

叶片间相角对蒸汽轮机叶片颤振的影响*

北京航空航天大学 张扬军** 李克俭 陶德平

【摘要】 蒸汽轮机叶片的气动力衰减和叶片间相位角存在密切关系, 叶片间相角是决定蒸汽轮机叶片的气动弹性稳定性的重要因素, 应用可考虑叶片间相角变化, 适用于蒸汽轮机失速颤振预测的变形激盘法, 考察了叶片间相角对蒸汽轮机叶片的气弹稳定性的影响。

主题词: 叶片间相角 叶片振动 气动力弹性 蒸汽透平

分类号: TK263.3

1 前言

蒸汽轮机长叶片颤振严重影响大机组的安全运行, 解决长叶片颤振问题是发展大型机组的技术关键之一^[1], 许多实验结果表明, 叶片间相角对蒸汽轮机叶片的气动弹性稳定性有极其重要的影响。文献[2]指出, 气动力衰减和叶片间相位角存在密切关系。文献[3]通过实验研究了相邻叶片间相位差对平面振荡叶栅的气弹稳定性的影响, 结果表明只是在某些特定相角下才发生了颤振。在蒸汽轮机叶片的气弹稳定性的研究中, 探索叶片间相角对蒸汽轮机叶片气弹稳定性的影响, 在现已发表的文献中, 虽然给出了几种相角的实验结果, 但是实验数据有限, 没有得出定量的规律。由于研究蒸汽轮机叶片的气弹稳定性的计算方法受到各种限制, 除文献[4]外, 目前尚未见到考虑整排叶片相角变化对分析叶片颤振影响的理论。至今, 研究者在预测蒸汽轮机长叶片的气弹稳定性时一直沿用的一个基本假设是, 在一个叶片排中, 叶片间相角沿周向分布为常数。

文献[4]所发展的变叶片间相角的蒸汽轮机叶片气弹稳定性预测方法, 解除了等叶片间相角这一基本假设, 为探索叶片间相角对蒸汽轮机叶片气弹稳定性的影响提供了有效的工具。本文应用可考虑叶片间相角变化, 适用于蒸汽轮机失速颤振预测的变形激盘法, 考察了叶片间相角对蒸汽轮机叶片的气弹稳定性的影响。

2 基本的物理假设与非定常流场的求解

本文采用变形激盘模型, 以转子上靠近叶尖的特征截面的二维叶栅来模拟转子叶片的非定常流动特征, 设叶片的弦长为零, 将叶栅假设为一个无厚度的盘, 振动叶片排蜕化为变形激盘, 不考虑叶片槽道内部的具体流动过程, 仅考虑激盘前后的流场。变形激盘振动是流场非定常性的唯一起因。假设激盘前后均为无粘流, 非定常振荡分量 $\tilde{f}$ 相对于定常分量 f 为小扰动, 且 $\tilde{f}$ 随时间呈简谐变化。过激盘间断面满足4个突跃条件: 质量守恒、能量守恒、气流角突跃

1993年6月收到; 1993年9月收到修改稿。 * 本文系国家自然科学基金资助项目

** 北京航空航天大学407教研室 100083

条件和总压损失突跃条件。进一步假设变形激盘的运动规律由叶片排的振动规律所决定,叶片间的相角根据实际情况输入,激盘的运动方程可以表示为:

$$\tilde{h}_{x,j} = \bar{h}_{x,j} e^{i(\varphi_j + kt)}, \quad \tilde{h}_{y,j} = \bar{h}_{y,j} e^{i(\varphi_j + kt)} \quad (1)$$

式中的 $\bar{h}_{x,j}$ 、 $\bar{h}_{y,j}$ 分别表示叶片的轴向弯曲振幅和横向弯曲振幅, j 表示第 j 个叶片; φ 为叶片的振动相位角, 叶片的振动相位角可根据实际情况输入。用叶片间相角 σ 来表示第 $(j+1)$ 叶片与第 j 叶片的振动相位差。

$$\sigma_j = \varphi_{j+1} - \varphi_j \quad (2)$$

根据基本假设, 对 Euler 方程组进行线性化, 可得本文所用的控制方程组为:

$$\bar{p}_\tau + u\bar{p}_x + v\bar{p}_y + \rho(\bar{u}_x + \bar{v}_y) = -ik\bar{p} \quad (3)$$

$$\bar{u}_\tau + u\bar{u}_x + v\bar{u}_y + (1/\rho)\bar{p}_x = -ik\bar{u} \quad (4)$$

$$\bar{v}_\tau + u\bar{v}_x + v\bar{v}_y + (1/\rho)\bar{p}_y = -ik\bar{v} \quad (5)$$

$$\bar{p}_\tau + u\bar{p}_x + v\bar{p}_y - a^2(\bar{p}_\tau + u\bar{p}_x + v\bar{p}_y) = -i(\bar{p} - a^2\bar{p}) \quad (6)$$

方程 (3)~(6) 即为非定常流场的控制方程。

进口边界: 当定常平均轴向分速为亚音时, 给定 3 个条件: $\bar{p} = \bar{u} = \bar{v} = 0$; 当轴向分速为超音时给定 4 个条件: $\bar{p} = \bar{u} = \bar{v} = \bar{p} = 0$ 。

出口边界: 当出口为亚音时, 给定反压 $\bar{p} = 0$, 其余的 3 个条件直接外推即可。

在计算域内有一个变形激盘间断面, 这一突跃面与激波不同, 由于突跃面内有叶片作用力的存在, 不满足激波的 R-H 条件。它满足如下 4 个突跃条件:

过激盘突跃面的流量连续:

$$(\rho_1 + \bar{\rho}_1)u_{1n}^{\text{rel}} = (\rho_2 + \bar{\rho}_2)u_{2n}^{\text{rel}} \quad (7)$$

这里的下标 “1” 与 “2” 分别代表激盘前、后, n 表示垂直于变形激盘的方向, 上标 “rel” 表示相对于变形激盘的运动。

过激盘的相对总焓守恒:

$$\gamma/(\gamma-1)](p_1/\rho_1) + (1/2)(q_1^{\text{rel}})^2 = \gamma/(\gamma-1)](p_2/\rho_2) + (1/2)(q_2^{\text{rel}})^2 \quad (8)$$

式中 γ 为比热比, q 为气流速度的模。

非定常落后角关系: 引入经验关系

$$\beta_2 = \theta + \delta_{m2} + g[\alpha(\beta_1, \theta, \delta_{m1}), M_{a1}, \theta(\theta, \partial x/\partial y)] \quad (9)$$

将上式进行线性化之后来表示非定常气流角的突跃条件, 式中 β_1 、 β_2 为激盘前后的气流角, δ_{m1} 、 δ_{m2} 为激盘前后的几何构造角。

过激盘的滞止压力损失条件: 对于大负攻角透平叶栅, 极少有现成的公式可用, 文献 [2] 利用平板问题的解加上修正系数 K_i 过渡到真实叶型, 得到速度损失系数 ψ 为:

$$1/\psi^2 = 1/\psi_0^2 + K_i(\Delta u_0/q_1)^2 \quad (10)$$

试验数据表明, K_i 随着攻角的改变而改变。根据实验数据可得:

$$K_i = -0.2\alpha/(\alpha - \alpha_0) \quad (11)$$

3 结果与讨论

采用某机组的叶片, 研究叶片间相角变化对该机组气弹稳定性的影响。应用可考虑叶片间相角变化, 适用于蒸汽轮机失速颤振预测的变形激盘法, 考察了叶片间相角对蒸汽轮机叶片的气弹稳定性的影响。

取 87.5% 叶高处的截面为特征截面, 图 1 给出了在相对容积流量分别为 $\overline{GV}_2 = 1.0, 0.75, 0.5, 0.3, 0.2, 0.15$ 等 6 种工况下, 叶片间相角在 -180° 至 $+180^\circ$ 之间变化时, 叶片的最小气动阻尼随叶片间相角变化的曲线。由图可知, 叶片间相角对蒸汽轮机工作叶片的气弹稳定性有比较大的影响。在相对容积流量 $\overline{GV}_2$ 一定的情况下, 叶片的最小气动阻尼随叶片间相角的变化呈非单调性变化。在 -180° 至 -90° 的范围内, 叶片的最小气动阻尼随叶片间相角的增加而增加; 在 -90° 左右, 随着叶片间相角的增加, 叶片的最小气动阻尼呈下降趋势, 在 -45° 左右时达到极小值, 之后叶片的最小气动阻尼随叶片间相角的增加而增加, 并在叶片间相角为 0° 时, 达到正的最大值。在 0° 至 $+90^\circ$ 的范围内, 叶片的最小气动阻尼随叶片间相角的增加而减少; 在 $+90^\circ$ 左右时达到负的最低值, 然后随着叶片间相角的增加, 叶片的最小气动阻尼呈上升趋势。

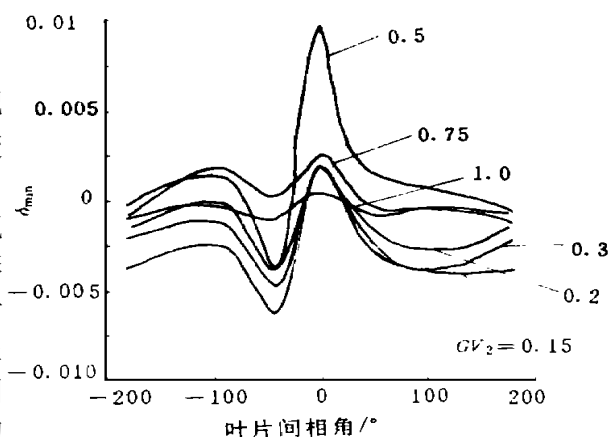


图 1 最小气动阻尼随叶片间相角的变化

在相对容积流量 $\overline{GV}_2$ 一定的情况下, 叶片间相角在 -45° 和 $+90^\circ$ 时, 气动阻尼均出现极小值, 叶片的气动弹性稳定性恶化, 相角在 $+90^\circ$ 附近最为严重。叶片间相角在 0° 左右时, 叶片的最小气动阻尼出现极大值, 叶片的气弹稳定性最好。拉筋使得叶片以相同的相位振动, 叶片间的相角为零, 使得叶片的机械阻尼变大, 也使得叶片的气动阻尼增加, 使其气弹稳定性变好。

从图中还可以看出, 在 6 种不同的相对容积流量工况下, 叶片间的相角对叶片的最小气动阻尼的影响趋势大致是相同的。在相对容积流量为 $\overline{GV}_2 = 1.0$ 时, 叶片间的相角对叶片的最小气动阻尼的影响较小, 但随着叶片的相对容积流量的减小, 不仅叶片的最小气动阻尼的数值变小, 而且叶片间相角对叶片的最小气动阻尼的影响也越来越大, 而蒸汽轮机的叶片颤振主要是在小的相对容积流量下发生, 因此, 叶片间相角对叶片的气弹稳定性有不可忽视的影响。

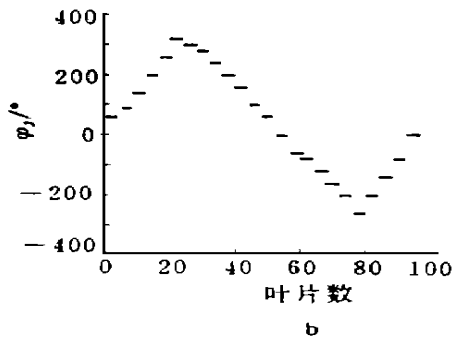
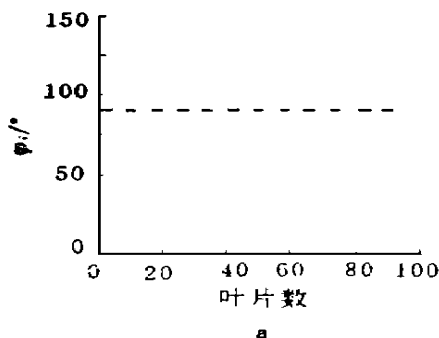


图 2 叶片的振动相位角分布

在相对容积流量为 $\overline{GV}_2 = 0.15$, 叶片的振动相位角均为 60° 时, 整个转子各个叶片的气动阻尼均为 0.0018, 说明当叶片以同一相位振动时, 叶片的气动阻尼基本上是相同的, 然而整个转子叶片排在实际上不可能均以同一相位振动。在相对容积流量为 $\overline{GV}_2 = 0.15$, 叶片的振动相

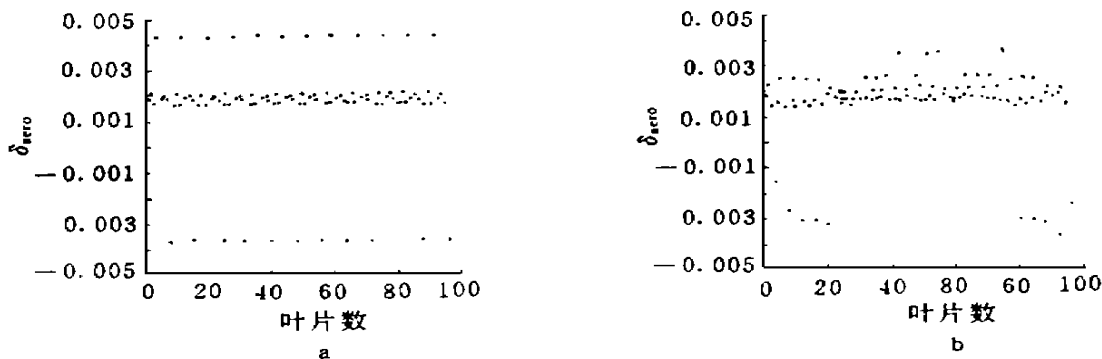


图 3 整个转子各叶片的气动阻尼分布

位角分别如图 2(a)、(b) 所示的分布时, 图 3(a)、(b) 表示整个转子各个叶片的气动阻尼的分布情况。有些叶片出现了负的气动阻尼, 而其他叶片的气动阻尼为正。负的气动阻尼对应于叶片间相角为正的叶片, 说明叶片的气弹稳定性与叶片的振动相位分布密切相关, 转子开始发生颤振时, 并不是所有的叶片同时损坏, 而是气动阻尼为负的那些叶片首先损坏, 这与实际故障发生时的情况相符。

4 结 论

(1) 在不同的相对容积流量下, 叶片间相角对叶片的最小气动阻尼的影响趋势大致是相同的。在相对容积流量为 $\overline{GV}_2 = 1.0$ 时, 叶片间相角对叶片的最小气动阻尼的影响较小, 但随着相对容积流量的减小, 叶片间相角对叶片的最小气动阻尼的影响也越来越大。(2) 在相对容积流量 $\overline{GV}_2$ 一定的情况下, 叶片的最小气动阻尼随叶片间相角的变化呈非单调性变化。当叶片间相角在 0° 左右时, 气动阻尼出现极大值, 其气弹稳定性最好。叶片间的相角在 -45° 和 $+90^\circ$ 时, 气动阻尼出现两个极小值, $+90^\circ$ 相角的叶片的气弹稳定性最差。(3) 对整个转子叶片排来说, 当所有叶片均以同一相位振动时, 叶片的气动阻尼基本上是相同的。实际上整个转子叶片排不可能均以同一相位振动。(4) 本文发展的方法能给出每个叶片的气动阻尼, 从而可以判断哪些叶片最容易失稳。通过研究不同的叶片间相角分布对叶片气动阻尼的影响表明, 正的叶片间相角是不稳定的, 与国外文献给出的实验结果一致。本文结果还说明各个叶片的气动弹性稳定性与叶片的振动相位角分布密切相关, 转子开始发生颤振时, 并不是所有的叶片同时进入颤振区, 而是气动阻尼为负值的那些叶片首先进入颤振区而发生损坏。

参 考 文 献

- 周盛, 陶德平, 许楚镇. 汽轮机末级气动弹性问题研究概论. 国家“七. 五”大型火电成套设备科技攻关项目 75- 50- 2- 01 - 13 提交报告, 1990
- 汽轮机叶片颤振研究译文集. 上海发电设备成套设计研究所, 1988
- 许玉妹, 严宏强. 振荡叶栅颤振的实验研究. 国家“七. 五”大型火电成套设备科技攻关项目 75- 50- 2- 01- 13 提交报告, 1990
- 张扬军, 陶德平. 变叶片相角的蒸汽轮机叶片颤振预测方法. 水利电力机械, 1993, (4)

(责任编辑 魏星禄)

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

condition.

Key words Interblade phase angle Blade vibration Aeroelasticity Steam turbines

WALL TEMPERATURE CALCULATION OF A NEW MAT-WALL COMBUSTOR STRUCTURE

Hei Liming

(*Air Force Engineering College, Xi'an 710089*)

Li Zhanglin

(*7th Dept., Northwestern Polytechnical University, Xi'an 710072*)

ABSTRACT

A seven-heat-flow model for calculating of the wall temperature of mat-wall has been established, in which internal axial heat conduction of flame tube wall is considered. A numerical method for solving the model is provided, which takes into account the effect of temperature on physical parameters. Therefore, the accuracy of the wall temperature calculation is enhanced. In order to provide required parameters for calculating the wall temperature, the calculation of axial distribution of mass flow and other flow parameters in the mat-wall structure have also been done. After that, the axial temperature distributions on the mat-wall of a single flame tube have been calculated at design and experimental conditions, and are compared with those of the prototype with air film cooling and its experimental data. It is demonstrated that the calculated results are principally consistent with experimental data and the mat-wall structure is possessed of excellent cooling performance. In comparison with the prototype, the cooling air mass is decreased by 20%, the wall temperature is obviously lowered. Furthermore, the axial wall temperature distribution is more uniform. Therefore, this structure will prolong the life of the flame tube or raise the gas temperature before the turbine.

Key words Flame tube Wall temperature Models Calculation

A MATHEMATICAL MODEL OF IMPINGING JETS ATOMIZATION AT LOW WEBER NUMBER

Shi Shaoping and Zhuang Fengchen

(*1st Dept., National University of Defense Technology, Changsha 410078*)

ABSTRACT

The characteristics of the spray produced by impinging jets atomization is theoretically analyzed. Depending on Weber number, two breakup regimes of sheet are considered with transition occurring at Weber numbers between 500 and 2000. In the low Weber number regime ($We \leq 2000$), the sheet breakup is due to Taylor cardioid waves. While in the high Weber number regimes, the sheet disintegration is caused by growth of Kelvin-Helmholtz instability waves. The model uses an analogy with the oblique impact of a liquid jet on a wall. At Weber number less than 2000, mathematical expressions to predict the sheet thickness and shape during formation of nonviscous liquid sheet are derived from mass conservation and momentum conservation equations by reference to theoretical results and empirical correlations reported in the literature. The numerical results are in good agreement with experimental data available in the literature. Predictions of the spray droplet size and distribution formed by the disintegration of the viscous liquid sheets are also obtained by use of the mathematical formulation developed for fan-spray injectors. This model can be used to estimate the size range and distribution of droplets formed by impinging jets atomization in quasi-static surroundings.

Key words Liquid drop Impinging jets atomization Mathematical models