文章编号:1001－2060(2001)05－0517－03

汽轮机真空系统严密性试验的静态模拟计算

李　勇,董玉亮,谷运兴,杨善让

(东北电力学院 动力系,吉林省 吉林市　132012)

摘　要:通过对汽轮机真空系统严密性试验的模拟计算,指出真空下降速度并不是漏入凝汽器空气量的单值函数,汽轮机负荷、冷却水流量、冷却水入口温度以及凝汽器管材等对真空下降速度均有不同程度的影响,并详细分析了这些参数对真空下降速度的影响,所得结论对于更准确地评价汽轮机真空系统严密性具有参考价值。

关　键　词:汽轮机;凝汽器;真空严密性试验

中图分类号:TK264.1　　文献标识码:A

1　前言

汽轮机真空系统的严密性失常是汽轮机运行中经常出现的故障之一。特别是随着汽轮机单机功率的增大,汽轮机排汽口数量增多,凝汽器体积增大,汽轮机真空系统的严密性更难以保证[1]。由于运行中无法准确地确定漏入真空系统的空气量,通常是采用真空严密性试验测得的真空下降速度来评价真空系统的严密性。目前,检验真空系统严密性的方法是进行真空严密性试验。在汽轮机带80%～100%额定负荷下,保持运行工况稳定,用关闭抽气器空气门或停止抽气设备的方法,观察真空表指示值下降速度。我国"固定式汽轮机技术条件"中对汽轮机真空系统严密性的标准规定,功率大于100 MW的汽轮机,真空下降速度不大于0.4 kPa/min;小于100 WM的汽轮机,真空下降速度不大于0.67 kPa/min;对于300 MW以上的汽轮机,要求真空下降速度不大于0.2 kPa/min。这种真空系统严密性试验方法除对试验时汽轮机的负荷有一定要求外,对凝汽器的其它运行条件如冷却水流量、冷却水入口温度、凝汽器管材以及凝汽器水侧清洁程度均未提出任何要求。

研究发现,在真空系统严密性试验过程中,即使在单位时间内漏入真空系统空气量一定的条件下,对应不同的凝汽器运行条件,真空下降速度也往往有不同值。本文中通过对真空系统严密性试验的静态模拟计算,定量分析了凝汽器各运行参数对真空下降速度的影响,为更准确地估计真空系统严密性提供了一个参照分析方法。

2　真空系统严密性试验的静态模拟计算

2.1　凝汽器的静态数学模型

在汽轮机真空系统严密性试验过程中,凝汽器压力 P_k 的变化不仅通过空气量对凝汽器端差的影响而导致蒸汽分压力的变化体现出来的,而且还通过空气分压力 P_a 的变化体现出来。亦即 $P_k = P_c + P_a$。

则真空严密性试验中凝汽器压力 P_k 的变化速度为

$$v_H = \frac{\Delta P_k}{\Delta\tau} = \frac{\Delta P_a + \Delta P_c}{\Delta\tau} \tag{1}$$

这里,近似用 $\frac{dP_c}{d\tau}$ 来表示 $\frac{dP_c}{\Delta\tau}$,则蒸汽分压力的变化情况为

$$\frac{\Delta P_c}{d\tau} = \frac{dP_c}{dt_c}\frac{dt_c}{d\vartheta}\frac{d\vartheta}{dk}\frac{dk}{d\alpha_s}\frac{d\alpha_s}{d\varepsilon}\frac{d\varepsilon}{dm_a}\frac{dm_a}{d\tau} \tag{2}$$

其中

$$t_c = t_{w1} + \Delta t + \vartheta \tag{3}$$

$$\vartheta = \frac{\Delta t}{e^{\frac{kA_C}{C_p D_w}} - 1} \tag{4}$$

$$k = \frac{1}{\frac{d_1}{d_2}\frac{1}{\alpha_w} + \frac{1}{\alpha_s} + \frac{d_1}{2\lambda}\ln\frac{d_1}{d_2} + R_f} \tag{5}$$

$$\alpha_s = 0.81\alpha_0(\varepsilon d_s)^{-0.04} \text{[2]} \tag{6}$$

$$\varepsilon = \frac{m_a}{m_a + m_s} \tag{7}$$

$$m_s = \rho''_s V'' \quad m_a = \rho_a V'' \tag{8}$$

空气的分压力借助于理想气体的状态方程来确定,即

收稿日期:2000－08－15

基金项目:国家电公司重点科技基金资助项目(SPKJ013－07)

作者简介:李　勇(1964－),男,辽宁本溪人,东北电力学院教授,博士

$$P_a = \frac{R(t_c + 273.15)m_a}{V''} \quad (9)$$

则 $$\frac{\Delta P_a}{\Delta \tau} = \frac{dP_a}{dm_a}\frac{dm_a}{d\tau} = \frac{R(t_c + 273.15)}{V''}G_a \quad (10)$$

上述诸式中：v_H 为凝汽器真空下降速度；$\Delta\tau$ 为试验时间；t_c 为凝汽器压力 P_c 对应的饱和温度；t_{w1} 为凝汽器冷却水入口温度；Δt 为冷却水在凝汽器中的温升；δ 为凝汽器端差；k 为凝汽器的传热系数；A_c 为凝汽器的冷却面积；C_p 为冷却水的比热；D_w 为冷却水流量；d_1、d_2 分别为冷却水管的外径和内径；R_f 为凝汽器水侧污垢热阻；α_w 和 α_s 分别为凝汽器水侧和汽侧的对流换热系数；λ 为冷却水管材的导热系数；α_0 为纯净水蒸气的平均凝结放热系数[3]；ε 为凝汽器中空气的相对含量；d_s 为凝汽器单位冷却面积的蒸汽负荷；m_a 和 m_s 分别为凝汽器中的空气和蒸汽的质量；ρ''_s、ρ_a 和 V'' 分别为对应各自分压力下的饱和蒸汽密度、空气密度和凝汽器汽空间的体积；$\frac{dm_a}{d\tau} = G_a$ 为单位时间漏入真空系统的空气量。

则由式(1) 可得真空下降速度为

$$v_H = KG_a \quad (11)$$

这里 K 为比例系数

$$K = \left(XY\frac{m_s}{m_s + m_a} + \frac{R(t_s + 273.15)}{V''}\right)$$

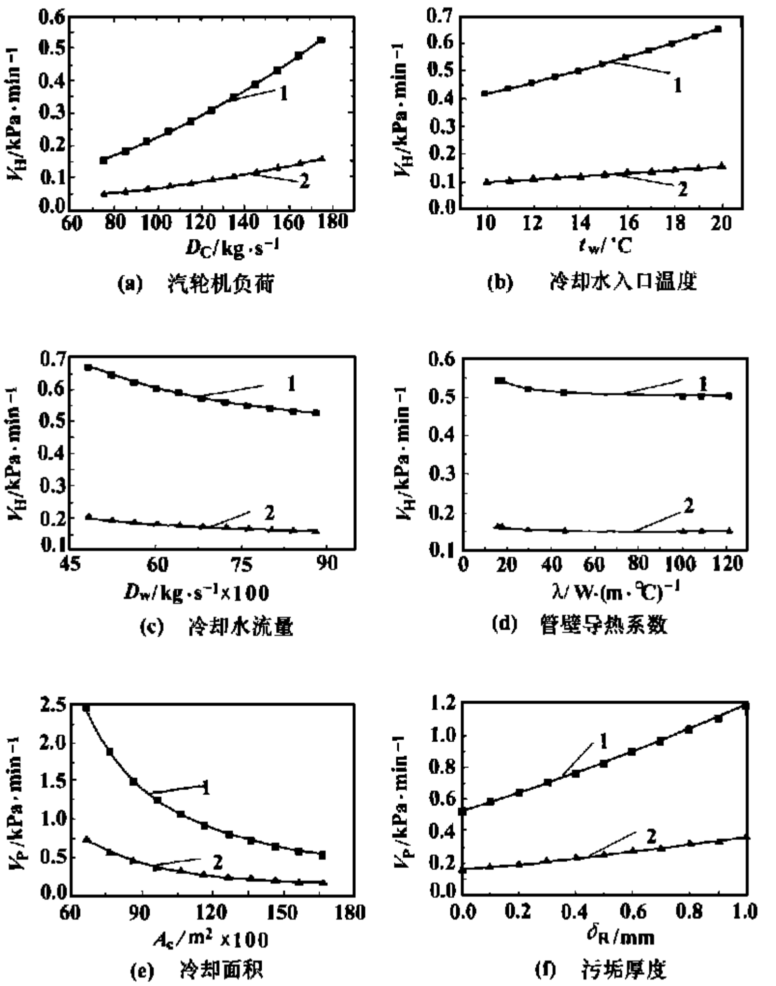
其中 $$X = 0.0324\frac{dp_c}{dt_c}\left(\frac{\Delta t A_c}{C_p D_w} \cdot \frac{e^{kA_C/C_pD_w}}{(e^{kA_C/C_pD_w} - 1)^2}\right) \quad (12)$$

$$Y = \frac{\alpha_0 d_s^{-0.04}\varepsilon^{-1.04}}{\left(\frac{d_1}{d_2}\frac{1}{\alpha_w} + \frac{1}{\alpha_s} + \frac{d_1}{2\lambda}\ln\frac{d_1}{d_2} + R_f\right)^2\alpha_s^2} \quad (13)$$

式(11) 表明，在单位时间漏入真空系统空气量 G_a 一定的情况下，真空系统严密性试验测得的真空下降速度还受到冷却水入口温度 t_{w1}、凝汽器单位冷却面积蒸汽负荷 d_s、冷却水流量 D_w、凝汽器管材导热系数 λ、凝汽器汽侧汽空间体积 V'' 及水侧污垢热阻 R_f 等因素的影响，亦即在真空系统严密性一定的前提下，不同的凝汽器运行参数对及不同的凝汽器结构或管材，对应不同的真空下降速度。

2.2　凝汽器的静态模拟计算实例

为了定量说明真空严密性试验中，凝汽器上述各参数对真空下降速度的影响，以某 300 MW 汽轮机凝汽器为例，该凝汽器的设计运行工况为：蒸汽负荷 $D_c = 175$ kg/s，冷却水入口温度 $t_{w1} = 15$ ℃，冷却水流量 $D_w = 8\,825$ kg/s，凝汽器管材为 $B5$，其导热系数 $\lambda = 29.1$ W/(m·℃)。在漏入真空系统空气量分别为 $G_a = 0.03$ kg/s 和 $G_a = 0.009$ kg/s 的条件下，分别改变汽轮机负荷、冷却水入口温度、冷却水流量、管材导热系数、凝汽器冷却面积及水侧污垢厚度，得到对应各工况下的真空下降速度如图 1 所示。



1— $G_a = 0.03$ kg/s；2— $G_a = 0.009$ kg/s

图 1　各参数对真空下降速度的影响

由图 1 可见，在真空严密性试验中测得的真空下降速度，随汽轮机负荷、冷却水入口温度、水侧污垢厚度的增加而增大，随冷却水流量、管材导热系数及冷却面积的增大而减小。亦即在漏入真空系统空气量一定的条件下，凝汽器的真空越低，则真空下降

速度越大。

同时，由图 1 还可以发现，漏入真空系统的空气量越多，则上述参数以真空下降速度的影响也越大。

通过上述对真空系统严密性试验的静态模拟计算可以看出，以真空下降速度作为判断真空系统严密性的指标存在一定的近似性。在漏入真空系统空气量一定的前提下，真空下降速度依当时的运行条件、凝汽器结构尺寸及管材的不同而不同。只有在相同的参数条件下，才能依据真空下降速度来判断真空系统的严密性。

3　结论

（1）在进行汽轮机真空系统严密性试验时，真空下降速度并不只是漏入真空系统空气量的单值函数，其数值还受到汽轮机负荷、冷却水入口温度、冷却水量、水侧脏污程度和凝汽器管材及本身结构的影响。

（2）真空严密性试验中测得的真空下降速度，随汽轮机负荷、冷却水入口温度、水侧污垢厚度的增加而增大，随冷却水流量、管材导热系数及冷却面积的增大而减小。而且漏入真空系统的空气量越多，这些参数对真空下降速度的影响也越大。

（3）分析真空严密性试验的结果，应该将上述这些因素对真空下降速度的影响进行修正，以便准确确定真空系统的严密性情况。

参考文献：

[1]　李 勇，陈梅倩. 汽轮机运行性能诊断技术及应用[M]. 北京：科学出版社，1999.

[2]　杨善让. 汽轮机凝汽设备及运行管理[M]. 北京：水利电力出版社，1993.

[3]　杨世铭. 传热学[M]. 第二版，北京：高等教育出版社，1996.

（何静芳　编辑）

经验交流

凝汽器铜管现场保洁措施

凝汽器铜管污染、结垢，致使传热系数降低，对数平均温差增大，真空下降。我厂采用深井地下水闭式循环系统，因自然水质钙、镁成份含量高，加之循环水蒸发浓缩和凉水塔风吹损失等因素，循环水处于不稳定状态，以致使铜管结垢，热交换能力下降，机组出力不足。在电厂条件下，我们采取了一系列铜管保洁措施，收到一定成效。

对于常温结垢较为严重的水质，首要的是根据水稳分析，采用连续加药阻垢、定期杀菌以及向循环水中定期加剥离剂的方法，控制污染及结垢。具体做法：在循环水中添加浓度为 50×10^{-6} 的阻垢缓蚀剂，使水中钙、镁离子形成不易析出的螯合物，避免铜管表面结垢。为防止藻类滋生繁衍，影响热交换，每运转班加氯 30 分钟，每月加剥离剂一次，常年控制异氧菌总量不大于 3×10^{5}，收到较好的使用效果。循环水中添加阻垢缓蚀剂，可使循环水浓缩倍率得以提高，减少排污水量及补充水量，减少循环水系统的清扫次数和运行费用，增加了设备年运行时数，从而提高了发电量，使发电煤耗下降。经考核，循环水加药处理以后，年增收 165 万元人民币。

现阶段水质稳定剂的生产水平和循环水加药除垢的实施工艺，很难达到 100％的阻垢效果。我们还辅以凝汽器胶球清洗的措施，采取每运行班清洗二次，且每次不少于 30 分钟的方法，将沉积铜管中尚处于松散状的钙、镁离子或杂质清扫、驱逐干净，免其在铜管内附着、生根，达到凝汽器铜管保洁的目的。当循环水浓缩倍数增高到 3 倍左右时，适时进行排污是克服循环水生成螯合物，减轻铜管结垢和异物沉积并减少补充水的另一措施。凝汽器铜管中若已形成较厚的垢层，停机后采用高压水射流逐根清洗，或采用化学方法清洗铜管，不失为有效的除垢方法。

机组停运后尽早放掉凝汽器中积水，利于克服铜管中杂物沉淀或结垢。利用停机的时机，对冷却塔内的淋水填料及喷溅装置进行清扫、维修，并辅以冷却塔周边环境的净化和有效治理，对凝汽器铜管保洁有着实际意义。

凝汽器铜管保洁不但提高了热传导能力，使机组热效率得以提高，是电厂安全经济运行的需要，另一重要意义在于可以防止凝汽器铜管垢底腐蚀，保全凝汽器有效寿命。

（淮北市热电厂　邮编 235029　王金明）

503～506
On the basis of the first and second laws of thermodynamics and under constant heat flux conditions an analysis was conducted of the influence of fouling on the thermodynamic performance of convection heat exchange process in a tube. A criterion for evaluating such an influence is put forward, the so-called relative increment number of dimensionless entropy generation. Also discussed is the effect of parameters such as in-tube fluid Reynolds number (in the absence of fouling) and dimensionless heat flux density on the above-mentioned increment number. The results of the study indicate that the above criterion reflects not only the effect of fouling on the in-tube heat transfer process but also the effect of fouling on the in-tube flow process. It is noted that the entropy generation caused by the fouling layer heat conduction plays an important role in the total entropy generation of the in-tube heat transfer process. In addition, the study results have been compared with those of a tube with constant wall temperature. **Key words**: constant heat flux, convection heat exchange, fouling, thermodynamic performance

先进汽轮机准三维设计方法研究与流型分析＝**A Study of the Quasi Three-dimensional Design Method for an Advanced Steam Turbine and an Analysis of Its Flow Patterns** [刊, 汉] / ZHANG Dong-yang, JIANG Hong-de (Institute of Engineering Thermophysics under the Chinese Academy of Sciences, Beijing, China, Post Code: 100080), LI Jiu-hua (Jiangxi Electric Power Testing Research Institute, Nanchong, Jiangxi Province, China, Post Code: 330006) // Journal of Engineering for Thermal Energy & Power. — 2001, 16(5). —507～509, 512
Presented is the development of a quasi three-dimensional design method for an advanced steam turbine. With a successfully modified 200 MW unit serving as an object of study the authors have performed the related analysis of its flow patterns. **Key words**: steam turbine, quasi three-dimensional design, flow pattern analysis

正交实验法在湿式烟气脱硫装置除水器实验中的应用＝**The Application of an Orthogonal Experimental Method in the Experimental Study of a Dehydrator for a Wet Flue Gas Desulfurization Unit** [刊, 汉] / QIU Zhong-zhu, ZHANG He-sheng, XU Ji-huan (Tongji University, Shanghai, China, Post Code: 200092) // Journal of Engineering for Thermal Energy & Power. — 2001, 16(5). —510～512
An orthogonal experimental method was employed to conduct the experimental study of the performance of a centrifugal dehydrator for a wet flue gas desulfurization unit. Through a variance analysis of the test data the effect of the various structural parameters of the dehydrator on water removal performance was ascertained along with an determination of the optimum structural parameters of the dehydrator. This had led to an optimum design of the latter. **Key words**: orthogonal experimental method, centrifugal dehydrator, dehydration performance, optimum design

大型油页岩循环床电站锅炉运行性能分析＝**Operating Performance Analysis of an Oil Shale-fired Circulating Fluidized Bed Boiler of the Highest Capacity Currently in Operation in China** [刊, 汉] / WANG Qing, HAO Zhi-jing, SUN Jian, et al (Power Engineering Department, Northeast Electric Power Institute, Jilin, China, Post Code: 132012) // Journal of Engineering for Thermal Energy & Power. — 2001, 16(5). —513～516
Presented are the design features, overall structure, operating performance and the test results of a 65 t/h oil shale-fired circulating fluidized bed boiler of low circulation ratio. The economic and social benefits of the power plant operating on oil shale were also analyzed. The test results show that the overall performance of the boiler has attained the advanced international level. **Key words**: oil shale, circulating fluidized bed, boiler, operation

汽轮机真空系统严密性试验的静态模拟计算＝**Static Simulation Calculation of the Air Tightness Test of a Steam Turbine Vacuum System** [刊, 汉] / LI Yong, DONG Yu-liang, YANG Shan-rang (Power Engineering Department, Northeast Electric Power Institute, Jilin, China, Post Code: 132012) // Journal of Engineering for Thermal Energy & Power. — 2001, 16(5). —517～519

Through a simulation calculation of the air-tightness test of a steam turbine vacuum system it is concluded that the vacuum decrease rate is not a single-valued function of the air leakage into the condenser. The vacuum decrease rate can be affected more or less by a multitude of factors. Among these one may list: steam turbine load, cooling water flow rate, inlet temperature of cooling water and condenser tube material, etc. A detailed analysis is peformed of the effect of the above factors on the vacuum decrease rate. The conclusions reached can be of some reference value for a more accurate evaluation of the steam turbine vacuum system. **Key words**: steam turbine, condenser, air-tightness test of a vacuum system

凝汽器喉部蒸汽流动的三维数值模拟＝**Three-dimensional Numerical Simulation of the Steam Flow at a Condenser Throat Section** [刊, 汉] / CUI Guo-min, CAI Zu-hui, LI Mei-ling (Thermal Energy Engineering Research Institute under the Shanghai University of Science and Technology, Shanghai, China, Post Code: 200093) //Journal of Engineering for Thermal Energy &Power. — 2001, 16(5). —520～522

With the help of a direct simulation Monte Carlo method incorporating a super-particle model and through a domain-decomposition and mathematical modeling of a steam turbine condenser throat a three-dimensional numerical simulation was conducted of the steam flow at the condenser throat of a specific structure. The simulation of the throat steam flow was undertaken with a focus on the analysis of its flow distribution. As a result, identified were the non-uniformity feature of the throat flow field and the underlying cause of the non-uniform flow field. **Key words**: condenser throat, numerical simulation, direct simulation Monte Carlo method

基于 MATLAB 的三轴燃气轮机动态仿真模型研究＝**Dynamic Simulation Modeling of a Three-shaft Gas Turbine Based on a Software MATLAB** [刊, 汉] / AO Chenyang, ZHANG Ning, CHEN Hua-qing (Naval Equipment Research Center, Beijing, China, Post Code: 100073) //Journal of Engineering for Thermal Energy &Power. — 2001, 16(5). —523～526

Simulation technology represents an effective means for the study of gas turbine performance. With the help of a quasi-nonlinear method set up was the mathematical model of a three-shaft gas turbine. An object-oriented dynamic simulation platform was developed for the three-shaft gas turbine on the basis of a dynamic simulation software MATLAB. The results of the simulation show that the simulation model is correct and rational, featuring simplicity and ease of use. **Key words**: software MATLAB, three-shaft gas turbine, simulation model, object-oriented approach

某型两级涡轮流场数值模拟＝**Numerical Simulation of the Flow Field of a Two-stage Turbine** [刊, 汉] / WU Meng, WANG Song-tao, FENG Guo-tai, WANG Zhong-qi, et al (Energy Science and Engineering Institute under the Harbin Institute of Technology, Harbin, China, Post Code: 150001) //Journal of Engineering for Thermal Energy &Power. — 2001, 16(5). —527～529

Through the use of a three-dimensional viscous flow calculation program a numerical simulation was performed of a two-stage turbine. The program adopts a Godunov scheme of third-order accuracy with a turbulent flow model being of a B-L algebraic one. During the calculation the effect of a change in specific heat has been taken into consideration. An analysis of the calculation results indicates that there lacks a proper reflection of the matching of gas flow angles. This comes about because the gas turbine was designed and calculated through the use of a stream surface S_2 and single row viscous flow with losses being taken account of. As a result, there emerged a relatively great positive incidence angle in the second stage stator, leading to an ineffective role of adopting a rear loading profile and a failure to achieve an decrease in secondary flow loss. In view of this it is necessary to conduct in the aerodynamic design a calculation of the matching of multi-stage viscous flows. **Key words**: three-dimensional flow, numerical simulation, two-stage turbine

半干式脱硫系统的热量物质衡算模型＝**Calculation Model of Heat and Mass Balance for a Semi-dry Flue Gas Desulfurization System** [刊, 汉] / GAO Ji-hui, WU Shao-hua, Qin Yu-kun (Energy Science and Engineering Institute