Numerical simulation of flow separation in a thrust optimized parabolic nozzle

Sina Afkhami[a], Nematollah Fouladi[*b], Mahmood PasandidehFard[c]

[a,c]Department of Aerospace Engineering, Ferdowsi University, Mashhad, Iran
[b]Space Transportation Research Institute,Iranian Space Research Center, Tehran, Iran

ABSTRACT

Complex flow separation in thrust optimized parabolic (TOP) nozzles in over-expanded condition is one of challenging issue of many numerical investigations. The correct estimation of a TOP nozzle performance extremely depends upon the accurate estimation of the onset of flow separation. Literature review indicates that conventional RANS turbulence models have a significant error in predicting the onset of flow separation in these types of nozzles due to the overestimating of turbulent kinetic energy production. Recently proposed GEKO (generalized model of k-ω) has made it possible to rectify numerical simulations based on governing physics and using limited experimental results. In the present study, the flow physics in the LEA_TOC nozzle has been investigated with the numerical simulation approach. At the first, the significant error of conventional RANS turbulence models is shown to simulate flow separation in this type of problem. Then, the GEKO parameters are modified based on limited experimental result of LEA_TOC nozzle, and the ability of this model has been evaluated to estimate the flow physics under different pressure ratios. Numerical investigations show that GEKO has a high capability for accurate estimating the onset of flow separation at wide range of nozzle pressure ratios. Applying the corrected GEKO has resulted in an improvement of about 30% in estimation of the onset of separation in over-expanded LEA_TOC nozzle compared to the k-ω-sst model.

KEYWORDS

Numerical simulation, Thrust optimized parabolic nozzle, flow separation pattern, GEKO turbulence model

1. Introduction

The contour of the thrust optimized parabolic (TOP) nozzles was developed by Rao in 1960 with the aim of achieving the maximum thrust with the minimum nozzle length. These nozzles are widely utilized for launch vehicles that experience different operating environments from the ground up to high altitudes because of flow separation delay at high back pressure condition in lower altitudes. In low altitudes, where the pressure at the exit of the nozzles with high expansion ratio is lower than the ambient pressure, compression waves occur to increase the pressure in the divergent part of the nozzle, which is mainly accompanied by flow separation in this area. In this mode, the nozzle operates in the so-called over-expanded condition. Numerical simulation is an appropriate alternative to costly experimental tests if it is sufficiently accurate. Also in addition to reducing the number of experimental tests, it makes it easier to study the physics of flow in different conditions. Accurately predicting the location of flow separation in TOP nozzles under over-expanded conditions is one of the challenging issue in numerical simulation with RANS[1] methods, So that in most studies, the location of the separation is predicted upstream or downstream of the actual location. In a study conducted by Allamaprabhu et al. [1], the weakness of RANS models was mentioned in accurately predicting flow separation in TOP nozzles. Nabacheh et al. [2] examined the two nozzle models TIC and TOP during hot gas and cold nitrogen gas, respectively. Their results had a significant error in predicting the separation location according to other RANS turbulence models. Recently, Fouladi et al. [3] and fouladi & farahani [4] conducted the numerical and experimental study of flow physics in TOP nozzles under both atmospheric and high-altitude conditions. In their research, the k-ω-sst turbulence model has been used, and the weakness of conventional RANS models has been pointed out in correctly predicting the onset of separation in the TOP nozzles.

2. Problem definition and Numerical method

The details of the numerical method applied in this research are presented according to Table 1. Ansys Fluent software (version 19/3) was used for numerical simulation. In the present problem, due to the high expansion ratio of the nozzle and its performance in atmospheric conditions, the interaction between the shock wave and the nozzle wall boundary layer occurs, and subsequently the flow in the nozzle wall is separated. Therefore, the turbulence model used should be suitable for estimating the separation onset. Menter et al. [5] developed a new turbulence model family called Generalize k-ω (GEKO) model with the goal of turbulence model consolidation. GEKO is a two-equation model, based on the k-ω model formulation, but with the flexibility to tune the model over a wide range of flow scenarios. This model offers six free parameters – two of them aiming at wall bounded flows, two for the calibration of free shear flows, one coefficient to improve corner flow simulations (corner separation) and finally a curvature correction term. It should be noted that the last two parameters are specific to three-dimensional flows.

As mentioned in the review of references, most researches have made significant errors in predicting the location of separation. Therefore, in the present study, with the aim of correctly predicting the onset of separation with altering the separation parameter (C_{sep}) by using limited experimental results, the suitable value of this parameter is achieved.

Table 1: Details of numerical method

List	Content
Dimension	2-D axisymmetric
Equation of state	Calorically perfect ideal gas
Solver	Density -based
Time	steady
Turbulence modeling	Menter's GEKO
Inviscid flux	Roe
Spatial discretization:	
gradient	Least squares cell based
flow	Second order
Turbulent kinetic energy	First order upwind
Specific dissipation rate	First order upwind
grid	Triangular unstructured

3. Results and discussion

Firstly, in order to evaluate the performance of conventional RANS turbulence models, the LEA_TOC nozzle were analyzed under atmospheric conditions and the nozzle pressure ratio (NPR) equal to 22.8. Figure 1 shows the results of dimensionless static pressure distribution along the nozzle wall with different RANS models (Spalart Almaras, RSM, Standard-k-ε, Realizable-k-ε, RNG-k-ε, k-ω-wilcox, k-ω-sst and GEKO). As can be seen from Figure 1, the use of RANS models has failed to accurately predict the actual separation location. Therefore, in order to achieve the correct results, the parameters of the GEKO turbulence model are changed. Appropriate values of GEKO model parameters for simulation of the desired nozzle are obtained with the results of an experimental test at a pressure ratio of 22.8 according to Table 2.

[1] Reynolds averaged navier stokes

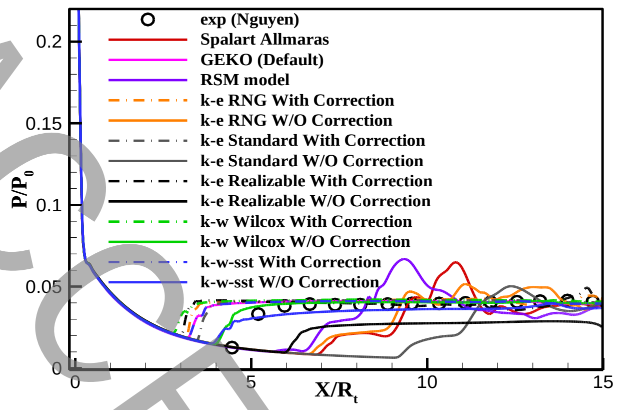

Figure 1. Comparison of conventional turbulence models with experimental results of Ref. [11], (NPR=22)

To evaluate the accuracy of the GEKO model with the parameter values specified in Table 2, numerical simulations in different pressure ratios were performed and compared with the experimental results of Nguyen [6]. Comparison of nozzle wall pressure distribution has been done in 2 different pressure ratios of 23.9 and 38. In these two pressure ratios, there were different physics of separation pattern (FSS and RSS). In the diagrams of Figures 2 and 3, it can be seen that in both separation patterns, the numerical results have acceptable accuracy. According to the pressure curves obtained from numerical simulation, it is clear that the pressure ratio of 23.9 belongs to free shock separation and also in the pressure ratio 38, the RSS separation pattern is established, which causes severe fluctuations in the pressure profile at the end of the nozzle wall.

Table 2: Details of numerical method

Parameter	Value
C_{sep}	0.82
C_{nw}	0.5
C_{mix}	-0.1485
C_{jet}	0.9

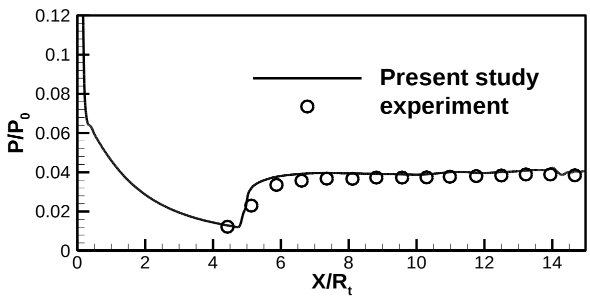

Figure 2 Comparison of wall pressure distribution of the present study with that of experimental result of Ref. [6], NPR=23.9

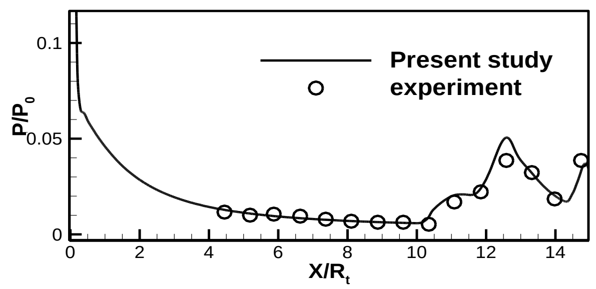

Figure 3 Comparison of the wall pressure distribution of the present study with that of experimental result of Ref. [11], NPR= 38

4. Conclusions

In this research, the numerical study of flow separation was conducted in the LEA-TOC nozzle. At First, RANS turbulence models were evaluated, and their weakness was shown in the estimation of separation onset in this type of nozzle under over-expanded conditions. After that, GEKO turbulence model was examined. It was shown that by applying the separation parameter (C_{sep}) equal to 0.82, the numerical results were in good agreement with the experimental available data. The utilizing of GEKO model with the new coefficients has discounted the error of about 30% in estimating the separation onset respect to the base k-ω-sst model.

5. References

[1] A. Yaravintelimath, B. Raghunandan, J.A. Moríñigo, Numerical prediction of nozzle flow separation: Issue of turbulence modeling, Aerospace Science and Technology, 50 (2016) 31-43.

[2] A. Nebbache, Separated nozzle flow, Comptes Rendus Mécanique, 346(9) (2018) 844-854.

[3] N. Fouladi, M. Farahani, A. Mirbabaei, Performance evaluation of a second throat exhaust diffuser with a thrust optimized parabolic nozzle, Aerospace science and technology, 94 (2019) 105406.

[4] N. Fouladi, M. Farahani, Numerical investigation of second throat exhaust diffuser performance with thrust optimized parabolic nozzles, Aerospace Science and Technology, 105 (2020) 106020.

[5] F.R. Menter, A. Matyushenko, R. Lechner, Development of a generalized k-ω two-equation turbulence model, in: Symposium der Deutsche Gesellschaft für Luft-und Raumfahrt, Springer, 2018, pp. 101-109.

[6] A.T. Nguyen, H. Deniau, S. Girard, T. Alziary de Roquefort, Unsteadiness of flow separation and end-effects regime in a thrust-optimized contour rocket nozzle, Flow, Turbulence and Combustion, 71(1) (2003) 161-181.