

轴流压气机进口小总压径向畸变 对失速颤振的影响

北京航空航天大学 冯毓诚^{**} 赵秀华 胡宗安 屠 棘 王雨渭

【摘要】 对进口流场畸变失速颤振的影响进行了实验研究。实验结果表明: 小的叶尖径向畸变能抑制颤振的发作。对此现象用三维变形激盘法进行了计算。计算结果与实验符合。

主题词: 径向畸变 颤振 轴流式压气机

分类号: V215.34 V235.113

1 前 言

进气畸变是限制飞机机动飞行包线的决定因素。目前, 进气畸变对压气机失速颤振性能的影响研究得极少, 已发表的研究大多集中在失速颤振产生的机理、预估方法上。P&W 公司的 FDRC 在高空舱中对 F100-PW-100 及 J85-GE21 进行了实验。所选择的模拟飞行条件是以发动机会出现过的气弹不稳定性为基础的。对 F100-PW-100 一级风扇, 通过改变进口气流参数得到 33 个颤振数据点。对应的进口压力在 85000~211000 Pa, 进口温度在 135~171 °C。发动机的折合转速变化范围为 64%~73%。实验结果表明, 进口压力与温度变化对颤振边界位置有显著影响。增加进口压力及温度都使失速颤振边界朝着共同工作线移动。

对进口流场畸变与叶片颤振之间的交互作用则研究得更少。在机理方面尚未见到哪怕是象平行压气机这类简单模型的发表。至于实验则只在文献[1]中报导了超音非失速颤振影响的一些结果。指出当使用畸变大的进气道时, 风扇发生颤振的折合转速反而高, 而气流均匀的进气道却对应着最低的折合转速。而对失速颤振的影响至今未见有报导。

本文主要是利用 BF-1-2 颤振转子, 在过去研究工作的基础上进行了进口总压叶尖径向畸变的研究。结果表明, 小的叶尖径向畸变可以改善气动弹性稳定性。用三维变形激盘法计算出的结果与实验是符合的。该结果对研究抑制颤振发作的工程措施, 以及开展主动控制的研究都是有重要参考价值的。

2 实验装置和测量仪器

有关试验台、颤振转子、常规气动参数的测点布置等, 可参看文献[2]。在进口截面畸变区内还布置了 3 处总静压测点。为了造成进口畸变流场, 在上游距进口 0.5 直径处装一支撑网, 1 目/英寸、丝径 2 mm。产生畸变的扰流网复盖于支撑网上, 10 目/英寸、丝径 0.5 mm。畸变网在叶尖位置复盖了 1/3 总面积。为了监测颤振的发作, 安排了 3 套测量系统: (1) 动应力测量系统, 应变片贴在 10 个叶片上, 位于进气边叶背距叶根 18 mm 处, 和转子轴线方向垂直。(2) 声

学测量系统监测颤振时发出的特殊啸声。在机匣叶尖截面处,相距 90 装两支 CH16 电容传声器。一支 CH13 装在进气防尘网上用以测量远场声。传声器输出信号进入 FDA-2 型传声放大器,记录在磁带机上。(3)叶尖振幅测量系统。在机匣叶尖截面利用间断相原理,采用非接触式方法测量全部 34 个叶片的叶尖振幅。间断相原理的实质是叶片相对位移的测量是不连续的,而是在某些瞬时进行。当叶片没有振动,每一转经过固定在机匣上的传感器时,发出的脉冲信号都重复在同一位置上。当叶片振动,叶尖存在位移时,每转的脉冲信号与静止位置存在时延差,此时差反应了位移的大小。将所得信号经放大、采集后,由微机进行处理即可得到叶尖的振幅值。

3 实验结果及分析

3.1 均匀来流条件下的实验结果

BF-1-2 转子是 BF 系列转子中最容易出现失速颤振的一个。在均匀来流条件下,经多次试验发现,每当颤振发作时,叶尖振幅增加,动应力加大,同时出现类似远处火车鸣笛的单调啸声。三者几乎同步发生^[2]。图 1 有实测出的颤振边界和旋转失速边界,可见在 $\bar{n} = 0.74 \sim 1.0$ 范围内都出现了失速颤振。

3.2 畸变条件下的实验结果

试验设备、试验件、测试系统都不改变,在畸变的条件下,实验结果却出现了令人意想不到的变化。从节气门全开直到发生旋转失速的整个过程中,没有颤振现象出现。

图 2(a) 是在均匀进气稳定状态运转时, $n = 11820 \text{ r/min}$, 34 个叶片的叶尖振幅图,最大振幅为 0.942 mm 。图 2(b) 是均匀进气条件下, $n = 14040 \text{ r/min}$, 当节流到 $G = 10.04 \text{ kg/s}$, 发生颤振时的振幅图,最大振幅为 3.96 mm , 应力为 $121 \times 10^6 \text{ Pa}$ 。在畸变流场条件下当转速相同, $n = 14040 \text{ r/min}$, 节流到 $G = 9.88 \text{ kg/s}$ 时,其振幅分布见图 3, 最大幅值为 1.44 mm , 应力为 $46 \times 10^6 \text{ Pa}$, 可见,叶尖畸变后振幅减小,应力下降,没有出现颤振。

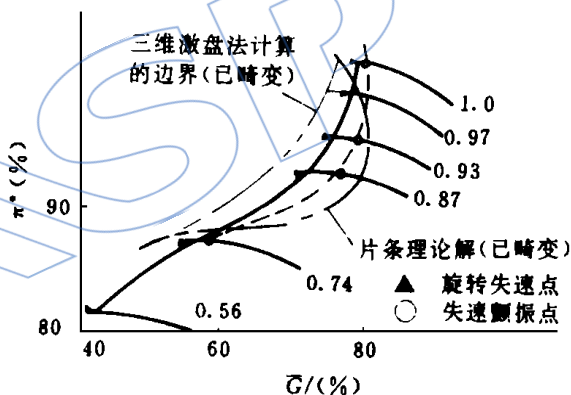


图 1 BF-1-2 转子失速颤振边界

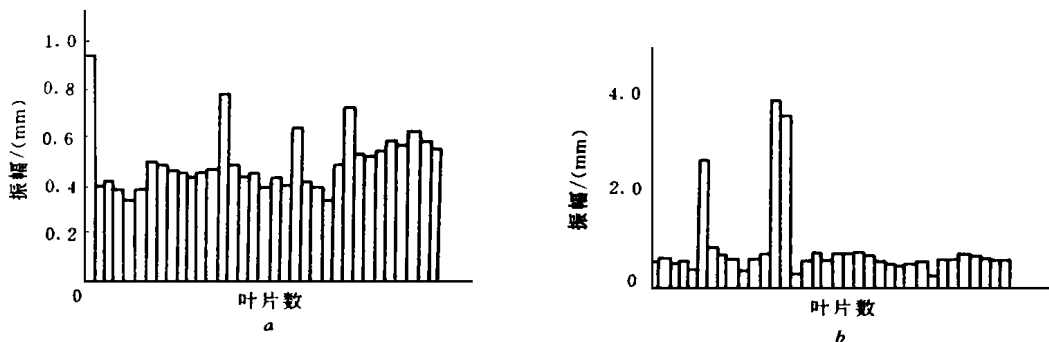


图 2 均匀进气 (a) 稳态运转 (b) 发生颤振时的叶尖振幅

气流对振荡叶片所作的功主要集中在叶尖。这一点已为实验所证实^[3]。因此,叶尖部分的流动情况对颤振的发作有很大影响。

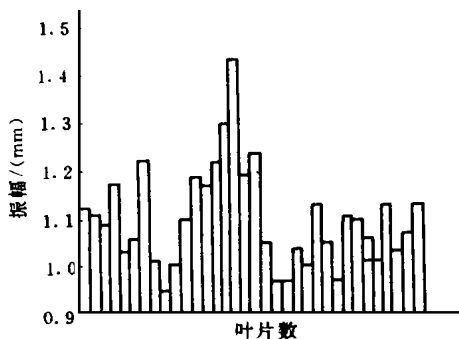
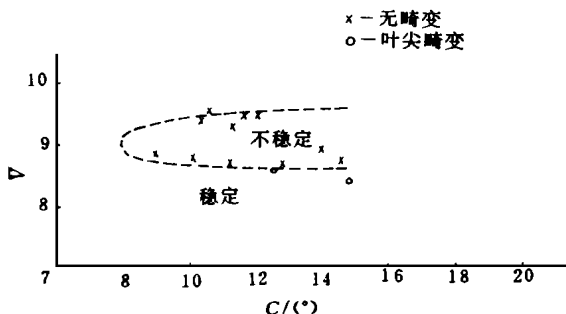


图3 径向畸变流场下的叶尖振幅图

图4 颤振时的 $\nabla-i$ 边界

径向畸变时,叶尖部分被扰流网堵塞,压气机内的流场沿径向发生变化。即在叶尖区进气总压和气流绝对速度降低。一方面造成叶尖区攻角增加,叶背分离更加严重。另一方面气流速度的降低会使气动阻尼增加而成为正值,有利于抑制颤振的发作。关于攻角的影响,文献[4]的实验表明,大攻角时所发生的颤振只复盖了折合速度的一个有限范围。高于及低于此折合速度都没有颤振发生。文中对此现象的解释是,颤振发作时,可能需要局部超音流,而低转速时,甚至在大攻角下也不能实现。在高转速,甚至喘振时,此攻角对颤振的发作还不够。BF-1-2 转子的折合速度 ∇ -攻角 i 表示的颤振边界(85%叶高特征截面处)见图4。可见同样存在一个有限的颤振区。当进气畸变时,由于来流速度下降,虽然攻角增加,但 ∇ 值却减小了。畸变区内大多数点都落在颤振区外。

本实验畸变指数 D 值较小,即叶尖存在小的径向畸变,造成小的速度梯度,此梯度可看作一个涡。因为颤振的发生与叶片定常负荷有关,畸变流场降低了叶片负荷,因此比起均匀流场来不易发生颤振。但若叶尖区畸变度较大,气动阻尼与流量之间则呈现出复杂的变化。是否仍能改善气弹稳定性,尚需进一步研究和验证。

4 三维变形激盘模型的计算结果

为了与实验结果比较。在实验的基础上用三维变形激盘模型^[5]计算了叶尖径向畸变下的失速颤振边界。计算时,将压气机分为两个区域,畸变区 I 和非畸变区 II,见图5。畸变指数 D

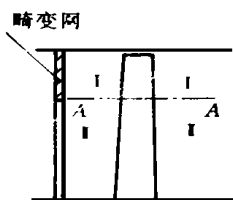
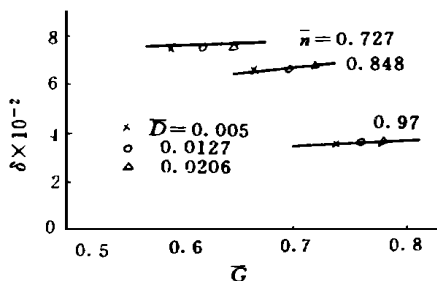


图5 畸变时分区示意图

图6 不同 D 下总阻尼的变化

定义为 $D = (P_{\text{imax}}^* - P_{\text{max}}^*) / P_{\text{imax}}^*$ 。为防止进口总压沿径向的突然变化给计算带来过大的扰动, 给定进口总压后用光滑处理将突跃适当抹平。

在 3 种 D 下, 即 0.005、0.0127、0.0206 对不同 n 沿均匀流场下得到的颤振边界作了计算, 结果见图 6。可见均未出现负阻尼。

图 1 上画出了用三维变形激盘法算出的畸变流场下的颤振边界。可见畸变后推迟了颤振的发作, 这与实验结果是符合的。遗憾的是不能实测出边界与计算值对比, 因为在此之前已发生了旋转失速。图上还有一条用片条理论计算的边界, 显然与实验结果有差距, 这是因为三维变形激盘模型考虑了三维效应的结果。

5 结束语

(1) 在 BF-1-2 颤振转子失速颤振实验的基础上, 进行了叶尖小的径向畸变实验。实验结果表明, 畸变后, 该转子的气动弹性稳定性好转。(2) 使用三维变形激盘法, 对有径向畸变的 BF-1-2 转子进行了颤振预测, 结果与实验符合, 并得到了预测的颤振边界。(3) 叶尖区小的径向畸变对抑制颤振发作是有利的, 但畸变度大, 则需要进一步研究和验证。精心控制进口流型对抑制失速颤振的发作是一种有效的措施。

参 考 文 献

- 1 Halliwell D G. The Effect of Intake Coditions on Supersonic Flutter in Turbofan Engines. AIAA Paper, 79-0044, 1979
- 2 冯毓诚, 胡宗安, 赵秀华, 王雨渭. 轴流压气机流体诱发振动的实验研究. 中国航空科技文献, HJB870474, 1987
- 3 周盛等. 叶轮机气动弹性力学引论. 国防工业出版社, 1989
- 4 Stargardter H. Subsonic Transonic Stall Flutter Study. NASA-CR-165256, 1979
- 5 杨晓东. 关于叶轮机气动弹性力学模型的若干问题探讨: [博士论文]. 北京航空航天大学, 1988

(责任编辑 魏星禄)

SURVEY PAPER

PRELIMINARY SEARCH FOR NEW FLOW- TYPES IN TURBOMACHINERY

Liu Jingmei and Zhou Sheng

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

ABSTRACT

As viewed from flow- types, the developing process of aircraft aerodynamics is compared with that of turbomachine gasdynamics. For the branch of turbomachine's gasdynamics, it is found that the steady body- fitted flow- type is still the searching object. However, the entire body- fitted flow- type is impossible. It means that the unexpected separations in turbomachinery can not be eliminated. And our comparison has shown that, as for the development of flow- types, the branch of turbomachine's gasdynamics lags considerably behind the branch of aircraft aerodynamics! Some of the suggestions, with regard to the possible ways of introducing new ideas of flow- types from aircraft aerodynamics into turbomachine gasdynamics, are presented in this paper.

Key words Turbomachines Vortex flow Non- steady flow Separated flow

FULL- LENGTH PAPERS

AEROACOUSTIC SOLUTION OF TURBOMACHINERY IN FREQUENCY DOMAIN

Ju Hongbin

(Dept. of Engineering Mechanics, Tsinghua University, Beijing 100084)

Zhong Fangyuan

(Shanghai University, Shanghai 20003)

ABSTRACT

According to the sound radiation solution of rotating source in frequency domain, formulae of noise sources in turbomachine have been derived. Then the detailed aeroacoustic solutions of the sound field in the turbomachine, the blades of which are thin and bent slight in the mid- chord alignment were obtained. The result is suitable for near sound field analysis. Moreover the influence of geometric parameters and operation conditions are explicitly expressed in the frequency domain solutions.

Key words Turbomachines Frequency domain analysis Aerodynamic noise

EFFECT OF SMALL RADIAL DISTORTION OF INLET FLOW ON STALL FLUTTER IN AXIAL FLOW COMPRESSOR

Feng Yucheng, Zhao Xiuhua, Hu Zongan, Ji Tu, Wang Yunwei

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

ABSTRACT

The effect of small radial stagnation pressure distortion of inlet

flow on the stall flutter has been investigated experimentally. The results of experiments show that the small radial distortion of the inlet flow at blade tip can eliminate the stall flutter. This phenomenon is verified by computation with three-dimensional actuator disk method. The computed results conform with the experimental data.

Key words Axial-flow compressors Radial distortion Flutter

NUMERICAL SIMULATION OF INTERNAL FLOW IN 2-D NOZZLES WITH THRUST VECTORING

Yong Honwei and Chun-Hian Lee

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

ABSTRACT Numerical simulation of internal flow with thrust vectoring in two-dimensional convergent nozzles is described, which utilizes multigrid technique in conjunction with MacCormack's explicit scheme. In computations, the Reynolds averaged Navier-Stokes equations are adopted for the five grids with $K-\epsilon$ turbulence model; and on the other hand, Euler equations are solved on the coarse grids. Numerical results provide good resolution to the flow fields, and the computed pressure distributions are found in good agreement with the experimental data. The influences of geometric parameters upon the flow quality and thrust coefficient are also analyzed. It is demonstrated that the multigrid technique adopted here is reliable and suitable for parametric study.

Key words Jet nozzles Thrust vector control Numerical simulation

COMPUTATION OF 2-D TRANSONIC AND SUPERSONIC FLOW FIELDS USING TVD SCHEME AND FOUR-STAGE RUNGE-KUTTA TECHNIQUE

Wang Shangjin, Ni Mingjiu, Xi Guang

(SER Research Center, Xi'an Jiaotong University 710049)

ABSTRACT Harten's TVD scheme is extended to 2-D general curvilinear coordinates by local rotation matrix translation. A new fully explicit solving method based on Four-Stage Runge-Kutta technique is developed to predict 2-D supersonic and transonic flow field. Comparison between computation and experiment is presented for both supersonic and transonic flow. The numerical results show that this scheme is not only convenient to solve the supersonic flow, but also capable of predicting the transonic flow.

Key words Euler motion equation Supersonic flow Transonic flow

INVESTIGATION ON FLOW RESISTANCE AND HEAT TRANSFER IN COOLING PASSAGES OF GAS TURBINE BLADE

Gu Weizao, Tu Jianping, Liu Wenyan, Wang Zhifang

(Institute of Engineering Thermophysics, Academia Sinica 100080)

Zhao Dongmei, Sheng Rongchang, Liu Xiaofeng, Ling Jinru

(Nanhua Powerplant Research Institute)

ABSTRACT An enlarged test blade of gas turbine has been investigated experimentally for flow resistance and heat transfer in its cooling passages. Air entered the radial cooling channel inside the leading part of the blade and contrary flowed downward into the passage between two plates while the air flow reached to the top. The bulk of the air stream