

二维对流—扩散方程的控制体有限元法数值模拟

西北工业大学 王 旭** 严传俊

【摘要】 推导了二维对流—扩散方程的控制体有限元公式,其主要特点是所得离散化方程适合于逐点迭代法及共轭梯度法进行求解,使得该方法更适用于求解复杂的几何域及节省更多的计算机内存。通过几个算例表明,该方法对于特别大的 Peclet 数其精度也是很高的。

主题词: 对流 扩散 有限元法
分类号: O242.21

1 计算域的离散化及控制体积分方程

稳态的对流—扩散方程

$$\alpha \rho_u \varphi / \alpha + \alpha \rho_v \varphi / \beta = (\partial \alpha) [\Gamma (\partial \varphi \alpha)] + (\partial \beta) [\Gamma (\partial \varphi \beta)] + S \quad (1)$$

其中 (u, v) 为流体流动速度, ρ 和 Γ 为流体密度及扩散系数。当取 Γ 为流体的粘性系数, S 分别为 $S_u - \partial \rho / \alpha$ 及 $S_v - \partial \rho / \beta$ 时,方程(1)便可分别代表 x 方向及 y 方向的动量方程;当取 Γ 为热传导数,方程(1)便可代表能量方程。但当其对流项显著时,传统的有限差分法及有限元法便会遇到解的振荡及内存不足问题。控制体有限元法(CVFEM)吸收了有限体积法及有限元法的相互优点,有效地解决了前者所遇到的问题^[1-3]。本文推导了方程(1)的 CVFEM 离散化方程及自然边界条件的处理方法,使其适合于逐点法及共轭梯度法求解,便于对复杂的几何域问题进行数值模拟。通过几个算例表明本文所述方法的计算精度及收敛速度都是比较理想的。

对于任一计算域,其网格的离散过程可分两步来完成:(1)将整个域离散为4结点的四边形单元;(2)再将每一四边形单元分为两个三结点的三角形单元。这样离散的目的在于能方便地建立各结点的相邻单元及相邻结点数。从每一单元的形心点向其三个边中点作连线,便对每一结点形成一个多边形域,即控制体。根据守恒原理,方程(1)在每个控制体上都是守恒的。设结点 P 的控制体为 V ,其表面为 ∂V ,则方程(1)的控制体积分方程为

$$\int_V \vec{J} \cdot \vec{n} ds = \int_V S dV \quad (2)$$

其中 $\vec{J} = J_x \vec{i} + J_y \vec{j} = [\rho_u \varphi - \Gamma (\partial \varphi \alpha)] \vec{i} + [\rho_v \varphi - \Gamma (\partial \varphi \beta)] \vec{j}$ 为对流—扩散通量, $\vec{n}$ 为 ∂V 上微元 ds 的单位外法向量。

2 插值函数及离散化方程

假设方程(1)中 ρ 和 Γ 在整个计算域上为常数,源项 S 为零,则方程(1)或(2)式的离散仅

涉及速度向量及标量 φ 。对速度在单元上采用线性插值, 对 φ 采用如下的流向型迎风方案。

在图 1 所示的 $P-K-L$ 典型单元上建立坐标旋转变换

$$M = \begin{bmatrix} \cos\alpha & \cos\beta \\ -\cos\beta & \cos\alpha \end{bmatrix}$$

其中 $\cos\alpha$ 和 $\cos\beta$ 为单元形心点 c 上平均速度的方向余弦。于是

$$\begin{bmatrix} X \\ Y \end{bmatrix} = M \begin{bmatrix} x - x_c \\ y - y_c \end{bmatrix}, \quad \begin{bmatrix} U \\ V \end{bmatrix} = M \begin{bmatrix} u \\ v \end{bmatrix}$$

假设单元上的流场速度恒等于其形心点上的速度, 则方程 (1) 在该单元上可变换为

$$\partial(\rho U_{av} \varphi) / \partial X$$

$$= (\partial \partial X) [\Gamma(\partial \varphi \partial X)] + (\partial \partial Y) [\Gamma(\partial \varphi \partial Y)] \quad (3)$$

由于 (3) 式仅在 X 方向上存在对流项, 故对 φ 在 X 方向上采用指数型迎风方案, 在 Y 方向上采用线性方案, 即

$$\varphi = A + B\xi + CY \quad (4)$$

$$\text{其中 } \xi = \frac{\Gamma}{\rho U_{av}} \left\{ \exp \left[\frac{Pe \Delta (X - X_{\max})}{(X_{\max} - X_{\min})} \right] - 1 \right\}, \quad Pe \Delta = \frac{\rho U_{av}}{\Gamma} (X_{\max} - X_{\min}) \quad (5)$$

式 (5) 中 $X_{\max}$ 和 $X_{\min}$ 分别为单元结点 X 坐标的最大、最小值, U_{av} 为在 X 方向的平均速度值。

对 $P-K-L$ 依逆时针顺序编号, 再将各结点的坐标及 φ 值代入 (4) 式即得

$$\begin{bmatrix} \varphi_1 \\ \varphi_2 \\ \varphi_3 \end{bmatrix} = \begin{bmatrix} 1 & \xi_1 & Y_1 \\ 1 & \xi_2 & Y_2 \\ 1 & \xi_3 & Y_3 \end{bmatrix} \begin{bmatrix} A \\ B \\ C \end{bmatrix}$$

记上述矩阵的逆为 $[a_{ij}]$, 则

$$J_x = \left[\rho U N_i - \Gamma \frac{\partial N_i}{\partial X} \right] \varphi, \quad J_y = \left[\rho V N_i - \Gamma \frac{\partial N_i}{\partial Y} \right] \varphi \quad (6)$$

其中 $N_i = a_{1i} + a_{2i}\xi + a_{3i}Y$ 。对 (2) 式左端在旋转坐标系下按 Simpson 方法积分

$$\begin{aligned} \int_2 \vec{J} \cdot \vec{n} dS &= (Y_2 - Y_1) [(J_x)_1 + 4(J_x)_{1/2} + (J_x)_2] / 6 - (X_2 - X_1) \\ &\quad \times [(J_y)_1 + 4(J_y)_{1/2} + (J_y)_2] / 6 \end{aligned} \quad (7)$$

其中 1, 2 分别表示积分线段的始点和终点。当 P 为内点时, (7) 式只在图 1 的 $\overline{ac}$ 和 $\overline{bc}$ 上进行积分。当 P 为自然边界点时, (7) 式还需视 $\overline{Pa}$ 和 $\overline{Pb}$ 是否为网格计算域的边界而确定是否作相应的积分。此时 (6) 式为

$$J_x = \rho U N_i \varphi - \Gamma (\partial N_i / \partial X), \quad J_y = \rho V N_i \varphi - \Gamma (\partial N_i / \partial Y)_{1/2}$$

$$(\partial \varphi \partial X, \partial \varphi \partial Y)^T = M^{-1} (\partial \varphi \partial \alpha, \partial \varphi \partial \beta)^T$$

在 P 的所有相邻单元上逐一完成上述坐标旋转、对流—扩散通量离散及线积分运算, 并根据相邻单元上局部结点与相邻结点的关系对其装配, 便可得到 P 点的离散化方程

$$a^P \varphi = \sum a_{nb} \varphi_b + b^P \quad (8)$$

其中 nb 取遍 P 的所有相邻结点。

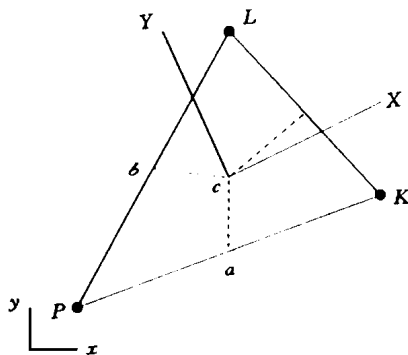


图 1 典型单元及其流向型坐标系

3 算例及结论

算例 1: 如图 2(a) 所示, 设流体沿 $\overline{ab}$ 方向均匀流动, 其粘性系数 $\Gamma \rightarrow 0$, φ 的边界值规定为在线段 $\overline{ab}$ 上方为 1, 在 $\overline{ab}$ 下方为 0, 在 $\overline{ab}$ 与边界的交点处为 0.5。由于 $\Gamma \rightarrow 0$, φ 只产生对流输运, 因而上游边界 φ 的台阶变化在流体流动方向上是应保持不变的。取 $\Gamma = 10^{-6}$, $\rho|\vec{u}| = 1$, 将边长为 10 的计算域剖分为 21×21 均匀网格。图 2(b) 给出了 4 种情况下计算域中垂线 $\overline{cd}$ 上 φ 的变化情况, 从中可以看出, $y_c = 0, 0.5$ 时, 模拟解与精确解是完全吻合的。而 $y_c = 0.2$ 及 0.4 时, 模拟解发生了轻微的振荡。

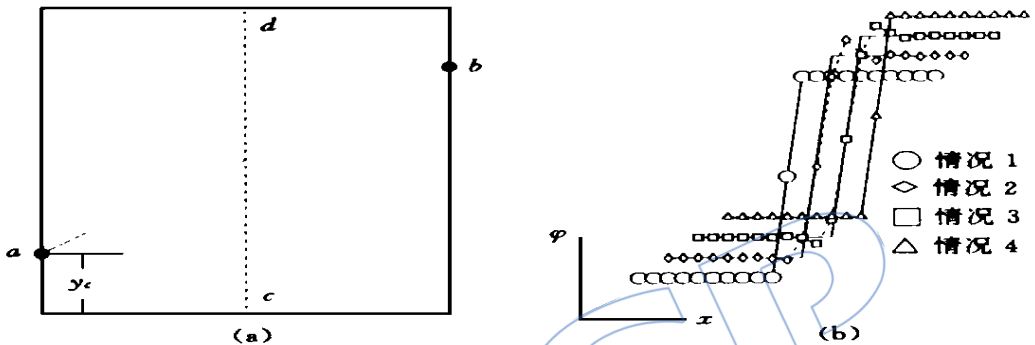


图 2 算例 1 的计算域及结果值

算例 2: 图 3(a) 表示流速分量为 $u = -y, v = x$ 的单位正方形循环流域, 取 $\Gamma = 10^{-6}$, $\rho = 1$, φ 在边界上取为 0。在 $\overline{OA}$ 上按图 3(b) 给出了流体流过 $3/4$ 圆周后 $\overline{OA}$ 上按实线规定的 φ 在 $\overline{OB}$ 上的变化情况。

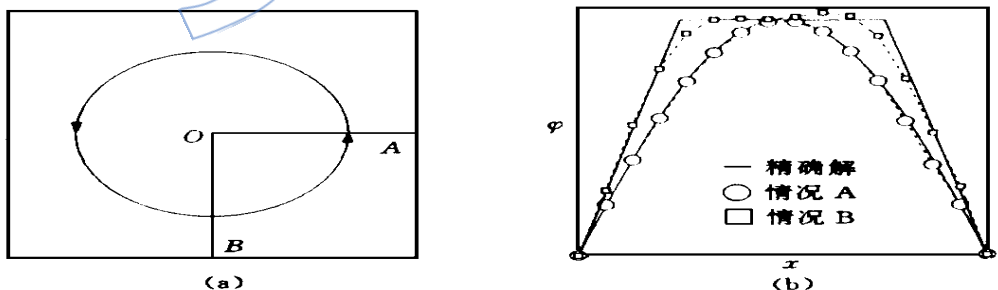


图 3 算例 2 的计算域及结果值

算例 3: 设计算域为四边形域 $-1 \leq x \leq 1, 0 \leq y \leq 1$, 流体流动速度为

$$\begin{aligned} u &= 2y(1 - x^2) \\ v &= -2x(1 - y^2) \end{aligned}$$

边界条件为

$$\varphi = 1 + \tanh[(2x + 1)\alpha]$$

$$\begin{aligned}
 y &= 0, \quad -1 \leq x \leq 0 \\
 \partial \varphi / \partial y &= 0, \quad y = 0, \quad 0 \leq x \leq 1 \quad (9) \\
 \varphi &= 1 - \tanh \alpha \quad \text{其余边界上}
 \end{aligned}$$

取常数 α 为 10。将计算域分为 21×21 均匀网格, 记 $Pe = \rho / \Gamma$, 图 4 给出了各种情况下出口边界 (9) 式上的计算结果, 可见本文与文献 [2] 在四边形单元上的计算结果是一致的。

从如上推导及 3 个算例可以看出, 控制体有限元方法不仅对大的 Peclet 数精度高, 而且可节约大量的计算机内存, 故可推广到三维对流—扩散及二、三维流体流动问题的计算中去。

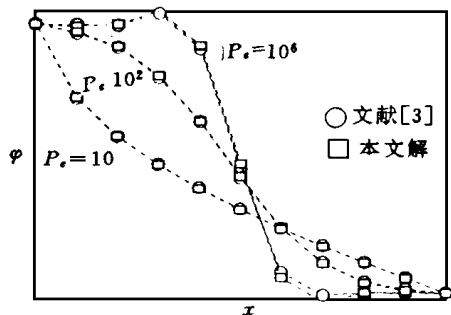


图 4 算例 3 的计算域及结果值

参 考 文 献

- 1 Baliga B R, Patankar S V. A Control Volume Finite-Element Method for Two Dimensional Fluid Flow and Heat Transfer. Numer. Heat Transfer, 1983, 6: 245- 262
- 2 Ramadhyani S, Patankar S V. Solution of the Convection- Diffusion Equation by a Finite- Element Method Using Quadrilateral Elements. Numer. Heat Transfer, 1985, 8: 595- 612
- 3 LeDain- Muir B, Baliga B R. Solution of Three- Dimensional Convection- Diffusion Problems Using Tetrahedral Elements and Flow- Oriented Upwind Interpolation Functions. Numer. Heat Transfer, 1986, 9: 143- 162

(责任编辑 魏星禄)

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