NUMEIRICAL SIMULATION OF SAJBEN DIFFUSER WITH A TURBULENCE MODEL

GAO LIN[1]

[1]*Gao Lin, College of Mechanical Engineering, Xi'an Shiyou University, gldut@126.com*

Keywords: *turbulence, diffuser, numerical simulation, OpenFOAM*

To avoid the separation in the flow field, the requirement of designing is always to reduce the size of subsonic diffuser. Therefore, it is very difficult to choose from meeting the requirements of length and size of diffuser and avoiding of separation. Sajben diffuser is a typical transonic nozzle, which is very fit for verifying the turbulence model for separated flow. In this paper, a turbulence model k-ξ model is adopted, with which better results are expected. For comparison, experimental data and several results of other turbulence models are given.

1. Governing equations and computational methods

There are two transport equations solved for two turbulent quantities in two-equation models, which are the turbulent kinetic energy k and the other variable to derive a turbulent length scale. Thus the eddy viscosity can be expressed as:

$$\nu_t = C_\mu \cdot k^m \cdot \psi^n \tag{1}$$

In which, C_μ, m, n are constants, ψ is considered as a generic length-scale variable. Based on numerical analysis, the second length-scale variable in turbulence modeling is very important, especially in prediction of separated flows. The larger the value of the sum of m and n, the larger the eddy viscosity since most turbulence variables are proportional to rate of strain in strong shear boundary layer regions. This will affect the ability of models to predict separated flows.
Table 1 presents the comparison of the sum of m and n of commonly used two-equation models.

Table 1 *m* and *n* value of different models

Turbulence model	ψ	m	n	$\Sigma = m + n$
$k-\varepsilon$ (Launder and Sharma)	ε	2	-1	1
$k-\omega$ (Wilcox)	ω	1	-1	0
$k-\zeta$ (Jiang)	ξ	1	-2	-1

In this paper, a new eddy viscosity turbulence model, k-ξ model, has been adopted. It is directly derived from standard k-ε and k-ω models based on a new standpoint.
The k-ξ model base on the standard model and a new variable ξ is introduced, which is the square root of specific dissipation $\xi = \sqrt{\omega} = \frac{\sqrt{\varepsilon}}{\sqrt{k}}$. Space lacks for a detailed description of the detailed deducing process.

The transport equations of k and ξ are as below:

$$\frac{\partial \xi}{\partial t} + U_j \frac{\partial \xi}{\partial x_j} = \alpha \frac{\xi}{k} P_k - \beta \xi^3 + \frac{\partial}{\partial x_j}\left[\left(\nu + \sigma_\xi \nu_t\right)\frac{\partial \xi}{\partial x_j}\right] + \sigma_c \frac{1}{\xi^2}\frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} \tag{2}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - k\xi^2 + \frac{\partial}{\partial x_j}\left[\left(\nu + \sigma_k \nu_t\right)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

Where the production rate P_k is $P_k = \nu_t S_{ij}{}^2$.With the strain rate S_{ij} is $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial u_i}\right)$.

The eddy-viscosity is defined as

$$\nu_t = C_\mu \frac{k}{\xi^2} \tag{4}$$

The constants for turbulence model used in this paper are given in Table2.

Table2 Constants for k-ξ turbulence model

α	β	σ	σ	C_μ	σ_c (to ensure to be non-negative)
0.26	0.4	1.0	1.0	0.09	0,if $\frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} \le 0$; 1.5,if $\frac{\partial k}{\partial x_j}\frac{\partial \xi}{\partial x_j} > 0$

In this paper numerical results obtained with k-ξ model had compared with experimental data to test and verify the new eddy viscosity turbulent model.
For both cases, the inlet condition of k and ξ can be set as followed:

$$k = \frac{3}{2}(U \cdot i)^2 \tag{5}$$

$$\xi = \sqrt{C_\mu k / \nu_t} \tag{6}$$

The eddy viscosity is nearly ten times of fluid viscosityν_t=10ν, and outlet variables can be set as zero gradients. No-slip wall conditions are used on the airfoils surface. If the mesh is coarse ($y^+ \geq 30$), standard wall-functions for the turbulent variables are adopted. When the mesh is fine enough ($y^+ \approx 1$), turbulent kinetic energy is set to zero and $\xi = \sqrt{\frac{6\nu}{0.075y^2}}$ are directly derived from the wall boundary condition for ω with a fine mesh .An under-relaxation method is used for stability and faster convergence. The pressure under-relaxation factor is 0.3 and a value of 0.7 is used for the other equations.
In this paper, all calculation and simulation are implemented on OpenFOAM 1.7.1. OpenFOAM is an Open Source library written in C++. It is a well-structured code, mostly used to implement CFD solvers, although it is also used in other applications. OpenFOAM is based on the finite volume method, but there are also implementations of the finite area and finite element methods. With regards to basic features, such as turbulence models and discretization schemes, OpenFOAM is a serious and high quality CFD tool that is constantly evolving. The solution is affected by an iterative pressure-correction (semi-implicit method for pressure-linked equations）SIMPLE algorithm for incompressible flow. The advective volume-face fluxes are approximated using second-order（total variation deminishing）TVD-limited linear differencing. Preconditioned (bi-) conjugate gradient matrix solver methods are used to solve the discretised matrix equations.

2.Numerical results and discussions

During 1979 to 1986, experiments and theoretical studies have been carried by several researchers. The flow-field in sajben diffuser with weak shock have been analyzed. Figure 1 is the geometric diagram of sajben diffuser, which is a kind of a converging-diverging diffuser.

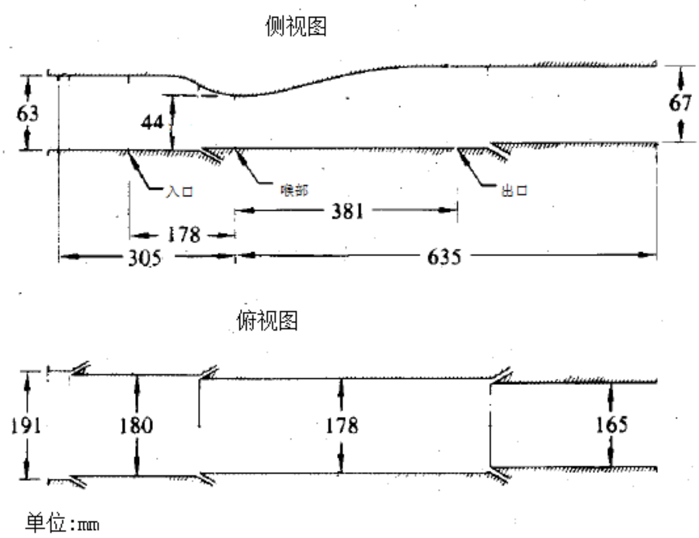


Figure 1: sajben diffuser

As seen, the ratio of the inlet width of the diffuser passage to the throat width (that is, the narrowest part of the pipe) is 1.4, and the ratio of inlet to throat is 1.5. The distance between throat and inlet is four times as long as the width of throat. In the experiment, in order to ensure the two-dimensional characteristics of the flow, suction joints are installed at the corners of the wall and the upper surface perpendicular to the flow direction.
In this case, mach number is 0.46 with uniform inflow and 1% turbulence. The conditions of fluid flows are subsonic inlet and 16.937pisa outlet pressure. The case is meshed as 307*51 hexahedron unstructured mesh, with proper enciphering of wall boundary layer. The simulated results are shown as below. The fluid flows into the diffuser from inlet with uniform condition, and the fluid pressure is varied as the pipeline shape changing. First, the flow pressure reduces and the velocity increases as the pipe constriction. And then the flow velocity decreases and the pressure rise after the throat. At last the flow is tending to balance gradually. It is experimental data that the mach number is 0.46 at inlet, 0.78 after weak shock, and 0.51 at outlet. Its highest value can be 1.3 around throat. The pressure distributions on upper and lower walls are shown in Figure 2 and 3. As seen it is got the nearly results with different turbulence models. But the k-ξ model gives the better pressure distribution and location of the weak shock near throat. The k-ε turbulence model has the late shock separation, compared with the experimental data. At the beginning of balance stage, k-ε turbulence model gives the higher pressure. The trend of fluid flow is better predicted with the k-ξ model. The results of it is closer to experimental data, especially on upper wall.

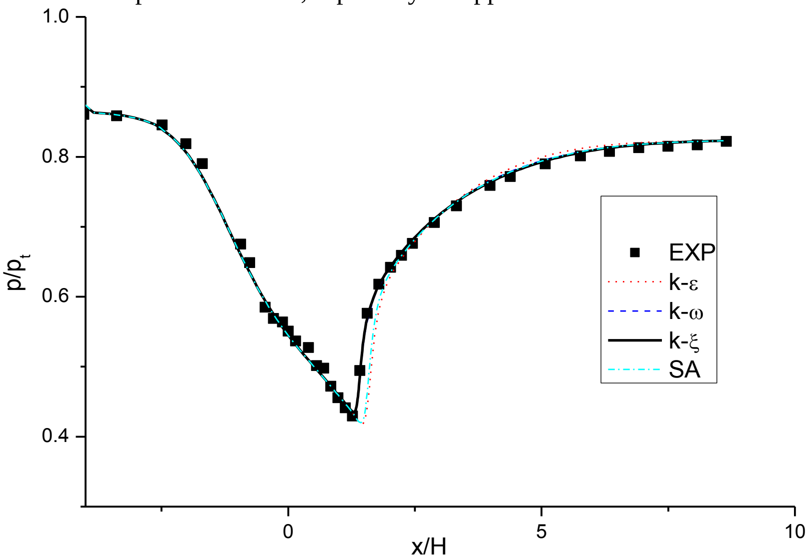


Figure 2: ma=0.46， pressure distributions on lower wall

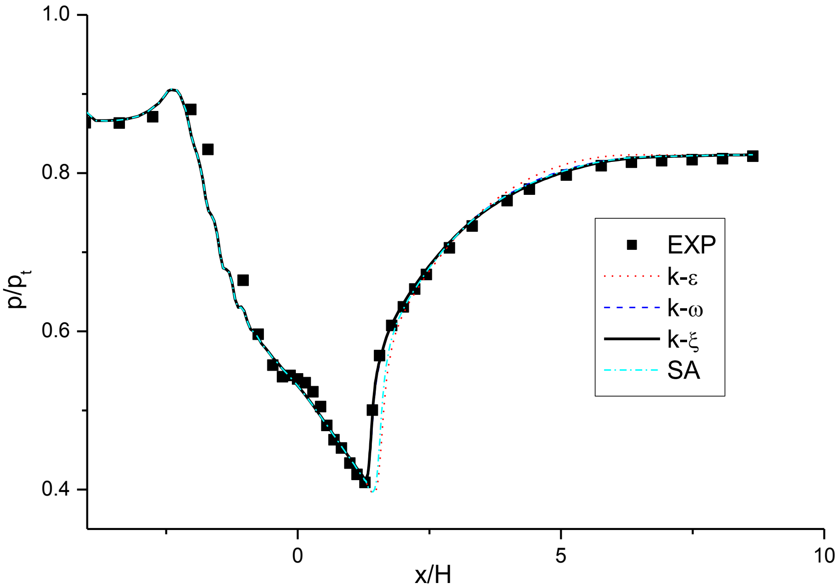
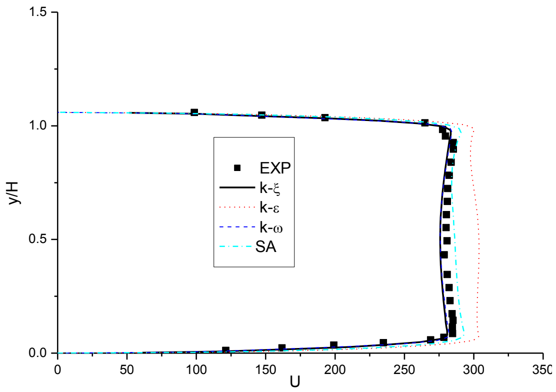
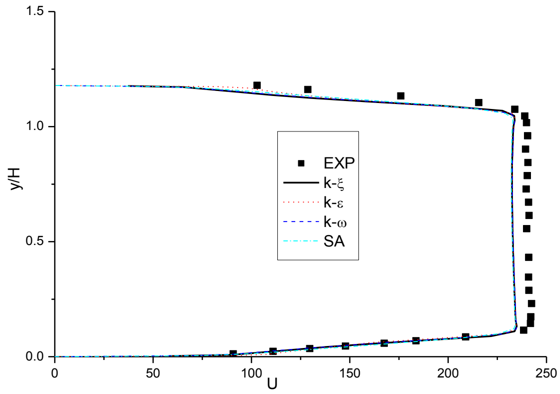


Figure 3:.ma=0.46， pressure distributions on upper wall

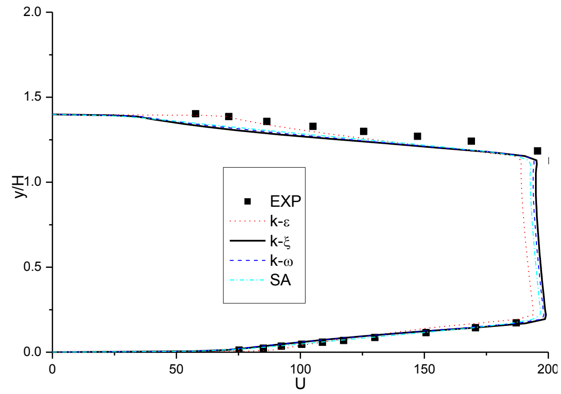
Diffuser wall pressure depends more on shape of wall and initial condition, so all pressure results with different model are close. The velocity profile on the cross section of the diffuser is more persuasive, it is shown the details inside flow field. The figures below show us the velocity profile on the different cross section, x/H=1.729，2.882，4.611，6.340. H is the height of throat of diffuser. It is seen that results with k-ε model have got the higher value on cross section 1 and lower value on cross section 3 and 4. Compared to other models, the results with k-ξ model are closer to experimental data and more in line with physical reality.



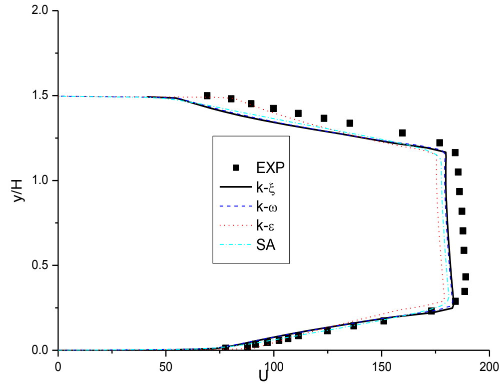
(a) velocity profile on the different cross section, x/H=1.729



(c) velocity profile on the different cross section, x/H=4.661



(b) velocity profile on the different cross section, x/H=2.882



(d) velocity profile on the different cross section, x/H=6.340

Figure 4:ma=0.46，velocity profile on the different cross section, x/H=1.729，2.882，4.611，6.340.

References

[1] W.Mayer, D. and W.D. kneeling, Evaluation of Two Flow Analyses for Subsonic Diffuser Design, in 30th Aerospace Sciences Meeting & Exhibit, A. 92-0273, Editor. 1992: Reno, Nevada.

[2] Bogar, T.J., Structure of Self-Excited Oscillations in Transonic Diffuser Flows. AIAA JOURNAL, 1986. 24(1): p. 54-61.

[3] Chen, C.P., M. Sajben, and J.C. Kroutil, Shock Wave Oscillations in a Transonic Diffuser Flow. AIAA JOURNAL, 1979. 17(10): p. 1076-1083.

[4] Hsieh, T., T.J. Bogar, and T.J. Coakley, Numerical Simulation and Comparison with Experiment for Self-Excited Oscillations in a Diffuser Flow. AIAA JOURNAL, 1987. 25(1): p. 75-81.

[5] Hsieh, T., et al., Numerical Investigation of Unsteady Inlet Flowfields. AIAA JOURNAL, 1987. 25(1): p. 81.

[6] Sajben, M., T.J. Bogar, and J.C. Kroutil, Forced Oscillation Experiments in Supercritical Diffuser Flows. AIAA JOURNAL, 1984. 22(4): p. 474.

[7] Salmon, J.T., T.J. Bogar, and M. Sajben, Laser Doppler Velocimeter Measurements in Unsteady, Separated Transonic Diffuser Flows. AIAA JOURNAL, 1983. 21(12): p. 1697.

[8] Liou, M.S. and T.J. Coakley, Numerical Simulations of Unsteady Transonic Flow in Diffusers, in AIAA/ASME 3rd Joint Thermophysics,Fluids, Plasma and Heat Transfer Conference. 1982: St. Louis, Missouri.

[9] Xiao, Q., H.M. Tsai, and F. Liu, Viscous Computation of Steady/Unsteady Transonic Diffuser Flows, in 32nd AIAA Fluid Dynamics Conference and Exhibit. 2002: St. Louis, Missouri.

[10] Bogar, T.J. and M. Sajben, Characteristic Frequencies of Transonic Diffuser Flow Oscillations. AIAA Journal, 1983. 21(9): p. 1240.