文章编号: 1001－2060(2007)05－0539－03

高压汽水两相流摩擦阻力特性的研究

朱玉琴[1], 李迓红[1], 毕勤成[2], 陈听宽[2]
(1. 西安石油大学 石油炼化工程技术研究中心, 陕西 西安 710065;
2. 西安交通大学 动力工程多相流国家重点试验室, 陕西 西安 710049)

摘　要: 研究国产 600 MW 直流锅炉水冷壁管在亚临界及近临界压力区的摩擦阻力特性。试验段为 ϕ25× 3 mm 的 1Cr18Ni9Ti 不锈钢管, 试验参数为: 压力 $p=12\sim21$ MPa, 质量流量 $G=400\sim1\ 200$ kg/(m^2·s), 质量含汽率 $x=0\sim1.0$。试验研究了工质压力、质量流速和质量含汽率对摩擦阻力的影响。根据所得的大量实验数据, 在理论分析的基础上得出了可供设计使用的水冷壁管摩擦阻力的关联式, 计算值与试验值的相对误差在 15% 以内。

关　键　词: 直流锅炉; 汽－水两相流; 水冷壁管; 摩擦阻力

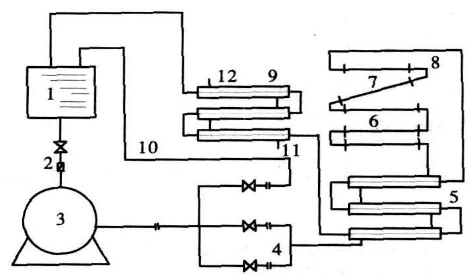
中图分类号: TK229.5　　文献标识码: A

引　言

近年来, 为了满足国民经济的发展对电力需求日益增加的需要, 我国正大力引进、研制和生产 600 MW 及以上的超临界变压运行直流锅炉。在锅炉设计的过程中, 为了确定水冷壁管的水动力特性, 必须具有水冷壁管摩擦阻力的数据。所以, 许多研究者对水冷壁管内单相和汽－液两相流体的阻力特性进行了实验研究[1]。但是, 这些研究一般局限在较低压力范围内, 对于高压和超高压下汽水两相流动阻力特性的研究还很少。为发展我国 600 MW 机组配套的超临界变压运行直流锅炉垂直部分的水冷壁管, 由西安交通大学与东方锅炉厂合作, 在西安交大动力工程多相流国家重点实验室的高压汽水试验台上, 针对实际锅炉水冷壁的结构参数和它的运行参数范围进行了传热与摩擦阻力特性的试验研究。本文主要阐述水冷壁管在高压和超高压下摩擦阻力特性的试验结果。

1　试验系统与试验参数

为模拟超临界锅炉水冷壁的实际工作条件, 试验在西安交通大学多相流国家重点试验室的高压汽－水两相流试验台上进行, 试验回路系统如图 1 所示, 试验系统的详细说明可参考文献[6]。高压柱塞泵出口额定压力可达 32 MPa, 流量可达 4.5 t/h, 与试验回路上的阀门配合, 可保证试验系统的压力和流量调节到预定值。



1-水箱; 2-滤网; 3-高压柱塞泵; 4-孔板流量计; 5-再生换热器; 6-预热器; 7、8-试验段; 9-冷却器; 10-旁路; 11-冷却水人口; 12-冷却水出口

图 1　试验系统回路简图

预热段采用交流电通过管子本身的电阻直接加热。通过自耦变压器调节电压可连续改变电加热功率, 最大加热功率可达 1 000 kW。

试验段采用 ϕ25× 3 mm 的 1Cr18Ni9Ti 不锈钢管, 其长度为 3 m。阻力测量段为水平布置, 目的在于消除重位压差, 以精确测量摩擦阻力。在试验段进口装有 Rosemont 压力变送器以确定压力; 采用 1151HP 电容式差压变送器配合锐边孔板来测量试验段流量; 电加热功率用测量到的电压和电流的有效值计算得到; 摩擦阻力采用环室取压方式, 用 Rosemont 差压变送器测量。用 7 支 ϕ3 mm 的 NiCr-NiSi 铠装热电偶测量系统工质温度, 其中 6 支布置在预热段, 1 支布置在试验段的进口。

本试验参数范围是: 压力 $p=12\sim21$ MPa; 质量流量 $G=400\sim1\ 200$ kg/(m^2·s); 质量含汽率 $x=0\sim1.0$。

收稿日期: 2007－01－18;　修订日期: 2007－02－14
基金项目: 国家自然科学基金资助项目(50323001)
作者简介: 朱玉琴(1965－), 女, 河南杞县人, 西安石油大学副教授.

2 试验结果及分析

我国在计算超临界压力锅炉的水动力特性时，单相摩擦阻力系数按下式计算[2]：

$$\lambda=1/4d\left(\lg 3700\frac{d}{k}\right)^2 \tag{1}$$

式中：d—管子内径，m；k—管子内壁绝对粗糙度，mm（碳钢及珠光体合金钢，$k=0.06$；奥氏体钢 $k=0.008$）。试验中，水冷壁管为 1Cr18Ni9Ti 不锈钢管，取 $k=0.008$ mm，代入式(1)得，$\lambda=0.016$。

通过试验可以测得两相摩擦阻力压降 Δp_{TP}，该值即为差压变送器的读数，液相(水)单独通过管道时压降 Δp_{LO} 为：

$$\Delta p_{LO}=\lambda\frac{L}{d}\frac{\rho_L u^2}{2}=\lambda\frac{L}{d}\frac{G^2}{2\rho_L} \tag{2}$$

式中：L—两取压环之间的距离，m；G—质量流速，kg/(m^2·s)。两相摩擦因数 $\phi^2_{LO}=\Delta p_{TP}/\Delta p_{LO}$。所以，确定了 Δp_{TP} 和 Δp_{LO} 之后，就可以计算出两相摩擦因数 ϕ^2_{LO}。

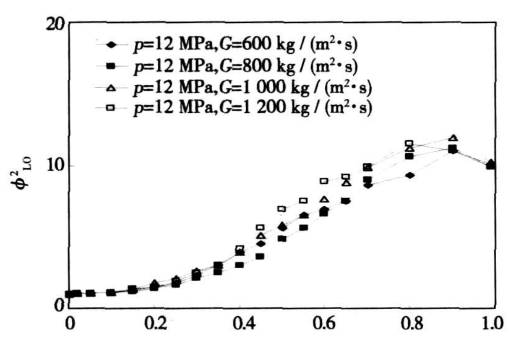


图 2　两相摩擦因数 ϕ^2_{LO} 与干度 x 的关系

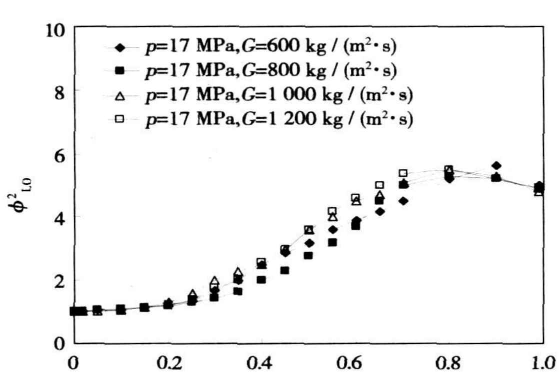


图 3　两相摩擦因数 ϕ^2_{LO} 与干度 x 的关系

图 2～图 4 是由试验得到的不同压力和质量流速下两相摩擦因数随质量含汽率的变化规律。试验结果表明，$\phi^2_{LO}\sim x$ 的关系是二次曲线，ϕ^2_{LO} 升到最大值后下降，当 $x=1.0$ 时，ϕ^2_{LO} 趋于该压力下水—汽两相密度比。在本试验的质量流速范围内，两相摩擦因数 ϕ^2_{LO} 与管内汽液两相流的质量流速关系不大。

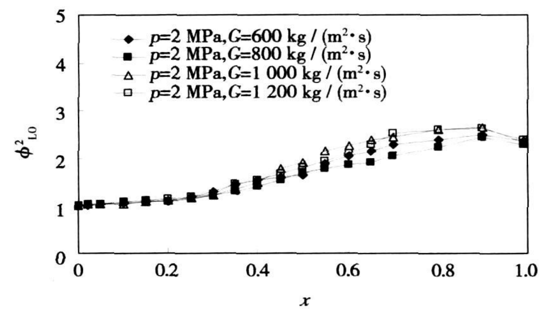


图 4　两相摩擦因数 ϕ^2_{LO} 与干度 x 的关系

3 试验数据的整理与分析

3.1 理论分析

根据 Chisholm 提出的“实际动压头”概念[5]，两相摩擦压降可表示为：

$$\frac{dp_{TP}}{dz}=\frac{\lambda}{d}\frac{1}{2}\left[xGu_G+(1-x)Gu_L\right] \tag{3}$$

液相单独通过管道时的压力梯度为：

$$\frac{dp_{LO}}{dz}=\frac{\lambda_L}{d}\frac{G^2}{2\rho_L} \tag{4}$$

根据连续性方程，汽相速度 u_G 和液相速度 u_L 分别为：

$$u_G=\frac{xG}{\rho_G\varphi},\ u_L=\frac{(1-x)G}{(1-\varphi)\rho_L} \tag{5}$$

将式(5)代入式(3)，得：

$$\frac{dp_{TP}}{dz}=\frac{\lambda}{d}\frac{G^2}{2\rho_L}\left[\frac{x^2\rho_L}{\varphi\rho_G}+\frac{(1-x)^2}{1-\varphi}\right] \tag{6}$$

式中：φ—截面含汽率。

假定流动进入自模化区，那么 $\lambda=\lambda_L$，两相摩擦因数 ϕ^2_{LO} 可表示为：

$$\phi^2_{LO}=\frac{dp_{TP}}{dp_{LO}}=\left(\frac{x^2}{\varphi}\right)\left(\frac{\rho_L}{\rho_G}\right)+\frac{(1-x)^2}{1-\varphi} \tag{7}$$

定义滑移比 $S=u_G/u_L$，由式(5)可推出截面含汽率 φ 为：

$$\varphi=\frac{x}{x+S(1-x)\rho_G/\rho_L} \tag{8}$$

将式(8)代入式(7)，得：

$$\phi^2_{LO}=1+x^2\left(\frac{\rho_L}{\rho_G}-1\right)+x(1-x)\left(\frac{1}{S}\frac{\rho_G}{\rho_L}+S-2\right)$$

$$= 1 + (\frac{\rho_L}{\rho_G} - 1)[Bx(1-x) + x^2] \quad (9)$$

其中 B 系数为:

$$B = \frac{(1/S)(\rho_G/\rho_L) + S - 2}{(\rho_G/\rho_L) - 1} \quad (10)$$

3.2　两相摩擦因数的经验关联式

由式(9)可得:

$$B = \frac{(\phi_{LO}^2 - 1)/[(\rho_L/\rho_G) - 1] - x^2}{x(1-x)} \quad (11)$$

为了便于关联试验数据, 将式(9)中 $Bx(1-x)$ 项用系数 C 表示, 即:

$$C = Bx(1-x) \quad (12)$$

则式(9)化简为:

$$\phi_{LO}^2 = 1 + (\frac{\rho_L}{\rho_G} - 1)[C + x^2] \quad (13)$$

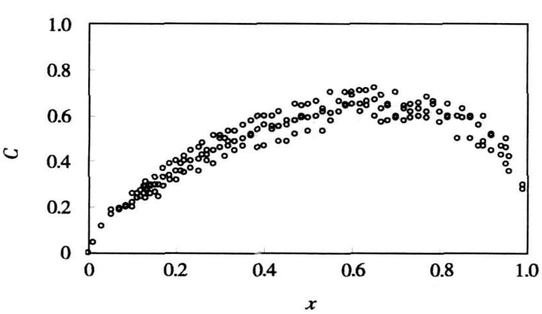


图 5　系数 C 与干度 x 的关系

图5是压力为 12～17 MPa 系数 C 随 x 的变化规律。系数 C 是两相密度比的函数, 因而系数 C 受压力的影响。在本文的临界压力以下, 试验范围大致可分为 12～17 MPa、19～21 MPa 两个区, 由图5同时考虑到式(12), 选下式为系数 C 的回归型线:

$$C = C_0 x^n (1-x)^m \quad (14)$$

式中: C_0—待定系数, m、n—待定指数。

根据多元回归分析可得: 当 $p = 12 \sim 17$ MPa 时:

$$C = 1.182x^{0.697}(1-x)^{0.308} \quad (15)$$

当 $p = 19 \sim 21$ MPa 时:

$$C = 0.890x^{0.567}(1-x)^{0.215} \quad (16)$$

根据回归式(15)和式(16)计算系数 C 再代入式(13)计算两相摩擦因数 ϕ_{LO}^2 与试验所得的两相摩擦因数 ϕ_{LO}^2 比较, 两者符合较好, 平均误差不超过 15%。

本文的计算较好地反映高质量含汽率区的两相摩擦阻力, 其型线与 Martinelli—Nelon 的计算曲线相似, 但避免了查线算图的不便, 为锅炉设计提供了方便。

4　结　论

(1) 随着压力的增加, 水冷壁管全液相两相摩擦因数减少; 在一定压力和质量流速下, 随着含汽率的增加, 全液相两相摩擦因数先持续增大, 当含汽率超过某一临界值后, 又随着含汽率的增加而减少。在相同压力条件下, 质量流速越小, 全液相两相摩擦因数越大, 但这种差别很小。

(2) 在本文试验参数范围内, 水冷壁管摩擦阻力可用式(15)和式(16)计算, 计算值与试验值的相对误差在 15%以内。

参考文献:

[1]　林宗虎. 气液两相流与沸腾传热[M]. 西安: 西安交通大学出版社, 2003.

[2]　电站锅炉水动力计算编写小组. 电站锅炉水动力计算方法标准[M]. 北京: 机械工业出版社, 1995.

[3]　举怀明. 载热质热物性计算程序及数据手册[M]. 北京: 原子能出版社, 1990.

[4]　郑建学, 陈听宽, 罗毓珊, 等. 内螺纹管高压汽水两相流摩擦阻力特性的研究[J]. 西安交通大学学报, 1994(5): 56—59.

[5]　CHISHOLM D. Two-phase flow in pipelines and heat exchangers[M]. London: George Godwin, 1983.

[6]　朱玉琴, 陈听宽, 毕勤成. 超临界压力下 600 MW 直流锅炉水冷壁管阻力特性的试验研究[J]. 动力工程, 2005(6): 786—789.

(编辑　滨)

ducted. The steam output of the boiler has increased by nearly 40% when compared with a 0.5 t/h conventional boiler having the same volume, thus exhibiting a comparatively good thermodynamic performance. **Key words**: pulsation combustion, pulsation heat transfer, pulsation frequency, boiler, self-excited burner

燃煤电站锅炉 NO_x 排放影响因素的数值模拟分析＝**Numerical Simulation and Analysis of the Influencing Factors of NO_x Emissions of Coal-fired Utility Boilers**[刊, 汉]/LIN Peng-yun, LUO Yong-hao, HU Li-yuan (Research Institute of Thermal Energy Engineering, Shanghai Jiaotong University, Shanghai, China, Post Code: 200240)//Journal of Engineering for Thermal Energy &Power. — 2007, 22(5). —529～533

A coal-fired utility boiler is one of the main sources of NO_x pollutants. By using a CFD (computational fluid dynamics) software platform and a numerical calculation method, a numerical simulation was performed of various combustion conditions of a utility boiler to study the different factors excercising an influence on NO_x emissions of the boiler in question. The calculation results show that the excess air ratio is one of the major factors influencing the production of NOx and the concentration of NO_x emissions will increase with an increase of the excess air ratio. A change of the secondary-air distribution mode can also influence the generation of NO_x. Among the three combustion conditions being calculated, the concentration of NO_x produced by equal air distribution is the lowest followed by that of a reverse-tower type air distribution. The girdled type air distribution has produced the highest concentration. To change the secondary-air deflection angle can influence the amount of NO_x produced. The concentration of NO_x emissions will decrease in case of an increase in the secondary-air deflection angle. **Key words**: coal-fired boiler, numerical simulation, NO_x emission, influencing factor

某200 MW四角切圆锅炉燃烧器改造降低 NO_x 数值模拟＝**Numerical Simulation of the Modification of Burners for a 200 MW Tangentially Fired Boiler to Reduce NO_x Emissions**[刊, 汉]/XING Fei, FAN Wei-jun (College of Energy Source and Power Engineering, Beijing University of Aeronautics and Astronautics, Beijing, China, Post Code: 100083), CUI Jin-lei, DENG Yuan-kai (Beijing Guodian Kehuan Clean Combustion Engineering Technology Co. Ltd., Beijing, China, Post Code: 100039)//Journal of Engineering for Thermal Energy &Power. — 2007, 22(5). — 534～538

To address such problems as comparatively high NO_x emissions and serious slagging in the 200 MW tangentially fired boiler of a power plant in China, a study has been conducted of the in-furnace combustion process before and after burner modification by using a CFD (computational fluid dynamics) software platform and numerical simulation method. The calculation results show that due to the effect of attached wall jet flow, high-temperature zones are concentrated in the central part of the furnace, effectively preventing boiler slagging. After modification, the furnace had a reducing atmosphere zone even greater than that of a boiler adopting conventional combustion modes, thus suppressing the generation of NOx emissions and reducing them by 34.6%. The calculation results after the modification correspond quite well with those of test ones. The numerical simulation can well provide a theoretical basis for the design, modification and operation of the boiler in question. **Key words**: boiler combustion, temperature distribution, NO_x emissions, numerical simulation

高压汽水两相流摩擦阻力特性的研究＝**A Study of Frictional Resistance Characteristics of High-pressure Steam-water Two-phase Flows**[刊, 汉]/ZHU Yu-qin, LI Ya-hong (Technology Research Center of Petroleum Refinery Engineering, Xi' an Shiyou University, Xi' an, China, Post Code: 710065), BI Qin-cheng, CHEN Ting-kuan (National Key Laboratory on Multi-phase Flows in Power Engineering, Xi' an Jiaotong University, Xi' an, China, Post Code: 710049)//Journal of Engineering for Thermal Energy &Power. — 2007, 22(5). —539～541

A study was performed of the frictional resistance characteristics of the water-wall tubes in subcritical and close-to-critical pressure zones of a 600 MW once-through boiler made in China. The test section consists of 1Cr18Ni9Ti stainless steel tubes having a diameter of $\phi 25 \times 3$ mm with the following experimental parameters: pressure $p = 12 \sim 21$ MPa, mass flow rate G= 400 ~ 1200 kg/(m^2·s) and mass steam content x = 0 ~ 1.0. Through the tests, studied was the impact of working medium pressure, mass flow velocity and mass steam content on the frictional resistance. On the basis of a great deal of data obtained from the tests and related theoretical analyses, a correlation formula of frictional resistance of water-wall tubes was derived, which is suited for design purposes. The relative error between the calculated values and test ones does not exceed 15%. **Key words**: once-through boiler, steam-water two-phase flow, water-wall tubes, frictional resistance

煤种对超细煤粉再燃脱硝效率影响的数值研究= **A Numerical Study of the Influence of Coal Ranks on Reburning-based Denitration Efficiency of Superfine Pulverized Coal**[刊,汉]/JIA Yan-yan, BI Ming-shu, LIU Zhi (College of Chemical Engineering, Dalian University of Technology, Dalian, China, Post Code: 116012)//Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —542~547

By using CFD (computational fluid dynamics) calculation software FLUENT6.1 a three-dimensional numerical simulation was performed of the superfine pulverized-coal reburning process in a full-size tangentially fired boiler. With five kinds of superfine pulverized coal having a comparatively big difference in coal quality serving as the reburning fuels, studied was the law governing the variation of NO_x emissions with the following factors: the length of the reburning zone, injection location of the reburning fuel, excess air ratio α_{op} in the reburning zone and reburning quantity. The study results show that for different coal ranks serving as reburning fuels there exists an identical optimal injection location of a reburning fuel. The greater the coal volatile content, the more notable the reburning effectiveness. The NO_x removal rate increases with an increase of the length of the reburning zone and the reburning coal quantity. The excess air ratio in the reburning zone has a major influence on the NO_x removal rate. Through an analysis of the calculation results, an empirical formula was obtained, showing the relationship between the volatile content V_d of the dry fuel basis of the reburning pulverized coal and the optimal value of excess air ratio α_{op} in the reburning zone, thus providing a convenient approach for the optimization of combustion parameters. **Key words**: superfine pulverized coal, reburning-based denitration, numerical simulation, full-size boiler

利用电容层析测量煤粉浓度的实验研究= **An Experimental Study of Pulverized Coal Concentration Measurement by Using Capacitance Tomography**[刊,汉]/SUN Meng, LI Zhi-hong, JIANG fan, et al (Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing, China, Post Code: 100080)//Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —548~550

The accurate measurement of pulverized coal concentration in air pipes and its proper regulation are very important to the safe and cost-effective operation of a boiler during its combustion process. By using the capacitance tomography, tested and studied was the concentration of solid conveyed by a dilute-phase pneumatic force at a normal temperature. To overcome the impact of the nonuniformity of the sensitive field of a capacitance sensor on the image formation, a cyclone separator was set up in the test system. The role of the cyclone separator is to concentrate the solid particles mainly in the wall-surface zone with the electrodes of the sensor being located on the straight pipe of the separator. The advantage of the above testing method consists in its not demolishing the operating characteristics of the original system. The method pertains to a non-intrusive type of on-line quick testing technology. The test results obtained from the test stand and relevant on-line measuring system show that the method under discussion is feasible. **Key words**: pneumatic conveyance, capacitance tomography, measurement of flow rate, gas-solid two-phase flow

三偏心快关阀的液压系统设计与动态特性仿真= **Design of a Hydraulic System for a Three-eccentricity Quick Closing Valve and Simulation of its Dynamic Characteristics**[刊,汉]/LAI Xi-de, HE Hai-bin (College of Energy Source and Environment, Xihua University, Chengdu, China, Post Code: 610039), YANG Jiong-bo (Chengdu Huaxi Chemical Engineering Science and Technology Stock Co. Ltd., Chengdu, China, Post Code: 611830), ZHANG ji-jun