文章编号: 1001-2060(2014)05-0503-06

板壳式换热器传热与流动特性研究

栾辉宝[1], 陈 斌[1], 郑伟业[1], 陶文铨[2]
(1. 中国船舶重工集团公司第七一一研究所 能源装备事业部, 上海 201108;
2. 西安交通大学 能源与动力工程学院 热流科学与工程教育部重点实验室, 陕西 西安 710049)

摘 要: 对圆形板片板壳式换热器内的传热流动特性进行了数值模拟及实验研究。采用SOLIDWORKS软件对单个流道进行了无简化、全尺度的实体建模, 采用ICEM软件进行了网格划分, 采用FLUENT软件进行求解, 并通过中试规模的水—水传热实验来验证模拟结果。实验样机采用与数值模拟几何结构完全相同的板片, 板片数目122片。通过对比发现, 在雷诺数 Re 为200-7 000的范围内, 数值模拟结果与实验结果误差在15%以内。模拟结果可以作为该类产品工业应用中设计选型及优化的依据。

关 键 词: 板式换热器; 数值模拟; 传热实验; FLUENT

中图分类号: TK123 **文献标识码**: A

引 言

板壳式换热器是板式换热器的一种。圆形板片板壳式换热器可以应用-200-900 ℃、0-20 MPa的场合, 越来越多的应用于工业生产当中[1]。板式换热器的传热流动研究成果可以应用于板壳式换热器。近年来板式换热器在传热流动性能方面得到了广泛研究, 研究主要有实验研究和数值模拟两种手段。板式换热器的单相流动实验已比较成熟, 有较多的文献可以参考。板式单相对流传热与流动特性主要跟几何结构(波纹角、特征长度)和流体物性有关。文献[2]详细总结了近年来关于不同板型的板式换热器单相换热的传热经验关联式。文献[3]总结了从1960年至2000年间板式换热器单相换热具有代表性的30余个传热经验关联式。

数值模拟研究不仅可以得到换热器的平均传热系数, 而且可以得到局部传热系数, 为板型结构优化提供借鉴。数值模拟离不开实验研究, 数值计算的准确性也需要通过与必要的实验结果做比较才能确认。近年来有大量的学者对板式换热器的传热流动进行了数值模拟研究[4-12]。文献[4]总结了计算流体力学软件(CFD)在换热器模拟中的应用。文献[5]对于波纹板片构成的流动通道, 从单个小单元(Unitary cell)进行模拟, 得到与实验吻合的结果。文献[6]通过研究不同形状的小单元, 得到较为优化的几何结构。文献[10]对复合波纹板式换热器流动传热进行了数值模拟, 分析了波纹几何参数对复合波纹板式换热器换热和流动特性的影响规律。文献[11]通过数值模拟研究了板式换热器内浮升力效应。文献[12]通过数值模拟研究了波纹板内的结垢情况。

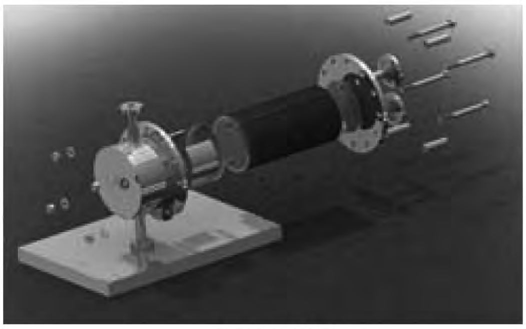
目前, 关于板式换热器的数值模拟多是基于简化的几何模型, 计算结果难以准确描述板式换热器内部流体完整的流动和换热特性。具体表现为: (1)截取流道的一部分作为计算区域, 采用充分发展流和周期性流动假设。(2)把波纹板片简化成平板, 与实际板片结构相差很大, 无法预测出真实湍流对强化换热和压降的影响。本研究对直径为440 mm的圆形板片通道进行全尺度实体建模, 采用高性能计算集群进行数值模拟。为了验证数值模拟结果的可靠性, 通过中试规模的试验平台, 对相同板型、板片数目122片的样机进行了实验测试。

1 板壳式换热器原理

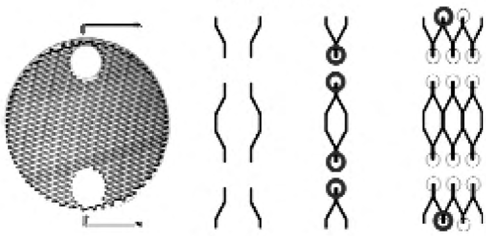
板壳式换热器是管壳式与板框式换热器的结合体, 如图1(a)所示。它具有传热效率高、末端温差小、耐高温高压、紧凑化以及重量轻等优点。板壳式换热器具有类似于板式换热器的波纹板片结构, 如图1(b)所示。板片外形为圆形, 片与片之间通过内孔焊接构成板片组, 板片组与板片组之间通过外圆

收稿日期: 2013-11-25; 修订日期: 2014-02-13
基金项目: 国家重点基础研究发展计划(973计划)基金资助项目(2013CB228304); 浙江省低碳脂肪胺工程技术研究中心开放课题资助项目(2012E10033)
作者简介: 栾辉宝(1983-), 男, 山西运城人, 中国船舶重工集团公司第七一一研究所工程师.

焊接，最终形成板芯。焊接板束类似于管束一样放置于壳体中。板壳式换热器板片之间不使用垫片，使得它可以用于普通板框式换热器无法应用的高温、高压场合。板壳式换热器的流动方式如图 2 所示，板程流体和壳程流体在板片的两侧纯逆向流动。



(a) 爆炸视图



(b) 芯体焊接示意图

图 1　圆形板片板壳式原理图

Fig. 1 Schematic diagram of round plate & plate-shell type heat exchangers

图 2　内部流动示意图

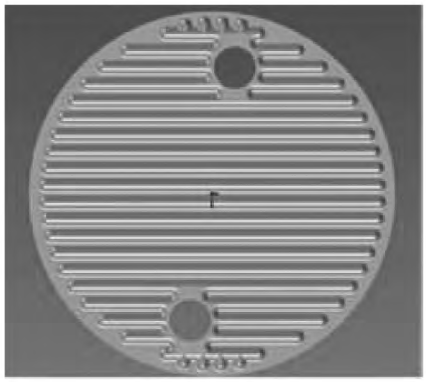
Fig. 2 Schematic diagram of the internal flow

2　数值模拟

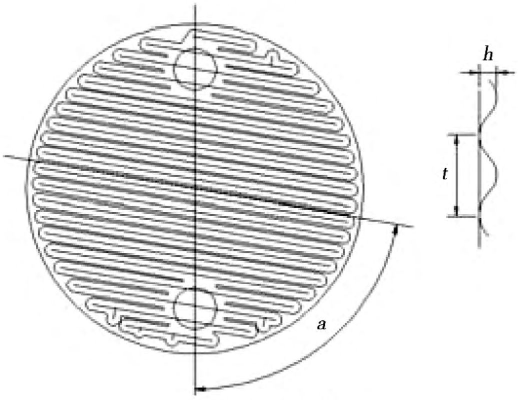
2.1　几何模型

采用 SOLIDWORKS 软件构建板片的三维实体模型，如图 3(a) 所示。板片特征尺寸如图 3(b) 所示，板片直径 440 mm，内孔直径 80 mm，两内孔中心距为 310 mm。波纹的倾斜角 $a = 80°$，波纹深度 $h = 4.0$ mm，波纹节距 $t = 9.0$ mm。两板之间构成的网状流道如图 4 所示。

根据板壳式换热器板程流道和壳程流道的特性，构建如图 5(a) 所示的板程流道几何模型和图 5(b) 所示的壳程流道几何模型。为了减少进出口位置对传热流动的影响，在板片进出口位置处均增加了直道延长段。



(a) 实体模型



(b) 板片尺寸图

图 3　板片几何模型

Fig. 3 Geometrical model for the plate

2.2　计算模型

采用 ICEM 软件直接对几何模型生成非结构化网格，网格类型为 Tetra/Mixed[13]。网格尺寸：进出口延长段直通道网格尺寸为 5 mm，板片内部网格尺

寸为 0.5 mm，直通道同板片网格之间有过渡区域。对于图 5(a) 所示的几何模型，网格数目 2720 万。对于图 5(b) 所示的流动模型，网格数目 2931 万。采用 ICEM 软件的 Smooth Mesh 功能，可使得 网格质量低于 0.3 的网格数目控制在 300 以下。

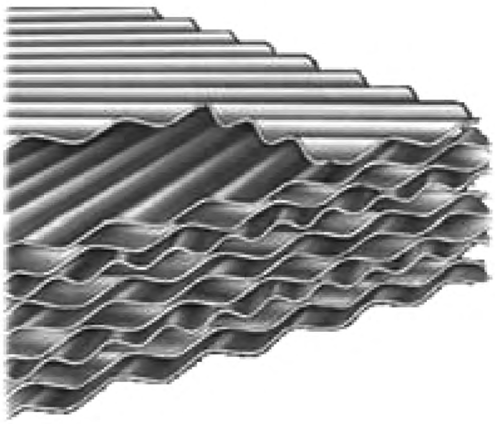
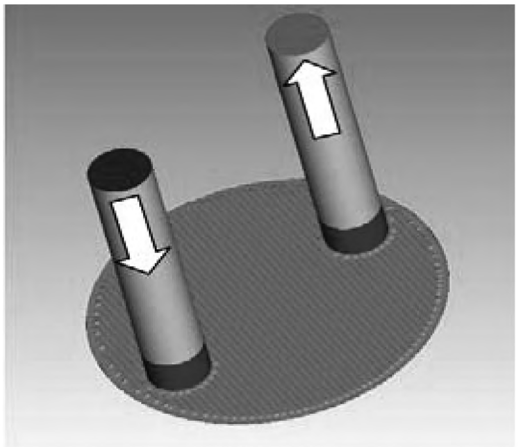


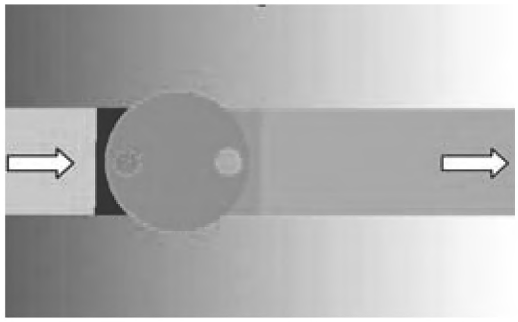
图 4　板片对构成的网状流道示意图

Fig. 4 Schematic diagram of the net-shaped flow passage formed by a pair of plates

通过计算发现，这些质量较差的网格，并未造成整个物理场计算的发散。以板程流体流动模拟为例，采用 FLUENT 计算的物理模型设置: (1) 流体为水，常物性; (2) 进口边界条件: 速度进口，流速范围为 0.003 – 0.15 m/s，温度 330 K; (3) 壁面边界条件: 壁面，温度 300 K; (4) 出口边界条件: 自由出流; (5) 湍流模型: Realizable $k - \varepsilon$ 模型; (6) 近壁面处理方法: 增强型壁面处理方法。



(a) 板程流道



(b) 壳程流道

图 5　CFD 几何模型

Fig. 5 CFD geometrical model

2.3　计算结果

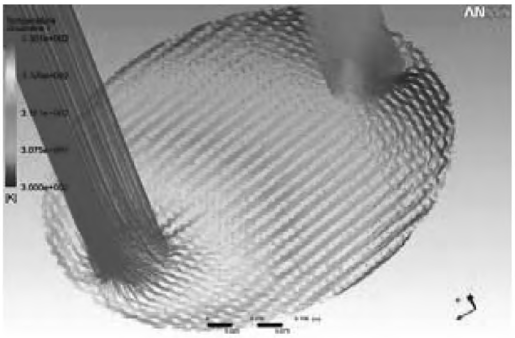
通过对换热器进口流速在 0.003 – 0.15 m/s 的范围内进行数值模拟，得到板程的传热结果如表 1 所示。从表中可以看出，随着流速的增加，流体出口温度增加，换热系数增加。

表 1　板程流动模拟结果

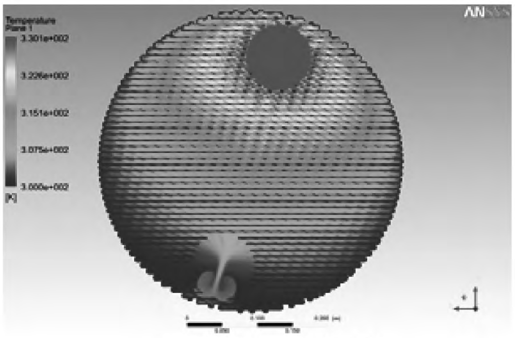
Tab. 1 Simulation results of the flow inside the plate side

进口温度 t_{h1} /K	壁温 t_w /K	进口流速 v_{in} / m · s^{-1}	体积流量 q_v /m^3 · h^{-1}	换热面积 A /m^2	换热量 Q /W	出口温度 t_{h2} /K	对数平均温差 LMTD/K	换热系数 h /W · (m^2 · K) $^{-1}$
330	300	0.003	0.06	0.320	1 915	300.0	4.1	1 459
330	300	0.006	0.12	0.320	3 891	300.1	5.2	2 319
330	300	0.010	0.18	0.320	6 191	300.3	6.3	3 065
330	300	0.013	0.23	0.320	7 688	300.5	7.1	3 397
330	300	0.020	0.36	0.320	11 983	301.2	8.9	4 185
330	300	0.025	0.45	0.320	14 713	301.7	9.9	4 663
330	300	0.050	0.90	0.320	26 610	304.4	13.4	6 215
330	300	0.075	1.36	0.320	37 136	306.2	15.1	7 687
330	300	0.100	1.81	0.320	46 903	307.5	16.2	9 030
330	300	0.150	2.71	0.320	65 430	309	17.4	11 722

图 6 给出了板程流动状态示意图。从图 6(a) 所示的三维流线图可以看出，在流速为 0.05 m/s 时，流体自板片内孔流入板程，并围绕内孔扩散开来。两内孔中心的连线处，流体的流动距离最短，而越靠近板片边缘流体的流动距离越长。从图 6(b) 板与板之间中心面的温度场分布可以看出，从进口处进入的流体沿孔四周扩散开来，在流动的前端基本上具有相同的温度梯度，然而随着流动距离的加长，离进口内孔较远的流股的温度梯度会慢慢下降。



(a) 三维流线图



(b) 两板中心面温度云图

图 6　板程流动示意图

Fig. 6 Schematic diagram of the flow in the plate side

图 7 给出了流速为 0.05 m/s 时，板程流动的速度分布图。从图 7 可以看出，进口内孔周边的流体流线较为均匀，而出口内孔处由于多股流体的汇聚，流动状态较为复杂。

同理，采用相同的物理模型来模拟壳程的流动。模拟结果如图 8 和图 9 所示。图 8(a) 给出了壳程进口流速为 0.185 m/s 时的流线图，可以看出，内孔的存在，对温度场分布的影响较为明显。图 8(b) 给出压力场标示的流线图，可以看出，压力场的分布受内孔的影响相对较小。图 8(c) 给出了壳程速度场标示的流线图。

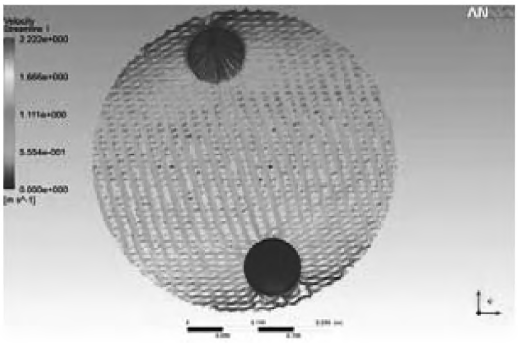
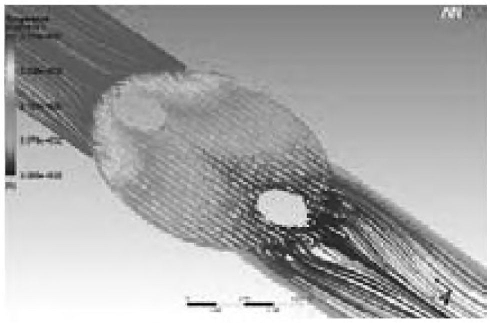
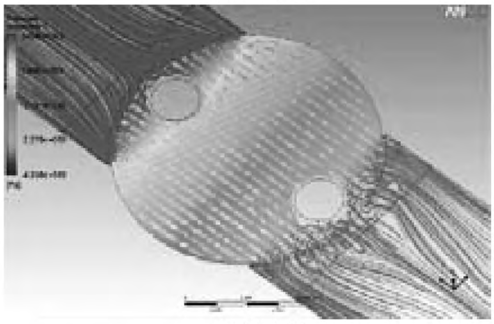


图 7　板程流动示意图

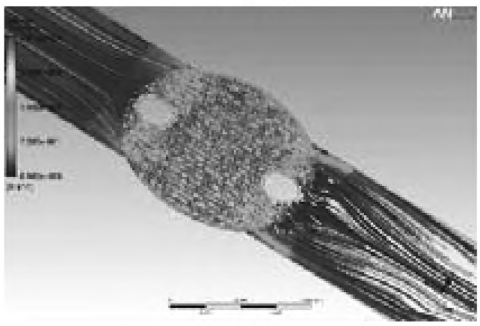
Fig. 7 Schematic diagram of the flow in the plate side (speed value indicated by the colors)



(a) 温度场



(b) 压力场



(c) 速度场

图 8　壳程三维流线图

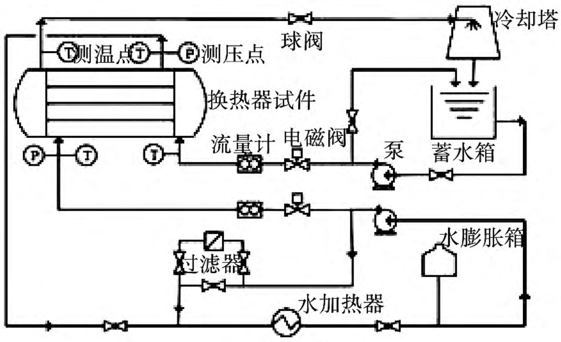
Fig. 8 3D streamline diagram of the shell side

3　实验测试

为了对数值模拟结果的可靠性进行验证。在中试规模的试验台上对由上述板型构成的换热器样机进行了传热性能测试,工质为水。样机板片数目为 122 片,其中板程通道 61 个,壳程通道 60 个。试验台原理如图 9(a) 所示,实测照片如图 9(b) 所示。水最大加热功率 800 kW; 热水流量范围 15 – 100 m^3/h,进口温度 50 – 80 ℃; 冷却水流量范围 20 – 150 m^3/h,进口温度 15 – 20 ℃。热水走板程,冷水走壳程。

采用 Wilson 热阻分离方法,从实验结果中分离得出板程的对流换热系数,并将其同数值模拟结果进行性对比,其中传热因子 j 定义为:

$$j = NuPr^{-\frac{1}{3}}\left(\frac{\mu}{\mu_w}\right)^{-0.17} \tag{1}$$



(a) 原理图

(b) 实验照片

图 9　中试规模的试验台

Fig. 9 Pilot-scale test rig

如图 10 所示,通过对比可以发现,在 Re 数 200 – 7 000 的范围内,数值模拟误差与实验的误差在 15% 以内。产生误差的原因: (1) 数值模拟研究了单个通道,而试验研究中板程流道有 60 个通道。流道间的流动分配不均匀性会产生影响。(2) 数值模拟壁面边界条件是恒定壁温,而试验中壁面温度是不均匀的。(3) 由于板片结构复杂,尺寸较大,数值模拟在复杂流道中的网格质量相对较差,一定程度上影响了模拟精度。(4) 试验测量精度也会引起一定误差。

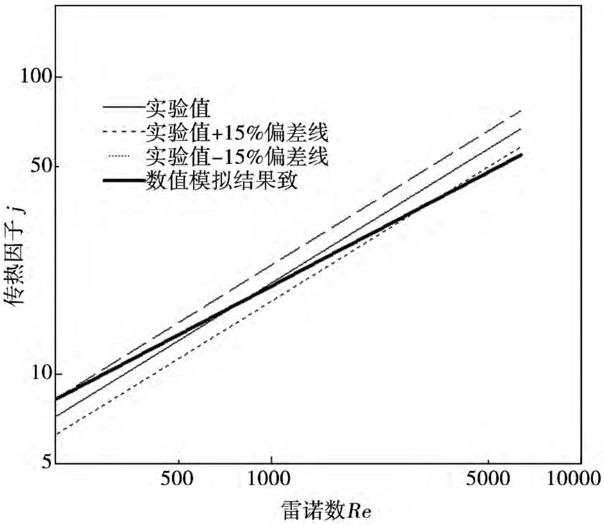


图 10　数值模拟值同实验值对比(15%误差棒)

Fig. 10 Contrast of the numerical simulation values with the test ones (15% error bar)

4　结　论

(1) 数值模拟过程并未对板片进行简化,采用 SOLIDWORKS 软件进行全尺度建模,在 ICEM 软件中进行网格划分,一次性生成非结构化网格。减少了传统的几何区域分区的步骤,大大节省了划分网格的时间。在 Re 数 200 – 7 000 的范围内,数值模拟结果同实验结果误差在 15% 以内,具有一定的工程指导意义。

(2) 在 $Re < 200$ 时,实验结果与 CFD 计算结果误差大于 15%。原因为实验测试中,工质为水,在较小的 Re 下,进入换热器的流量过小,测量误差较大。下一步需开展以油为工质的换热实验,可以保证在较小的 Re 下,具有较大的流量,提高测量精度。

(3) 在 $Re > 7000$ 时,实验结果要与 CFD 计算结果误差大于 15%。分析原因可能在于网格过粗。下一步将进一步加密网格,以期精准捕捉近避免处流体的湍流特性,提高计算精度。

参考文献:

[1] 栾辉宝,陶文铨,朱国庆,等.全焊接板式换热器发展综述[J].中国科学 E 辑:技术科学,2013,43(9):1020-1033.
LUAN Hui-bao,TAO Wen-quan,ZHU Guo-qing,et al. Survey of the advances in all-welding plate type heat exchangers[J]. China Science E Edition: Technical Science,2013,43(9):1020-1033.

[2] LI Q,FLAMANT G,YUAN X,et al. Compact Heat Exchanger: a review and future applications for a new generation of high temperature solar receivers[J]. Renewable & Sustainable Energy Reviews,2011(15):4855-4875.

[3] AYUB Z H. Plate heat exchanger literature survey and new heat transfer and pressure drop correlations for refrigerant evaporators[J]. Heat Transfer Engineering,2003,24(5):3-16.

[4] BHUTTA Aslam M M,HAYAT N,BASHIR M H,et al. CFD application in various heat exchanger design: a review[J]. Applied Thermal Engineering,2012(32):1-12.

[5] CIOFALO M,STASIEK J,COLLINS M W. Investigation of flow and heat transfer in corrugated passages-II. Numerical simulations[J]. International Journal of Heat and Mass Transfer,1996,39(1):165-192.

[6] ZHANG L,CHE D F. Influence of corrugation profile on the thermal hydraulic performance of cross-corrugated plates[J]. Numerical Heat Transfer,Part A,2011,59(4):267-296.

[7] KANARIS A G,MOUZA A A,PARAS S V. Optimal design of a plate heat exchanger with undulated surfaces[J]. International Journal of Thermal Science,2009,48:1184-1195.

[8] 蔡 毅.板式换热器性能的数值模拟及实验研究[M].北京:北京化工大学硕士学位论文,2008.
CAI Yi. Numerical simulation and experimental research of the performance of plate type heat exchangers[C]. M. D. dissertation,Beijing University of Chemical Technology,Beijing,2008.

[9] 邱小亮.人字形板片结构参数对板式换热器传热与流阻特性影响研究[M].广州:华南理工大学硕士学位论文,2013.
QIU Xiao-liang. Study of the influence of the parameters of the chevron plate structure on the heat transfer and flow resistance charactreristics of a plate type heat exchanger[M]. M. D. dissertation,South China University of Technology,Guangzhou,2013.

[10] 栾志坚.新型板式换热器内高粘性流体传热与流动特性研究[D].山东:山东大学博士学位论文,2009.
LUAN Zhi-jian. Research of the heat transfer and flow characteristics of the high viscid fluid in new type plate type heat exchangers[D]. Ph. D. dissertation,Shandong University,Shandong,2009.

[11] GHERASIM I,TAWS M,GALANIS N,NGUYEN T. Numerical and experimental investigation of buoyancy effects in a plate heat exchanger[J]. Applied Thermal Engineering,2013(51):347-363.

[12] LI W,LI H X,LI G Q,YAO S C,Numerical and experimental analysis of composite fouling in corrugated plate heat exchanger[J]. International Journal of Heat and Mass Transfer. 2013(63):351-360.

[13] 纪兵兵,陈金瓶. ANSYS ICEM CFD 网格划分技术实例详解[M].北京:中国水利水电出版社,2012.
JI Bing-bing,CHEN Jin-ping. Case-by-case explanations of the Ansys ICEM CFD Grid-division technology[M]. Beijing: China Water Conservation and Hydropower Press,2012.

(丛 敏 编辑)

新技术、新工艺

额定功率 550 MW 联合循环和抛物面太阳能发电场

据《Gas Turbine World》2013 年 11-12 月刊报道,苏丹电力公司正在开发 Duba(杜巴)1 独立电站项目,该项目将坐落在苏丹阿拉伯西北海岸塔布克区域杜巴以北 50 km 处。

混合式电站将设计一个额定功率铭牌 550 MW 天然气联合循环和集成 20-30 MW 浓缩燃料太阳能发电设备。燃气轮机将主要用天然气和浓缩燃料运行,并利用阿拉伯超轻蒸馏油作为备用燃料。

沙特电力公司将签订购买电力的长期协议。按照该合同,苏特电力公司将购买来自该电站的全部电力并且将供应矿物燃料和该电站用地。

(吉桂明 摘译)

to the pretighening force designed was obtained, thus establishing a finite element model for tie-rod rotors of gas turbines with the contact effect of the wheel disks being taken into account. In combination with the traditional finite element model, the critical speed calculation results were compared. It has been found that the first two-order critical speeds of the two models are basically equivalent and both relative errors are lower than 1%. Both relative errors of the three-order critical speeds are relatively big, being 1.66%. The improved finite element model can more truly reflect the contact status of the wheel disks on the tie-rod rotor of a gas turbine. **Key Words**: contact model, spring unit, tie-rod rotor, joint surface rigidity, critical speed

吸收式湿热空气余热回收系统的模拟分析 = Analysis of the Simulation of an Absorption Type Humidified Hot Air Waste Heat Recovery System [刊，汉] WU Yong-ping, ZHENG Jiao, LI Jian-xin (Energy Source and Environment Engineering Research Institute, Ningbo College of Science and Technology, Zhejiang University, Ningbo, China, Post Code: 315010), CHEN Guang-ming (Refrigeration and Cryogenics Research Institute, Zhejiang University, Hangzhou, China, Post Code: 310027) //Journal of Engineering for Thermal Energy & Power. -2014, 29(5). -498-502

Through analyzing the typical drying thermodynamic process, the authors put forward a scheme for a humidified hot air waste heat recovery system based on the principles for dehumidification by solutions and absorption type heat pumps. The system in question produced saturated steam at a temperature of around 120℃ when it was jointly driven by the waste heat from the exhaust gases of a typical dryer and solar energy-produced hot water, partially substituting the drying heat source to realize the energy saving. Through establishing a thermodynamic model, the scheme for the system was verified with the influence of the evaporator temperature and generator temperature on various performance indexes of the system being discussed. The calculation results show that with the lithium bromide solution being chosen as an absorbent, under the condition of the evaporation temperature being 80 ℃ and the working temperature of the generator being 63 ℃, the energy saving of the dryer at an efficiency of 33.45% can be realized with the COP (coefficient of performance) of the system being 0.43, among which 79% of the heat quantity serving as the driving force comes from the humidified hot exhaust gas and the total sprinkling quantity of the absorption solution is 6.33 times of the mass of the exhaust gas. **Key Words**: waste heat recovery, absorption type, thermodynamic model, dryer

板壳式换热器传热与流动特性研究 = Study of the Heat Transfer and Flow Characteristics of a Plate and Shell Heat Exchanger [刊，汉] LUAN Hui-bao, CHEN Bin, ZHENG Wei-ye (Energy Source and Equipment Cause Department, CSIC No. 711 Research Institute, Shanghai, China, Post Code: 201108), TAO Wen-quan (Education

Ministry Key Laboratory on Thermo-fluid Sciences and Engineering, College of Energy Source and Power Engineering, Xi´an Jiaotong University, Xi´an, China, Post Code: 710049) //Journal of Engineering for Thermal Energy & Power. -2014, 29(5). -503-508

A numerical simulation and experimental study were performed of the heat transfer and heat exchange characteristics of a round plate and shell heat exchanger. By using the software Solidworks, a full scale real-entity model for a single flow passage was established during the numerical simulation without any simplification, dividing the grid by employing the software ICEM and seeking solutions by adopting the software Fluent. In addition, the simulation results were verified through a pilot-scale water-water heat transfer test. The prototype machine for the test used the plates fully identical to the geometrical structure during the numerical simulation with the number of the plates being 122. Through a comparison, it has been found that when the Reynolds number is in a range from 200 to 7 000, the error between the numerical simulation results and the test ones is within 15%. The simulation results can be used as an underlying basis for type selection and optimization during design of the products of the same kind in industrial applications. **Key Words:** plate heat exchanger, numerical simulation, heat exchange test, Fluent

螺旋槽管束管外对流换热特性的数值模拟 = Numerical Simulation of the Convection-based Heat Exchange Characteristics Outside Spirally Grooved Tube Bundles [刊, 汉] WANG Ying-hui, SUN Ning (College of Energy Source and Power Engineering, Jiangsu University, Zhenjiang, China, Post Code: 212013), GUI Ke-ting (College of Energy Source and Environment, Southeast University, Nanjing, China, Post Code: 210096) //Journal of Engineering for Thermal Energy & Power. -2014, 29(5). -509-514

In the light of the problem of the flow and heat transfer outside the spirally grooved tube bundles in an in-line arrangement swept across by flue gases, by using the CFD technology and through changing the structural parameters such as the lateral and longitudinal spacing, pitch and groove depth of the spirally grooved tube bundles in an in-line arrangement etc., the influence of the multiple geometrical parameters on the heat transfer characteristics of the flow outside the spirally grooved tubes was analyzed with the theoretical cause of the heat transfer enhancement and rational structural parameters being determined. It has been found that the heat transfer characteristic number *Nu* outside spirally grooved tubes is 7% to 20.6% greater than that outside bare tubes. With an increase of the lateral spacing, the heat transfer characteristic number *Nu* outside tubes will decrease and the flow resistance of flue gases will decrease accordingly. An increase of the longitudinal spacing will make both the heat transfer characteristic number outside tubes and the flow resistance of flue gases increase. To increase the pitch and decrease the groove depth can both enhance the heat exchange, however, the flow resistance of flue gases will also increase. With all the