

多斜孔冷却技术分析

北京航空航天大学 孟祥泰** 刘高恩

【摘要】 通过燃烧室火焰筒壁一种先进冷却技术(多斜孔壁)的计算分析表明,采用这种冷却技术的火焰筒壁温要比纯气膜冷却的火焰筒壁温约低 150K,并且指出这种冷却技术中的内对流冷却项是使壁温降低的一项重要因素。

关键词: 空气喷气发动机 环形燃烧室 燃烧 冷却 火焰筒

分类号: V235 V233.5

1 比较用的假定环形燃烧室的基本参数及纯气膜冷却计算

在近期公布的美国 GE 公司^[未公开发表]GE-90 发动机燃烧室资料中的火焰筒采用了多斜孔冷却技术(见图 1),该发动机的压比为 40(起飞状态),燃烧室进口的温度及压力均比 E³(Energy Efficient Engine)^[1]的要高,且温升亦比 E³ 燃烧室的高 100K 左右,对火焰筒冷却要求有很高的有效性。因此,本文主要是用计算对比的方法来研究多斜孔冷却技术的特点及其冷却有效性。分析比较的基础是在相同的进口条件及相同的环形燃烧室,一种是采用纯气膜冷却,一种是采用多斜孔冷却技术。

燃烧室进口压力 P_2 为 3040kPa,进口温度 T_2 为 880K,空气流量 $\dot{m}_a = 78\text{kg/s}$,燃烧区油气比 $f/a = 0.065$,外火焰筒直径 $D_{t,o} = 0.7\text{m}$,火焰筒长度 $l = 180\text{mm}$,内火焰筒直径 $D_{t,i} = 0.46\text{m}$,火焰筒高 $H_i = 0.12\text{m}$,外环腔气流通道面积 $A_{an} = 0.065\text{m}^2$ 。气量分配:火焰筒头部旋流空气占总空气量的 15%,火焰筒头部冷却空气占总空气量的 8%,外火焰筒冷却气量占总空气量的 14.0%,内火焰筒冷却气量占总空气量的 11.0%,外主燃孔空气量占总空气量的 15.2%,内主燃孔空气量占总空气量的 15.2%。火焰筒壁材料的幅射出射度 $\epsilon_w = 0.7$,火焰筒壁材料的导热系数 $K_w = 26\text{W/(m} \cdot \text{k)}$,燃烧室机匣(铝)的 $\epsilon_c = 0.4$,火焰(煤油加空气)的亮度因子 $L = 1.7$ 。

纯气膜冷却的外火焰筒壁温计算^[2]。按前述气量分配,外火焰筒冷却气量占总空气量的 14.0%,设计为 4 段冷却气膜,而 1 至 4 段气膜的气量占燃烧室总量的分配比例分别为 3.36%、4.62%、3.36%及 2.52%。因此相应这 4 段气膜的气量分别为 2.81kg/s、3.60kg/s、2.81kg/s 及 1.97kg/s。重点计算第 2 段气膜,设这段气膜的长度为 45mm(注:按燃烧室长度 180mm 除以 4),假定冷却缝槽高度 s 为 2mm。规定 x 是离气膜进口的轴向距离, x_p 是冷却气流的核心区长度,因此该段气膜末端处的 $x = 0.044\text{m}$, $x/s = 22$ 。冷却槽缝环面积 $A_s = 43.98$

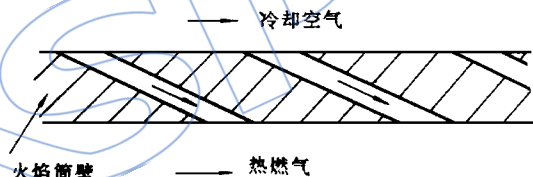


图 1 多斜孔冷却火焰筒壁简图

$\times 10^{-4} \text{ m}^2$, 冷却槽缝环内气流量为 $\dot{m}_s = 3.6 \text{ kg/s}$ 。冷却空气密流 $\rho_{a,u} = 818 \text{ kg}/(\text{m}^2 \cdot \text{s})$, $Re_s = \rho_{a,u} \cdot s / \mu_a$, 而空气粘性系数 $\mu_a = 3.89 \times 10^{-5} \text{ kg}/(\text{m} \cdot \text{s})$, $Re_s = 4.2 \times 10^4$, $Re_x = 92.5 \times 10^4$, $\dot{m}_g = 41.34 \text{ kg/s}$, 火焰筒面积 $A_l = 0.22 \text{ m}^2$, $\rho_g u_g = 187.85 \text{ kg}/(\text{m}^2 \cdot \text{s})$, 密流比 $M = 4.35$, $x/Ms = 2.2$, 气膜有效性 $\eta = 1.0 - 0.12 s_N^{0.65}$, 其中 $s_N = [(x - x_p)/Ms] \cdot Re_s \cdot (\mu_a/\mu_g)]^{-0.15} \cdot (A_s/A_{eff})$, 式中 A_{eff} 是流动有效面积, 取 $A_s/A_{eff} = 2.5$, 经计算, $s_N = 2.29$, $\eta = 0.79$ 。

计算燃烧区的燃气温度 T_g 。按油气比、进口压力及温度查得燃烧区的理论温升为 1722 K , 假定燃烧区效率为 90% , 因此实际温升为 $1722 \times 0.9 = 1550 \text{ K}$, 再考虑燃油热分解及蒸发消耗 50 K , 所以最终的实际温升为 1500 K , $T_g = T_a + \text{温升} = 2380 \text{ K}$, $\eta = (T_g - T_{w,ad}) / (T_g - T_a)$, $T_{w,ad}$ 为绝热壁温, 这样算得的 $T_{w,ad} = 1195 \text{ K}$ 。

热燃气对火焰筒壁的单位面积的对流换热为 $C_1 = 0.1 \times (K_a/x) \cdot Re_x^{0.8} (x/s)^{-0.36} \cdot (T_{w,ad} - T_w)$, 式中的 K_a 为空气导热系数。热燃气对火焰筒壁的单位面积的辐射换热为 $R_1 = 0.5\sigma(1 + \epsilon_g)\epsilon_g T_g^{1.5}(T_g^{2.5} - T_w^{2.5})$, 式中的 σ 为玻尔兹曼常数 $5.67 \times 10^{-8} \text{ W}/\text{m}^2 \cdot \text{K}^4$, ϵ_g 是热燃气的辐射射度, $\epsilon_g = 1 - \exp[-290PL(q/b)^{0.5} \cdot T_g^{-1.5}]$, 式中的 P 为燃烧室压力, 单位 kPa , q 油气比, L 亮度因子, b 辐射路程长度 $= 0.6 \times$ 火焰筒高度, 单位 m 。火焰筒壁对机匣的辐射换热 $R_2 = 0.4\sigma(T_w^4 - T_a^4)$ 。而火焰筒壁与外环腔空气的对流换热 $C_2 = 0.02 \times (K_a/D_{an}^{0.2}) \cdot (\dot{m}_{an}/A_{an}\mu_a)^{0.8} (T_w - T_a)$, 式中 D_{an} 是外环腔高度的 2 倍, K_a 是空气导热系数, $\dot{m}_{an}$ 是外环腔内的空气流量, 而 A_{an} 是外环腔通道面积, 其中的 μ_a 是空气粘性系数。

热平衡方程为 $R_1 + C_1 = R_2 + C_2$, 算得 $T_w = 1428 \text{ K}$, 或 $t_w = 1155^\circ \text{C}$, 这就是第 2 段气膜末端处的外火焰筒壁温, 当然这里假定了壁厚很薄亦即内外壁的壁温是一样的。

2 多斜孔火焰筒壁的壁温计算

同样, 对于上述环形燃烧室, 在相同的冷却气量及进口条件下按多斜孔火焰筒设计方案来计算其火焰筒的壁温。要计算合适的多斜孔几何, 即角度、孔径及孔距等参数, 先假定多斜孔与 x 轴的夹角为 30° , 外火焰筒壁厚为 2 mm , 外火焰筒壁的冷却空气量仍按 $14\% \dot{m}_a = 10.92 \text{ kg/s}$, 外火焰筒壁的侧表面积为 0.40 m^2 , 跨过火焰筒的 $\Delta P/P = 3.5\%$, $\Delta P = 30 \text{ kPa} \times 3.5\%$, 根据柏努利方程, $\Delta P = \zeta \cdot (1/2) \rho V^2 = 1/2 \rho V^2$, 式中 ρ 为密, V 为孔内流速, ζ 是流过小孔口及小孔时的水力损失系数, 取 $\zeta = 0.15$, 而 ρ 可由进口压力及温度算得, 这样算出的 $V = 123 \text{ m/s}$ 。取小孔直径为 0.5 mm , 则小孔流通面积 F_d 为 $19.63 \times 10^{-8} \text{ m}^2$, 孔数 $n = \text{冷却气量}/(F_d \rho g v) = 38425$, 假定孔的轴向间距为 a , 周向间距为 $a/2$, 则按孔数及火焰筒长度 l 及火焰筒周长 $\pi D_{l,o}$ 可以算出 a , 即 $(l/a) \times (\pi D_{l,o}/(a/2)) = n$, 算得的 $a = 4.54 \text{ mm}$ 。这样的多斜孔外火焰筒沿轴向共有 40 排孔, 每排孔的孔数为 969 个孔, 考虑排间以交叉方式排列。

对于多斜孔火焰筒壁的热平衡应考虑壁内小孔中内对流这一项, 即 $R_1 + C_1 = R_2 + C_2 + C_{内}$, 式中的 $C_{内}$ 就是内对流项, 但上述平衡方程是对火焰筒单位侧表面积写的, 因此 $C_{内}$ 亦必需折算到单位侧表面积来, 为此可以这样来折算: 小孔的总侧表面积(换热的孔表面积)应为 $n \cdot \pi d_{小孔} l_{小孔} = n \pi \cdot d_{小孔} \cdot \text{壁厚}/\sin 30^\circ$; 而外火焰筒侧表面积为 $\pi D_{l,o} \cdot l$, 因此折合因子 f 为总的小孔侧表面积除以外火焰筒的侧表面积, 即 $f = (\pi d_{小孔} (\text{壁厚}/\sin 30^\circ)) / (\pi D_{l,o} \cdot l) = 0.610$, 所以 $C_{内} = 0.0185 (K_a/d^{0.2}) (\dot{m}_{小孔}/A_{小孔}\mu_a)^{0.8} (T_w - T_a) \cdot f$ 。式中 d 为小孔直径(m)。

关于 C_1 的计算。多斜孔喷出的多股射流在火焰筒的燃气侧形成了一层保护膜, 类似于气膜。对于这样的保护气膜, 其有效性 η 是需要加以研究的, 但从分析比较的目的来讲, 我们可以

假定不同的 η 值, 且把这样多孔射流折合成二维的环形槽, 折合关系可以这样考虑: 前已述共有 40 排(轴向)孔, 每排 969 个孔, 因此对每排的冷却气量 $\dot{m}_s = 14\% \cdot \dot{m}_{al} / 40 = 0.273 \text{ kg/s}$, 折合的冷却缝高 s 假定为 0.5 mm , 所以 $\rho_a u_a = \dot{m}_s / \pi \cdot D_{l,o} \cdot s = 248.28 \text{ kg/(m}^2 \cdot \text{s)}$, $Re_x = \rho_a u_a \cdot x / \mu_a$, 按前述多斜孔几何尺寸, $x = 4 \text{ mm}$, 求得 $Re_x = 2553.01$, 而

$C_1 = 0.1(K_a/x) \cdot Re_x^{0.8}(x/s)^{-0.36}(T_{w,ad} - T_w) = 3484(T_{w,ad} - T_w) = \alpha(T_{w,ad} - T_w)$
设 $\eta = 0.79$, 按 $\eta = (T_g - T_{w,ad}) / (T_g - T_a)$ 可以算出 $T_{w,ad} = 1195 \text{ K}$, 根据热平衡关系 $R_1 + C_1 = R_2 + C_2 + C_{内}$, 可算出 $T_w = 1130 \text{ K}$, 同样假定 $\eta = 0.65$, 算得的 $T_w = 1230 \text{ K}$; $\eta = 0.50$ 时的 $T_w = 1330 \text{ K}$ 。

将以上的多斜孔计算的 T_w 与纯气膜冷却的 T_w 作个比较, 纯气膜冷却 $\eta = 0.79$, $T_w = 1428 \text{ K}$; 多斜孔冷却 $\eta = 0.79$, $T_w = 1195 \text{ K}$, $\eta = 0.65$, $T_w = 1230 \text{ K}$, $\eta = 0.50$, $T_w = 1330 \text{ K}$ 。可见多斜孔冷却的壁温要比纯气膜冷却的低很多, 估计多斜孔冷却的气膜有效性相对比纯气膜冷却的有效性 η 要低些。所以粗略地讲, 多斜孔的壁温比纯气膜的壁温要低 $150 \text{ K} \sim 200 \text{ K}$, 在计算过程中, 比较热平衡方程中各项量的数值, 明显可以看出其中 $C_{内}$ 项是非常有效的, 亦就是在多斜孔冷却技术中的内对流项是使壁温降低的重要因素。

对于多斜孔冷却方案取了 4 个 η 值及 2 个 α 值, η 为 $0.9, 0.7, 0.5, 0.3$, 而 α 取 3484 和 2000, 以利于比较并分析 α 的影响, 计算结果见表 1。从表 1 中可以看到, 当 $\eta > 0.70$ 时, α 的

表 1 多斜孔冷却方案的 T_w 计算结果

η	0.90	0.90	0.70	0.70	0.50	0.50	0.30	0.30
α	3484	2000	3484	2000	3484	2000	3484	2000
T_w	1060	1070	1200	1170	1330	1265	1468	1360

影响不大, 在 $\eta < 0.70$ 时的 α 对 T_w 有较大影响。多斜孔冷却方案 η 太低亦是不行的, $\eta < 0.3 \sim 0.4$ 是不理想的。从具体的计算 C_1 项时发现, 当 η 高达 0.8 到 1.0 时的 C_1 是负的, 即气膜冷却火焰筒壁; 而当 η 低时(< 0.8)的 C_1 是正的, 即气膜对壁加热。由此亦可推论, 当 η 高时换热系数 α 高较为有利, 而 η 低时 α 低较为有利(壁温 T_w 低)。

3 结 论

以上的计算是基于二维的方法并假定壁温沿火焰筒周向及径向是均匀的等等, 这些因素都会使计算的结果在一定量上不太准确, 但终究这样的计算比较方便, 易于从物理上了解与分析多斜孔冷却的主要优点。根据计算结果可以得到以下的结论: (1) 在现代先进燃烧室条件下采用多斜孔冷却技术的火焰筒壁温可以比纯气膜冷却的火焰筒壁温低 150 K 左右, 对此主要是内对流项作出了主要贡献。(2) 多斜孔喷出的多股射流在火焰筒内壁形成的气膜, 其气膜冷却有效性 η 必须保持有较高的值($> 0.4 \sim 0.5$), 因此有必要对于多斜孔冷却的各种方案进行研究, 亦即对绝热壁温 $T_{w,ad}$ 展开必要的研究。(3) 多斜孔喷出的多股射流所形成的气膜, 当 $\eta > 0.8$ 时, 对火焰筒壁起冷却作用; 而当 $\eta < 0.8$ 时, 对火焰筒壁起加热作用。

参 考 文 献

1 E³ 文集 4 分册. 航空工业总公司 628 所, 1992
2 金如山. 航空燃气轮机燃烧室. 北京: 宇航出版社, 1988

(责任编辑 杨再荣)

ty under bending load is obtained from researching the event. Two new random variables K_w and D_w are proposed to describe the random event of bending fatigue failure of the gear. K_w is the fatigue damage strength which indicates the inherent ability of the gear to resist the fatigue damage under bending load. D_w is the overall fatigue damage quantity which indicates the fatigue damage degree of the gear under bending load in the life span. A cumulative fatigue damage probabilistic model of the gear fatigue strength estimation under bending load is established by means of K_w and D_w . Based on the model, a predicting method is provided for the gear bending fatigue reliability.

Key words Gears Rotational bending fatigue Computational methods Reliability

EXPERIMENTAL INVESTIGATION ON CONVECTIVE BURNING IN A BED OF GRANULAR PROPELLANT BED

Yang Tao

(1th Dept., National University of Defense Technology, Changsha 410078)

Jin Zhiming

(Nanjing University of Science and Technology, Nanjing 210016)

ABSTRACT

A semi-closed bomb with 100MPa pressure limits was designed to investigate the ignition transient, flame spreading, and pressure wave propagation in a packed bed of granular propellant charge. The influences of parameters such as propellant properties, porosity, and blow-out diaphragm thickness on the convective burning characteristics were examined. The 1-D pressure wave propagation was observed in the experiment. The dynamic impaction of the bed exists during the flame spreading process, which is a main factor to cause the pressure wave. The 1-D combustion wave can not be formed in the bed with low loading density.

Key words Propellant combustion Pressure waves Internal ballistic tests
Combustion stability

EXPERIMENTAL INVESTIGATION ON FULL COVERAGE DISCRETE HOLE FILM COOLING

Wu Weiming

(Aero-gas Turbine Establishment)

Zhang Qingfan and Li Yafeng

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

ABSTRACT

A heat shield with full coverage discrete hole film cooling has practical application in some afterburners. Effects of geometric parameters and flow parameters of the heat shield on effectiveness of film cooling were investigated experimentally. A convective heat transfer coefficient of the film cooling was defined as $h_g = q_c / (T_g - T_w)$, where q_c is heat flux obtained by solving heat balance equation for the tested plate with discrete holes, and T_g and T_w are gas temperature and tested plate wall temperature, respectively. A semiempirical expression for calculating heat transfer coefficient is provided for a given diameter of the discrete holes and their distribution. It might be useful to calculating heat shield temperature.

Key words Airfilm cooling blades Cooling Experiment Combustion chambers

MULTI-HOLE COOLING TECHNOLOGY ANALYSES

Meng Xiangtai and Liu Gaoen

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

ABSTRACT

This paper is to analyse the advantage of the multi-hole cooling technology. The calculation showed that the combustor liner temperature with multi-hole cooling technology would be 150K to 200K less than the wall temperature with film cooling technology, and the term

of inner convection within the multihole is the main factor to make the wall temperature being lower.

Key words Combustion Cooling Flame tubes

APPLICATION OF DIACTUATOR DISC THEORY TO AERODYNAMIC COMPUTATION OF AXIAL- FLOW FAN WITH SKEWED- SWEPT ROTOR BLADES

Zhong Fangyuan, Wang Jianming, Cai Na, Li Zhong

(*Dept. of Power Machinery Engineering, Shanghai Jiaotong University, Shanghai 200030*)

ABSTRACT

In order to spread the advanced technology of aeropropfan, skewed-swept rotor blades are applied to axial flow fan and compressor. The diactuator disc theory model is adopted to calculate the main flow of the skewed-swept rotor blades in such a manner, as all blades forces affect on the first actuator disc, but the radial forces of blades vanish abruptly on the second actuator disc. Then the radial distribution of the boundary layer thickness (wake thickness) on the outlet surface of the rotor blade is taken as aerodynamic and aeroacoustic characteristic vectors. The effects of the secondary flow, endwall boundary layer, blades rotation and skewed-swept regularity (mainly radial forces of blades) are corrected according to plane cascade testing statistical curves. In addition, the results of four calculations demonstrate the effects of the second actuator disc, radial force and corrected outlet flow angle on magnitudes of the calculation results separately. The aerodynamic and acoustic tests show a good agreement of the calculation results with test data.

Key words Blades Blade forces Aerodynamic computations Diactuator disc theory

PROPELLER PERFORMANCE PREDICTION WITH FREE WAKE ANALYSIS METHOD

Chen Zemin and Pan Jieyuan

(*5th Dept., Beijing University of Aeronautics and Astronautics, Beijing 100083*)

ABSTRACT

The calculations of propeller performance at different advance ratios have been completed with the free wake analysis method. The wake is divided into near and far regions. In the case of heavy load when the advance ratio of propeller is small, the contraction of wake has large effect on the performance of the propeller. The free wake analysis method takes the wake contraction into account. According to lifting lines for the blade loading distribution, propeller performance calculated shows a good agreement with experimental data.

Key words Aircraft propellers Trailing vortex Lift line theory Performance analysis

HIGH- SPEED CAMERA MEASUREMENT OF A GAS- SOLID TWO- PHASE FLOW IN AN AXIAL- FLOW FAN

Wang Yunliang, Xu Zhong, Miao Yongmiao

(*Dept. of Power, Xi'an Jiaotong University, Xi'an 710049*)

ABSTRACT

A high-speed camera is used to measure the motion characteristics of particles in an axial-flow fan. In the experiment the measurements of the motion characteristics of the particles are carried out on three circular surfaces. The employed particles are 1mm diameter foamed plastic spheres. From photo processing some valuable conclusions are drawn as following: (1) In the impeller channel, the concentration of the particles nearby the pressure side is higher, while the suction side is lower. And there is an area where no particle exists. (2) The trajectories of the particles approach to straight lines. Some particles collide with the pressure side and rebound