

重气体离心压缩机三元叶轮的设计及实验

西安交通大学 王尚锦 席 光 刘士学
重庆通用机器厂 袁 治 王世池

【摘要】 利用一种基于准三元流动理论 S_2 流面的逆命题方程, 研究了重气体离心压缩机三元叶轮的设计方法及提高性能的措施。对设计的两个三元叶轮模型级进行了实验测量, 获得了优良的性能曲线。还研究了不同工况下叶轮出口动压分布, 分析了各部件的损失特性。

主题词: 离心压缩机 三元叶轮 设计及实验

分类号: HT452

1 设计方法及实例

作者在文献[1]中提出了一种径、混流式三元叶轮逆命题设计方法。采用该方法已设计制造了多种低负荷三元叶轮^[2], 获得了十分满意的结果。本文主要研究采用该方法设计大负荷(局部跨音)三元叶轮的可行性, 探索设计高性能、大负荷三元叶轮的方法及措施。为验证本文的设计方法, 在一个以氟里昂(R11)为介质的闭式试验台上对设计叶轮进行了详细的测量分析, 提供了有益的实验数据。

1.1 设计方法概述

所采用的逆命题方程^[1], 是建立在准三元流动理论基础上的方法。设计叶片前需输入叶轮子午流道尺寸。叶片造型采用如下方程:

$$\begin{aligned}dW_m/dq &= AW_m + B + C/W_m, \quad A = \cos(\alpha - \psi)/r_c \\B &= (d\lambda/dm)(d\theta/dq) - (d\lambda/dq)(\lambda - \omega^2)/(r^2W_m) + (dW_m/dq)\sin(\alpha - \psi) \\C &= dh_{in}^*/dq - \omega d\lambda_{in}/dq - T ds_e/dq \\ \theta &= \int_{out} \left[\frac{\lambda - \omega r^2}{r^2W_m} \right] dm + \theta_{out}\end{aligned}$$

式中, $\lambda(r_{c0})$ 为角动量或称为涡, $\theta = \theta(r, z)$, 为叶片的角坐标。 α, ψ 分别为子午面上绝对速度角及 q 线与半径方向夹角, r_c 为流线曲率半径。该方程采用流线曲率法求解, 设计时须输入如下参数: 质量流量、转速、叶轮进口滞止参数、叶轮子午流道坐标、叶片出口角坐标、涡分布及估计的叶轮效率。气体状态方程采用 M-H 方程^[3]。

1.2 提高大负荷三元叶轮性能的几点措施

设计中尽可能减小激波的强度及区域范围。由于叶轮出口马赫数很高, 改善叶轮出口流场对提高叶轮及扩压器效率将产生重要影响。因此, 设计中应尽可能减小叶片进口马赫数及叶片表面流动的局部扩压度, 这可通过增大轮盖侧曲率半径及采用合理的涡分布来实现。另外, 跨

盘、盖采用不等功分布,即靠近盖侧比盘侧适当多加一些功;盖侧叶片出口边向旋转方向倾斜等措施对改善叶轮出口流场亦有明显效果。

1.3 设计实例

为检验设计方法,本文设计了二个高负荷三元叶轮及其配套的静止原件,分别称为A、B叶轮。设计目标为:A叶轮,流量系数 $\Phi(\Phi=4Q_{in}/\pi D_2^2 u_2)=0.108$,级压比 $\epsilon=2.3$,级绝热效率 $\eta_t=0.81$ 。B叶轮,流量系数0.05,级压比2.2,级绝热效率0.81,各级的基本几何尺寸由一元方案计算得到。分别为:A叶轮,叶轮外径 $D_2=360\text{ mm}$,叶轮出口宽度 $b_2=19.4\text{ mm}$,叶片出口平均气角 $\beta_{2A}=45^\circ$;叶轮叶片数 $Z_2=14$,叶轮相对马赫数 $M_{u_2}=1.26$,转速 $n=9713\text{ r/min}$,扩压器进出口宽度 b_3, b_4 均为 19.4 mm ,扩压器出口直径 $D_4=594\text{ mm}$ 。B叶轮, $D_2=360\text{ mm}$, $b_2=12.9\text{ mm}$ 。 $\beta_{2A}=40^\circ$, $Z_2=14$, $M_{u_2}=1.06$, $n=9252\text{ r/min}$, $b_3=b_4=12.9$, $D_4=594$ 。图1为设计叶片采用的涡分布函数。图2为叶轮内部相对速度分布,可以看出叶轮内速度分布非常光滑,

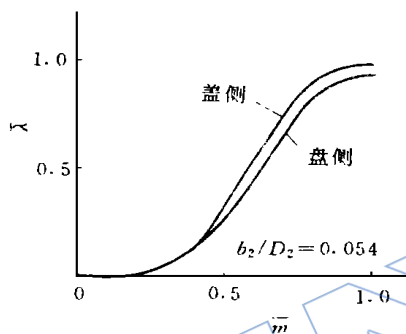


图1 涡分布函数

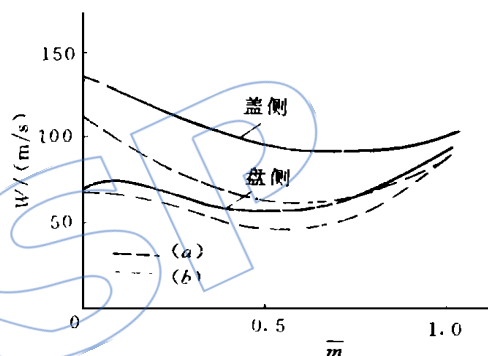


图2 S_2 流面相对速度分布

无大的局部扩压度,而且叶片进口区盘与盖处的相对马赫数均小于1.0。这表明叶轮内部流动得到了较好地控制。叶轮效率采用文献[4]中边界层方法计算。

2 实验验证

对所设计的两个三元叶轮进行了实验测量。实验在一个介质为(R11)的闭式离心式压缩机试验台上进行。为研究叶轮和扩压器的性能,在叶轮出口及扩压器出口增加了测量截面。实验中,采用水银温度计和带滞止腔的热电偶测量温度,用三孔空气动力探针测量速度和压力场。测量前所有的探针均在与实验介质相同的闭式风洞中进行了动态标定。

模型级A和B均在设计马赫数($M_{u_2}=1.26, 1.06$)下实验,对应转速分别为9713和9252 r/min。图3中(a)线为模型级A($b_2/D_2=0.054$)的性能曲线。由图可知,在设计工况点($\Phi=0.108$),实验压比为2.38,比设计值高3.5%,级绝热效率约为82%,比设计值高1%,图3中(b)线为模型级B($b_2/D_2=0.036$)的性能曲线。可以看到,在设计工况点($\Phi=0.05$),实验压比2.22,级绝热效率为81%,与设计值基本相同。

变工况下($\circ: \Phi/\Phi_d=1.0$; $\Delta: \Phi/\Phi_d=1.25$, $\cdot: \Phi/\Phi_d=0.38$)叶轮、扩压器、能量损失分析。本实验对模型级B的叶轮出口及无叶扩压器出口($1.05D_2, 1.65D_2$)流场进行了测量。图4为3个不同工况下叶轮出口动压 $\bar{P}_{dy}((P_{tot}-P_s)/(P_{dy})_m)$ 跨盘盖分布。由图可知,设计工况下,动压分布相当平坦。这表明设计中采用的跨盘、盖不等功分配及叶片出口边缘向转动方向倾斜的措施是有益的,获得了比较均匀的叶轮出口速度分布。大流量工况下,盖侧动压比盘侧低,这是因

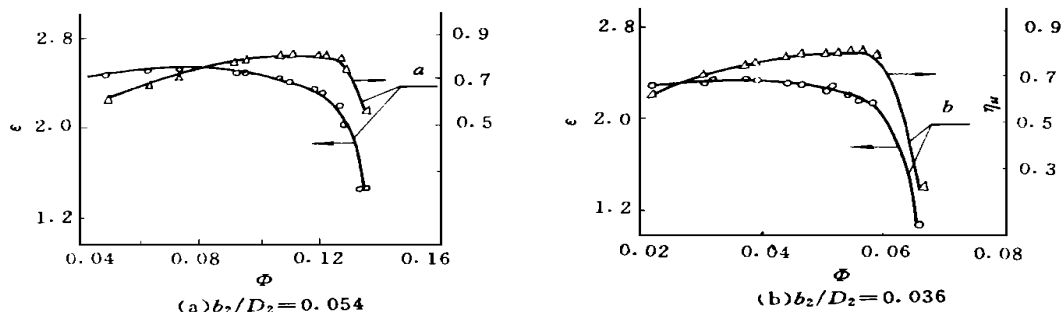


图3 实验模型级性能曲线

为盖侧叶片进口相对马赫数增高、流动损失增大的原因。小流量工况下, 盘侧的动压比盖侧低, 可能是因为盘侧叶片压力面速度过低, 分离损失增大的原因。由设计工况下叶轮绝热效率跨盘、盖分布可以看出, 盖侧的效率比盘侧低, 说明盖侧的损失比盘侧大, 这与设计中的判断是一致的。图 5 为模型级不同截面上绝热效率分布 (Δ 叶轮出口, $\cdot$ 扩压器出口, $\circ$ 蜗壳出口), 它反映各原件的损失情况。由实验知, 设计工况下, 叶轮损失使效率下降 5%, 扩压器损失使效率下降 7%, 蜗壳下降 6%。小流量工况下扩压器效率下降的速度比蜗壳快, 因为此时气流角很小, 扩压器内摩擦损失增加很大。大流量工况下, 蜗壳损失下降很快, 主要由于冲角过大, 分离损失迅速增加的原因。

实验表明本文设计的两个大负荷三元叶轮具有优良的性能; 这表明本文的设计方法及提高大负荷三元叶轮性能的措施是正确可行的。本文的实验结果亦为设计高压比离心压缩机三元叶轮提供了有益的参考。

参 考 文 献

- 1 王尚锦等. 离心压缩机三元叶轮的设计方法及计算机程序. 西安交大科技报告, 1976, 76~006
- 2 Wang S J and et al Development and industrial Application of the All-over Controlled Vortex Distribution Method for Designing Radial and Mixed Flow Impellers A S M E Paper, 92-GT-262
- 3 刘桂玉等编. 工程热力学. 高等教育出版社, 1989
- 4 王尚锦等. 径、混流式节能转子考虑边界层厚度的全可控涡设计的研究. 应用力学学报, 1985, 2(2)

(责任编辑 魏星禄)

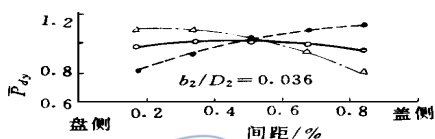


图4 叶轮出口动压分布

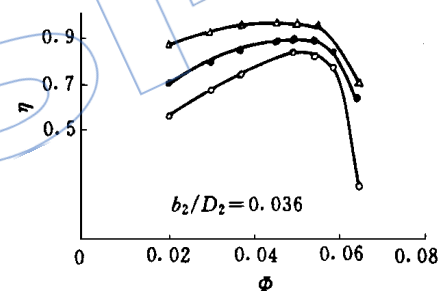


图5 效率分布

TECHNICAL NOTES

A MULTIGRID SCHEME FOR CALCULATING THREE-DIMENSIONAL FLOWS THROUGH COMPRESSOR BLADE ROW

Ning Wei and Zhou Xinhai

(Aeroengine Department, Northwestern Polytechnical University Xi'an 710072)

ABSTRACT Based on the MacCormack explicit difference Scheme, a multigrid scheme has been developed to solve the three-dimensional Euler equations expressed by weak conservation law in cylindrical coordinate system. The distribution and interpolation formulas of corrections for the presented multigrid scheme have been derived. The present Euler solver was used to calculate the flows of two sets of transonic compressor rotor blade rows, and proven to be a valid and useful design tool. The computed results indicate that this scheme is enable to accelerate the computation convergence greatly, and the method is accurate and reliable.

Key words Compressors Three-dimensional flow Finite difference methods

DESIGN AND EXPERIMENT OF CENTRIFUGAL COMPRESSOR 3-D IMPELLERS FOR HEAVY GASES

Wang Shangjin, Xi Guang, Liu Sixue

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

Yuan Yan, Wang Siyi

(Chongqing Machinery Manufacturing Factory, Chongqing)

ABSTRACT An inverse problem method based on s_2 flow surface theory of 3-D flow field has been developed to design centrifugal compressor impellers for heavy gases and some techniques to improve the performance of high pressure ratio impellers have been presented too. Two impellers with Mach number $M_{u_2} = 1.26, 1.06$ and relative width $b_2/D_2 = 0.054, 0.036$ respectively have been designed by use of this method. Experiments of the impellers have been performed in a closed-loop test rig with Freon gases (R11) to verify the design method. The dynamic pressures distribution from shroud to hub at the exits of impellers and the losses of vaneless diffusers and volutes have been measured and analyzed under different work points. Their good performance obtained from tests shows the design method and techniques presented in this paper is feasible and favourable.

Key words Centrifugal compressor 3-D impeller Design and experiment

A MODEL FOR ESTIMATING TIP CLEARANCES IN GAS TURBINE ENGINE TRANSIENTS

Zhu Zhili and Liao Kuo

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

ABSTRACT During the gas turbine engine transient, the clearances of the compressor and turbine blade tips differ from those in stable operation because the heat change between gas and components causes the different radial movements of the rotor and case. In order to simulate the real thermodynamics process in the gas turbine engine transient as possible, a model for estimating the variations of the tip clearance and efficiency with time is given. In this model, the typical stage chosen from a multi-stage turbomachinery will be decomposed into three components (case, blade and disc), the instantaneous heat growth of each component is calculated individually by means of the different heat transfer model, and the instantaneous centrifugal growths of blade and disc are also estimated, therefore the instantaneous tip clearance are obtained by summing the heat and centrifugal growths of all compo-