

航空发动机圆套状尾喷管 流场温度场数值模拟

南京航空航天大学 夏春林* 刘德彰

【摘要】 对某机型双收缩圆环套状尾喷管内, 冷热气流相互掺混的流动传热过程, 作了数值模拟。用 SIMPLE 思想在曲线系中进行程序设计。该程序可以对任意形状的两股流有(无)掺混时的流场、质量场、温度场、辐射通量场进行模拟, 对二维流动传热问题比SIMPLE程序更具有通用性。最后对某机型的实际情况(形状尺寸), 用区域拼装法生成曲线网格作了模拟。结果说明, 引射的冷空气与热主流在双收敛喷管内掺混, 确实能降低尾喷管内气流温度、降低红外辐射。

主题词: 空气喷气发动机 数值模拟 尾喷管 数值计算 对流传热

分类号: V235 V231.3

1 建立模型

对某型号的尾喷管试采用数值计算的方法, 利用发动机排气对外界冷气流的引射作用, 使冷热两股流在收缩圆环状喷管内掺混, 来解决喷管出口排气的红外辐射量增大问题, 并数值地求得各特征量场。

实际发动机尾喷管及延伸段的结构比较复杂, 加上引入外界冷气流与主燃气流掺混后, 其内部气体流动是一个包括湍流流动、对流传热与辐射传热等的复杂过程。为了能预测这个过程, 需将对象简化为二维轴对称问题, 通用控制方程如下:

$$\frac{\partial}{\partial \alpha}(\rho u \Phi) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho v \Phi) = \frac{\partial}{\partial \alpha} \left[\Gamma_{\Phi} \frac{\partial \Phi}{\partial \alpha} \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[r \Gamma_{\Phi} \frac{\partial \Phi}{\partial r} \right] + S_{\Phi} \quad (1)$$

其中 Φ 是因变量, Γ_{Φ} 是名义扩散系数, S_{Φ} 是源项, 它们在各方程中的表达式见表 1。其中湍流模型用的是考虑浮力影响的代数应力模型^[1], 辐射通量模型采用四热流模型模拟。

方程(1)在贴体曲线坐标系下变换为:

$$\frac{\partial}{\partial \xi}(\rho U \Phi) + \frac{\partial}{\partial \eta}(\rho V \Phi) = \frac{\partial}{\partial \xi} \left[D_1 \frac{\partial \Phi}{\partial \xi} \right] + \frac{\partial}{\partial \eta} \left[D_2 \frac{\partial \Phi}{\partial \eta} \right] + JS_{\Phi} + S_R \quad (2)$$

方程(2)加上边界条件、壁面函数^[2]构成定解封闭条件。

表 1 源项表

Φ	Φ	Γ_{Φ}	S_{Φ}
u	1.0	μ_{eff}	$(\partial/\partial x) [\hat{f}_u(\partial u/\partial x)] + (\partial/\partial r) [\hat{f}_u(\partial v/\partial r)] + (\Gamma_u/r)(\partial/\partial x + \partial/\partial r)$
v	1.0	μ_{eff}	$(\partial/\partial x) [\hat{f}_v(\partial u/\partial r)] + (\partial/\partial r) [\hat{f}_v(\partial v/\partial r)] + (\Gamma_v/r)(\partial/\partial r - v/r)$
T	0.7	μ_{eff}/σ_T	$2K_a(R_x + R_y - 2E)]/C_p + (\Gamma_T/r)(\partial T/\partial r)$
K	0.9	μ_{eff}/σ_K	$G_k + G_b - \rho \epsilon + S_k'(K) + (\Gamma_K/r)(\partial K/\partial r)$ 其中 $G_k = \mu_{eff} \{ 2 [(\partial u/\partial x)^2 + (\partial v/\partial r)^2 + (v/r)^2] + (\partial u/\partial r + \partial v/\partial x)^2 \}$ $G_b = -\beta \rho [\hat{g}_x(\mu_T/\sigma_T)(\partial T/\partial x) + \hat{g}_r(\mu_T/\sigma_T)(\partial T/\partial r)]$
ϵ	1.1	$\mu_{eff}/\sigma_{\epsilon}$	$(\epsilon/K)(C_1 G_k - C_2 \rho \epsilon) + (\Gamma_{\epsilon}/r)(\partial \epsilon/\partial r) + S_k'(\epsilon)$ $C_{\epsilon_1}(\epsilon/k)(G_k + G_b)(1 + C_{\epsilon_3} R_f) - (C_{\epsilon_2} \rho \epsilon^2/K) + (\Gamma_{\epsilon}/r)(\partial \epsilon/\partial r) + S_k'(\epsilon)$
R_x	1.0	$1/(K_a + K_s)$	$K_a(E - R_x) + (K_s/2)(R_r - R_x) + (\Gamma_{R_x}/r)(\partial R_x/\partial r)$
R_y	1.0	$r/(K_a + K_s + 1/r)$	$K_a(E - R_y) + (K_s/2)(R_x - R_r) + (\Gamma_{R_y}/r)(\partial R_y/\partial r)$

$\mu_{eff} = \mu + \mu_T \quad \mu_T = C_u \rho K^2 / \epsilon \quad K_s \doteq 0.01 \quad E = \sigma T^4 \quad S_k'(\varphi) = (\partial/\partial x_k) [\ell C_k (K/\epsilon) v_i v_j (\partial \varphi / \partial x_i)] - (\partial/\partial x_i) [\hat{f}_{\varphi} \partial \varphi / \partial x_i]$
 $K_a = [(0.78 + 1.6 P_{H_2O}/P) / ((P_{H_2O} + P_{CO_2}) / P - 0.1)] \times [1 - 0.37 \times (T_g / 1000)] \times (P_{H_2O} + P_{CO_2})$

2 计算方法及结果分析

双收缩尾喷管复杂形状的曲线坐标生成,采用分块生成,最终拼装法。在贴体曲线坐标系中,引入压力校对方程^[3],还需引入交错网格选取最佳迭代次序、松弛因子与源项最佳线性处理方法。针对某型号航空发动机双收敛圆环套状尾喷管内按实际尺寸大小、工况条件掺混的情况,计算了流场、温场等特征量的分布。

2.1 流场分布

喷管内流场矢量分布如图 1 所示,从中可见,主辅气流之间存在一个很强的卷吸作用,使

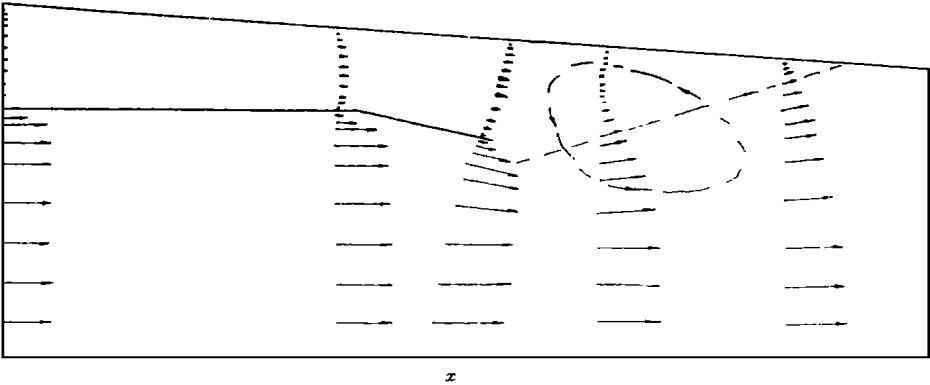


图 1 喷管内流速矢量分布图

延伸段近外壁处产生一个很强的涡,从而导致冷热气流之间的强烈掺混效应,主气流对离出口 0.12m 附近的外壁面有冲击作用,与实验结果是吻合的。

2.2 温度分布

从图 2 双收敛尾喷管内等温线分布可见, 温度边界层在内喷管壁面附近是存在的, 且向下

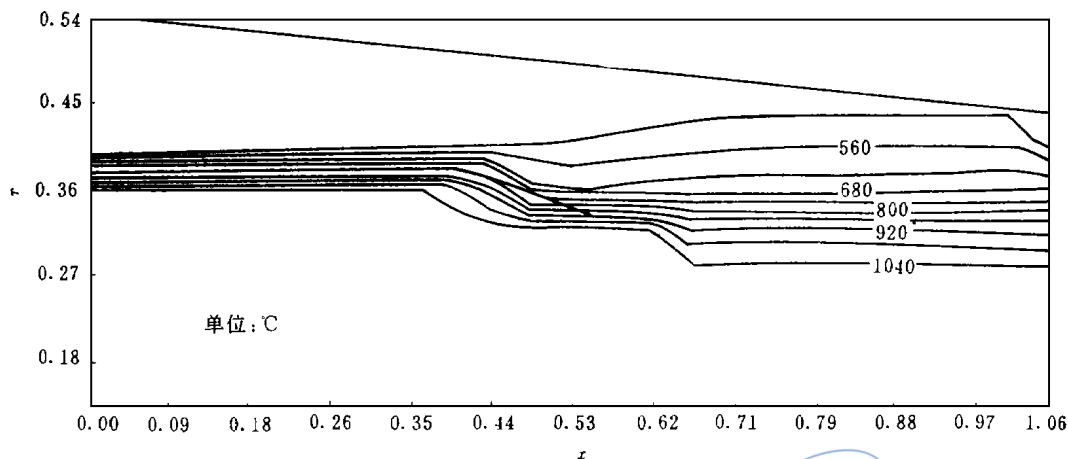


图 2 双收敛尾喷管内等温线分布

游有一定发展但不显著。在内喷管收缩及出口处, 由于壁面收缩作用, 内喷管的等温线向中心轴收缩; 外喷管的等温线近内喷管壁面附近的, 开始向中心轴线收缩; 流动一定距离之后, 又向外扩展; 不靠内喷管壁面的便向外扩展。这说明在内喷管出口处之后, 内外喷管热冷两股流出现了强烈的流动、温度掺混效果, 热量从内喷管的热气流迅速传输给引入的冷气流射流, 温度下降, 其红外辐射量必定下降。与圆环喷管的结果进行比较, 以发现双收敛喷管的掺混降温效果要比圆环喷管的要好。在外喷管出口处附近, 因外喷管是微收敛型, 等温线又有些向中心线收缩, 将再次显示内喷管出口处那种强烈掺混降温效果。

参 考 文 献

- 1 周力行. 湍流两相流动与燃烧的数值模拟. 北京: 清华大学出版社, 1991
- 2 Launder B E, Spalding D B. Numerical Computation of Turbulent Flows. Computational Methods in Appl. Mech. Eng., 1974, 3
- 3 Rubin S G, Hairis J E. Numerical Studies of Incompressible Viscous Flow in a Driven Cavity. NASA SP- 378, 1975

(责任编辑 杨再荣)

MODIFICATION OF STATOR IN AN AXIAL FLOW COMPRESSOR— STAGE

Cao Renjing and Tao Deping

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

ABSTRACT

According to the detail analysis of the experimental results for the stator of an axial flow stage, the main factors of the stator losses have been found. The end treatment technique— hub swept technique, which can enhance the efficiency in stator hub under high blade loading, has been used in the modification. The redesigning scheme for the stator and its end treatment are presented, and the detailed through-flow calculation and blade profiling have been carried out. Optimization of stator parameters has been made in the modification. The results of the modification show that the end treatment technique and blade profiling help improve the efficiency in end-wall region.

Key words Axial flow compressors Stator blades Discharge current capability
Blade shape construction

NUMERICAL SIMULATION OF WAVES CONSTRUCTION IN TRANSONIC NOZZLE FLOW

Jiang Xinghong and Wu Xingping

(College of Aerospace, Northwestern polytechnic University, Xi'an 710072)

Chen Fulian

(Mechanical Engineering Dept., Zhejiang University of Technology, Hangzhou)

ABSTRACT

The Split—Coefficient Matrix (SCM) method is manipulated here for the numerical computation of Euler equations so as to simulate the two-dimensional axisymmetric transonic nozzle flow. The coefficient matrix is splitted directly by a set of simplified formula. A second-order accurate explicit scheme is constructed to compute inner grid points. The scheme is free of artificial viscosity and is economical in terms of computer memory. 5151 grid points are embedded into the nozzle flowfield and the computational results are in good agreement with experiments. Once the flow field is known, the Mach waves characteristic relations are numerically solved by Euler method. Based on the graphic output of the Mach number and Mach waves in nozzle, it is possible to demonstrate not only the formation and reflection of an oblique shock within a tested nozzle geometry but also its peculiar zonal distribution of expansion and compression waves. As a result excellent consistency between parameter distribution and waves construction is revealed.

Key words Transonic nozzles Numerical simulation Expansion waves Compression waves

NUMERICAL SIMULATION OF FLOW AND TEMPERATURE FIELDS IN DOUBLE CONVERGENT NOZZLES

Xia Chunlin and Liu Dezhang

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

ABSTRACT

A numerical simulation was employed to study the flow and temperature fields of hot exhaust gas mixing with cold air in exhaust ejector. In the calculation the algebraic stress model (ASM) with the effect of buoyancy forces as turbulence model was adopted. Four heat radiation flux model was taken as heat radiation model. Due to complicated geometry of the nozzles the boundary-fitted curvilinear coordinates were employed and curvilinear mesh was generated with the help of TTM method, then the blocks were put together under the given boundary conditions. A new general numerical method is developed on the basis of SIMPLE (Semi-Implicit Method for Pressure Linked Equations), and a general software was compiled in the boundary-fit-

ted curvilinear coordinate system. This software can be used widely for numerical simulation of flow fluid, heat and mass transfer, combustion, radiation in mixed flow with arbitrary boundary shapes. The results for some aeroplanes illustrate that the double convergent walls of the nozzles can intensify the mixing of hot and cold flows, and the violent mixing can greatly reduce the heat radiation flux and the temperature of aeroengine exhaust gas in double-convergent nozzles.

Key words Air breathing engines Numerical simulation Exhaust nozzles
Numerical computation Convective heat transfer

EXPERIMENTAL STUDY ON ATOMIZATION PERFORMANCE OF COAXIAL INJECTORS AT BACK VESSEL PRESSURE

Wu Jinxiang, Liu Weidong, Zhou Mong, Wang Zhenguo

(1st Dept. National University of Defence Technology, Changsha 410078)

Zhuang Fengchen

(Institute of Command Technology CONDST)

ABSTRACT

A MF Laser Scatter Sizer and an experimental system designed by the authors were employed to measure the sprays produced by coaxial injectors of LO_x/LH_2 liquid rocket engines at back pressure in the vessel. Experimental results show that injectors work as both air-blast and pressure nozzles; keeping the air/liquid mass flux ratio (ALR) constant, the atomization performance is improved when the air pressure drop (ΔP_g) is increased; and enhanced back pressure results in bad atomization of the liquid. Experimental correlations between spray mean size (SMD) and its main influential parameters have been obtained by fitting experimental data. Several injectors with different configurations have been investigated and the results obtained from the experiments provide valuable reference for injector design.

Key words Injector Atomization back pressure

APPLICATION OF DETECTION FILTER TO SENSOR FAULT DETECTION AND ISOLATION

Du Kuanwen, Zhang Hongyue, Wang Yulin

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

ABSTRACT

Detection filter, which is actually a specific class of observers, can fix the output error direction of the system, so that it can be associated with a particular failure mode and its known design failure direction. The detection filter algorithm developed by White & Speyer is applied to a sensor fault detection and isolation system in an aeroengine control system. A nonlinear engine computer simulation model is set up and used to process the close-loop simulation which includes sensor fault detection and isolation system. An isolation method is presented to isolate the fault sensor location. The results of the simulation show that the present fault detection and isolation system is very effective.

Key words Fault detection Linear systems Sensors Detection filters