The First National Aerodynamics & Hydrodynamics Conference -HAC2012
16-17 October, 2012, Institute of Aviation Industries Organization, Tehran

Evaluation of different turbulence models for simulation of shock train in a convergent-divergent nozzle

Seyed Mahmood Mousavi[1], Ehsan Roohi[2]

[1*]M. SC. Student of aerospace engineering; Ferdowsi university of Mashhad; sm.mousavi.1985@stu.um.ac.ir
[2]Assistant professor; Ferdowsi university of Mashhad;e.roohi@um.ac.ir

Abstract
In this paper, shock train behavior in two-dimensional convergent-divergent nozzle (CDN) has been investigated numerically using various turbulence models including k-ε renormalized group (k-ε RNG), shear stress transport k-ϖ model (K-ϖ SST), Spalart-Allmaras (S-A) and Reynolds stress model (RSM). The results are compared with experimental data and analytical relations. The results show that the data obtained by k-ϖ SST and S-A models are appropriate to the cornet case. The results of RSM better than the K-ε RNG model, so that squared correlation coefficient (R^2) for RSM is an average 15.5% more than k-ε RNG model. Suitable agreement between the experimental and numerical results validates the numerical method and confirms its ability to model the similar cases.

Keywords: *shock train– convergent-divergent nozzle (CDN) - turbulence models- Reynolds stress model (RSM)*

Introduction

The recompression of supersonic gas flow is a very common phenomenon in modern aerodynamics. This phenomenon occurs in different of applications for such as supersonic ramjet or scramjet inlets, internal diffusers and supersonic ejectors. The actual mechanism of recompression can be very different. All of them coincide with compression shocks and shock boundary layer interaction.

The interaction between a normal shock wave and a boundary layer along a wall surface in internal compressible flows causes a very complicated flow. When the shock is strong enough to separate the boundary layer, the shock is bifurcated and one or more shocks appear downstream of the bifurcated shock. A series of shocks thus formed, called ‘’shock train’’, is followed by an adverse pressure gradient region, if the duct is long enough. Thus the effect of the interaction extends over a great distance. The flow is decelerated from supersonic to subsonic through the whole interaction region. In this sense, the interaction region including the shock train in it is referred to as ‘’pseudo-shock’’ [1]. In contrast to other shock systems, the supersonic flow is decelerated at first through a shock system and followed by a mixing region as shown in Fig. 1. Therefore, the flow undergoes successive changes form supersonic to subsonic. In the mixing region, the flow consists of a double tong like supersonic flow near the center line and a subsonic outer region. However, the supersonic flow does not exhibit any compression shocks [2].

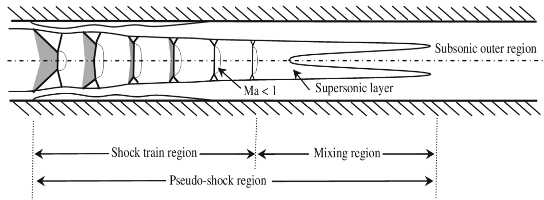

Fig. 1 Sketch of a pseudo shock system [2]

Many investigations have been performed on the shock train topic since the middle of the last century. Comprehensive reviews on related publications as well as fundamentals concerning the so-called pseudo-shock theory are given in Ref. [3]. A brief overview on nozzle flow separation is published by Hadjadj and Onofri [4]. Papamoschou et al. [5] numerically and experimentally investigated the symmetry and asymmetry of the pseudo-shock system in a planar nozzle.

In the percent study, shock train in two-dimensional CDN is numerically investigated using various turbulence models. The results are compared with the experimental data and analytical models. Our results show that the application of RSM for the simulation of CDN will result in less error in comparison with results models. The finite volume technique with coupling pressure and velocity fields of SIMPLE algorithm is used for solving the governing equations.

Geometry of model

The nozzle applications considered in this work is schematically shown in Fig. 2, and the boundary conditions of table 1 are specified to compare experimental and numerical results.

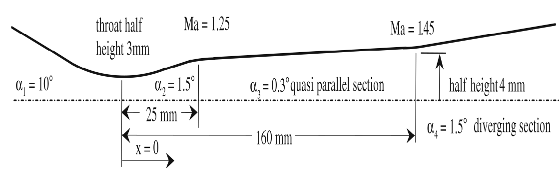

Fig. 2 Sketch of the nozzle contour

Table.1 boundary condition for the nozzle [2]

	P_0 (kPa)	T (k)	V (m/s)
Inlet	480	293	67
outlet	320	293	-

Analytical equation

Waltrup and Billig [6] presented an empirical Eq. 1 for the pressure distribution p(x) in the shock train region.

This equation was derived from experiments in a constant area duct.

$$\frac{x'\left(M_1^2-1\right)\mathrm{Re}_\theta^{1/4}}{D^{1/2}\theta_1^{1/2}}=50\left(\frac{p}{p_1}-1\right)+170\left(\frac{p}{p_1}-1\right)^2 \qquad (1)$$

where x′ is the axial distance in the downstream direction measured from the first shock wave, M1and θ1are the Mach number and momentum thickness at just upstream of the first shock wave, respectively, Re_θ the Reynolds number with $\theta_1$as a length scale, $p/p_1$the wall static pressure in the pseudo-shock wave against that of just upstream of the first shock wave. Initially, the above equation was derived from experiments with circular ducts; therefore Billig [7] has adapted the empirical model also for square ducts given in Eq.2.

$$\frac{x'\left(M_1^2-1\right)\mathrm{Re}_\theta^{1/5}}{H^{1/2}\theta_1^{1/2}}=50\left(\frac{p}{p_1}-1\right)+170\left(\frac{p}{p_1}-1\right)^2 \qquad (2)$$

The static pressure obtained by Eq. 2 agrees well with the experimental data, not only for the flows in circular cross-sectional area but also the flows in squared cross-sectional area [2].

Results and discussions

Fig. 3 illustrates Mach number variation in axial direction (x-coordinate) calculated using different turbulence models including k-ε RNG, k-ϖ SST, RSM and S-A model. The numerical results are compared with Weiss experimental results. It can be observed the data resulted by k-ϖ SST and S-A models are not appropriate for the case. Results of RSM and k-ε RNG models coincide on each other before the shock train generation starting point. The results of RSM are in a better accordance with the experiments compared to k-ε RNG results during the shock train phenomena. (R^2 for RSM and k-ε RNG model are 91% and 78% respectively)

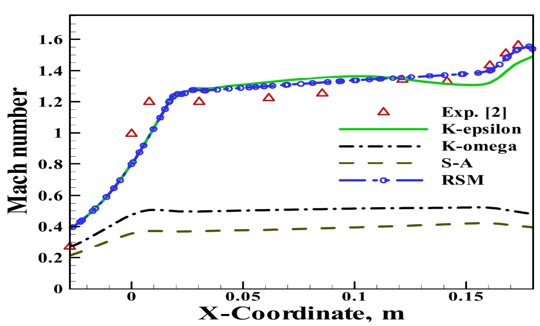

Fig. 3 Comparison between experimental and numerical values of Mach number

In Fig. 4, the pressure distribution on diverging duct's wall in axial direction and the comparison between analytical (Billig equation), experimental data and numerical results are shown using various turbulence models. The error of k-ϖ SST and S-A results are too high, which recommends that these models are not applicable to this case. The application of RSM and k-ε RNG model result in good set of data, and also the precise location of shock train can be detected using the RSM. From the point before the shock train generation to the outlet point of diverging duct the results of RSM closely coincided on experimental results compared to k-ε RNG model. R^2 for RSM and k-ε RNG model are 0.93 and 0.75 respectively. Also it is clear that the static pressure obtained by Eq. (1) agrees well with the experimental data.

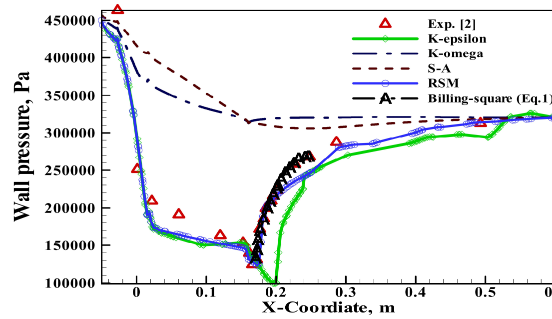

Fig. 4 Comparison between experimental, theoretical and numerical values of static pressure

From two illustrations it can be concluded that the application of RSM turbulence model for the simulation of a diverging duct will result in lower error values in comparison with other turbulence models.

Conclusion

In the percent study, shock train in two-dimensional CDN has been investigated for various turbulence models including RSM, K-ϖ SST, S-A and k-ε RNG model using CFD. The finite volume methods with coupling pressure and velocity fields of SIMPLE algorithm are used for solving the governing equations. The results are compared with experimental data and analytical relation. The results of k-ϖ SST and A-S models are not appropriate to the experimental case. The application of RSM for the simulation of a diverging duct resulted in less error in comparison with other turbolence models, so that R squared for RSM is an average 15.5% more than k-ε RNG model. The good agreement between the experimental and numerical results validates the numerical method and confirms its ability to model the similar cases.

Symbols

D Diameter or equivalent diameter of the duct

H Duct height

Refrences

[1]-Matsuo K, Miyazato Y, Kim H-D: Shock train and pseudo-shock phenomena in internal gas flows. Progress in Aerospace Sciences 35:33-100, 1999..

[2]-Weiss A, Grzona A, Olivier H: Behavior of shock trains in a diverging duct. Exp Fluids 49:355–365, 2010.

[3]-J.M D: Shock wave/turbulent boundary layer interaction and its control. Prog Aerosp Sci 22(4):209–280, 1985.

[4]-Hadjadj, A., Onofri, M.: Nozzle flow separation. Shock Waves 19(3), 163–169. 2009.

[5]-Papamoschou, D., Zill, A., "Fundamental investigation of supersonic nozzle flow separation," 42nd AIAA Aero-space Sciences Meeting, Reno, Nevada, USA. 2004.

[6]-PJ W, FS B: Structure of shock waves in cylindrical ducts. AIAA J 11(10):1404–1408, 1973.

[7]-Billig F Research on supersonic combustion. J Propuls Power 9(4):499–514. 1992