

高阶各向异性 $k-\epsilon$ 模型及在直通道 湍流流场中的数值研究

西安交通大学 刘正先** 宋保军 谷传纲 苗永森 苏莫明

【摘要】 以各向异性 $k-\epsilon$ 模型为基础,确定得出新的模型:高阶各向异性 $k-\epsilon$ 模型(简称 MAKE),MAKE 模型的主要特点在于雷诺切应力表达式中出现速度梯度的三次函数。使其适应于强各向异性的湍流流场。应用该模型和标准 $k-\epsilon$ 模型(SKE)、各向异性 $k-\epsilon$ 模型(AKE)对二维直管湍流流场进行数值分析,与实验数据比较,结果表明,MAKE 模型对主流速度场没有更大影响,和另外两种模型一样能较好地与实验值吻合。对雷诺正应力的估测,MAKE 模型比 SKE 和 AKE 模型更为精确。

主题词: 湍流 各向异性 数值模拟

分类号: O357.5

1 前言

湍流流场的数值研究目前主要依赖于湍流模型的建立,70 年代期间,由 Launder 和 Spalding^[1]提出的标准 $k-\epsilon$ 两方程模型作为代表性的模型,然而,由于内在的前提假设条件的限制,使其在强各向异性湍流流场中的应用受到了限制。近年来,Nisiziwa 和 Yoshizawa^[2]另辟湍流模型的新途径,提出各向异性 $k-\epsilon$ 湍流模型。这种模型主要区别于标准 $k-\epsilon$ 模型之处在于:在雷诺正应力表达式中加入速度梯度的二次项,以估测湍流流动的强各向异性。Youheimorinshi 和 Toshio Kobayashi^[3]在后突台流动中用到了此模型,Hyoniook 和 Toshio Kobayashi^[4]以及宋保军用此模型估算三维直管道发展中湍流流动。该模型的缺点是各向异性在雷诺切应力中的影响没有体现出来。本文是在文献[5]的基础上,进一步化简得到高阶各向异性雷诺应力涡粘性表达式。其特点是雷诺切应力是平均速度的一次和三次式,以适应湍流流场的强各向异性。对表达式中新增系数确定出较为恰当的系数值。应用该模型和 SKE、AKE 模型对二维直管湍流流动作数值计算。结果表明,适当的系数值使 MAKE 的数值结果比另两种模型在雷诺应力方面与实验值能更好的符合。

2 高阶各向异性 $k-\epsilon$ 模型

Nisiziwa 和 Yoshizawa 依统计理论,提出 AKE 模型,与标准 $k-\epsilon$ 相比,各向异性 $k-\epsilon$ 模型虽然在雷诺应力表达式中多了两项,但各项都是平均速度的一阶梯度表达式,不会增加计算困难。且模型中包含有 SKE 模型,适应性更广。Hyonkook 和 Toshio Kobayashi^[4]及宋保军用该模型估算三维直管道发展的湍流流动,较好地预测出湍流驱动二次流,这是 SKE 模型做不

到的。对于二维直通道湍流流动,有 $V_2=V_3=0$, $\partial V_1/\partial x_1=\partial V_1/\partial x_3=0$, 化简 AKE 模型的应力方程式得到雷诺应力公式:

$$R_{11} = - (2/3)\rho k - (1/3)(-2C_{r1} + C_{r3})(\mathcal{U}/\partial y)^2 k^3/\varepsilon^2 \quad (1)$$

$$R_{22} = - (2/3)\rho k - (1/3)(C_{r1} - 2C_{r3})(\mathcal{U}/\partial y)^2 k^3/\varepsilon^2 \quad (2)$$

$$R_{33} = - (2/3)\rho k - (1/3)(C_{r1} + C_{r3})(\mathcal{U}/\partial y)^2 k^3/\varepsilon^2 \quad (3)$$

$$R_{12} = \mu_t(\mathcal{U}/\partial y) \quad (4)$$

由以上 4 式可知,表达式右边第二项的存在使雷诺正应力体现各向异性,但在切应力中并未得到体现,而湍流流动不仅使雷诺正应力表现出各向异性,同时使切应力发生变化。针对 AKE 模型的不足,文献[5]提出适用于各种流动状况的 MAKE 模型。

$$-\rho \overline{u_i u_j} = -\frac{2\rho}{3}k\delta_{ij} + \mu_t S_{ij} - \sum_{m=1}^3 \tau_m \left(S_{ij}^m - \frac{1}{3}S_{aa}^m \delta_{ij} \right) + \sum_{m=1}^4 \tau_m \left(T_{ij}^m - \frac{1}{3}\tau_m^{aa} \delta_{ij} \right) - \sum_{m=5}^6 \tau_m E_{ij}^m \quad (5)$$

式中 $S_{ij}^m = S_{mij}$, $\tau_m = C_{\tau m} k^4/\varepsilon^3$, $T_{ij}^1 = \frac{1}{2} \frac{\mathcal{U}_i}{\partial x_a} \frac{\mathcal{U}_j}{\partial x_a} e_{ab}$, $C_{\tau m}$ ($m=1-6$) 为常数

$$T_{ij}^2 = \frac{1}{2} \frac{\mathcal{U}_a}{\partial x_b} \left(\frac{\mathcal{U}_i}{\partial x_a} \frac{\mathcal{U}_b}{\partial x_j} + \frac{\mathcal{U}_j}{\partial x_a} \frac{\mathcal{U}_b}{\partial x_i} \right), \quad T_{ij}^3 = \frac{1}{2} \frac{\mathcal{U}_b}{\partial x_a} \left(\frac{\mathcal{U}_i}{\partial x_a} \frac{\mathcal{U}_b}{\partial x_j} + \frac{\mathcal{U}_j}{\partial x_a} \frac{\mathcal{U}_b}{\partial x_i} \right)$$

$$T_{ij}^4 = \frac{1}{2} \frac{\mathcal{U}_a}{\partial x_i} \frac{\mathcal{U}_b}{\partial x_j} e_{ab}, \quad E_{ij}^5 = \frac{1}{2} \frac{\mathcal{U}_a}{\partial x_b} \frac{\mathcal{U}_a}{\partial x_a} e_{ij}, \quad E_{ij}^6 = \frac{1}{2} \frac{\mathcal{U}_a}{\partial x_b} \frac{\mathcal{U}_a}{\partial x_b} e_{ij}, \quad e_{ij} = \frac{\mathcal{U}_i}{\partial x_j} + \frac{\mathcal{U}_j}{\partial x_i}$$

等式(5)右边第一、二项与标准 $k-\varepsilon$ 模型相同,为各向同性的涡粘性项,说明 MAKE 模型与 AKE 模型一样具有包含 SKE 的优点。第三项与 AKE 模型的相应项相同,为由于各向异性的存在引起的雷诺应力的改变。第四、五项是 MAKE 模型雷诺应力表达式与其它模型显著区别之处,为涡与涡之间强烈作用引起的速度梯度高阶项。该高阶项的存在将对雷诺应力数值起一定影响。值得注意的是,模型由于新增加两项,随之产生待定系数 C_u 和 $C_{\tau m}$ ($m=1-6$),由于 MAKE 模型适应于各种流动状况,因此这些系数确定后适应各种简单和复杂的流动,为讨论问题方便,本文从简单的二维直通道流动开始引入张量形式化简(5)式得到:

$$R_{12} = \mu'_t(\mathcal{U}/\partial y) - (1/2)(C_{\tau 3} + C_{\tau 6})(\mathcal{U}/\partial y)^3 k^4/\varepsilon^3 \quad (6)$$

R_{11}, R_{22}, R_{33} 同式(1), (2), (3)。 $\mu'_t = C_u \rho k^2/\varepsilon$ 。

由上述 4 式可知,雷诺切应力公式不仅含 AKE 模型的 $(\mathcal{U}/\partial y)$ 一次项,更包含了 $(\mathcal{U}/\partial y)$ 三次项,从而使湍流的各向异性在雷诺正应力和切应力中均得到体现,弥补了 AKE 模型各向异性在雷诺切应力中的缺点,改善了切应力的数值分布,由于动量方程中含有切应力项,故通过解动量方程对速度场的分布也起一定作用。

在二维直通道流动中,系数 $C_{\tau m}$ ($m=1, 2, 4, 5$) 没有出现,其余的两个系数 C_u 和 C'_1 [$C'_1 = (C_{\tau 3} + C_{\tau 6})/2$] 文献[5]依统计理论推导出系数公式并初步确定 $C_u = 0.85$, $C'_1 = -0.0005$, 显而易见,应用到实际流动中还要作进一步调整。以文献[5]所给系数值为依据,选取不同数值应用于计算中,定量考察系数的大小对计算结果的影响,同时确定与 MAKE 模型匹配的系数值。由于标准 $k-\varepsilon$ 模型中 $C_u = 0.09$, 故作者对 $C_u = C'_1 = 0.09$ 情况下,3 种模型的数值计算首先做过讨论,发现对主流速度的估测 3 种模型的计算结果值与实验值吻合得均非常好。对湍流强度的计算,MAKE 模型结果相比之下令人不太满意,物别对 v' 的预估。数值结果最差,且结

果曲线凹凸起伏,没有规律,定性、定量均与实验值相差较大。究其原因,是模型的不完善造成,还是系数选取不当引起,不能断定,为此,我们改变系数 C_u 取值做相同算例的数值比较。结果确定 $C_u=0.081, C_1=-0.0001$ 。

3 种模型中公用常数取值分别为: $C_u=0.09, \sigma_k=1.0, \sigma_\epsilon=1.3, C_a=1.44, C_b=1.92, C_{r1}=-0.042, C_{r2}=-0.028, C_{r3}=0.014$ 。

3 数值计算及结果分析

采用 SIMPLE 算法实施关于 U, V, P 代数方程的求解,乘方格式离散方程。源项采用局部线性化方法离散。在修正压力 P' 时采用 ADI 方法,并用块修正技术加速其收敛。入口边界:给定进口处的速度值, k 值取来流平均动能的 0.5%。入口截面的 ϵ 按式 $\epsilon=C_u \rho k^2/\mu_t$ 计算, μ_t 按 $\rho UL/\mu_t=100-1000$ 确定。

出口边界:在没有回流情况下, k 和 ϵ 的边界条件按坐标局部单向化方式处理。

近壁区:采用壁面函数法给出该区的经验公式 $U^+=U_p/U^*=(1/k)\ln(Y_p U^*/\nu)+B$ 。

采用 3 种模型对 Laufer^[6] 的二维直通道流场进行数值分析。Laufer 通道内 $Re=3.08 \times 10^4$, 计算网格取 (33×35) 。3 种模型计算得到的主流速度 U 分布见图 1(图 1-4 中—SKE 计算值,---AKE 计算值,—◇—MAKE 计算值,* * Laufer's 实验值; $C_u=C_u^*=0.081$) 可看到,虽彼此稍有差别,但均与实验值符合很好,说明 C_1 的取值对主流速度分布影响不明显。

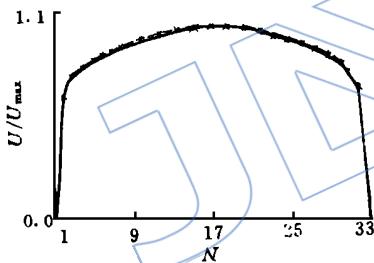


图 1 二维直通道主流速度分布

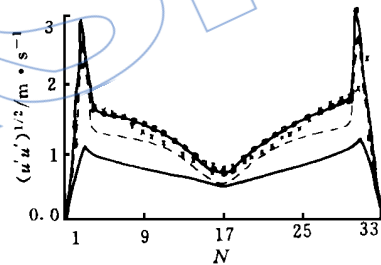


图 2 二维直通道脉动速度 u' 分布

图 2 为脉动速度 u' 分布图。采用确定系数值后的 MAKE 模型计算结果与实验结果比较,两者吻合很好。AKE 模型的结果曲线稍低于 MAKE,与实验点较符合,两种模型在壁面附近与实验值有一定差距。SKE 模型的计算曲线低实验点较大,误差由中间向两壁面从 20% 增到 40%。在图 3 脉动速度 v' 的分布中,MAKE 模型的计算结果比采用 $C_u=0.09$ 计算结果有突破性改进,特别在流道中间附近,改善了 SKE 模型普遍偏低与实验值的状况,与实验结果吻合较好。AKE 模型的计算结果比 $C_u=0.09$ 时更差,最大误差达到 60%。脉动速度 w' 的分布见图 4,与实验值符合程度最好的是 MAKE 模型,仅在两壁面附近,略高于实验值,SKE 和 AKE 模型均低于实验点约 0.1—0.3。

由图 2-4 可知,确定系数取值后的 MAKE 模型,计算结果明显得到改善, u' 和 w' 分布图中曲线与实验点吻合非常好, v' 的数值虽与实验值有局部的偏差,但比其它系数下的结果更令人满意。说明:(1)先前 MAKE 模型的计算结果不好不是模型本身的问题,而是需要有恰当的系数与之匹配。(2)系数的大小对数值计算结果影响较大,有必要对系数的选取进行充分调试。

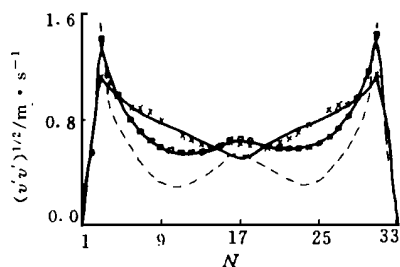
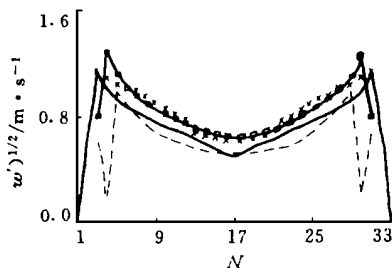
图 3 二维直通道脉动速度 v' 分布图 4 二维直通道脉动速度 w' 分布

图 2—4 中脉动速度的计算值在壁面附近与实验值误差较大,原因可能为:(1)网格不是足够密。壁面处速度梯度大,速度脉动更加聚烈,网格太疏使起重要作用的脉动值被忽略,引起误差。(2)壁面函数法处理近壁区本身就具有假定性,有待进一步完善,采用之进行数值计算不可避免引起一定误差。

4 结 论

(1)本文推导化简得到高阶各向异性 $k-\epsilon$ 模型,主要特点是雷诺切应力表达式中包含速度梯度的三次项,使其适用于湍流流动的强各向异性。同时与 SKE 和 AKE 模型作了定性分析。(2)分别用 SKE、AKE 及 MAKE 模型对二维直管道湍流流场进行数值分析,结果表明,MAKE 模型比 SKE 模型对脉动速度的预估有很大改进(最大误差从 40%降为 10%),比 AKE 模型在 u' 和 w' 方面改进较大(最大误差从 30%降为 10%),在主流速度和 v' 方面均与实验值吻合很好。

参 考 文 献

- 1 Laufer B E, Spalding D B. The Numerical Calculation of Turbulent Flows. Comput. Methods in Applid Mech. and Eng. , 1974, 3(1): 269—289
- 2 Shoit Nisizima, Akira Yoshizawa. Tubulent Channal and Couette Flows Using an Anisotropic K— ϵ Model. AIAA J. , 1987, 25(3): 414—420
- 3 Youhei Morinshi, Toshio Kobayashi. Assessment of K— ϵ Model Using LES Data on the Flow over Backward Facing Step. ASME Fluid Mech. Forun FED, 1991, 117
- 4 Hyon K M, Toshio Kobayashi. Numerical Simulation of Three Dimensional Developing Turbulent Flows in a Square Duct with the Anisotropic k— ϵ Model. ASME Fluid Mech. Forum, FED, 1991, 117
- 5 Song Baojun. On Tow—Equation Turbulence Model for Complex Turbulen Flows by Using DI Approximation (1) Theory. 9th Inter. Conf. of Turbulent Shear Flow, Japan, 1993
- 6 Laufer J. Investigation of Turbulent Flow in a Two Dimension Channal. NACA Report 1053, 1951

EXPERIMENTAL INVESTIGATION ON PERFORMANCE OF A LARGE S- SHAPED DIFFUSER WITH " CARET" INLET

Zhong Yicheng, Yu Shaozhi, Chen Xiao

(2nd Dept. Nanjing University of Aeronautics and Astronautics, Nanjing 210016)

ABSTRACT

A " caret" inlet was designed for Mach number 2.0 according to theory of waverider. The experimental investigation on flow characteristics of a large S- shaped diffuser with such an inlet has been carried out. The flow characteristics of the inlet and the S- shaped diffuser have been analysed by means of flow visualization on the walls, measurements of total pressure recovery distribution and velocity vector plots in the sections concerned. The performance of this large S- shaped diffuser was compared with that of the same diffuser but with conventional inlet. It is shown that, under takeoff condition and at the same Ma^2 of the diffusers, the total pressure recovery of the former is a little higher but the circumferential total pressure distortion index $\Delta\sigma$ and DC60 of the former are larger than that of the latter.

Key Words Engine inlets Diffusers Total pressure Separated flow

A CVFEM FOR 2- D CONVECTION- DIFFUSION EQUATION

Wang Xu and Yan Chuanjun

(7th Dept. Northwestern Polytechnical University, Xi'an 710072)

ABSTRACT

The formulation of a control- volume- based on finite- element method (CVFEM) for two- dimensional convection- diffusion equations is presented. It has a main advantage in capability of the obtained discretization equation to be solved with point - by- point and conjugate gradient iteration method, so that it is more suitable to complex geometrical fields and can save much more computer memory. Some numerical examples proved that the accuracy to any high Peclet number is quite encouraging.

Key Words Convection Diffusion Finite element methods

A HIGH- ORDER ANISOTROPIC K- ϵ MODEL AND ITS NUMERICAL INVESTIGATION

IN TWO DIMENSIONAL TURBULENT FLOW FIELDS

Liu Zhengxian, Song Baojun, Gu Chuangang, Miao Yongmiao, Su Moming

(Energy and Power Engineering Dept. Xi'an Jiaotong University, Xi'an 710049)

ABSTRACT

A new anisotropic K- ϵ model (MAKE) is deduced and simplified based on the basis of anisotropic K- ϵ model. It has been applied to numerical analysis of two dimensional turbulent flows. Its main feature is the presence of the first and third order functions of velocity gradient in the Reynolds shear stress formula, so as to be applicable for strong anisotropism of turbulent flow fields. The new coefficients of the MAKE model are analyzed and selected. The straight two dimensional turbulent flows are numerically studied by using MAKE standard K- ϵ model (SKE) and anisotropic K- ϵ model (AKE). The numerical results obtained from the three models are compared with previous experimental data. It is shown that the main flow velocity field is predicated by MAKE as well as the other two models, but MAKE is more accurate than SKE to prediction of the Reynolds normal

stress.

Key Words Turbulence Anisotropy Numerical simulation

A STUDY ON FLOWRATE CHARACTERISTICS OF GAS- LIQUID COAXIAL ATOMIZERS IN HYDROGEN/OXYGEN ROCKET ENGINE

Zhou Jin, Hu Xiaoping, Huang Yuhui, Wang Zhenguo

(1st Dept. National University of Defense Technology, Changsha 410073)

ABSTRACT An experimental and theoretical study on the flowrate characteristics of gas- liquid coaxial atomizers in hydrogen/oxygen rocket engine are presented in this paper. The dependence of the gas and liquid flowrates of atomizers with different recess ratio on the injection pressure drops was determined by the tests with air and water. The effect of the interaction between gas and liquid flows within atomizer on the flowrate characteristics was investigated experimentally. The results show that the interaction between gas and liquid flows has a certain effect on the flowrate characteristics, and the effect is greater for the atomizers with larger recess ratio. A simplified theoretical model is developed to calculate the flowrates of gas- liquid coaxial atomizers. The comparison between the theoretical and experimental results shows a reasonable agreement.

Key Words Atomizers Flow rate Flow theory Flow characteristics

TRANSITION SIMULATION USING GLM- EM BASED CLOSED EQUATIONS OF TURBULENCE

Gao Ge

(4th Dept. Beijing University of Aeronautics and Astronautics, Beijing 10083)

Y. Yong

(Florida Atlantic University U. S. A.)

ABSTRACT The half- wave Generalized Lagrangian Mean (GLM) method is applied to deriving the closed equations of incompressible turbulent flow. The set of equations does not contain any correlation terms and empirical coefficients. The eddy- viscosity is determined by the Stokes drift velocity, its length scale and divergence. Without using the wall function and any empirical treatments, the transition process of a plan boundary layer has been successfully simulated by the equations. All the transition details, such as the change of the mean parameters, the transformation of the velocity profiles from laminar to turbulent, the evolution of the thickness of the boundary layer, the length of transition zone, and the curve of friction coefficients, are well consistent with experimental results.

Key Words Turbulence equations Transition Simulation