第 6 卷 第 2 期 航空动力学报 Vol.6 No.2
1991 年 4 月 Journal of Aerospace Power Apr. 1991

平板边界层转捩的数值模拟

中国科学院工程热物理研究所 蒋洪德
美国弗吉尼亚理工学院机械系 J. G. Moore, J. Moore

【摘要】 对于自由湍流引起的压力梯度为零的平板边界层转捩过程的数值模拟，本文提出了一个单方程湍流模型（VLM 模型），对湍流粘性系数和湍流长度尺度都进行了修正以正确引入低雷诺数效应。数值计算表明，该模型在自由来流湍流度为 0.03～8% 的大范围内能正确模拟边界层转捩全过程，包括转捩区起点位置和转捩区长度、边界层内的平均及脉动参数分布等。本文还提出了上述转捩过程的时均化五阶段模型。

一、前　　言

对层流到湍流转捩的深入理解并准确预测仍是当代流体力学面临的难题之一。近年来用湍流模型来模拟转捩过程的方法日益受到重视。但通常使用的湍流模型是对充分发展的湍流建立的，因而都不能正确模拟转捩过程。例如目前广泛使用的 k-ϵ 模型用于自由湍流引起的平板边界层转捩预测时，计算结果与实验数据有很大差别。本文对单方程湍流模型进行了研究，在综合实验数据的基础上，对湍流粘性系数和湍流长度尺度都进行了修正。数值计算表明，本文提出的单方程湍流模型在模拟自由湍流引起的边界层转捩时具有良好的能力。

二、充分发展湍流的单方程湍流模型

单方程湍流模型使用湍流动能方程，它包括湍流动能 k 的对流、扩散、生成和耗散四部份。目前常用 Hassid 和 Poreh 于 1975 年[1] 提出的下列方程：

$$\rho U\partial k/\partial x+\rho V\partial k/\partial y=\partial[(\mu_l+\mu_t)\partial k/\partial y]/\partial y+\mu_t(\partial U/\partial y)^2-[\mu_t\cdot C_D/C_\mu+2\mu_l]k/L^2 \tag{1}$$

在高雷诺数和各向同性湍流的假定下，湍流粘性系数 μ_t 和湍流长度尺度 L 可分别表示为：

$$\mu_t=C_\mu\rho k^{0.5}L \tag{2}$$

$$L=\begin{cases}\kappa y\,[1-\exp(A\rho k^{0.5}y/\mu_l)] & \text{内层}\\ (L/\delta)_{\text{outer}}\cdot\delta & \text{外层}\end{cases} \tag{3}$$

方程(1)～(3) 构成了充分发展湍流的单方程湍流模型（简称 HP 模型），其中的常数 $C_D=0.17$，$C_\mu=0.536$，$A=0.0292$，$\kappa=0.41$ 和 $(L/\delta)_{\text{outer}}=0.08$。图 1 是 HP 模型用于压力梯度为零的平板边界层转捩的计算结果以及与实验数据[2,3]的比较。可以看出计算求得的转捩起始位置均大大超前于实验结果、转捩区长度远小于实验值，且转捩后壁面摩擦系数明显低于充分发展湍流状态值。图 2 则表明 HP 模型的计算结果强烈地依赖于平板前缘的初始雷诺数 R_{xo}。

本文于 1990 年 10 月收到。

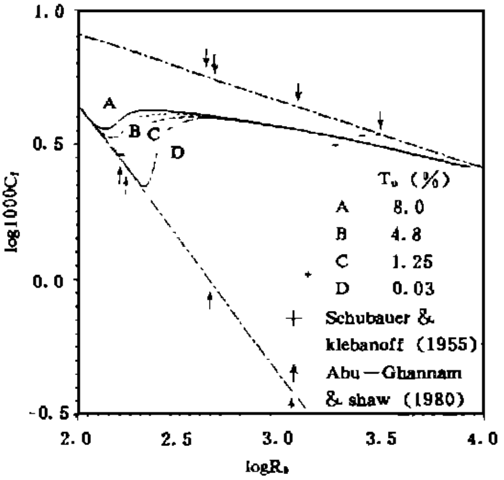

图 1 来流湍流度变化时 HP 模型的计算与实验值的比较

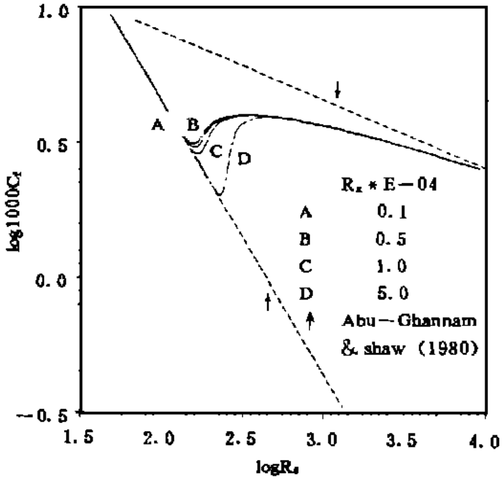

图 2 计算初始雷诺数变化时 HP 模型的计算与实验值的比较

总之，高雷诺数单方程湍流模型不能直接用于转捩模拟。

三、湍流粘性—长度尺度修正（VLM）模型

单方程湍流模型中主要变量是湍流粘性系数 μ_t 和湍流长度尺度 L。对已有实验数据[3]进行综合分析后，本文对 μ_t 和 L 进行如下修正：

1. 使用一个分段光滑的函数 F_{tu} 修正 μ_t，即

$$\mu_t = F_{tu} \cdot C_\mu \rho k^{0.5} L \tag{4}$$

$$F_{tu} = \begin{cases} 0.05 & \delta^+ \le \delta_1^+ \\ 0.05 + 0.95(\delta^+ - \delta_1^+)/(\delta_2^+ - \delta_1^+) & \delta_1^+ < \delta^+ \le \delta_2^+ \\ 1.0 & \delta_2^+ < \delta^+ \end{cases} \tag{5}$$

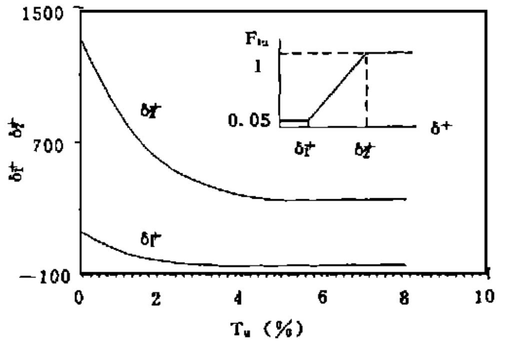

图 3 μ_t 的修正函数 F_{tu} 和 δ_1^+，δ_2^+ 曲线

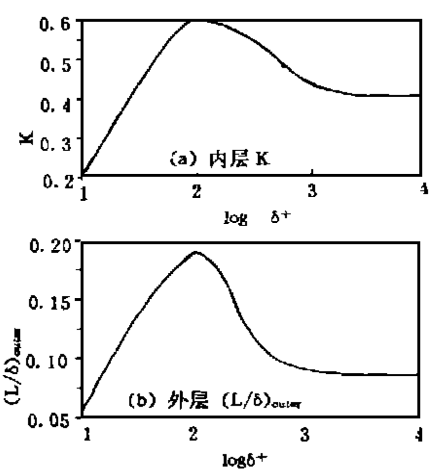

图 4 湍流长度尺度修正曲线

δ^+（$= \rho u_\tau \delta / \mu_l$）是以边界层厚度 δ 和壁面摩擦速度 u_τ 表示的无量纲参数。δ_1^+ 和 δ_2^+ 两个特征量

是根据文献 [2] 的实验数据来确定的, 见图 3。

2. 采用图 4 所示的低雷诺数下湍流长度尺度 L 的两条修正曲线。其特征是内外层的 L 均呈峰值分布且外层的相对增加量大于内层。

3. 为了减少在计算边界层厚度 δ 时的误差, 采用先求 δ_{90} (边缘平均速度为自由流速 90% 的边界层厚度) 再求 δ 的方法。参照的实验数据[3], 本文提出了低雷诺数条件下 δ_{90}/δ 关系曲线 (图 3)。

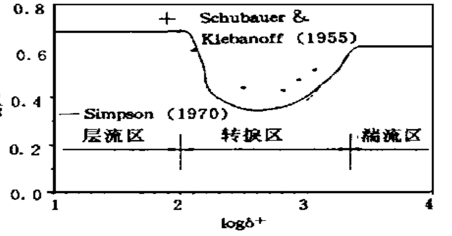

图 5　σ_{90}/σ

四、数值计算结果与讨论

本文求解的基本方程组是二维不可压边界层的连续、动量方程以及由方程 (1) ～ (5) 给出的 VLM 模型。用 Crank-Nicolson 二阶精度空间推进格式求解。计算网格为 350×60, 流向步长 Δx 不大于当地边界层动量厚度的 4～6 倍, 法向第二站设在 $y^+ = 0.25$ 处。所有的计算都从层流状态开始, 除注明者外, 初始截面 R_{xo}均为 10000, 速度分布由 Blasius 解确定。转捩起终点分别取壁面摩擦系数 C_f 的极小与极大值拐点。

图 6 是来流湍流度为 0.03～8% 时 VLM 模型的计算结果与实验数据的比较。可看出, 该模型准确地预测了转捩的起点和终点位置。图 7 则表明 VLM 模型的计算结果与 R_{xo}无关, 这对实际应用十分有利。为进一步核查 VLM 模型的性能, 给出 $T_u = 0.03\%$ 时计算结果与文献 [3] 的详细测量结果的比较。图 8 是形状因子 H 和壁面摩擦系数 C_f 的变化(作为参考, $T_u = 1.25$, 4.8 和 8% 的计算结果亦在图中标出), 图 9 是转捩区内六个不同截面 ($\bar{L}$ 表示该截面在转捩区内的相对位置) 上的平均速度分布。可看出, VLM 模型准确模拟了转捩边界层的平均参数。

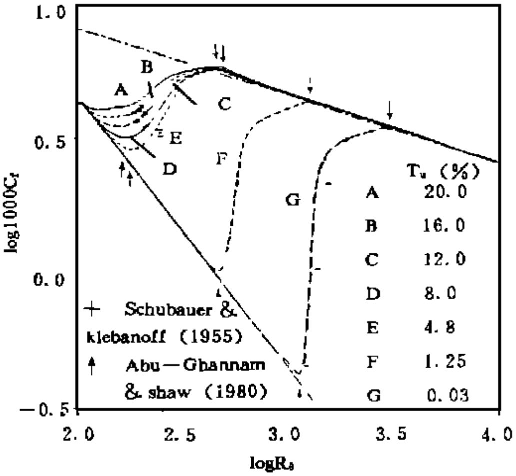

图 6　来流湍流度变化时 VLM 模型的计算与实验值的比较

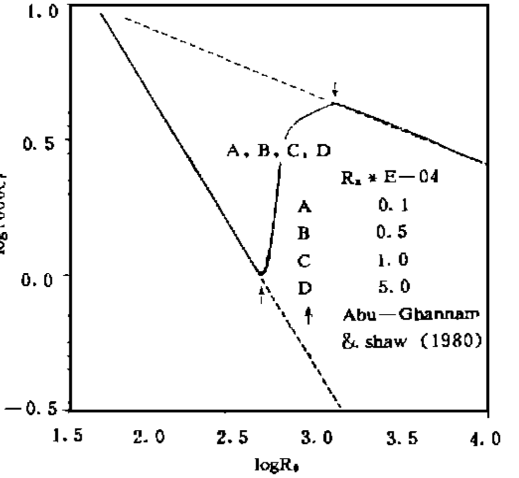

图 7　计算初始雷诺数变化时 VLM 模型的计算与实验值的比较

沿流动方向边界层内湍流脉动速度 (均方根平均值, 下同) 最大值的分布见图10, VLM 模型的计算值除转捩区内的峰值偏低以外, 与实验数据基本吻合。图 11 是转捩区内六个不同截面上湍流脉动速度的分布, 它表明 VLM 模型正确地模拟了转捩的初始和结束阶段边界层内湍流动能的分布, 但在转捩中段未能模拟出近壁面区湍流动能十分陡峭的峰值。

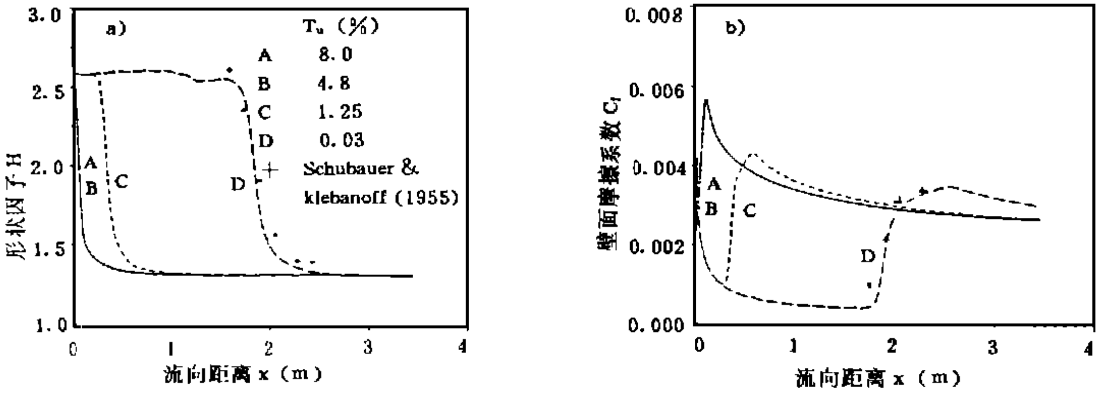

图 8 形状因子和摩擦系数的计算与实验值（T_u= 0.03%）的比较

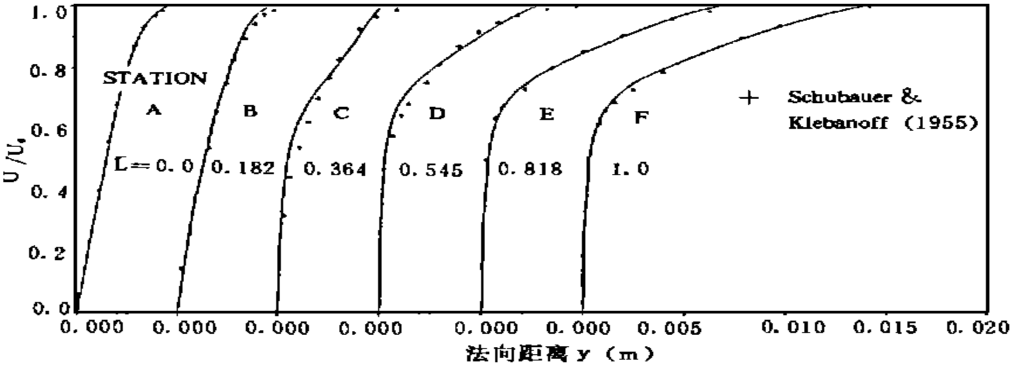

图 9 转捩区内的平均速度与实验值（T_u= 0.03%）的比较

实验研究已表明转捩过程是十分复杂的，它包括二维线性小扰动（TS 波）、三维非线性不稳定及破裂、湍流斑的出现与扩展等一系列过程，本文的数值模拟则是对上述过程作时均化处理。综合本文计算结果，可将由层流到湍流的全过程在时均化的意义上分为五个阶段（图 12）：(1) 完全层流区　在远离转捩起点的上游，在自由湍流作用下层流边界层内小扰动开始形成，但绝对值很小且增长缓慢，数值计算求得的 k 可忽略不计，流动具有完全层流特征；(2) 转捩前区　在自由湍流的持续作用下，边界层外层首先出现了湍流动能的较快增长，其最大值可达充分发展湍流动能值的 20～30%，但内层则呈现一定程度的滞后，即壁面摩擦系数沿流向仍减小，故此阶段仍属传统意义上的层流区；(3) 转捩急变区：从壁面摩擦系数开始偏离层流状态到湍流动能达到其峰值这一区域，其特点是湍流动能的急剧增长且其峰值大大超过充分发展湍流的水平，以及壁面摩擦系数的迅速同步增长；(4) 转捩发展区：湍流现象已生成并逐步进入自持状态，湍流动能开始下降但内层再次出现滞后，即壁面摩擦系数继续增长；

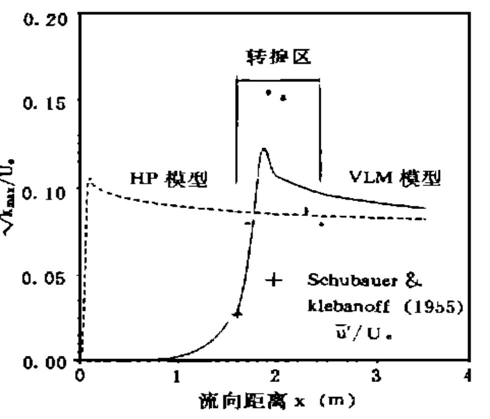

图 10 湍流动能最大值与实验值（T_u= 0.03%）的比较

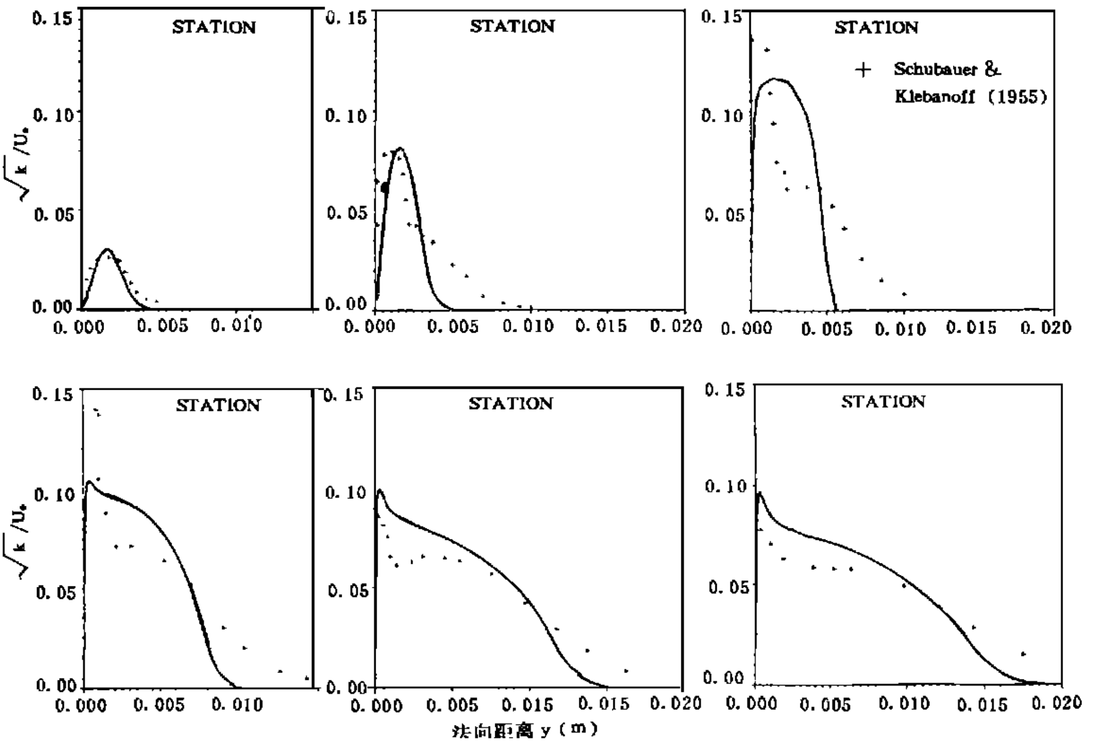

图 11　转捩区内不同截面上湍流动能计算结果与实验值（T_u= 0.03%）的比较

(5) 完全湍流区：壁面摩擦系数达充分湍流值并开始下降，边界层进入平衡湍流状态，转捩过程结束。

结论　在综合实验数据的基础上本文提出一个单方程湍流模型，即湍流粘性—长度尺度修正（VLM）模型，数值计算结果表明：(1) 在来流湍流度为 0.03～8% 的范围内，该模型能准确预测压力梯度为零的平板边界层转捩的起点和转捩区长度；(2) VLM 模型的计算结果与层流边界层入口处的初始雷诺数无关；(3) 来流湍流度为 0.03% 的计算表明，该模型不仅可很好地模拟转捩边界层的平均速度，还可相当准确地模拟湍流脉动参数的分布。

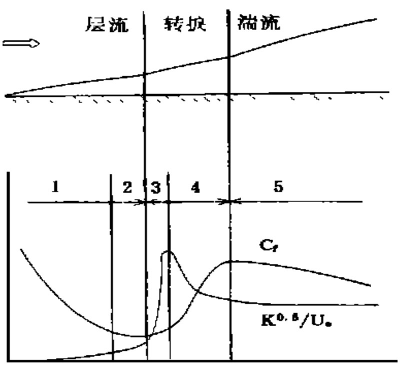

图 12　平板边界层转捩的时均化模型

参　考　文　献

[1]　Hassid, S. and Poreh, M., "A Turbulent Energy Model for Flows with Drag Reduction", ASME Journal of Fluids Engineering, Vol. 97, pp. 234-241, 1975.

[2]　Abu-Ghannam, B. J. and Shaw, R., "Natural Transition of Boundary Layers the Effects of Turbulence, Pressure Gradient, and Flow History", Journal Mechanical Engineering Sciences, IMechE 1980, Vol. 22, No. 5, pp. 213-228, 1980.

[3]　Schubauer, G. B. and Klebanoff, P. S., "Contribution to the Mechanism of Boundary-Layer Transition", NACA TN3489, 1955.

by integrating the momentum equation and then the densities are obtained from the energy equation. Nine numerical examples of three types of the centrifugal compressors show that the solution obtained by the present method is in good agreement with experimental data, provided that there is no large flow separation in compressors.

PREDICTION OF LONG BLADE FLUTTER IN A STEAM TURBINE

Tao Deping, Yang Xiaodong, Zhou Shen

(***Beijing University of Aeronautics and Astronautics***)

ABSTRACT The flow characteristics of the last stage of the steam turbine are analysed on the basis of the results obtained from the cascade experiments and flow field calculation. In the case of low volumetric flow rate, the large scale separation occurs in the last stage rotor hub, and the height of separated flow region increases as the volumetric flow rate decreases. Flow separation appears at the blade pressure surface as the bulk flow enters the rotor with large negative incidence. The flow in the rotor can be divided into three regions — a separated hub flow region, an attached-flow region and a separated-flow region. The dimensions of the three regions vary with the flow rate. The flow pattern in the last stage is very complicated. Aseries of deforming actuator disk methods and a numerical method have been developed to predict the blade flutter. The deforming actuator disk method system is applicable to calcalating 2-D and 3-D unsteady flow fields and also to calculating the bending mode and the torsional mode blade flutters. The analytical and numerical deforming actuator disk methods for predicting the blade flutter in a compressor or a steam turbine have been developed in consideration of variability of or invariabiity of the interblade phase angle. The small-disterbance potential amplitude equation is adopted in the numeried method for unsteady flow in the cascade, while the airfoil thickness and bending degree are arbitrary and the rotating finite difference scheme is employed in the calculaton of transonic flow field. The comparison between the computational and the experimental results shows that the presented methods are feasible in prediction of the flutter in blade design.

NUMERICAL SIMULATION OF TRANSITIONAL FLOWS OVER A FLAT PLATE

Jiang Hongde (***Institute of Engineering Thermophysics***)

J. G. Moore and J. Moore (***Virginia Polytechnic Institute and State University***)

ABSTRACT A new version of one-equation turbulence model for predicting the incompressible, transitional flow over a flat plate is developed. Based on an analysis of the existing one-equation models and some physical features of the transitional flow, both the eddy viscosity and turbulence length scale in the turbulence kinetic energy equation are modified in order to bring the low Reynolds number effect into the new model. The capability of this model for predicting the transitional flow is demonstrated by an extensive numerical testing for flat plate flow under free stream turbulence intensity ranging from 0.03 to 8 percent. Both the start and length of the transition are correctly predicted. Comparison of the calcutated results of both the mean and fluctuating flow properties with experimental data indicates the agreement between them is very good. A time-averaged picture for a complete boundary flow from purely

laminar flow to fully turbulent flow over a flat plate is given

NUMERICAL SIMULATION OF 3-D VISCOUS FLOW IN NON-AXISYMMETRIC NOZZLE

Ji Yongchun (*Tsinghua University*)

Ma Tieyou and Cui Jiya (*Beijing University of Aeronautics and Astronautics*)

ABSTRACT The three-dimensional viscous flow in non-axisymmetric nozzle are simulated numerically. MacCormack finite difference scheme is extended to solving three-dimensional Navier-Stokes equations. Local time step technique is adopted in numerical iteration for rapid convergence. The computer memory and CPU time are saved by proper programming. Two examples of simulation are presented. Favorable agreement is shown between computed and measured surface static pressure distributions.

EFFECT OF HEAT-TRANSFER LAW ON FINITE-TIME EXERGOECONOMIC OPTIMAL PERFORMANCE OF A CARNOT ENGINE

Chen Lingen, Sun Fengrui, Chen Wenzhen

(*Naval Academy of Engineering*)

ABSTRACT The relationship between optimal efficiency and profit of an endoreversible carnot engine is derived. According to this relationship the finite-time exergoeconomic optimal peroformance of the engine has been considered for three common heat-transfer laws. A number of new universal conclusions have been drawn therefrom.

EXPERIMENTAL INVESTIGATION ON LIQUID-COOLING MECHANISM FOR TURBINE VANE

Liu Dezhang and Ouyang Ning (*Nanjing Aeronautical Institute*)

ABSTRACT The experimental investigation on liguid cooling mechanism was carried out with water through an even-heated adiabatic straight pipe. The flow pattern, the heat transfer coefficient along full-flow-path, and the correlation between them, have been studied. In order to imitate the actual working conditions of the cooling passage of turbine vane, the heat transfer flux, the coolant inlet parameters, and the pipe angle are changeable. Some empirical formulas for come-to-saturated point Z_B and come-annular point Z_N have been derived and several useful conclusions have been drawn for engineering application.