratio, the flow is subsonic. The temperature and

The 10th International Meeting of Advances in Thermofluids (IMAT 2018)
AIP Conf. Proc. 2062, 020015-1–020015-8; https://doi.org/10.1063/1.5086562
Published by AIP Publishing. 978-0-7354-1790-8/$30.00

density also would change follow the pressure with the gas ideal correlation and determined with the k constant as the specific heat ratio.

Several studies conducted to improve the mass flow and velocity to obtain the high momentum flux, as well the flow properties change affecting them. Kuttan B. and Sajesh M. [1] investigated the optimization divergent angle of Rocket nozzle, to increase the Mach number using 2D CFD model with the various Ma number, static pressure, and turbulent intensity. Dhingra M.et. al [2] analyzed flow behaviour through converging diverging nozzle and plotted the pressure ratio between axial length for different chamber pressure. Factor expansion in diverging section had been the concern of investigation by Kundu B. and Lee KS. [3] and compared to non isentropic flow condition. The effect of cavitation water jet analyzed by Yu LY., et.al (4), and found the bubble phenomena as the indication factor in turbulent model simulation of CFD. Noh MHM., et.al. [5] investigated numerically the chocked converging diverging nozzle of thrust application with the variables of changing geometry to analyze the flow characteristics and performance, and the results showed the separation flow at divergence angle of 24^{o}, 14^{o}, and 10^{o} , where normal shock occured and degraded performance. The study pressure forces from venturi nozzle also done by Olaru I. [6] in the application of metal cutting cooling system . Patel KS. [7] used CFD model of supersonic rocket engine nozzle with boundary condition parameters like velocity, static pressure, turbulent intensity, and temperature, to analyze divergent angle and oblique shock phenomena. The fluid properties like pressure, velocity, temperature, as dependent variables on cross section, simulated computationally by Rao AVR. and Akhil L. [8] at region velocity of sub sonic, sonic, and supersonic. Concerning with the flow behavior also studied using CFD to analyze parameters such as static pressure, temperature, and velocity by Rai R., et.al. [9,10], as well compared analytically. Reji AK., et.al. [10] designed the supersonic convergent divergent nozzle for cold spraying and analyzed using CFD, as well validated where the geometry or shaped had been a main consideration.

The internal fluid properties and characteristics in curved converging-diverging nozzle would be studied analytically and computationally in 2D model. The properties change of saturated steam, occurred at gradually cross section area and it characteristics is analyzed to obtain the distribution of parameters such as pressure, Mach number (Ma), temperature, thereby could be known the verification from simulation model.

MATERIAL AND METHODS

This study of saturated steam flow in curved converging-diverging nozzle has the input parameters such as geometry and size of nozzle those are at inlet, throat, and outlet cross sections, steam pressure known as stagnation pressure, specific heat ratio (k) and tools calculation using Engineering Equation Solver (EES™). The calculation and simulation models assumed the flow is steady, isentropic process; steam behaves ideal gas, and 2 dimensional turbulent modeling.

Analytical Model

Nozzle has the function to convert thermal or flow energy into kinetic energy through the gradually smaller cross section area. Pressure and temperature are the main variables in obtaining the fast velocity, comply the ideal gas the density is the function of its parameters.

$$P = \rho.R.T \tag{1}$$

where : P is fluid pressure, ρ is density , R is universal gas constant, and T is temperature.

Meanwhile the speed of sound could be found using the temperature.

$$c = \sqrt{k.R.T} \tag{2}$$

So the Mach number is the ratio of gas velocity (v) to speed of sound.

$$Ma = \frac{v}{c} = \frac{v}{\sqrt{k.R.T}} \tag{3}$$

From isentropic process, the flow properties is determined using the specific heat ratio (k, for steam k=1.3) exponent in isentropic expansion equation. For compressible fluid flows in nozzle, this properties ratio defined using the equation divided with that stagnation parameters (subscript-0, indicates the zero velocity).

$$\frac{P_0}{P_{ds}} = \left[1 + \left(\frac{k-1}{2}\right).Ma^2\right]^{k/(k-1)} \tag{4}$$

$$\frac{T_0}{T_{ds}} = \left[1 + \left(\frac{k-1}{2}\right).Ma^2\right] \tag{5}$$

$$\frac{\rho_0}{\rho_{ds}} = \left[1 + \left(\frac{k-1}{2}\right).Ma^2\right]^{1/(k-1)} \tag{6}$$

where subscript-ds is downstream side.

The chocked condition at throat area occurred at Ma=1, therefore the properties and geometry could be found by input it in above equations, and indicated by using the subscript-*, as well, the downstream area calculated using Ma variable. For steady state condition, the mass flow could be determined using the area, pressure, Ma, and temperature suing the critical correlation.

$$\frac{P^*}{P_0} = \left(\frac{2}{k+1}\right)^{k/(k-1)} \tag{7}$$

$$\frac{T^*}{T_0} = \left(\frac{2}{k+1}\right) \tag{8}$$

$$\frac{\rho^*}{\rho_0} = \left(\frac{2}{k+1}\right)^{1/(k-1)} \tag{9}$$

$$\frac{A_{ds}}{A^*} = \frac{1}{Ma}\left[\left(\frac{2}{k+1}\right)\left(1 + \frac{k-1}{2}\right)\right]^{(k+1)/[2(k-1)]} \tag{10}$$

$$\dot{m} = \frac{A.Ma.P_0\sqrt{\frac{k}{R.T_0}}}{[1 + (k-1).Ma^2/2]^{(k+1)/[2(k-1)]}} \tag{11}$$

Computational Model

The built model is 2D, steady flow, and adiabatic using the CFDPRE© and CFDSOF© as CFD tools. Firstly geometry and meshing defined to make computational domain, cell discretization, and give boundary conditions for inlet, wall, and exit side. Solving and displaying the simulation result to obtain the contour distribution of pressure, temperature, density, and Ma where as the working fluid is saturated steam 200 kPa and the geometry as shown in figure-1. Discretization for numerical cells using the Body Fitted Coordinate (BFC) and k-ε turbulent model.

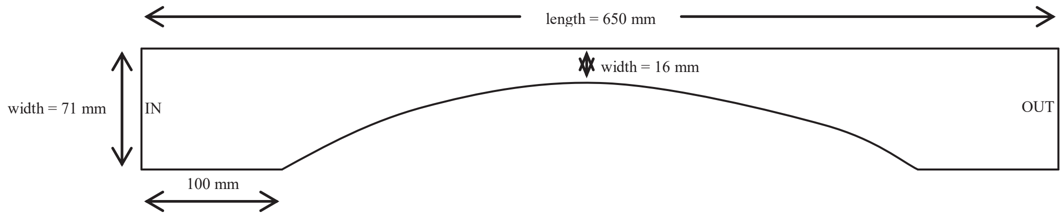


FIGURE 1. Supersonic Converging-Diverging (CD) Nozzle – Geometry 2D Model

RESULT AND DISCUSSION

The calculation results are presented in the graph of the flow properties change, and the simulation one given in the contour distribution of pressure, Ma, and density. Each data for calculation and simulation taken from the same data as previously provided such as stagnation pressure 200 kPa, saturated steam, steady flow, and isentropic process. Verification of simulation results comparing with analytical is done at chocked condition.

Compressible Fluid Flow Analysis of CD-Nozzle Model

The change of flow properties and its critical ratio are based on the trend exponentially, as described in the equation. Ma determined by the back pressure, and will affect the temperature, as well density. The higher Ma, the properties will change inversely by exponential, with the same manner the downstream cross section also changes positive exponentially.

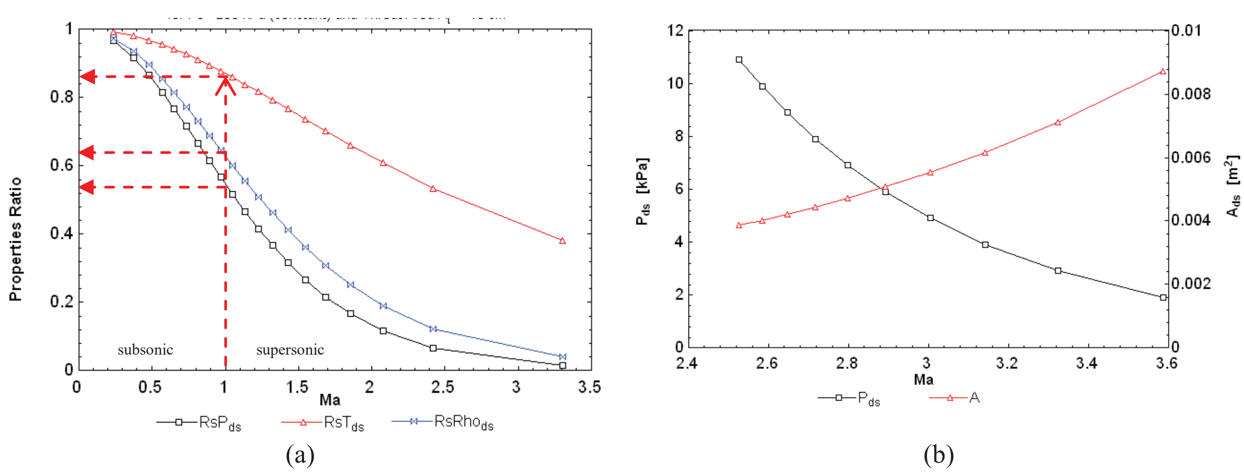


FIGURE 2. The flow properties change analytically of CD Nozzle (Pressure, Density, and temperature) (a), the change of back pressure and downstream area for various Ma (b)

Based on the critical properties ratio, from Critical Pressure Ratio (CPR) is 0.5455, and this point is going to be the reference to find the sub or super sonic region. From calculation also can be found at this chocked condition, those are the magnitude for temperature ratio is 0.8696 and for density ratio is 0.6276. Therefore for P_0 of saturated steam is 200 kPa, the critical properties are 109.1 kPa; 341.9 ^{o}K; and 0.4092 kg/m^3. The correlation backpressure upon the downstream cross section is inversely by exponential, as can be seen on figure-2 (b) the larger area the pressure gets lower, because the total pressure changes into the dynamic pressure. To achieve the hypersonic Ma more than 3, for exit area 0.0071 m^2, the back pressure is 3 kPa. The change of back pressure along cross section area also is presented in figure-3 below.

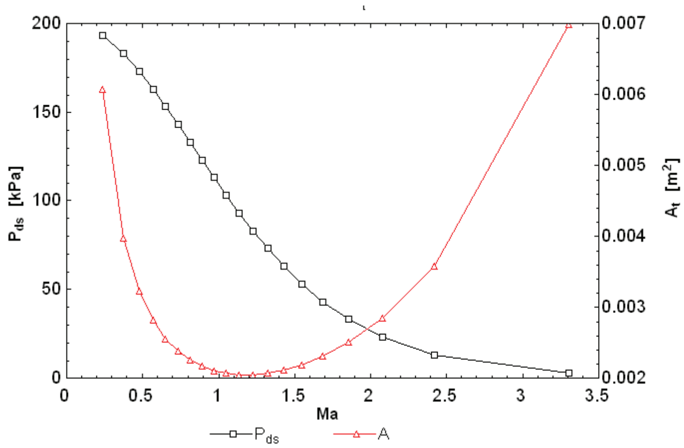


FIGURE 3. The profile corresponding of back pressure to cross section area for 200 kPa saturated steam

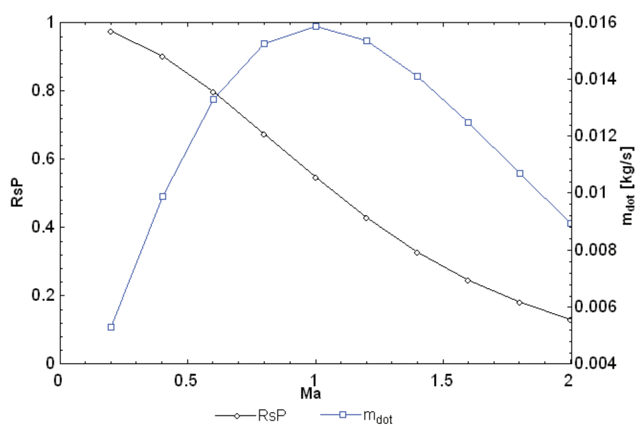


FIGURE 4. The corresponding profile of pressure ratio to mass flow for 200 kPa saturated steam

The mass flow of saturated steam gets the optimum value at Ma =1, when the stagnation inlet pressure of nozzle kept constant on 200 kPa. For supersonic region the mass flow is going to be more lower because the cross section area increases gradually and the density getting more lower. The magnitude of the saturated steam mass flow for this nozzle is about 0.01585 kg/s and occurred at the throat section.

Flow Characteristics and Properties from Computational Modeling

The computational results consist of pre-processing model of grid generation and the plot of distribution contours of flow properties. The mesh generation using BFC model with the non distributed uniformly, and the verification with the way of comparison the parameters of flow like backpressure, temperature, and Ma with the boundary condition given. For the first in the table-1 presents the verification results.

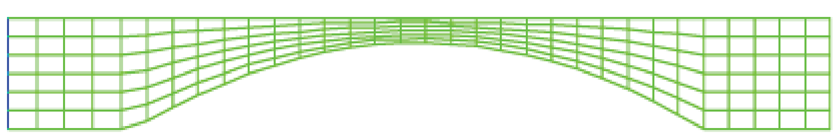


Figure 5. Mesh generation and distribution for Curved CD-Nozzle (34x8 cells)

Grid independency has been tested for cells with dimension are 34x8 ; 76x 16; and 131x28. Contour distribution of flow property, that is pressure below atmospheric, has shown the similarity among these three cell dimensions. Accordingly for analysis is chosen the cell dimension of 34x8, as given in the figure 5. The comparing of pressure distribution below 1 atm, for three cells dimension is shown in the figure 6 below.

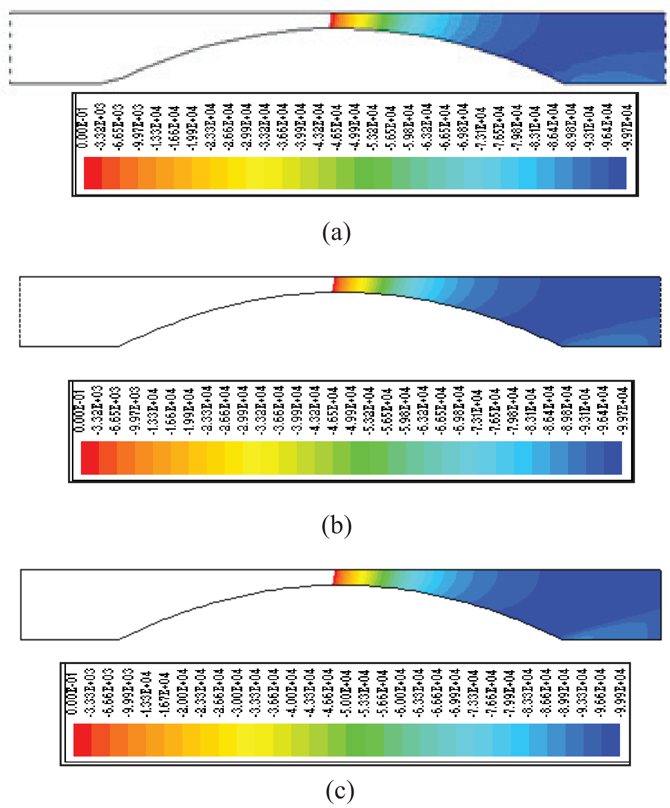
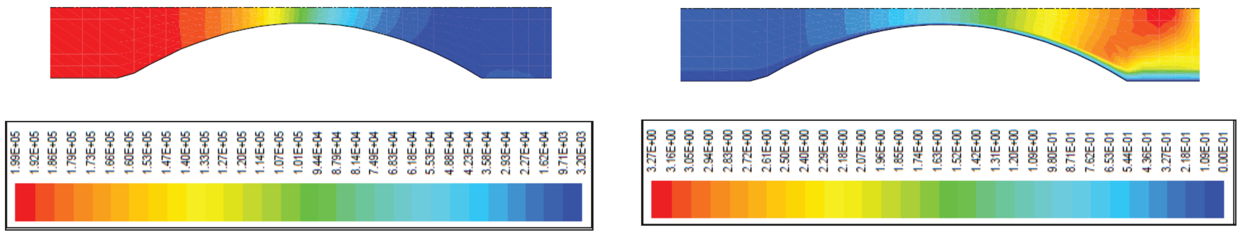


Figure 6. Contour of Vacuum Pressure in CD Nozzle for cells sizing 34x8 (a); 76x16 (b); and 131x28

.

TABLE 1. Verification Results of Simulation at given boundary condition

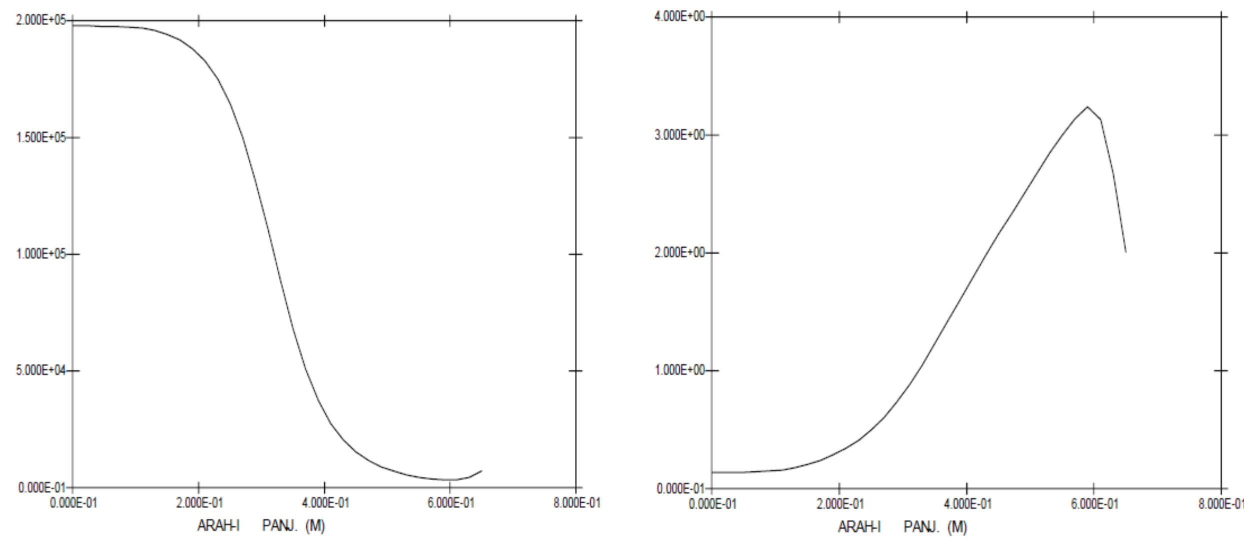
Verified Parameters	Analytical	Computational	Bias (% error)
BackPressure, kPa	3	3.1977	0.0659
Downstream temperature, ^{o}K	149.2	154	0.032
Ma	3.302	3.267	-0.0106



(a) (b)

FIGURE 7. The contour distribution of flow properties : Steam Pressure (a) and Ma number (b)

The verification shows that the occurred bias is less than 7%, and the contour distribution presents that velocity will increase when flows passing through the throat section, as well the pressure decrease as consequence the increasing of dynamic pressure.



(a) (b)

FIGURE 8.The profile distribution of flow properties : Steam Pressure (a) and Ma number (b)

The stagnation pressure 200 kPa gradually will decrease through the cross section area until below atmospheric. As the consequence of compressibility, the velocity gets increasing and after passing the throat the flow changes into supersonic. At a position along axial, the simulation give the prediction that the flow getting out reaches minimum pressure 3 kPa and velocity 3 Ma. The calculated mass flow has the magnitude 1.55 kg/s, meanwhile from simulation gives the mass flux magnitude 1.22 kg/s.

CONCLUSION

The simulation of curved converging-diverging nozzle for saturated steam analytically and computationally has been done. The simulation results give the good acceptance with the bias les than 7% compared to analytical. Saturated steam with stagnation pressure 200 kPa. and downstream area 71 cm^2 with throat area 16 cm^2, could achieve the Ma about 3 analytically and computationally. At chocked condition, the maximum mass flow has the magnitude less than 0.016 kg/s, with the critical properties are 109.1 kPa; 341.9 ^oK; and 0.4092 $kg.m^3$. The calculated mass flow has the magnitude 1.55 kg/s, meanwhile from simulation gives the mass flux magnitude 1.22 kg/s.

ACKNOWLEDGMENTS

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. Biju Kuttan P, Sajesh M. Optimization of divergent angle of a rocket engine nozzle using computational fluid dynamics. Optimization. 2013;2(2):196-207.
2. Dhingra M, Husain M, Rehman S. Development of an Experimental Setup to Study the Flow through a Convergent-Divergent Nozzle.
3. Kundu B, Lee K. Existence of Subsonic Flow in Divergent Section Adjacent to Throat of a Convergent-Divergent Nozzle for Actual Flow. Journal of Applied Fluid Mechanics. 2018;11(2).
4. Lu Y-y, Liu Y, Li X-h, Kang Y, Zhao J-x. Numerical simulation on turbulent flow field in convergent-divergent nozzle. Journal of Coal Science and Engineering (China). 2009;15(4):434.
5. Noh MHM. NUMERICAL INVESTIGATION OF CHOCKED CONVERGING-DIVERGING NOZZLES FOR THRUSTER APPLICATION. IIUM Engineering Journal. 2011;12(3).
6. Olaru I, editor A study on fluid flow simulation in the cooling systems of machine tools. IOP Conference Series: Materials Science and Engineering; 2016: IOP Publishing.
7. Patel KS. Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD). IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). 2014.
8. Rao AR, Akhil I. INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY CFD ANALYSIS OF CD NOZZLE AT TWO DIVERGENT ANGLES FOR PREDICTION OF MACH NUMBER (M< 1, M= 1 & M> 1) AND FLOW PARAMETERS OF NEWTONIAN FLUID BY USING ANSYS-FLUENT 14.0.
9. Rao GR, Ramakanth U, Lakshman A. Flow analysis in a convergent-divergent nozzle using CFD. International Journal of Research in Mechanical Engineering. 2013;1(2):136-44.
10. Reji AK, Kumaresan G, Menon AS, Parappadi J. Simulation and validation of supersonic flow through a convergent divergent nozzle.