

带有腔室旁通道的进气道 气动性能试验研究*

南京航空航天大学 陈晓* 李潞龙 马若龙

【摘要】 分析了支线民航机一种新型亚音速带有腔室旁通道的进气道的几何形状设计和气动试验结果。针对某型发展型飞机设计并制造了缩尺比例为1:2的该型式进气道模型,进行了流态观察,气动性能试验和投掷外来物试验。结果表明,这种带有腔室旁通道的进气道虽然总压恢复系数比某型发展型的带有管状旁通道的进气道稍低,但具有许多优点,尤其是具有很强的排除外来物的能力,对适航性有严格要求的民航飞机来说,这种进气道将有很好的应用前景。

主题词: 进气道 气动特性 性能试验 旁通道

分类号: V228.7

1 进气道结构几何形状分析

当前国外支线民航机大多采用了具有一定排除外来物能力的带管状旁通道的进气道。80年代初,加拿大德里维兰飞机公司提出了一种新型的支线民航机进气道方案,即带有腔室旁通道的进气道方案,并在Dhc-8支线民航机上成功地投入实用,性能优越。现某型发展型支线飞机已采用了带有管状旁通道的进气道。为了使我国的民用航空发展跟上国际先进水平,必须了解和分析带有腔室旁通道的进气道流动特性、气动性能和排除外来物的能力,以便能正确地自行设计该型式的进气道。由于没有实物和完整的结构图纸作依据,我们只能在分析方案图的基础上进行几何形状的设计,并通过试验选择某些主要的几何尺寸。为了与某型发展型的带有管状旁通道的进气道进行比较。按其进出口尺寸的大小设计并制造了缩尺比例为1:2的该型进气道模型,进行了流态观察、气动性能试验和投掷外来物试验。

带有腔室旁通道的进气道几何形状设计是围绕着减小损失和消除外来物进入发动机的机会两个目标来进行的。其基本思想是消除进气道下唇口到发动机压气机进口截面的直接视线和在腔室中造成一个大的旋涡,使吞入进气道的外来物进入腔室中,同时进入腔室中的气流则通过次流段回到主流形成循环流动。图1为带有腔室旁通道的进气道示意图(1.进气口,2.喉道,3.扩压器,4.扩压器出口弯曲段,5.捕涡装置,6.腔室,7.滑油冷却空气引出口,8.旁通道出口,9.次流段,10.压气机前进气段,11.测量截面,12.钟形入口)。为了与某型发展型飞机带有管状旁通道的进气道进行比较,所设计进气道的进气唇口、喉道和压气机前进气段都相同于某型发展型飞机进气道的有关部分。现对下列主要结构几何形状进行分析。

在扩压器出口有一弯曲段, 其上半部分通过一弧形弯曲使气流向上进入发动机, 下半部分以一入口通向腔室, 如图 2。弧形弯曲板的弯曲程度直接与进气道进出口之间的偏心距有关, 也就是与能否消除从进气唇口到压气机进口截面的直接目视线有关, 而且弧形弯曲板还是主流道与次流通道的分隔板, 其形状、位置影响了主次流的流道、流量比例和掺混。因此进气道的总压恢复系数、总压畸变指数和排除外来物的能力都与其形状和位置有关。在弧形弯曲板的两侧根部各有一条缺口, 以用来形成两侧次流。弯曲板下部的腔室入口面积比管状旁通道的入口面积大得多 ($A_{腔}/A_{管} = 1.4$), 因此腔室吞咽外来物的可能性大于管状旁通道。

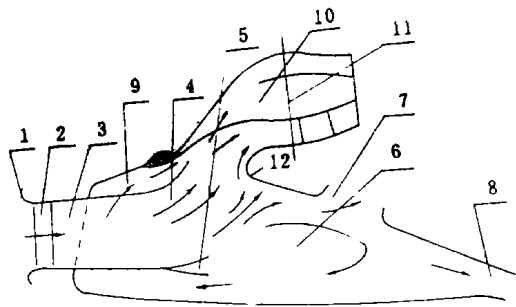


图 1 带有腔室旁通道的进气道

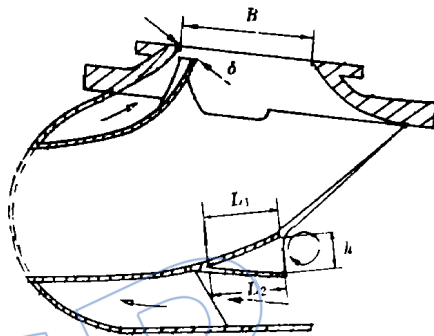


图 2 进气道部分结构

捕涡装置如图 2 所示。在弯曲段出口的下部和两侧有一捕涡装置, 其目的是在弯曲段出口下部和两侧形成固定的涡, 从而减小气流进入腔室的突扩损失, 并保持腔室内气流的稳定性, 其几何尺寸通过试验选择。腔室有两个作用: (1) 吞咽进入进气道的任何外来物, 并将其留在腔室内, 而管状旁通道的出口一定要有一个常开的排除外来物门, 这会造成排气阻力。但在腔室的尾部设有一个不常开的旁通道门, 以便排除堆积的外来物和冰雪等。(2) 腔室中的气体可引起冷却滑油散热器。而对于管状旁通道的进气道, 用于冷却滑油散热器的气体则需从主进气道下方的小进气道提供。

次流段 (见图 2) 是腔室中的气流返回主流的气流通道, 它由扩压器外壁和腔室内壁面构成, 二次流在这一段中被加速从扩压器出口弯曲段的弧形弯曲面和压气机进气段的钟形入口之间的缝隙进入压气机进气段和主流混合。次流段有两个作用: (1) 造成腔室中的大旋流; (2) 由于在扩压器出口弯曲段和压气机前进气段钟形入口之间的二次流, 允许主流作较大角度的转弯而没有大的损失或气流分离。次流出口间隙 δ 值是较重要的几何参数, δ 值大小影响了次流的损失和次流量的大小, 因此影响主次流掺混损失的大小, 其值将通过试验来选择。

压气机进气段前钟形入口段的型面对进气道气动性能也是至关重要的, 要精心设计。

2 试验结果与分析

用真空泵作为吸气源。测量截面取在压气机前进气段出口环形面, 沿圆周均布 8 个总压耙, 每个总压耙上有 5 根总压管按等环面积分布, 在通道内壁沿圆周均布 4 个壁面静压孔。

2.1 唇口和内通道流动情况

在某型发展型进气道地面起飞状态下, 通过贴丝线对模型的腔室, 捕涡装置和部分次流段进行了流谱观察, 发现除了在某些气流方向不稳定处, 丝线有大幅度摆动外, 大部分气流呈图

3 所示的有规律流动。通过吸入红墨水对钟形入口段和腔室滞止线附近作流谱观察, 没有出现墨水堆积现象, 流动状态良好, 在滞止线以上流线进入进气段, 在滞止线以下流线沿腔室上壁面进入腔室。在唇口和扩压器处流动稳定, 没有气流分离。

2.2 δ 对进气道气动性能的影响

选择次流出口间隙 δ 与通道宽度 B 的比值 $\delta/B = 0.043, 0.061, 0.092$ 和

0.123。图 4 表示在进气道出口截面(即压气机前进气段出口)平均马赫数 $Ma = 0.28$ 时(接近于某型发展型进气道地面起飞状态), 进气道总压恢复系数 σ 随 δ/B 值的变化曲线, 从图看出 $\delta/B = 0.061$ 时, σ 值较高。而总压畸变指数 D 值随 δ/B 值变化不大。

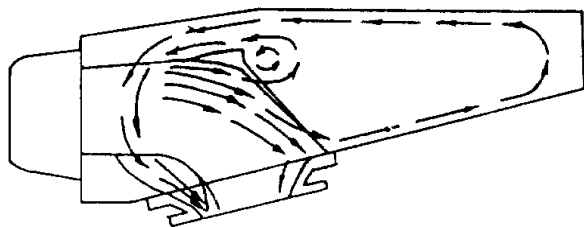


图 3 流谱

2.3 捕涡装置几何参数对进气道性能的影响。

上下长度比值的影响。选择 $h/L_1 = 0.93$, 取 $\Delta = L_1/L_2$ (见图 2) 为 0.81、1.04 和 1.3 三种情况对进气道进行气动性能试验。图 5 为 $Ma = 0.28$ 时, σ 随 Δ 变化曲线, 可以看出 $\Delta = 1.04$ 时, σ 较高。 Δ 对 D 影响不大。

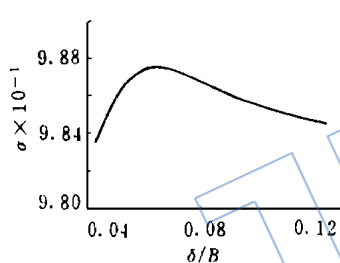


图 4 σ 随 δ/B 的变化

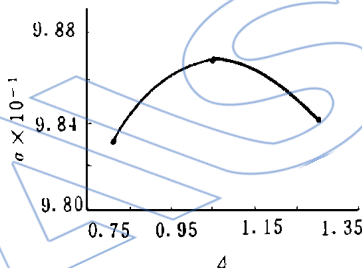


图 5 σ 随 Δ 的变化

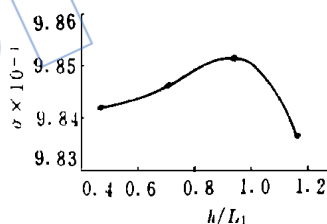


图 6 σ 随 h/L_1 的变化

以无因次量 h/L_1 表示高度 h 对性能的影响。图 6 是 $Ma = 0.29, \Delta = 1.04$ 时, σ 随 h/L_1 的变化曲线。从图看出 $h/L_1 = 0.93$ 时, σ 值较高, h/L_1 对 D 的影响也不明显。

2.4 G_b 对进气道气动性能的影响

图 7 为进气道 σ 随旁通道抽吸流量 G_b 和主通道流量 G 比值的变化曲线。 σ 值随着 G_b 的增加略有下降, 这是由于当 G_b 增加时, 进气道进口总流量增加, 从而使进口和扩压器内的流速增加。此外, 由于腔室的稳定作用, 抽吸不影响主流的流动, 因此 D 基本不变。

2.5 带腔室旁通道与带管状旁通道的进气道比较

图 8 和图 9 分别为两种型式进气道在地面起飞状态时, σ 和 D 值随 Ma 变化曲线(图 10 为管状旁通道的进气道示意图), 从图看出带管状旁通道的进气道气动性能略优于带腔室旁通道的进气道, 在地面起飞状态进气道出口设计马赫数 $Ma_{\text{设}} = 0.28$ 时, 前者 $\sigma = 0.991, D = 0.025$, 后者 $\sigma = 0.987, D = 0.035$ 。后者 σ 值比前者低 0.4%, D 也大一些。带腔室旁通道的进气道气动性能略差, 主要是由于二次流的损失和压气机前进气段钟形入口处流态不理想。

由于试验模型的比例为 1:2, 按要求采用重 0.226kg、63.5mm 直径的圆球作投掷试验。圆球用简单投掷物器从模型进口投入。圆球初速约为 9.3m/s, 可改变投射方向, 试验状态为进气

道出口最大 Ma 值。对于带腔室旁通道的进气道, 用标准球投射 20 次, 全部进入腔室旁通

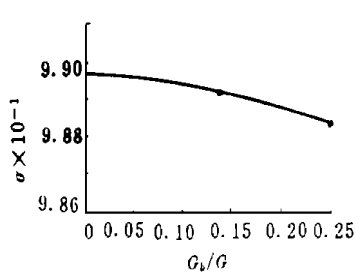


图 7 σ 随 G_b/G 的变化

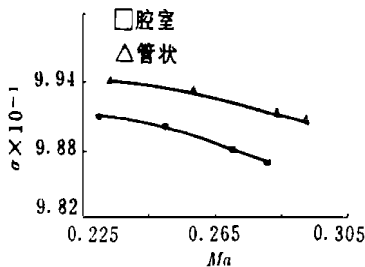


图 8 σ 随 Ma 的变化

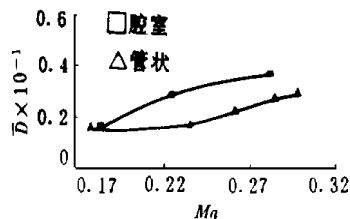


图 9 D 随 Ma 的变化

道, 用 0.18kg 球投射 8 次, 也全部进入腔室旁通道。对于带管状旁通道的进气道, 当标准球从正前方投入进气道时, 圆球可从旁通道排走。但当圆球向进气道上侧投射时, 圆球就进入主通道。显然腔室旁通道比管状旁通道具有较强的排除外来物能力, 这是由于: (1) 前者有利于消除进气道进口到压气机进口面的直接目视线, 后者如增加主通道的弯曲度, 则将增大损失。(2) 前者的腔室入口面积大于后者的旁通道入口面积。(3) 前者无论在任何飞行状态, 腔室均存在有对外来物的吸力, 而后者在外来物很容易进入进气道的地面起飞和低 Ma 数爬升状态时, 由于旁通道出口的静压高, 这时旁通道均处于进气状态, 这对外来物吸入旁通道是不利的。



图 10 管状旁通道的进气道

4 结 论

- (1) 经过试验研究设计的带腔室旁通道进气道模型(1:2)在起飞状态设计流量下, 总压恢复系数 $\sigma = 0.987$, 总压畸变指数 $D = 0.035$, 可满足飞机和发动机要求。
- (2) 在要求的范围内, 从腔室抽吸流量对进气道的气动性能(σ 和 D) 影响很小。
- (3) 在地面起飞状态发动机设计流量下, 腔室中的总压恢复系数很高, 为 0.9965 。
- (4) 腔室具有很强的吞咽外来物能力。

(责任编辑 魏星禄)

HOT- WIRE MEASUREMENT OF 3- D FLOW FIELD IN A RECTANGULAR VOLUTE

Huang Shujuan, Hu Zhiwei, Miao Yongmiao

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

ABSTRACT

3-D turbulent flow fields in a rectangular volute of a small centrifugal compressor were measured with a hot-wire probe at three across sections along its circumference at three different flow regimes. The space positions were adjusted with a three-dimensional coordinate frame. Time average velocities and Reynolds stress tensors were calculated by means of a hot-wire response equation with third order of precision. The measured results illustrated two recirculation regions near both side walls due to abrupt expansion at the inlet of the rectangular volute. The turbulent kinetic energy and Reynolds stresses in the recirculation regions were reduced as the flow rate increases and as the radius increases. In addition, asymmetry of the rectangular inlet induced the vortices at different cross sections of the volute, and made the trace of the flow as a curved spirochaeta. In consequence, the abrupt expansion and the asymmetry of the inlet must be avoided.

Key words Turbulent flow Flow field Hot-wire probe Measurement

EXPERIMENTAL STUDY OF 3- D TURBULENT FLOW FIELD IN A 90 ° BEND PIPE

Shang Hong, Wang Shangjin, Xi Guang, Yuan Minjian

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

Kong Qingli

(*Institute of Shenyang Blower Plant, Shenyang*)

ABSTRACT

The 3-D turbulent flow field in a 90 ° strongly curved pipe ($D = 280\text{mm}$, $Re/D = 0.87$) has been experimentally studied under different Reynolds numbers ($Re = 2.78 \times 10^5 \sim 5.40 \times 10^5$) with a single slant hot-wire measurement system and a five-hole dynamic probe. The detailed experimental results are given, including the distributions of 3-D time-averaged velocities, turbulent energy, total pressure and static pressure along the curvature direction at five different cross-section. It is shown that the loss in this strongly-curved pipe mainly occurs near the inner-side wall, and increases rapidly near the 90 ° cross-section due to flow separation. Static pressure distribution near the outer-side wall is higher than that near the inner-side wall throughout. The flow near the inner-side wall accelerates from the 90 ° to the 30 ° cross-section and decelerates after 30 ° cross-section. But near the outer-side wall, the tendency becomes adverse, and the pressure gradient here is much smaller than that near the inner-side wall.

Key words Turbulent flow Flow field Pipe flow

EXPERIMENTAL STUDY ON GASDYNAMIC PERFORMANCE OF INTAKE WITH PLENUM BYPASS

Chen Xiao, Li Lulong, Ma Rulong

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

ABSTRACT

The intake with plenum bypass is a new-type subsonic intake for civil medium-range aircraft. Its geometry is considered. A model of this type has been designed, which scale is 1:2. Experiments of flow visualization, gasdynamic performance and foreign object throwing have been done. The experimental results show that its perfect gasdynamic performance and flow steadiness satisfy the requirements of aeroplane and engine. Compared to the intake with pipe by-pass, this new-type intake has many advantages, especially the improved ability of eliminating

the foreign object, although the total pressure coefficient of this type intake slightly lessens. This type intake has extensive application in the future to the civil aviation with the strict requirements of airworthiness.

Key words Air Intake Gasdynamic performance Performance tests Bypass

A RESTORING ANALYTIC NUMERICAL METHOD FOR SOLUTION OF VISCOUS FLOW FIELD IN TURBINE

Zhang Chiping, Feng Guotai, Cui Minggen

(Dept. of Mathematics, Harbin University of Technology, Harbin 150001)

ABSTRACT

A new restoring analytic numerical method based on application of the method of restoring nucleus is developed for solution of viscous flow field in turbine. It makes the problems of storage, speed and accuracy in calculation easy to be solved. This paper offers ideas and methodology of application the method of restoring nucleus to solving viscous flow field in turbine, as well as in curvilinear coordinate system, and demonstrates the procedure of solving flow function integral equations. Finally, a numerical experiment for the two-dimensional cascade viscous flow mentioned in reference 1 by the aid of this method has been performed and the results are consistent with the data in reference. 1.

Key words Turbine Viscous flow Restoring Numerical computation

NUMERICAL SIMULATION OF 3- D FIELD WITHIN A COMPRESSOR ROTOR WITH C- H GRID

Wang Licheng

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

ABSTRACT

A 3-D C-H grid generation technique and 3-D Euler governing equation are described briefly. The highly loaded transonic compressor rotor designed by ONERA is taken as a numerical example. ONERA's numerical results and ours with H grid and C-H grid are compared respectively in respect of the pressure distributions along profiles at hub and tip of the rotor. The numerical technique presented is applied to evaluating the influence of inlet pre-swirl on its downstream flow field in the rotor, in result a new numerical approach is obtained for design optimization of pre-swirl distribution at compressor design stage. As numerical examples, two kinds of pre-swirl distributions are examined to show its influence on M_{w1} , β and α etc. distributions along radius.

Key words Euler's equation Three-dimensional flow Transonic compressor
Numerical simulation

EFFECT OF INTERBLADE PHASE ANGLE ON BLADE FLUTTER OF STEAM TURBINE

Zhang Yangjun, Li Kejian, Tao Deping

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

ABSTRACT

The interblade phase angle is one of key factors to act on the aeroelastic stability of steam turbine. A two-dimensional numerical deforming actuator disk method on non-constant interblade phase angle along circumferential direction has been developed to predict the stall flutter of the steam turbine. The effect of the interblade phase angle on the blade flutter of the steam turbine has been studied with the help of the developed method. The results indicate that when the interblade phase angle is between -45° and $+90^\circ$, the stall flutter may occur at a certain