

# 350 MW 超临界四角切圆锅炉冷态空气动力场试验及仿真研究

李智<sup>1</sup>, 王义松<sup>1</sup>, 宋振宇<sup>2</sup>

(1. 沈阳工程学院 辽宁 沈阳 110136; 2. 辽宁电力科学研究所 辽宁 沈阳 110006)

**摘要:** 本文结合华润电力盘锦热电厂 350 MW 机组新建工程的调整试验工作, 针对该锅炉的炉内空气动力学特性进行了仿真研究。利用计算流体力学 Fluent 软件, 以试验工况参数作为边界条件, 对炉膛内部进行了全范围三维仿真模拟。计算得到了炉内速度场, 并将此模拟计算结果与其中某一炉膛截面实测的速度场进行了对比验证。计算结果基本符合实际测量值, 验证了仿真模型和模拟计算方法的合理性。同时, 计算结果表明炉内一次风、二次风和 SOFA 风截面速度分布合理, 气流相交位置良好。此外, 可模拟得到实际较难测量的炉内湍流强度分布, 计算结果显示其分布合理, 有利于炉内的燃烧稳定和提高燃烧效率。该研究成果应用于该机组调试工作中, 在一定程度上简化了试验过程, 减少了工作量, 也得到了更丰富、更直观的试验结果。

**关键词:** 超临界锅炉; 冷态空气动力场; 仿真模拟

中图分类号: TK229.2 文献标识码: A

DOI: 10.16146/j.cnki.rndlge.2016.09.019

## 引言

电站锅炉新建工程建设中, 由于没有运行经验数据, 需要针对不同类型的锅炉, 进行大量的调试工作来将锅炉调整到最佳的运行状态, 以保证机组顺利通过试运行。采用四角切圆燃烧方式的锅炉炉内燃烧特性、传热过程和流体流动状态等均比较复杂, 且对四角热负荷的均匀性要求较高<sup>[1]</sup>。新建机组通常通过对此类锅炉进行通风试验, 来检验系统及转机整体运行情况, 以及整个烟风系统冷态运行特性及调节特性。尤其需进行炉内空气动力场试验, 来了解炉内空气动力学特性, 为锅炉的启动运行及热态燃烧调整提供参考依据。

本文根据华润电力盘锦热电厂 2 × 350 MW 新建工程 1 号机组锅炉的调试需求开展了研究, 并将研究成果应用于调试中。传统的冷态空气动力场测

量方法过程复杂, 耗时费力, 炉内测量环境恶劣。而且, 若想得到整个炉膛的速度场分布, 则工作量巨大。本文结合了锅炉冷态空气动力场试验工作, 用实际试验工况参数进行全炉膛仿真研究, 只需将仿真结果与一层实测的速度场进行对比验证, 即可得到认为合理的全炉膛三维流动特性, 从而在一定程度上简化了试验过程。

## 1 锅炉本体介绍

本文研究对象为哈尔滨锅炉厂有限责任公司自主开发设计制造的超临界褐煤锅炉, 型号为 HG-1150/25.4-HM2。此锅炉为一次中间再热、超临界压力变压运行, 采用不带再循环泵的大气扩容式启动系统的直流锅炉, 锅炉采用单炉膛、平衡通风、固态排渣、全钢架、全悬吊结构、π型和紧身封闭布置。炉宽 14.627 m, 深 14.627 m, 高 58.9 m。采用中速磨直吹式制粉系统, 每炉配 6 台 MPS190HP-II A 磨煤机, 额定负荷下 5 台磨煤机运行, 1 台磨煤机备用, 煤粉细度 R<sub>90</sub> = 35%。燃烧器采用四角切圆布置方式, 每层 4 只燃烧器对应 1 台磨煤机, 4 支射流形成直径为 754 mm 的假想切圆。SOFA 风喷口布置在主燃烧器区上方水冷壁的四角, 以实现分级燃烧, 降低 NO<sub>x</sub> 排放。燃烧器布置方式如图 1 所示。

## 2 炉内冷态模化试验

### 2.1 冷态模化原理

在设计新型锅炉时, 改造原有锅炉后, 或新建锅炉投入运行之前, 可通过冷态模化试验来了解、掌握

收稿日期: 2015-08-21; 修订日期: 2015-11-16

基金项目: 国家自然科学基金资助项目(61371200)

作者简介: 李智(1964-), 男, 辽宁沈阳人, 沈阳工程学院教授。

其流动规律 验证和修改设计、改造、调试及运行优化方案<sup>[2]</sup>。应用锅炉冷态模化原理,可以在等温冷态工况下模拟运行时炉内热态工况下的流动情况,但两者是有差别的。因为实际炉膛内部在燃烧状态下显然是存在温度梯度的,冷态模化只能得到关于炉内流动特性的一些定性结果。如果根据相似原理,在一定的边界条件下进行冷态试验,可以在一定程度上接近真实炉内情况,特别是在燃烧器区域的燃烧器喷口处较为符合。因此,进行冷态试验要满足以下的模化条件:

(1) 模型与实物的几何相似:冷炉试验是在所研究的原锅炉上进行,因此满足此条件;

(2) 保持气流运动状态进入自模化区:在流动过程中,起主要作用的是表示惯性力与粘性力之比的雷诺数( $Re = \mu d/\nu$ )。在等温流动时,它决定了流动的阻力特性,以欧拉数来表示压力与惯性力比值,即:

$$Eu = \frac{\nabla p}{\rho \omega^2} = f(Re) \tag{1}$$

式中:  $Eu$ —欧拉数;  $\nabla p$ —压差, Pa;  $\rho$ —气体密度,  $kg/m^3$ ;  $\omega$ —气流速度,  $m/s$ ;  $Re$ —雷诺数。

当  $Re$  大于某值后,  $Eu$  不再与  $Re$  有关而不变化,即惯性力起决定作用,此时气流运动状态进入自模化区;

(3) 边界条件相似:要求满足冷态试验工况与设计的热态工况下各层气流动量比相等,以确定各喷口试验风速来满足该条件。

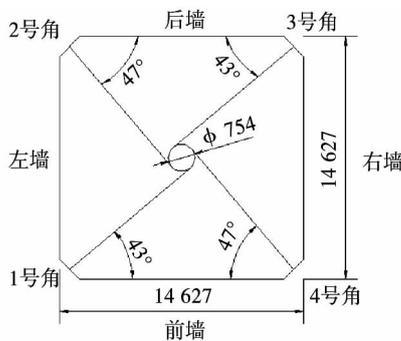


图 1 燃烧器布置方式

Fig. 1 Arrangement modes of burners

### 2.2 冷态模化工况的确定

根据以上原理,可以确定试验的一、二次风速和风量。试验值和设计值的一、二次风动量比为:

$$\frac{m_{1M}\omega_{1M}}{m_{2M}\omega_{2M}} = \frac{(1+k\varphi)m_{10}\omega_{10}}{m_{20}\omega_{20}} \tag{2}$$

式中:  $m$ —质量流速,  $kg/s$ ;  $\omega$ —速度,  $m/s$ ; 以角标  $O$ 、 $M$  分别代表设计值和冷态试验值,角标  $1$ 、 $2$ 、 $p$  分别代表一、二次风和煤粉颗粒。 $\varphi = m_p/m_{10}$ ,为一次风煤粉的质量浓度;  $k = \omega_p/\omega_{10}$ ,为考虑煤粉与一次风流速的滑移系数,经计算得  $0.83$ ,一般情况也可以取经验值  $0.8$ 。

在试验工况下,炉内温度  $t_{1M} = t_{2M}$ ,因此得到试验的一、二次风速比为:

$$\frac{\omega_{1M}}{\omega_{2M}} = \frac{\omega_{10}}{\omega_{20}} \sqrt{\frac{t_{20} + 273}{t_{10} + 273}} (1 + k\varphi) \tag{3}$$

本文利用试验与设计工况二次风射流的雷诺数相等,来先确定二次风速,同时其欧拉数相等。雷诺数相等而得到的二次风速计算表达式:

$$\omega_{2M} = \frac{\omega_{20}\rho_{20}m_{2M}}{\rho_{2M}m_{20}} \tag{4}$$

欧拉数相等而得到的二次风速计算表达式:

$$\omega_{2M} = \omega_{20} \sqrt{\frac{\rho_{20}}{\rho_{2M}} \cdot \frac{\Delta P_{2M}}{\Delta P_{20}}} \tag{5}$$

式中:  $\Delta P$ —炉内气体流动阻力,与风机运行工况有关。

本锅炉是超临界燃煤直流锅炉,可适用于各种变压工况运行,具有较高的锅炉效率和可靠性。在设计条件下任何 5 台磨煤机运行时,均能长期带满锅炉最大连续出力工况 (BMCR) 负荷运行,以 BM-CR 参数为设计值进行确定试验工况的计算。锅炉 BMCR 时燃烧器的主要设计参数如表 1 所示。

表 1 燃烧器主要设计参数

Tab. 1 Main design parameters of the burners

参数	数值
单个喷嘴热功率/MW	45.1
单个喷嘴煤量/ $t \cdot h^{-1}$	11.8
一次风率/%	33.79
一次风速/ $m \cdot s^{-1}$	27
一次风温/ $^{\circ}C$	65
二次风率/%	60.21
二次风速度/ $m \cdot s^{-1}$	48
二次风温/ $^{\circ}C$	379

通过计算得到的试验工况最低风速和风量,并校核模化条件和风机出力,计算结果如表 2 所示。

计算结果确定了此最低风速工况可满足模化条件,且风机容量能够满足试验所需风量的要求。

表 2 满足模化条件所需最低风速的工况  
Tab.2 Minimum wind speed conditions required by the modeling conditions

参数	数值
一次风速 /m · s <sup>-1</sup>	10
二次风速 /m · s <sup>-1</sup>	11
一次风雷诺数	4.39 × 10 <sup>5</sup>
一次风临界雷诺数	1.48 × 10 <sup>3</sup>
二次风雷诺数	1.80 × 10 <sup>5</sup>
二次风临界雷诺数	7.5 × 10 <sup>4</sup>
炉膛气流最小雷诺数	2.90 × 10 <sup>5</sup>
炉膛临界雷诺数	7.5 × 10 <sup>4</sup>
一次风风量 /t · h <sup>-1</sup>	339
两台一次风机总额定出力 /t · h <sup>-1</sup>	856
二次风风量 /t · h <sup>-1</sup>	494
两台送风机总额定出力 /t · h <sup>-1</sup>	892

但为了在试验时便于观察测量,通常尽量将风机调整至较大出力工况,且必须满足一、二次风动量

比相等的条件。试验中将所有二次风和 SOFA 风挡板调整至 60% 开度,实际试验工况如表 3 所示。试验采用的是飘带法,试验时在最下层一次风高度处搭建脚手架平台,用铁丝拉线形成一个米字架,从炉膛中心开始,每隔 400 mm 系 1 根飘带,调整锅炉系统各设备到该试验工况,使一次风和二次风速度达到计算得到的冷态模化风速,然后用热线风速仪测量铁丝拉线上各点和近壁面处的风速。

### 3 冷态空气动力场仿真

#### 3.1 仿真模型

本文在锅炉冷态空气动力场试验的基础上建立仿真模型,包括几何模型、数值模型和边界条件。按照锅炉设计图建立的几何模型大小与实际锅炉相等,满足模型与实物几何相似的条件。作为边界条件的一次风和二次风速度、温度和炉膛出口压力等参数,都来源于上文所述的试验工况参数。因此,此模型不仅能满足了冷态模化原理的条件,又能使其仿真结果更真实合理。

表 3 实际试验工况数值

Tab.3 Values obtained under the actual test conditions

项目	一次风速 /m · s <sup>-1</sup>	二次风速 /m · s <sup>-1</sup>	二次风风量 /t · h <sup>-1</sup>	总风量 /t · h <sup>-1</sup>	温度 /℃	炉膛压力 /Pa
数值	15	17	750	1 328	3	-20

#### 3.1.1 几何模型

利用 FLUENT 的前处理软件 Gambit,建立从炉膛底部到炉膛出口的三维模型,如图 2 所示。根据炉膛结构和工作特点,为了方便对炉膛进行非一致计算来减少网格数量,将其分成 5 个计算区域,并分别在横截面上划分面网格,然后在各自区域沿高度方向延伸成体网格。从下至上分别是:冷灰斗区,冷灰斗区与燃烧器区连接区,燃烧器区,燃烧器区与炉膛上部区连接区,炉膛上部区。

炉内燃烧器区域的物理量变化剧烈,采用合理的网格划分方法可以使计算更精确。De Vahl Davis 和 Mallinson 曾在二维直角坐标系下给出了流动的伪扩散系数的表达式<sup>[3]</sup>,即:

$$\Gamma_f = \frac{\rho U \Delta x \Delta y \sin 2\theta}{4(\Delta x \cos^3 \theta + \Delta y \sin^3 \theta)} \quad (6)$$

式中:  $\Delta x$  和  $\Delta y$ —网格尺寸;  $U$ —流体在网格处的流

动速度;  $\theta$ —速度与网格边界线之间的夹角。

从式(6)可以看出,对于燃烧器四角切圆布置的矩形炉膛截面,若气流方向与网格边界线夹角约成 45° 时,则在数值计算时易出现伪扩散现象,使模拟结果不准确<sup>[4]</sup>。本文采用图 3 的星形网格结构,使流动方向与网格边界线基本一致,有效的避免了伪扩散。同时为了提高计算精度,在物理量变化较剧烈的区域,如折焰角处、燃烧器区域,特别是喷口附近进行网格加密。

#### 3.1.2 数值模型

在冷态空气动力场试验工况下,炉内的气流为湍流流动。本文采用更符合炉内流动的需要对正应力进行数学约束的 Realizek  $k - \epsilon$  模型。其湍动能和耗散率的输运方程表示为:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M \quad (7)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 E \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{K} C_{3\varepsilon} G_b \quad (8)$$

式中:  $G_k$ —由于平均速度梯度引起的湍动能产生;  $G_b$ —由于浮力影响引起的湍动能产生;  $Y_M$ —可压缩湍流脉动膨胀对总的耗散率的影响;  $C_{1\varepsilon}$ 、 $C_{2\varepsilon}$ 、 $C_{3\varepsilon}$ 、 $C_2$ —经验常数;  $\sigma_k$ 、 $\sigma_\varepsilon$ —湍动能和湍动耗散率对应的普朗特数。

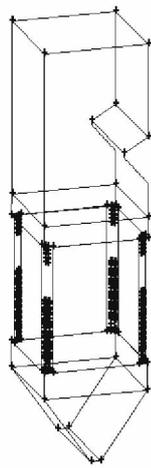


图 2 炉膛几何结构

Fig. 2 Geometrical structure of the furnace

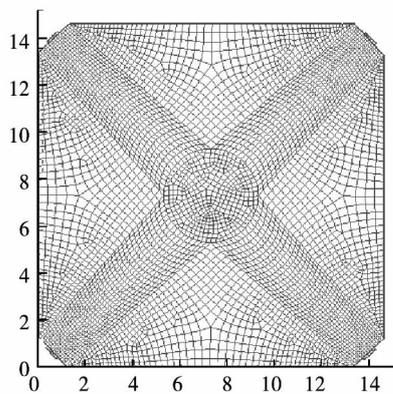


图 3 燃烧器区域截面网格

Fig. 3 Sectional grid in domain of the burners

然而 Realizable  $k - \varepsilon$  模型计算充分发展的湍流运动时才是有效,只能用于求解雷诺数较高的湍流

核心区域的流动。因此,在壁面区域采用标准壁面函数(Standard Wall Functions)模型来配合湍流模型进行计算,它是一组描述在近壁区流动的半经验公式。这样不但保证了计算精度,又避免了在炉膛壁面附近过细的划分网格。

数值计算采用一阶迎风离散格式,迭代求解方法使用压力修正法的 SIMPLE 方法,收敛判据为所有变量的残差  $\leq 10^{-4}$ ,并且进出口流量应相等。

### 3.1.3 边界条件

燃烧器喷口和二次风喷口的截面设置为速度进口,将试验工况的风速输入到进口边界条件中。矩形喷口的当量直径  $H$  和湍流强度  $I$  按下式计算:

$$H = 4A/x \quad (9)$$

$$I = 0.16 (Re_H)^{-1/8} \quad (10)$$

式中:  $A$ —喷口截面积,  $m^2$ ;  $x$ —喷口截面周长,  $m$ ;  $Re_H$ —喷口处的雷诺数。

炉膛出口截面设置为压力出口,假设出口截面的流动为充分发展流动,将炉膛出口处压力测点的数值输入到出口边界条件中。炉膛壁面的边界条件设置为固定壁面、无速度滑移。

## 3.2 仿真结果及分析

### 3.2.1 切圆直径

图 4 为模拟出的炉膛最下层 A 层一次风喷口截面速度矢量,可以看到切圆直径约为 7 m,与冷态试验时测量的实际切圆直径相同。根据冷态试验时的观察,飘带的飘动方向代表气体流动方向,俯瞰方向为逆时针方向,仿真结果符合实际试验情况。因此,对炉膛截面切圆的模拟,可以判断炉内旋流合理,切圆大小正常,无冲刷壁面现象。

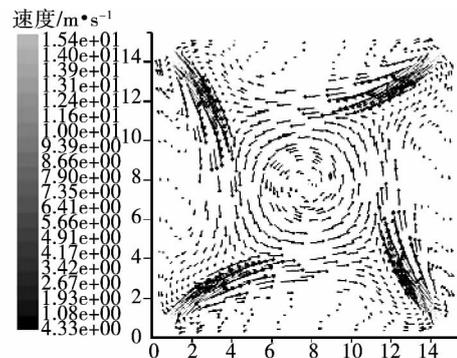


图 4 A 层一次风喷口截面速度矢量

Fig. 4 Velocity vector in the section of the primary air spout in the Layer A

### 3.2.2 速度场分析

炉膛 A 层一次风喷口截面速度等势图如图 5 所示,气流从四角燃烧器喷口进入炉膛后,速度逐渐衰减,并在中心形成切圆等势图形。图 6 为实测的此截面各喷口对角线速度分布数值与仿真结果的对比,实测和仿真得到的壁面处速度都为 1 m/s,表明仿真结果基本符合实际的速度分布。

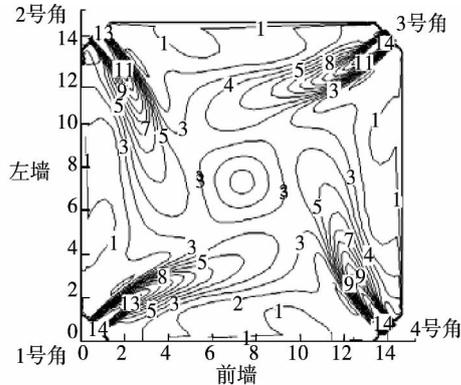


图 5 A 层一次风喷口截面速度等势图

Fig. 5 Chart showing the contours of the velocities in the section of the primary air spout in the Layer A

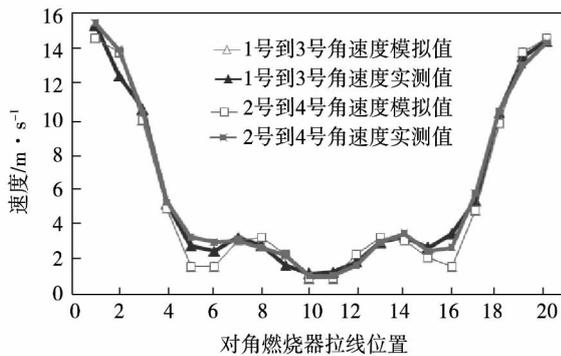


图 6 A 层一次风截面速度

Fig. 6 Primary air speed in the section in the Layer A

通过模拟,还可以得到如图 7 ~ 图 9 所示的 CD 层二次风、E 层一次风和 SOFA 风炉膛截面的速度等势图,并且可以模拟出如图 10 所示的三维图形,可更直观的进一步了解炉膛各层截面流动特性。四角喷射的气流相交位置良好,将有利于热态运行时煤粉的着火和燃烧稳定。SOFA 风的刚性较强,可有效的降低  $NO_x$  生成。

### 3.2.3 炉内湍流特性

湍流强度表征了流体的脉动程度<sup>[5]</sup>。湍流强度影响着燃烧过程中燃料与氧化剂的混合、扩散和

燃烧反应,强度分布合理将有利于炉内的燃烧稳定和提高燃烧效率。炉膛 A 层一次风截面和沿炉膛高度方向的湍流强度分布如图 11、图 12 所示。

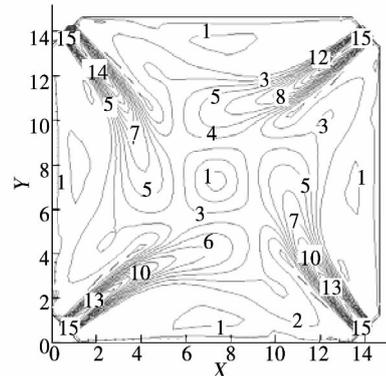


图 7 CD 层二次风截面速度等势图

Fig. 7 Chart showing the contours of the velocities in the section of the primary air spout in the Layer CD

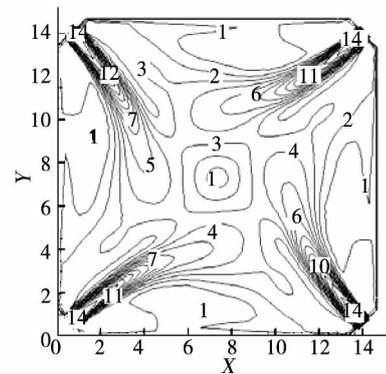


图 8 E 层一次风截面速度等势图

Fig. 8 Chart showing the contours of the velocities in the section of the primary air spout in the Layer E

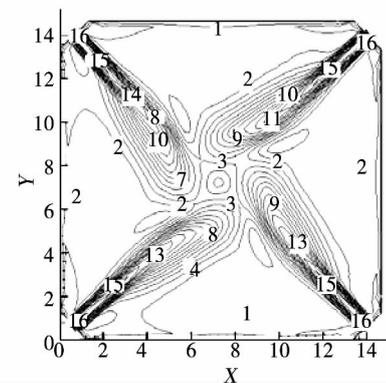


图 9 SOFA 风截面速度等势图

Fig. 9 Chart showing the contours of the SOFA air velocities in the section

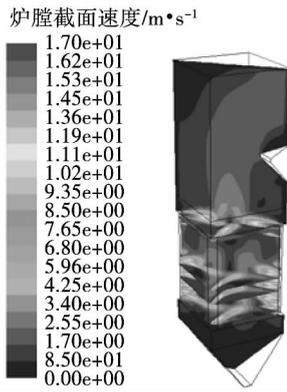


图 10 炉膛三维速度分布图

Fig. 10 Chart showing the distribution of the three-dimensional velocities in the furnace

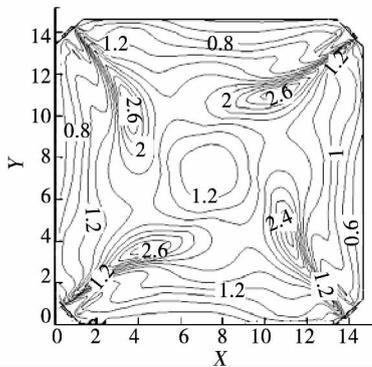


图 11 炉膛 A 层一次风截面湍流强度

Fig. 11 Primary air section turbulent flow intensity in the Layer A in the furnace

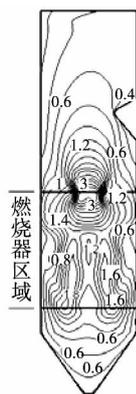


图 12 沿炉膛高度方向湍流强度

Fig. 12 Turbulent flow intensity along the height direction of the furnace

在一次风界面上,炉膛的近壁面处,湍流强度较低,高强度集中在距离燃烧器喷口一定距离处,能使

燃烧集中在切圆区域,火焰充满度良好,更可有效避免炉内结焦。沿炉膛高度方向,湍流强度大的区域集中在燃烧器区域,最大处在燃烧器区域末端,而后逐渐减小,有利于煤粉燃尽。此分布是由于四角的射流在离喷口一定距离区域混合,产生的旋流向上流动,不断推动上层气流,并与其混合,加强了湍流强度。

### 4 结 论

冷态空气动力场仿真结果与试验测量基本一致,验证了该模型可以合理的、有效的描述炉膛内部的流动特性。模拟得到的三维图形比试验测量数据表达的更直观,且数值计算得到的湍流强度分布更有助于分析炉内的流动特性。可以将试验和仿真相结合来研究炉内流动和燃烧,来简化工程中繁琐的工作量。此方法同样可以应用于锅炉的研发设计、调整试验和优化改造,可在生产实际中带来巨大利益。

### 参考文献:

[1] 岑可法. 锅炉燃烧试验研究方法及测量技术 [M]. 北京: 水利电力出版社, 1987.  
 CEN Ke-fa. Methods for studying the combustion tests in boilers and measurement technologies [M]. Beijing: China Water Conservation and Electric Power Press, 1987.

[2] 孙学信. 燃煤锅炉燃烧试验技术与方法 [M]. 北京: 中国电力出版社, 2002.  
 SUN Xue-xin. Coal-fired boiler combustion test technologies and methods [M]. Beijing: China Electric Power Press, 2002.

[3] Patankar S V. Numerical Heat Transfer and Fluid Flow ( Hemisphere Series on Computational Methods in Mechanics and Thermal Science) [M]. John Benjamins Publishing Co, 1980.

[4] 潘 维, 池作和, 斯东波, 等. 四角切圆燃烧锅炉炉膛网格生成方法的研究 [J]. 动力工程, 2005( 03) : 359 - 363.  
 PAN Wei, CHI Zuo-he, SI Dong-bo, et al. Study of the method for generating mesh grid in the furnace of a tangentially fired boiler [J]. Journal of Power Engineering, 2005( 03) : 359 - 363.

[5] 衡丽君, 段坤杰, 何长征. 220t/h 锅炉冷态数值试验研究 [J]. 热能动力工程, 2006( 06) : 573 - 577.  
 HENG Li-jun, DUAN Kun-jie, HE Chang-zheng. Cold state numerical experimental study of a 220 t/h boiler [J]. Journal of Engineering for Thermal Energy and Power, 2006( 06) : 573 - 577.

( 刘 瑶 编辑)

performed of the movement and deposition of pollutant particles in the atmosphere around the insulators to analyze the influence of the wind speed, mass concentration and particle diameters on the foul deposition speed and areas on the surface of the insulators. It has been found that the simulation results are in relatively good agreement with the test ones and the mathematical model thus established can relatively good reflect the law governing the foul deposition on the surface of the insulators. The density of fouls deposited on the surface of the insulators will increase with an increase of the wind speed and such an increment speed will gradually increase. The foul density ratio of the top and bottom surface will gradually decrease with an increase of the wind speed and the foul density and mass concentration on the surface of the insulators will increase in a linear relationship. The particles in various particle diameters will assume their different laws in the foul deposition areas on the surface of the insulators. On the top surface of the insulators, the number of particles at a diameter ranging from 0 to 20  $\mu\text{m}$  deposited on the surface of the insulators will be almost not influenced by the particle diameter while that at a diameter bigger than 20  $\mu\text{m}$  deposited on the surface will increase with an increase of the particle diameter and such an increment speed will gradually increase. On the bottom surface of the insulators, the number of particles at a diameter ranging from 0 to 20  $\mu\text{m}$  deposited on the surface of the insulators will decrease rapidly with the particle diameter but that at a diameter ranging from 20 to 50  $\mu\text{m}$  will go up and that at a diameter falling in a range from 50 to 100  $\mu\text{m}$  will decrease approaching to zero until the particle diameter reaches 100  $\mu\text{m}$ . **Key words:** insulator in transmission lines, foul deposition law, computational fluid dynamics, numerical simulation

350 MW 超临界四角切圆锅炉冷态空气动力场试验及仿真研究 = **Experimental and Simulation Study of the Cold-state Aerodynamic Field in a 350 MW Supercritical Tangentially-fired Boiler** [刊, 汉]/LI Zhi, WANG Yi-song (Shenyang Engineering College, Shenyang, China, Post Code: 110136), SONG Zhen-yu (Liaoning Electric Power Science Research Institute, Shenyang, China, Post Code: 110006) // Journal of Engineering for Thermal Energy & Power. -2016, 31(9). -113 ~ 118

In combination with the commissioning test of a 350 MW unit newly built in Huarun Electric Power Corporation Panjin thermal power plant, a simulation study was performed of the in-furnace aerodynamic characteristics of its boiler. By making use of the CFD software Fluent, with the parameters under the actual test conditions serving as the boundary conditions, a full spectrum of three-dimensional simulation of the internals inside the furnace was performed. The in-furnace speed field was obtained through calculation and the simulation and calculation results were contrasted and verified with the speed field actually measured in a section of the furnace. It has been found that the calculation results are basically in agreement with the values actually measured, therefore, the rationality of the simulation model and the simulation calculation method are verified. In the meantime, the calculation results show that the distribution of the primary air, secondary air and SOFA air in various sections is rational and the air flow intersection locations are good. In addition, the distribution of the turbulent flow intensity in the furnace relatively difficult to be actually measured can be obtained through the simulation calculation and the calculation results indicates that the distribution is rational, thus instrumental to the stable combustion inside the furnace and enhancing the com-

bustion efficiency. The research results have been applied in the commissioning work of the foregoing unit and the test run processes were simplified to a certain extent, thus making the work load lessened and an even richer and intuitive test results obtained. **Key words:** supercritical boiler, cold-state aerodynamic field test, numerical simulation

1 000 MW 超超临界机组给水泵汽轮机轴振增大原因分析与探讨 = **Analysis and Exploratory Study of the Causes for an Increase in the Vibration of the Shaft of a 1000 Ultra-supercritical Unit Feedwater Pump Steam Turbine** [刊, 汉] / ZHANG Xiu-feng, YONG Gan, HONG Ding-hua (Shenhua Guohua Xuzhou Power Generation Co. Ltd., Xuzhou, China, Post Code: 221166) // Journal of Engineering for Thermal Energy & Power. - 2016, 31(9). - 119 ~ 124

In the light of the problem that an increase in the vibration occurs to a feedwater pump steam turbine equipped in a 1000 MW ultra-supercritical unit during its normal operation, an analysis and exploratory study of the steam flow excitation, oil whipping, unbalance of the rotor in mass, rubbing between the rotor and stator and faults in the measurement elements etc. possibly causing an increase in the vibration were performed and an on-line inspection and measurement of the vibration of the feedwater pump steam turbine and the feedwater pump itself were also conducted through using test instruments, therefore providing an underlying basis for correctly judging the causes for an increase in the vibration. In combination with the actual conditions of the feedwater pump steam turbine and the feedwater pump undergone an increase in the vibration, phenomena of the faults and the installation of the equipment items on the spot, a further analysis and study of the causes were carried out. A series of prevention and improvement measures were proposed, including the inspection of the locations of the water drain valves in the feedwater pump steam turbine, thorough isolation of the high pressure steam sources and installation of vibration measuring devices on the pedestals of the bearings. Finally, the root causes of the problem were identified and the problem of an increase in the vibration of the feedwater pump steam turbine was successfully solved, thus offering reference for all persons concerned. **Key words:** 1000 MW ultra-supercritical unit, feedwater pump, steam turbine, bearing, vibration

ZGM 型中速磨煤机堵塞故障的分析与预测 = **Analysis and Prediction of the Jam of ZGM Medium-speed Coal Pulverizer** [刊, 汉] / Zhang Jing-zheng (Guangzhou Hengyun Enterprises Holdings Limited, Guangdong, Guangzhou, China, Post Code 510730) // Journal of Engineering for Thermal Energy & Power. - 2016, 31(9). - 125 ~ 128

**Abstract:** Based on ZGM-95N medium-speed Coal Pulverizer an example used by 330 MW unit, in the normal working condition and working condition of the clogging of the import export wind pressure, wind pressure and the two corresponding relations has carried on the comparative analysis. Results show that the ratio about export and import wind pressure between 0.4 ~ 0.6 is normal operation condition, the value is less than 0.4, the Coal Pulverizer beginning to clog. The ratio about export and import wind pressure is better than the export wind pressure and the difference of import wind pressure and export wind pressure about the jam condition of Coal Pulverizer. **Key words:** medium-speed Coal Pulverizer, Jam, Predict