

燃烧室流动的数值计算

北京航空航天大学 徐行** 黄勇

【摘要】提出了多连通域流场统一计算的处理方法,采用三维贴体曲线网格,非交错网格法和 $k-\epsilon$ 湍流模型。对包括扩压器,内外环通道和火焰筒的整个燃烧室流场进行了数值计算。可以计算流过主燃孔、掺混孔和气膜孔的流量。计算结果定性合理。

主题词: 数值计算 燃烧室 湍流

分类号: V 231. 21

1 引言

航空发动机主燃烧室整个流动区域是由4个相互影响的区域组成的多连通区域,前置扩压器出口的流场便是内外环通道和火焰筒的进口流场,而内外环出口的流场又是火焰筒的进口流场。因此要准确模拟燃烧室内的流动,就必须把燃烧室作为一个整体同时求解。燃烧室的各区域都具有复杂的几何形状,需要采用贴体曲线网格,以获得准确的边界条件。采用贴体曲线网格后,如再采用交错网格求解方程,则需要四套网格,各速度分量不在同一网格点,在计算逆变速度时需要插值,而且当气流有大约为90°弯角时,如内外环通道中的流动,交错网格仍然会导致压力波动^[1]。采用非交错网格有许多方法^[2,3],采用文献[2]中的方法,并进行了改进。

2 曲线坐标系下的控制方程和方程的求解

针对燃烧室为环形的特点,选取柱坐标系下的三个速度分量作为求解动量方程的变量。因此要把柱坐标系下的方程,转换成曲线坐标下的方程。方程的通用形式为:

$$\frac{1}{J} \frac{\partial}{\partial \xi} (JU \varphi) + \frac{1}{J} \frac{\partial}{\partial \eta} (JV \varphi) + \frac{1}{J} \frac{\partial}{\partial \zeta} (JW \varphi) = \frac{1}{J} \left\{ \frac{\partial}{\partial \xi} (J \Gamma_{\varphi} g^{\xi\xi} \varphi) + \frac{\partial}{\partial \eta} (J \Gamma_{\varphi} g^{\eta\eta} \varphi) \right\} + S_{\varphi} + S_{\varphi} \\ S_{\varphi} = \frac{1}{J} \left\{ \frac{\partial}{\partial \xi} [J \Gamma_{\varphi} (g^{\xi\eta} \varphi + g^{\xi\zeta} \varphi)] + \frac{\partial}{\partial \eta} [J \Gamma_{\varphi} (g^{\eta\xi} \varphi + g^{\eta\zeta} \varphi)] + \frac{\partial}{\partial \zeta} [J \Gamma_{\varphi} (g^{\zeta\xi} \varphi + g^{\zeta\eta} \varphi)] \right\} \quad (1)$$

式中, φ 取 1, u, v, w, k, ϵ 表示连续方程, 轴向、径向和周向动量方程, 湍流动能和耗散率方程。 U, V, W 是逆变速度, J 为 Jacobi 行列式, $g^{\xi\xi}$ 是度量张量。

S_{φ} 为非正交项产生的源项。当网格的正交性较好时, 这一项就较小。 S_{φ} 为其它项产生的源项, 如压力梯度, 离心力, 哥氏力等项构成的源项。

方程采用非交错网格法求解。节点取为控制体的中心点, 方程离散化后的形式为:

$$a_P \varphi_P = a_W \varphi_W + a_E \varphi_E + a_S \varphi_S + a_N \varphi_N + a_B \varphi_B + a_T \varphi_T + S_{\varphi} \quad (2)$$

差分格式采用混合差分格式。节点上的逆变速度为(由动量方程得到) $\bar{u}$

$$JU_P = H^U + B^U P_{\xi}, \quad JV_P = H^V + C^V P_{\xi}, \quad JW_P = H^W + D^W P_{\xi}$$

$$B^U = -J^2 g_{\xi\xi}/a_P, \quad C^V = -J^2 g_{\eta\eta}/a_P, \quad D^W = -J^2 g_{\zeta\zeta}/a_P \quad (3)$$

上式忽略了非交错项,并假设动量方程的系数对三个速度分量是相同的。

压力校正方程的推导:假设在控制体表面, $JU = JU^* + B^U P'_\xi$, $JV = JV^* + C^V P'_\eta$, $JW = JW^* + D^W P'_\zeta$ 代入连续方程可得:

$$a_P P'_P = \frac{a_E P'_E + a_W P'_W + a_N P'_N + a_S P'_S + a_B P'_B + a_T P'_T + m_b}{m_b} \quad (4)$$

$$m_b = \frac{JU^*}{JU^*} - \frac{JU^*}{JU^*} - \frac{JV^*}{JV^*} - \frac{JV^*}{JV^*} + \frac{JW^*}{JW^*} - \frac{JW^*}{JW^*}$$

控制体边界上的逆变速度,用动量平均,例如:

$$JU_{i+1/2}^* = (1/2)(H_i^U + H_{i+1}^U) + BU_{i+1/2}^* P_{\xi+1/2} = JU_{i+1/2}^* + B_{i+1/2}^U (P_{\xi+1/2} - \bar{P}_{\xi+1/2}) \quad (5)$$

上标—表示线性平均, * 表示动量平均。

3 节点速度分量的校正

在推导压力校正方程时,用了 $JU = B^U P'_\xi$ 式时,忽略了非正交项。因此如果直接由 $u = u^* + u$ 等式来修正节点的 U 便会产生误差。因此采用的方法是用 $JU_P = JU_P^* + B^U P'_\xi$ 式直接求出节点的 JU, JV 和 JW 的值,然后反算出节点的 u, v, w 值:

$$u = UX_\xi + VX_\eta + WX_\zeta, \quad v = Ur_\xi + Vr_\eta + Wr_\zeta, \quad w = Ur_\theta_\xi + Vr_\theta_\eta + Wr_\theta_\zeta \quad (6)$$

在求离散方程的系数时,对对流项采用了动量平均的逆变速度。这是因为动量平均的逆变速度满足连续方程,而线性平均的逆变速度不满足连续方程。

4 多连通区域流场的求解方法和边界条件

整个燃烧室是由 4 个区域构成的。各区域之间用连接面相连。在求解连接面一边区域内的值时,要用到连接面另一边区域内的值。采用的方法是把连接面两边区域的网格向外扩展。即原来把连接面作为区域的边界,而现在把连接面作为区域内的控制体表面。

对动量方程和其它标量方程,四个区域顺序求解,各区域单独求解若干次。而对压力校正方程,压力校正值得在迭代之初为零。而各区域的 P 的求解用到其它区域的 P' , 为保证流量连续,4 个区域的应同时求解,然后对整个区域速度、压力进行校正。

燃烧室有以下几类边界条件: (1) 进口条件: 给定进口的速度、湍流动能及湍流动能耗散率。(2) 出口条件: 假定出口参数的梯度值为常数。(3) 固壁条件: 采用无滑条件和壁面函数。(4) 扇形区侧面条件: 给定周期条件。(5) 连接面条件: 按上述方法处理。

对于非交错网格法,需要知道边界面上的压力和压力校正值得,这两个值可按垂直边界面方向梯度为零的条件,由内点的值外推得到。

5 算 例

计算了中推发动机燃烧室的流场。由于扰流器的复杂性,目前还无法模拟其内部的流动。只能假设其出口速度分布。在扰流器出口用圆环形的网格比较准确描述出口速度分布。火焰筒头部各孔的进气流量,按其流通面积和给定的流量系数确定。除了火焰筒头部,本文的方法,可以计算其余各孔的流量分配。图 1~ 图 4 为扩压器,内外环通道和火焰筒的速

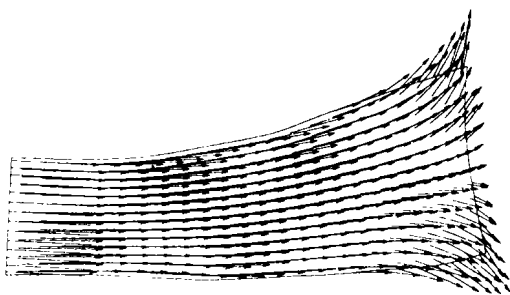
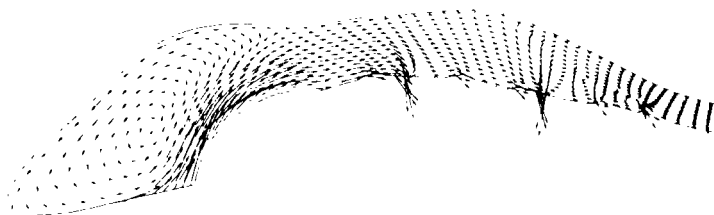
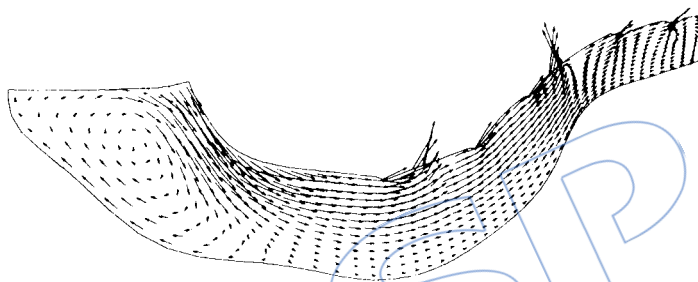
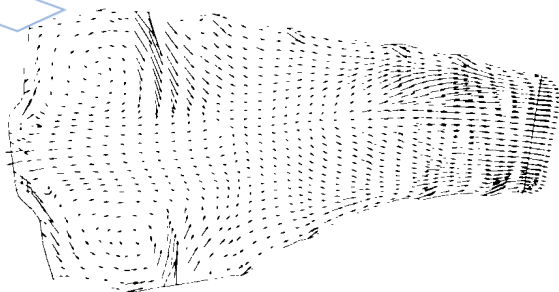


图 1 扩压器的速度矢量图(ξ-2 截面)

图 2 内环通道的速度矢量图($\xi=2$ 截面)图 3 外环通道的速度矢量图($\xi=2$ 截面)

度矢量图。方程的收敛准则为 $1 \cdot E-2$, 各区域进出口流量的误差小于 10%。扩压器的网格数为: $25 \times 16 \times 26$, 内外环为: $45 \times 16 \times 26$, 火焰筒: $31 \times 28 \times 26$ 。燃烧室的三维贴体网格见文献[4]。

结论 本文研究了燃烧室三维流场的计算方法, 计算结果定性是合理的。本文的研究方法还需进一步改进, 使之能考虑到头部喷油嘴对流场的影响和扰流器的内部流动。

图 4 火焰筒的速度矢量图 ($\xi=13$ 截面)

参 考 文 献

- 1 SHYYW. Three- Dimensional Analysis of the Flow in a Curved Hydraulic Turbine Draft Tube Int J. for Numerical methods in fluids, 1986, 1(6): 861- 882
- 2 Rhie C M. Numerical Study of the Turbulent Flow Past a Airfoil with Trailing Edge Separation A IAA Journal, 1983, 21
- 3 ABDALLAH S. Numerical Solutions for the Incompressible N- S Equations in Primitive Variables Using a Non- Staggered Grid II Journal of Computational Physics, 1987, 70: 193- 202
- 4 徐行, 杨榕 燃烧室三维贴体曲线网格计算机交互生成方法的研究 CSAA 96- pc- 33, 1996

(责任编辑 杨再荣)

A TWO- PHASE AFTERBURNER COMBUSTION MODEL

Zhu Zuojin and Chen Yiliang

(13th Dept University Science and Technology of China, Hefei 230026)

Zhang Xiaochun

(Shenyang Aeroengine Research Institute, Shenyang)

ABSTRACT According to the fundamental concept of two- fluids, the governing equations for description of gas- droplets turbulent combustion in an afterburner of jet engine have been formulated, in which the important injection effect of the droplets in afterburner has been taken into detailed consideration. The Favre average method has been employed to represent the effects of turbulent combustion in the afterburner. The multiple time scale turbulent k - ϵ model proposed by Kim is applied to calculating the eddy viscosity of the turbulent field. The heat flux radiation model has been taken to consider the effect of radiation at high temperature, the simple one step chemical reaction model is used to approximate the combustion of kerosene. The chemical reaction rate is assumed to be controlled by eddy break up process, and the reaction rate is modified by premixed reaction model in Arrhenius form. The application of SIMPLE algorithm indicates that the present two fluid model can calculate the two- phase turbulent combustion in the jet engine afterburner and obtain reasonable results.

Key Words Aircraft engines Gas- liquid flow Afterburners Combustion

NUMERICAL SIMULATION OF COMBUSTOR FLOWS

Xu Hang and Huang Yang

(4th Dept Beijing University of Aeronautics and Astronautics, Beijing 100083)

ABSTRACT A method has been developed for calculation of three- dimensional flows in aeroengine combustor including diffuser and flame tube, inner and outer annuluses. The 3D body- fitting grid has been generated to deal with the complex geometry of the combustor. The non- staged grid scheme and k - ϵ turbulence model has been adopted. The mass flowing through primary holes, dilution holes and film cooling holes has been calculated. The calculated results are consistent reasonably with the experimental measurements.

Key Words Numerical simulation Combustion chambers Turbulence

AEROENGINE TEST AUTOMATION OF ADAPTIVE CONTROL WITH RESTRAINTS

Fan Ding

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

Li Runzhi

(Southern Powerplant Machinery Corporations, Zhuzhou)

ABSTRACT An automatic control system of aeroengine test has been designed, based on the method of model reference adaptive control with restraints. According to its simulation results, this method has good adaptability when the controlled member contains non- linear elements or the controller member varies in a large range. The simulation results also show that, the controlled system still has satisfactory performance index by regulating