Proceedings of the 6th JFPS International
Symposium on Fluid Power, TSUKUBA 2005
November 7-10, 2005

Calculation of Flow Field of Diffused Pneumatic Silencer

Zhang Xi-wen Zhang Yan Li Heng He Feng

SMC-Tsinghua University Pneumatic Technology Center
School of Aerospace, Tsinghua University, 100084, Beijing, China
(E-mail: zhangxiw@tsinghua.edu.cn)

ABSTRACT

Diffused pneumatic silencer has been widely used in pneumatic fields because of its small dimensions and high performance on noise reduction. Numerical simulation of its interior and exterior flow field is important to studying the gas flow in the silencer and the flow structure outside the silencer, understanding the mechanism of its noise reduction. Porous media model and Darcy-Forchheimer principle are used as the basic theoretical frame in this paper. The unified governing equations are used to describe the compressible flow in and out of the silencer. Robust numerical scheme is used to discritize the equations and the TDBC(Time-dependent boundary conditions) is used to treat the non-reflect boundaries. The detailed structures of the inner and outer flow field of the diffused pneumatic silencer are gotten. The results of the simulation display the characteristics of the flow in the silencer. And the structure of the flow outside the silencer that can be compared with the experimental data is gotten.

KEY WORDS

Diffused pneumatic silencer, Numerical simulation, Flow field

INTRODUCTION

Pneumatic technology is widely applied in many fields of industrial production, but its application brings exhaust noise, which contaminates circumstance, disserves health, and reduces efficiency of production. Diffused pneumatic silencer is widely used in pneumatic fields because of its small dimensions and high performance on noise reduction. Numerical simulation of its interior and exterior flow field is significant for the study of the gas flow in the silencer and the flow structure outside the silencer, understanding the mechanism of its noise reduction, optimum design of the structure of the silencer.

Porous anechoic material is the main component in a diffused pneumatic silencer, and the gas flow through the porous media is compressible gas flow in which pressure, density and velocity change conspicuously. As a result, the flow field in diffused pneumatic silencer is so difficult that few numerical investigations have been made so far.

There are two kinds of conventional numerical methods which simulate the flow in the district where porous materials and pure fluid couple: one uses Darcy or Forchheimer equations and slip boundary conditions, and the other uses Brinkman or Brinkman-Forcheimer equations[1]. However, the governing equations in these methods aim at special flow problems, so the scope to use them is limited. In addition, there is no uniform method to choose governing equations, that is, governing equations are chosen empirically.

The numerical simulation method basing on porous media model was initially used to simulate the flow and heat exchange in nuclear reactors and exchangers. There are many pipes and flakes in a exchanger, so a great number of meshes are needed to simulate the flow in detail. Patankar and Spalding[2] (1974) gave out a method which adopts distributive resistance, it was called porous media model method. In this method, the influence of solid structure (pipes in a exchanger) on mass is reflected with porosity and surface permeability, the influence on momentum is reflected with distributive resistance; the influence on energy is reflected with distributive heat source, then the flow can be simulated with sparser meshes. Using this method, Sha[3] simulated the flow in a vapor generator and core of a nuclear reactor, Karayannis and Markatos[4] simulated the flow in a exchanger, Prithiviraj and Andrews[5][6] simulated the flow in a three-dimensional exchanger.

In this paper, porous media model is adopted, compressible gas flow in and outside a diffused pneumatic silencer is described with uniform and modified N-S equations, and the source terms in equations are determined with Darcy-Forcheimer law. Equations are discretized by using NND format which has good stability and high accuracy, no-reflect boundary conditions are dealt with by using time-dependent boundary conditions. Numerical simulation on the flow field in and outside a diffused pneumatic silencer is made, inner and external flow fields of a diffused pneumatic silencer are got, and the results of numerical simulation on external flow field can be compared with experiment.

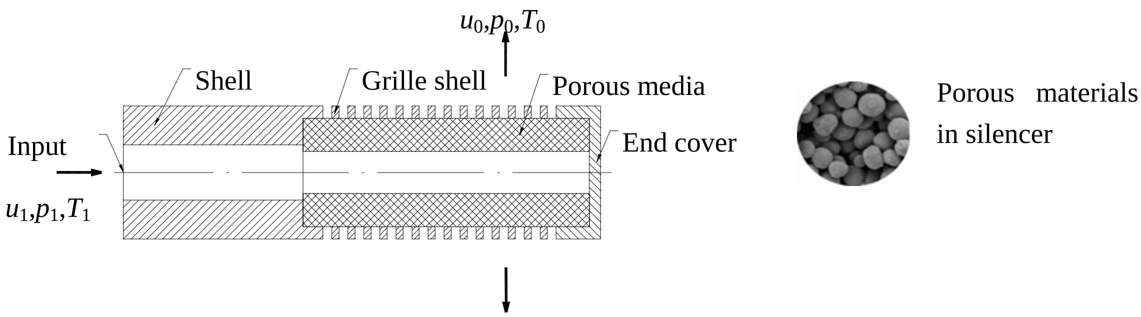

Figure 1.Structure of a diffused pneumatic silencer

DIFFUSED PNEUMATIC SILENCER

A diffused pneumatic silencer is composed of three parts: a silencer shell, porous materials and a end-cover. Its structure can be seen in figure 1.

PHYSICAL MODEL

In figure 1, high pressure gas which has the temperature of T_1, density of ρ_1, pressure of p_1 goes through a silencer at the axial velocity of u_1, because of the resistance of the end cover, the gas will effuse in the radial direction, and the velocity turns to u_0, pressure turns to p_0, density turns to ρ_0, temperature turns to T_0. This is the compressible axis-symmetrical flow which goes through a zone coupling porous media and pure fluid.

EXPERIMENT PARAMETERS

Experiment about the gas flowing in a silencer was performed, the purpose was to provide necessary parameters for numerical simulation and validate the result of numerical simulation. The velocity of the gas flowing into a silencer is 65.1m/s, and the pressure is 1.6602×10^5 Pa, the gas pressure turns to 1.013×10^5 Pa after effusing the silencer, the temperature is 298K, porosity of porous materials is 0.86, permeability K is $1.21\times10^{-11}\text{m}^2$, and the inertia coefficient C_F is $0.91\times10^5\text{m}^{-1}$.

GOVERNING EQUATIONS

N-S equation for integral unsteady axis-symmetrical compressible fluid is shown below [7]

$$\frac{\partial}{\partial t}\iiint_V \boldsymbol{Q}dV + \oiint_S \vec{n}\cdot\vec{\boldsymbol{A}}dS + \iiint_V \frac{\boldsymbol{S}}{r}dV = 0 \qquad (1)$$

where $\boldsymbol{Q}$ is a variable to be resolved, $\vec{\boldsymbol{A}} = (\boldsymbol{E} - \boldsymbol{E}_v)\vec{e}_x + (\boldsymbol{F} - \boldsymbol{F}_v)\vec{e}_r$, $\boldsymbol{E}$ *and* $\boldsymbol{F}$ are non-viscous fluxes in the directions of x and r respectively, $\boldsymbol{E}_v$ and $\boldsymbol{F}_v$ are viscous fluxes in the directions of x and r respectively, $\boldsymbol{S}$ is the source term in the axis-symmetrical N-S equation.

According to the porous media model [2], modified N-S equation for unsteady axis-symmetrical compressible flow in porous media can be gotten from (1)

$$\frac{\partial}{\partial t}\iiint_V a_v \boldsymbol{Q} dV + \oiint_S a_n \vec{n} \cdot \vec{\boldsymbol{A}} dS + \iiint_V a_v \frac{\boldsymbol{S}}{r} dV + \iiint_V a_v \frac{\boldsymbol{S}_{por}}{r} dV = 0 \quad (2)$$

where a_v is volume porosity, a_n is surface permeability in the direction of $\vec{n}$. $\vec{\boldsymbol{A}}$, $\boldsymbol{Q}$ and $\boldsymbol{S}$ have the same definitions as in (1). $\boldsymbol{S}_{por}$ is the source term of distributive resistance in the porous media and heat source. When $a_v = a_n = 1$, $S_{por} = 0$, (2) will be (1), which is normal N-S equation. Source term $\boldsymbol{S}_{por}$ is shown below, distributive resistances R_x and R_r, heat source Q_{rb} is relative to investigated flow.

$$\boldsymbol{S}_{por} = \begin{pmatrix} 0 \\ R_x \\ R_r \\ Q_{rb} \end{pmatrix} \quad (3)$$

Perform the coordinate transformation below on equation (2)

$$\begin{cases} \xi = \xi(x, r) \\ \eta = \eta(x, r) \end{cases} \quad (4)$$

then the physical plane can be transformed to computational plane, resolve the equation on computational plane, detailed formulations for coordinate transformation, fluxes and the source term can be found in reference [7].

IDENTIFICATION OF SOURCE TERM

To simulate the flow in porous media correctly, the source term $\boldsymbol{S}_{por}$ must be identified appropriately. The flow in porous media is very complex, the flow characteristics are different along with the variety of flow status, so the source term should be relative to the flow status of the simulated flow. To identify the expression of source term, the flow status should be known first. Hitherto most researchers regard the Reynolds Number based on aperture (Re_P) [8] and the Reynolds Number based on permeability (Re_K) [9] as the criterion of flow status. Usually, when $\mathrm{Re}_K < 1$, the flow is normal Darcy flow, that is, there is linear relation between pressure gradient and velocity. When $1 < \mathrm{Re}_K < 10$, the flow is not Darcy flow, the linear relation between pressure gradient and velocity is not tenable, Forcheimer modification to flow equation is needed, but the flow is still laminar flow [9], this kind of flow is the so-called Darcy-Forcheimer flow. Along with the increase of velocity, the flow in porous media will turn to turbulent flow. How to differentiate the turbulent flow in porous media has been a problem in the research domain, there is not a definite criterion so far. Seguin's recent experiment[10][11] indicated that the laminar flow in porous media can maintain until $\mathrm{Re}_P < 150 - 200$, when Re_P is larger than a given value, flow will turn to turbulent flow, the given value is relative to the structure of porous media, usually it is above 300. Equation (2) can simulate all kind of flows in porous media, when simulating laminar flow, only the expression of source term need to be identified; when simulating turbulent flow, it is necessary to introduce appropriate turbulence model and corresponding governing equations. There is only study on numerical simulation for laminar flow in this paper.

Steady Darcy-Forchheimer flow in porous media complies with Darcy-Forchheimer law [12] below:

$$\nabla p = -\frac{\mu}{K}\vec{V} - C_F \rho_f \left|\vec{V}\right|\vec{V} \quad (5)$$

where K is permeability, C_F is Forchheimer inertia resistance coefficient, ρ_f is the density of fluid, $\vec{V}$ is Darcy velocity, μ is the dynamic viscous coefficient, $\left|\vec{V}\right|$ is the modulus of velocity, $\left|\vec{V}\right| = \sqrt{u_x^2 + u_r^2}$.

The relation between Darcy velocity $\vec{V}$ and fluid velocity $\vec{u}$ is:

$$\vec{V} = a_v \vec{u} \quad (6)$$

then the expression (5) can be written as

$$\nabla p = -\frac{\mu}{K} a_v \vec{u} - C_F \rho_f a_v^{\,2} \left|\vec{u}\right| \vec{u} \quad (7)$$

In porous media model, distributive resistances [2] are identified with the expression below

$$\nabla p + \begin{pmatrix} R_x \\ R_r \end{pmatrix} = 0 \quad (8)$$

Comparing (7) with (8), expressions of distributive resistances can be given out below.

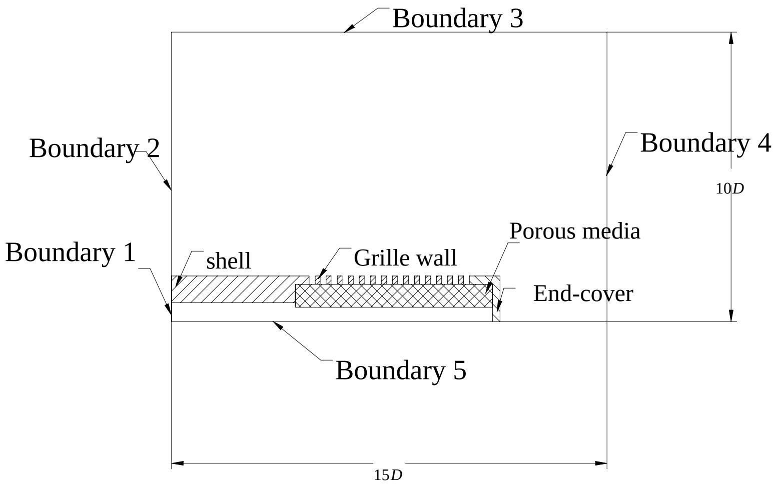

Figure 2.Computational zone

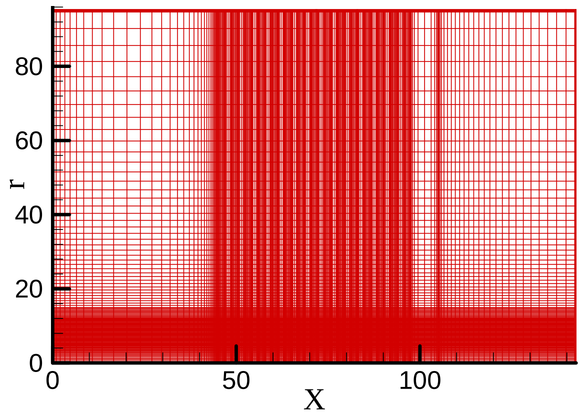

Figure 3.Computational grid

$$R_x = \frac{\mu}{K} a_v u_x + C_F \rho_f a_v^2 |\vec{u}| u_x$$

$$R_r = \frac{\mu}{K} a_v u_r + C_F \rho_f a_v^2 |\vec{u}| u_r \qquad (9)$$

For the flow in porous media without heat source, Q_{rb}=0.

For Darcy flow in porous media, set C_F in the expressions of distribute resistances in (9) to 0, porosity and inertia coefficient in (9) are determined in experiment.

NUMERICAL METHOD

Finite volume method was adopted to resolve equation (2). After the coordinate transformation, the equation was discretized on structural meshes. The viscous flux terms of the governing equations are computed by second-order central difference scheme and the non-viscous flux terms are discretized by the second order NND scheme [13]. The NND scheme belongs to TVD up-wind scheme, it has good stability and artificial viscosity is not needed in calculations. After spatial discretization, equations are integrated by the TVD type 3-stage Runge-Kutta scheme[14], which has third-order accuracy and its effectiveness is well proven in the calculation of time dependent problems.

COMPUTATIONAL ZONE AND GRID

As shown in figure 2, computational zone comprises inner zone, porous materials zone and external zone, although two solid zones, silencer shell and end cover are in computational zone, they are not calculated during computation. The computational zone has an axial length of 15D and a radial width of 10D, D is the inner radius of anechoic materials. Grid networks of 302(x)×121(y) were chosen to use, as shown in figure 3.

BOUNDARY CONDITIONS

There are five kinds of boundaries in computational zone: subsonic inflow boundary, subsonic outflow boundary, axisymmetric boundary, far-field non-reflecting boundary and adiabatic non-slip wall boundary condition. Time dependent boundary conditions (TDBC) which was given by Thompson and Lele were used to deal with far-field non-reflecting boundary, details can be found in references[15] [16] [17]. Five kinds of boundaries are shown in figure 2.

Boundary 1 is subsonic inflow boundary, subsonic inflow conditions are given, including temperature, axial velocity, and radial velocity.

Boundary 2 and 4 are far-field non-reflecting boundaries, far-field non-reflecting boundary conditions are given.

Boundary 3 is subsonic outflow boundary, subsonic outflow condition is given, which is outflow pressure.

Boundary 5 is axisymmetric boundary, axis-symmetrical conditions are given, that is, set radial velocity to be 0; for other parameters, their one-step differential coefficient in the direction of r are set to be 0 as well.

Surfaces on the solid structures in computational zone are given adiabatic wall boundary conditions, that is, set axial and radial velocities to be 0, and $\frac{\partial T}{\partial n} = 0$.

TURBULENCE MODEL

The flows studied in this paper comprise three flow domains: flow in the silencer, flow in anechoic materials, and flow outside the silencer. Flow in the silencer has high velocity and large Re, so it belongs to turbulent flow. Flow in anechoic materials has low velocity (<10m/s), and $\mathrm{Re}_P < 30$, $\mathrm{Re}_K > 2$, according to the criterion[8][9][10][11], the flow belongs to no-Darcy laminar flow in porous materials. Flow outside the silencer is considered to be turbulent flow, too. In this paper, RNG k-ε turbulence model[18][19] given by Yokhot and Orszag is used to deal with turbulent flow.

NUMERICAL RESULTS AND DISCUSSION

Pressure and velocity distributions on the axis in silencer are shown in figure 4 and figure 5, respectively. As shown in these figures, pressure and velocity are constant in the entrance section (domain enclosed by shell), after entering domain enclosed by porous materials, because the flow section reduces, velocity increases and pressure decreases. Gas in silencer is blocked by the end cover, so it effuses through anechoic materials in the radial direction, as a result, axial velocity decreases gradually and pressure increase meantime. In the domain outside the silencer, axial velocity was reduced to be 0, pressure turns to be atmospheric pressure. It can be seen that the pressure and velocity in the silencer are non-uniform.

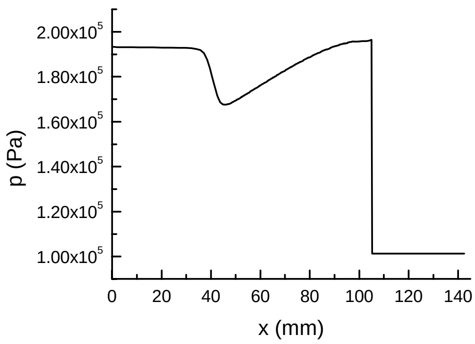

Figure 4. Pressure distribution on the axis

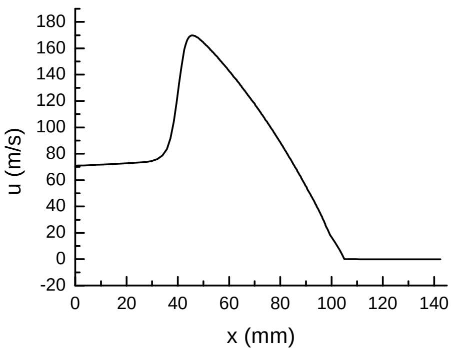

Figure 5. Axial velocity distribution on the axis

Flow fields distribution outside the silencer are shown in figure 6, 7, 8, which denote the distributions of radial velocity at three positions, which has a distance of 1mm, 3mm and 5mm from the out surface of anechoic materials. In these figures, velocity outside the silencer is very non-uniform, compare the experimental result with the result of numerical simulation, it can be seen that numerical simulation describes the flow field well.

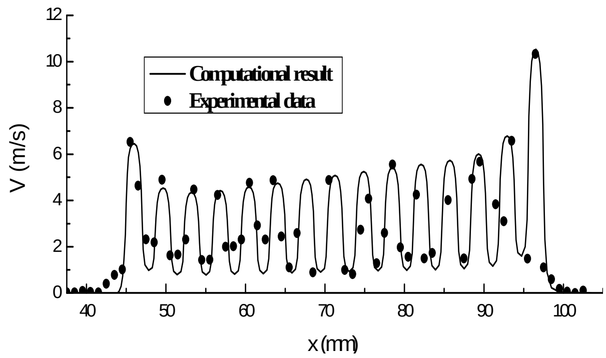

Figure 6. Radial velocity distribution (1mm from the surface of anechoic materials)

In addition, the characteristics of the flow field outside the silencer are: velocities of gas beside the silencer's two ends are higher than those in the middle of the silencer, and the velocities in the middle increase from left to right (figure 6). The flow field has the special structure due to the pressure distribution of the flow in the silencer. Radial velocity distribution on inner surface of anechoic materials is nearly uniform, it is shown in figure 10. There are a pressure drop and a minus radial velocity of large magnitude in figure 9 due to the contractibility of inflow area section. If presume that porosity and permeability of the porous materials are constant, there should be high outflow velocity at the position where pressure in anechoic materials is high, shown in figure 10.

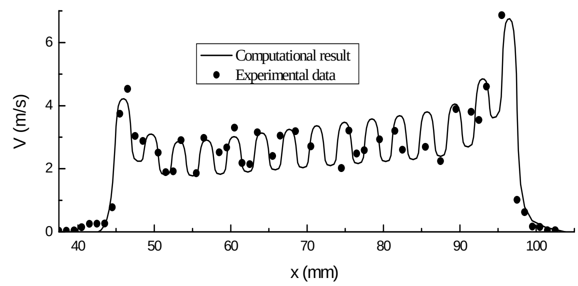

Figure 7. Radial velocity distribution
(3mm from the surface of porous media)

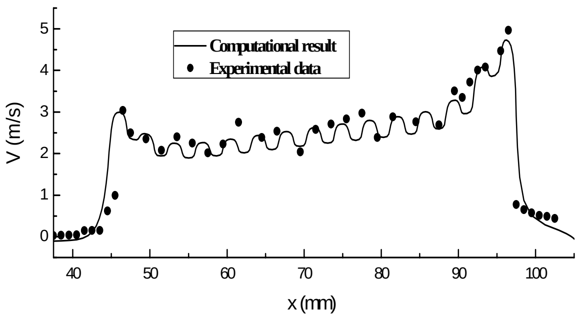

Figure 8. Radial velocity distribution
(5mm from the surface of porous media)

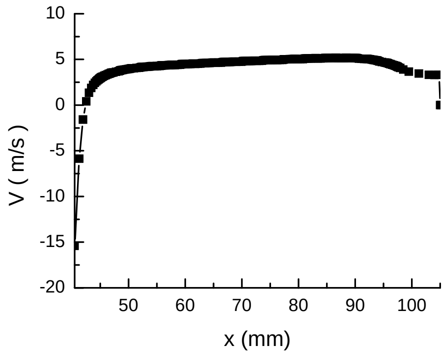

Figure 9. Radial velocity distribution (inner surface)

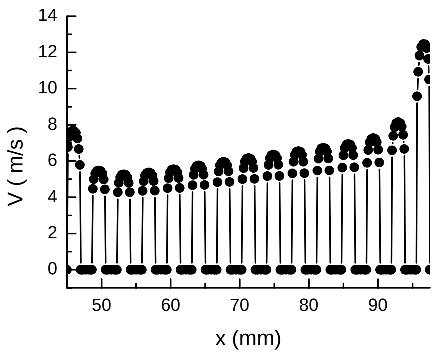

Figure 10. Radial velocity distribution
(external surface)

CONCLUSIONS

(1) With the use of porous media model, modified N-S equations, time-dependent boundary conditions(TDBC) for non-reflecting boundary, and NND schemes are adopted to numerically simulate the inner and external flow fields of a diffused pneumatic silencer, results of numerical simulation agrees well with experimental results.

(2) Results of simulation disclosed the characteristics of inner flow field and the distribution of external velocity of the silencer.

(3) Outflow velocity distribution is special non-uniform, it is determined by pressure distribution in the silencer. Inner pressure distribution and outflow velocity distribution can be improved by improving the structure of the silencer, such as modifying inner area section, reducing the area of shell and end-cover which cover anechoic porous materials. Velocity distribution's change can observably affect the noise elimination of a silencer.

REFERENCES

1. Li Heng, Zhang Xiwen, Journal of the University of Petroleum. 2000, 24(5):111-116 (in Chinese)
2. Patankar S.V., Spalding D.B., Heat exchangers: Design and Theary Source Book, McGraw-Hill, New-York, 1974,155-176.
3. Sha W.T., Yang C.I., Kao T.T. J.Heat Transfer,1982,104:417-425
4. Karayannis N. and.Markatos N.C.G, Proceedings of the 10 th International Heat Transfer Conference, Brighton, UK,The Industrial Sessions Papers,1984:13-18
5. Prithiviraj M., Numerical Heat Transfer Part A-Applications, 1998,33(8):799-816
6. Prithiviraj M., Andrews M.J., Numerical Heat Transfer Part A-Applications, 1998,33(8):817-828
7. Guo Yanhu, Ph. D Dissertation in Tsinghua University ,1996(in Chinese)
8. Marcos H.J.Pedras, Marcelo J.S. de Lemos, Int.Comm.Heat Mass Transfer, 2000,27(2):211-220
9. Nield D. A., Bejan A., Convection in Porous Media, New York, Springer-Velag Inc.,1999
10. Seguin D., Montillet A., Comiti J., Chemical Engineering Science, 1998,53(21):3751-3761
11. Seguin D., Montillet A., Comiti J., Chemical Engineering Science, 1998,53(22):3897-3909
12. Forchheimer P., Hydraulik(3rd ed.), Leipzig, Berlin: Teubner,1930
13. Zhang HX, Zhuang FG. Advances in Applied Mechanics, 1992, 29:143-165
14. Chi-Wang Shu., SIAM Journal on Scientific & Statistical Computing, 1988, 9:1073-1084
15.Thompson K.W. J Comput Phys, 1990, 89: 439-461.
16. Poinsot T.J., J Comput Phys, 1992, 101: 104-129.
18. Yakhot V., Orszag S.A.. J Sci Comput, 1986, 1: 39-51.
19. Yakhot V., Orszag et al. Phys Fluids, Ser. A, 1992, 4 (7): 1510-1522.
20. Patankar S.V., Sparrow E.M., Ivanovic M. Int. J. Heat Mass Transfer, 1978, 24: 269-274.