

S 弯进气道中自动定位叶片 对气流流动影响的研究*

上海大学 翁培奋**

【摘要】对 S 弯扩压管道内安置自动定位叶片后的气流流动特性进行了实验研究。结果表明,位于模型进气口处的可转动叶片在气流作用下能有两个自动稳定的位置。如果忽略可转动叶片转动轴的摩擦力矩,当叶片处在这两个自动稳定位置时,气流对叶片的作用力矩为零。研究结果还进一步表明,自动定位叶片对 S 弯进气道出口压力场与速度场起着重要作用。本研究为以后如何采取叶片技术抑制 S 弯进气道出口的旋流流动,以及模拟旋流流动提供了重要技术基础。

主题词: 进气道 流动畸变 流动特性 控制问题

分类号: V 228. 71 C 231. 3

1 引言

随着航空与隐身技术的迅猛发展,推进系统中的 S 弯或蛇形进气管道已越来越受到人们青睐。然而以往研究^[1]表明,大攻角进气时管道内的气流流动特性十分复杂,进气道出口速度场存在很大横向分量,十分不利于发动机正常工作。因而如何有效地控制 S 弯管道内气流流动就显得十分重要。文献[1]中用一种扰流器改变气流压力场方法来控制 S 弯管道出口旋流,它可使旋流明显减弱,或出现反向旋转旋流流动。文献[2]中研究了各种导流装置或栅栏改善 30 攻角下 S 弯进气道内旋流场的情况,得出了最佳栅栏安装位置。文献[3]则采用了有源涡控方法即通过抽吸进气口附近涡源处低能量气流来改善旋流的生成与发展。文献[4]采用了可调唇口技术改善了大攻角下(可至 80 攻角) S 弯进气道内的气流流动品质,并得出了攻角与最佳唇口角的单调函数规律。文献[5]中,作者研究了 S 弯进气道中的复杂气流流动控制问题。

本文研究 S 弯扩压进气道中自动定位叶片对管内气流流动特性的影响。其目的在于不但为叶片技术控制气流流动打下良好基础,而且还期待能探索出一条新的 S 形进气道内旋流模拟技术。

2 实验结果和讨论

本研究采用矩形截面 S 弯扩压进气道的实验模型如图 1 所示。它由前直管段、第一弯段、中间直管段、第二弯段和后直管段等 5 个主要部分所组成。有关模型详细介绍可参考文献[5]。

自动可调叶片是安装在模型进气口附近的。它具有 NACA 0012 系列的几何外形,其弦长为 50 mm,最大厚度 3 mm,叶片沿翼展长度为 109 mm。在离前缘 20 mm 处设置一个叶片转

动轴。该转动轴通过螺栓连接模型和叶片,这样既可以固定叶片,还能自由地转动叶片。叶片端面与模型上、下内壁面间存在一个很小间隙(不到 0.5 mm),以利于可调叶片的自由转动。模型进气口装有一角刻度盘,能方便地读出自动可调叶片转动角。自动定位叶片被安置在模型的中心线,好象把进气道分成了两个进气口。这里不妨约定:当自动定位叶片和进气道中心线相切时为 0° 叶片角,叶片后缘靠向小曲率半径侧时为“+”叶片角,反之

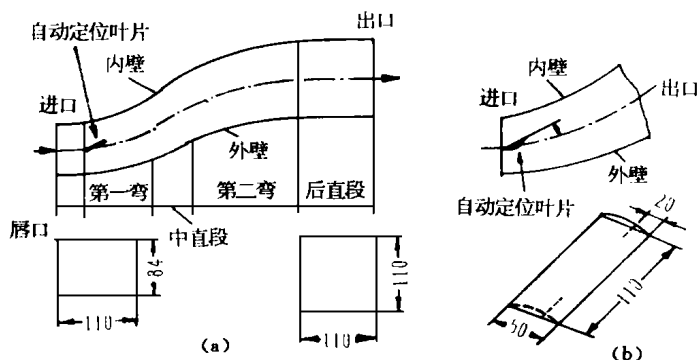


图 1 S 弯进气道模型和自动定位叶片

为负。整个模型实验是在南航进气道实验室里的吸气式风洞中完成的,实验来流速度保证为 100 m/s 。

实验时分别测量了 45 和 60 两种进口导流角下 S 弯进气道在安装自动定位叶片以后的沿程静压、出口截面上的气流压力读数。沿程静压由分布在模型四壁中心线上的压力小孔测出,而出口气流压力则通过三孔探针耙测得。一切压力信号最后都通过两台 DPT-6400 型电子扫描仪自动采集到微机中,并数据处理成所需图形。

主要结果讨论分析如下:

(1) 叶片自动定位现象及其机理初步探索。研究中发现,叶片在 S 弯进气道复杂气流流动中能自动地稳定在两个位置上,其一是 $+15^\circ$ 叶片转动角,另一是 -12° 叶片转动角。当叶片处于这两个位置时,叶片所受的气动力矩出现了平衡,此时的合力矩为零(当然可以忽略叶片转动产生的微小摩擦力矩)。就是说,要使叶片处于这两个自动定位位置时,无需任何外界装置就可实现。出现这种重要物理现象和叶片造成进气口压力场变化有着极为密切的关系,如图 2 所示。图 2(a) 表示当气流流过叶片,由于存在攻角致使在叶片产生了气流分离,形成一个负压力区,而叶盆则生成一个正压区,从而产生了叶片升力。用叶片产生升力机理可能解释这两个自动定位位置存在的可能性。图 2(b) 中可以看到,一方面进口气流造成顺时针方向气动力矩,另一方面由管道弯曲产生横向压力梯度造成逆时针方向气动力矩。这两个气动力矩正好相反,就有可能出现一个叶片位置,那里合气动力矩为零,这就是叶片的自动稳定位置。图 2(c) 则分析

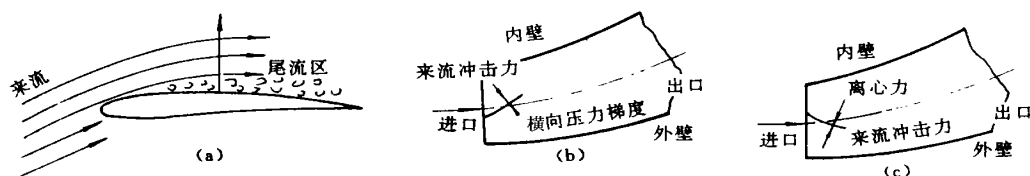


图 2 叶片两个稳定位置的机理分析

了另一个自动稳定位置存在的可能性。进口气流产生逆时针气动力矩,管道弯曲产生的气流离

心力形成顺时针方向的气动力矩, 这两个力矩也正好方向相反, 也有可能找到另外一个自动定位叶片位置, 使叶片没有受到外界合气动力矩的作用。

(2) 自动定位叶片对气流流动的作用。实验研究中还发现改变进口导流角对 S 弯进气道中的自动定位叶片稳定性有影响。当攻角为 45° 时, 自动定位叶片可以很好稳定在 15° 叶片角度以及 -12° 叶片角度上。而当 60° 攻角时叶片在 -12° 角度下稳定性依然很好, 而在 15° 角度下叶片稳定性较差。因而本实验中没有详细地测量 60° 攻角下当叶片处于 15° 角度位置时 S 弯进气道内的气流流动特性。以下主要以 45° 攻角下的实验结果进行分析讨论。

图 3 给出了无措施时 45° 或 60° 攻角下 S 弯进气道出口旋流流动横向速度矢量图谱。从图可见, 由弯曲管道造成横向压力梯度, 使得进口下壁面气流分离区生成并发展成了一个较大单涡式旋流。进口导流角越大, 出口旋流强度也越大, 这和以往研究结果是一致的^[1]。

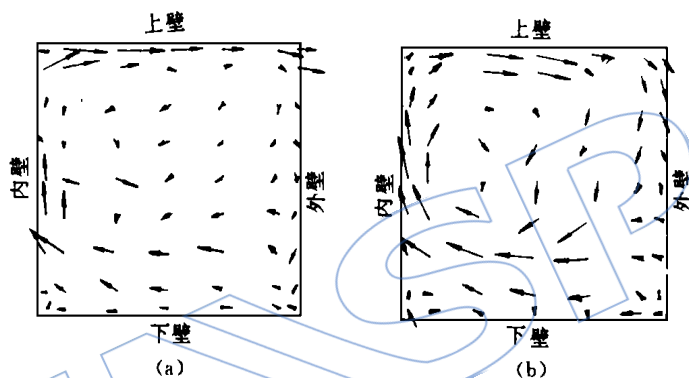


图 3 出口截面速矢图[无叶片(a) 45° 导流角(b) 60° 导流角]

通过测量自动定位叶片处于这两个稳定位置时管道出口气流流动特性, 特别是出口横截面旋流场, 发现叶片在不同稳定位置的出口旋流流动方向相反。当叶片处于 15° 角时, 出口基本上是逆时针方向旋转的单涡旋流。当处于 -12° 角时, 出口是一个顺时针方向旋转的单涡或“S”涡旋流, 如图 4 所示。出现上述现象也和叶片改变了进口附近的压力场有着密切关系。当

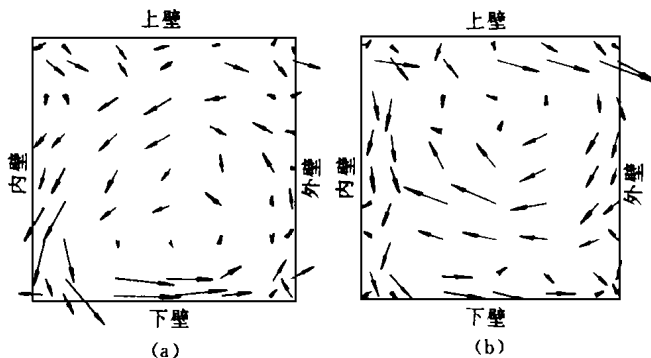


图 4 出口截面速矢图[45° 导流角(a) 叶片角 15° (b) 叶片角 -12°]

叶片处于 15° 叶片角度时(参见图 2b), 进口气流在叶背、叶盆上产生了压力梯度, 这个压力梯

度与弯曲管道产生的截面横向压力梯度正好方向相反,起着彼此抵消的作用。一旦叶片造成的压力梯度比弯度造成的压力梯度要大,那么分离区内低能量气流就会按叶片压力梯度作用生成固体式单涡,它的生成与发展并且与管道中的对称对涡合并,会在进气道出口出现一个逆时针方向旋转的不对称单涡。另外,当叶片处于 -12° 角度时(参见图 2c),上述两种压力梯度作用方向一致,会进一步地增强原来顺时针方向旋流流动。比较图 4(b)和图 3(a)可以得出,一方面顺时针横向流动增大,另外还出现了“S”形流动。

自动定位叶片还对沿 S 弯进气道壁面静压分布以及管道出口的总压场分布有明显影响。就壁面静压分布来说,和无措施相比较,整个变化趋势没有多大改变,但是压力值却有所下降。由于自动定位叶片置于进气道进口附近主流区,对流动总压损失影响较大^[5]。

3 结 论

(1) 可自由转动叶片在 S 弯进气道中气流作用下有两个自动稳定位置,其一为 15° 叶片角,另一是 -12° 叶片角。还发现两个稳定位置对应的出口旋流流动方向相反。

(2) 本研究结果对 S 弯进气道旋流抑制技术以及旋流模拟技术具有重要意义。

参 考 文 献

- 1 Guo R W, et al Swirl Characteristics of an S- Shaped Air Intake with both Horizontal and Vertical Offsets Aeronautical Quarterly, 1983, 32: 130- 146
- 2 Seddon J. Understanding and Countering the Swirl in an S- Duct, Testing on the Sensitivity of Swirl to Fences Aeronautical Journal, 1984, 88(874): 117- 127
- 3 林麒等 S 弯进气道中旋流的有源涡控研究 航空学报, 1989, 10(1): 35- 40
- 4 Weng P F, et al Swirl Control in an S- Duct at High Angle of Attack AIAA Journal, 1994, 32(6): 1321- 1322
- 5 翁培奋 弯曲管道气流流动控制及分离流数值模拟 [学位论文] 南京航空航天大学, 1991

(责任编辑 魏星禄)

depressed by the above measure. The results obtained from numerical simulation of one-dimensional and axisymmetric nozzle flows are presented.

Key Words Rocket engines Nozzle flow Numerical simulation

NAVIER- STOKES PREDICTIONS OF 2- D VECTORED NOZZLE FLOWS

Feng Jianhu

(Dept. of Applied Mathematics, Northwestern Polytechnical University, Xi'an 710072)

ABSTRACT In this paper, the viscous flows in the two-dimensional vectored and unvectored nozzles are numerically simulated by three-dimensional time-dependent Navier-Stokes equations in conservation forms. Finite volume discretization is used in spatial form and Runge-Kutta method in time marching direction. A new viscous terms' discretization in the frame-work of the finite volume method is used which has the advantages of less operation counts and full conservation. A modified Baldwin-Lomax algebraic turbulent model and wall function law are also adopted in the prediction. Comparison of numerical predictions with experiments shows that the Navier-Stokes code can accurately predict the flow fields in the vectored and unvectored nozzle where the flow is predominantly two-dimensional at the designed nozzle pressure ratios or above. Discrepancies between predictions and experiment are found at lower nozzle pressure ratios where separation typically occurs in a large portion of the nozzle.

Key Words Jet nozzles Vector fields Numerical computation

EFFECTS OF GEOMETRIC PARAMETERS OF SUBMERGED DELTA VORTEX GENERATOR ON VORTEX

Zhong Yicheng and Chen Xiao

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

ABSTRACT The effects of the geometric variables of a submerged deltaic vortex generator on the parameters of the vortex have been experimentally studied and a type of half deltaic vortex generator has been developed. A small-size five-hole pressure probe was used to measure the velocity profiles at several transverse sections behind the vortex generators and according to the velocity profiles, the Gauss model of vortex core and quasi-Newton non-linear least square method were used to determine the parameters of single vortex such as vorticity, radius and height of the vortex core. The variation of these vortex parameters along the streamwise was analysed. In the experimental investigation the vortex parameters between the submerged deltaic vortex generator and the submerged concave vortex generator were also compared.

Key Words Swirler Velocity distribution Geometric parameters Vortex

EFFECTS OF AUTOMATICALLY POSITIONING BLADE ON FLOW CHARACTERISTICS IN AN S- SHAPED AIR INLET

Weng Peifen

(2nd Dept. Shanghai University, Shanghai 200072)

ABSTRACT An automatically positioning blade with a NACA 0012-series profile was placed upstream near the throat of an S-shaped diffuser and experimented to investigate the characteristics of the flow behind it. The results show that two angular positions have been found where the blade experiences no resultant aerodynamic moment, i.e. 15° and

- 12°. Moreover, the swirls at the duct exit have different senses of rotation when the blade is held at these two angles. The study lays the foundation for suppression of the duct swirl and its simulation.

Key Words Air inlets Flow distortion Flow characteristics Control problems

DETERMINATION OF REVOLUTION DIFFERENCE LINE FOR TWIN-SPOOL TURBOJET ENGINE

Shang Yi and Guo Yufei

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

ABSTRACT The revolution difference of the low and high pressure rotors is a parameter for acceptance tests of a twin-spool turbojet engine. It indicates the relation between the reduced revolution of the high pressure rotor and the reduced turbine inlet temperature and also shows the matching between the low and high pressure rotors. In order to determine the revolution difference for a twin-spool turbojet engine it is necessary to define a standard revolution difference line for this type of engine. This paper analysis considers various possible approaches for the revolution difference line computation and suggests how to define the standard revolution difference line in practice.

Key Words Twin-spool jet engines Rotation speed Differences Computation

STUDY ON AIRCRAFT ENGINES USED IN COAL-FIRED DIFFERENT-VELOCITY BED COMBINED CYCLE SYSTEM

Zhao Li, Xu Xiangdong, Zhu Weimin

(Dept. of Thermal Power Engineering, Tsinghua University, Beijing 100080)

ABSTRACT A new combined cycle system for generating electrical power from coal-carried heat Partial Gasification Coal-fired Combined Cycle (CGCC) is introduced by Tsinghua University, Which adopts a gas turbine modified from FT-8 aeroengine. Through analyzing the function of the gas turbine, using modeling technique, the design and off-design regimes of the gas turbine are considered comprehensively. It is proved that application of the FT-8 to CGCC system has a great advantage.

Key Words Combination Cycles Gas turbines Performance analysis

QUASIDYNAMIC ANALYSIS OF HIGH-SPEED THRUST BALL BEARINGS

Luo Zhushan, Wu Linfeng, Sun Xinde

(5th Dept. Nanjing University of Aeronautics and Astronautics, Nanjing 210016)

Gao Xiangqun

(Nanhua Powerplant Research Institute, Zhuzhou 412002)

ABSTRACT A quasi-dynamic analysis of high-speed thrust ball bearings is proposed to predict the kinematic characteristics. The rolling velocity and relative sliding velocity of each point in contact zone of balls and rings in a ball bearing are determined at first. According to the elastohydrodynamic and hydrodynamic lubrication theories, the reactions between the rolling pairs and the effects of heat and oil starvation on the thickness of oil film are considered. Then the principles for setting up the basic equations of the balls, the cage and the inner ring are established for the quasi-dynamic analysis of the thrust bearings. The kinematic characteristics, the load distribution, the thickness of the oil film and the fatigue life of the ball bearings can be obtained from solution of those equations. It is shown that the