DOI:10.16082/j.cnki.issn.1001-4578.2005.04.004

◀专题研究▶

折流板换热器的数值模拟及场协同分析

严良文 王志文
（华东理工大学化工机械研究所）

摘要 在PHOENICS—3.5.1程序的基础上，采用多孔介质模型，以及体积多孔度、表面渗透度和各向异性的分布阻力来处理换热器内的管束；用分布热源考虑管侧流体对壳侧流体的影响，对单弓形折流板换热器的壳程流场和温度场做了数值模拟。结果表明：（1）采用换热器三维流动计算模型和k—ε湍流模型能较好地模拟折流板换热器内的流场分布；（2）通过数值模拟可直观地了解换热器内的流动状态，确定换热器的高、低速区和旋涡区。低速区和旋涡区换热效果差，管子易结垢，而高速区换热效率高，但管子易被冲蚀，且阻力较大，应予改进；（3）换热器中间段的场协同性较好，出入口处的场协同性较差，应尽量减小其结构尺寸，或采用导流筒式结构。

关键词 折流板换热器 数值模拟 场协同理论

引 言

自从过增元教授提出场协同理论[1,2]以来，国内许多学者对已有的一些强化传热结构进行了场协同分析，通过数值模拟的方法得出场量分布，再结合场协同理论解释了一些结构强化传热的原因，对于强化传热结构的改进和开发起到了促进作用。

笔者在大型通用商用软件PHOENICS—3.5.1程序的基础上，采用多孔介质模型，用体积多孔度、表面渗透度和各向异性的分布阻力来处理换热器内的管束，用分布热源考虑管侧流体对壳侧流体的影响，并选用工程上广泛采用的湍流模型，对单弓形折流板换热器的壳程流场和温度场进行了模拟，并应用场协同理论对其进行了分析，目的在于探索换热器壳侧的场协同效果，寻求换热器壳侧强化传热的理论基础。

多孔介质模型下的守恒方程

多孔介质模型[3,4]就是将流体、固体划入同一个控制体，通过对守恒方程及差分方法的修改来表现固体的影响。用体积多孔度即流体体积与整个控制体体积的比值表示固体构件对控制体内流体体积的影响，用表面渗透度即流体表面与控制体表面的比值表示固体构件对控制体表面作用力的影响。而固体构件带来的动量及能量交换的影响在方程中引入分布阻力和分布热源表示，这样守恒方程[5]表示如下。

1. 质量与动量守恒方程

三维流体的质量守恒方程为

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho U)}{\partial x}+\frac{\partial(\rho V)}{\partial y}+\frac{\partial(\rho W)}{\partial z}=0 \quad (1)$$

三维流体的动量守恒方程为

$$\rho\frac{\partial U}{\partial t}+\rho U\frac{\partial U}{\partial x}+\rho V\frac{\partial U}{\partial y}+\rho W\frac{\partial U}{\partial z}=-\frac{\partial p}{\partial x}+\mu_{eff}\left(\frac{\partial^2 U}{\partial x^2}+\frac{\partial^2 U}{\partial y^2}+\frac{\partial^2 U}{\partial z^2}\right)-f_x \quad (2)$$

$$\rho\frac{\partial V}{\partial t}+\rho U\frac{\partial V}{\partial x}+\rho V\frac{\partial V}{\partial y}+\rho W\frac{\partial V}{\partial z}=-\frac{\partial p}{\partial y}+\mu_{eff}\left(\frac{\partial^2 V}{\partial x^2}+\frac{\partial^2 V}{\partial y^2}+\frac{\partial^2 V}{\partial z^2}\right)-f_y \quad (3)$$

$$\rho\frac{\partial W}{\partial t}+\rho U\frac{\partial W}{\partial x}+\rho V\frac{\partial W}{\partial y}+\rho W\frac{\partial W}{\partial z}=-\frac{\partial p}{\partial z}+\mu_{eff}\left(\frac{\partial^2 W}{\partial x^2}+\frac{\partial^2 W}{\partial y^2}+\frac{\partial^2 W}{\partial z^2}\right)-f_z \quad (4)$$

式中 U、V和W——壳侧流体的平均速度分量；
t——时间；
p——流体的压力；
ρ——流体的密度；
f_x、f_y、f_z——分布阻力系数；

μ_{eff}——等效动力粘度，Pa·s。

若压力损失系数记为 ξ_x、ξ_y、ξ_z，则有

$$f_x = \xi_x \rho U U_R \tag{5}$$

$$f_y = \xi_y \rho V U_R \tag{6}$$

$$f_z = \xi_z \rho W^2 \tag{7}$$

其中 $U_R = \sqrt{U^2 + V^2}$，压力损失系数 ξ_x、ξ_y、ξ_z 分别表示为

$$\xi_x = 2\left(\frac{C_x}{e}\right)\left(\frac{\rho\beta}{e - D_h}\right)^2\left(\frac{1-\beta}{1-\beta_0}\right) \tag{8}$$

$$\xi_y = 2\left(\frac{C_y}{e}\right)\left(\frac{\rho\beta}{e - D_h}\right)^2\left(\frac{1-\beta}{1-\beta_0}\right) \tag{9}$$

$$\xi_z = \left(\frac{2C_z}{e}\right)\left(\frac{1-\beta}{1-\beta_0}\right) \tag{10}$$

式中　D_h——壳侧流体的水力直径；

e——管间距；

β 和 β_0——壳侧局部多孔度和整体多孔度；

C_x、C_y、C_z——摩擦因子。

摩擦因子可由下面的式子求出。

对于流体横掠管束：

$$C_x = 0.619 Re_x^{-0.198} \quad (Re_x < 8\,000) \tag{11}$$

$$C_x = 1.156 Re_x^{-0.2647} \quad (8\,000 \leqslant Re_x < 2\times 10^5) \tag{12}$$

$$C_y = 0.619 Re_x^{-0.198} \quad (Re_y < 8\,000) \tag{13}$$

$$C_y = 1.156 Re_x^{-0.2647} \quad (8\,000 \leqslant Re_y < 2\times 10^5) \tag{14}$$

对于流体纵掠管束：

$$C_z = 31 / Re_z \quad (Re_x < 2\,250) \tag{15}$$

$$C_z = 0.131 Re_z^{-0.294} \quad (2\,250 \leqslant Re_x < 2.5\times 10^4) \tag{16}$$

$$C_z = 0.066 Re_z^{-0.227} \quad (Re_x \geqslant 2.5\times 10^4) \tag{17}$$

其中 Re 准数由各速度分量、管子外径及动力粘度算出。

2. 能量守恒方程

对于管侧，可以认为是一维流动，因此能量方程中只有对流项，即

$$\rho_T C_{VT}\frac{\partial T_T}{\partial t} + C_{PT} g_T \frac{\partial T_T}{\partial z} = \alpha_T (T_W - T_T) \tag{18}$$

式中　ρ——密度，kg/m^3；

C_V——定容比热，kJ/(kg·℃)；

T——温度，℃；

C_P——定压比热，kJ/(kg·℃)；

g——质量流速，kg/(m²·s)；

α——给热系数，W/(m²·℃)。

下角标 T 表示管侧，W 表示管壁。

对于壳侧

$$\rho C_{ps}\left(\frac{\partial T_S}{\partial t} + U\frac{\partial T_S}{\partial x} + V\frac{\partial T_S}{\partial y} + W\frac{\partial T_S}{\partial z}\right) - \left(\frac{\partial p}{\partial t} + U\frac{\partial p}{\partial x} + V\frac{\partial p}{\partial y} + W\frac{\partial p}{\partial z}\right) = \lambda_s\left(\frac{\partial^2 T_S}{\partial x^2} + \frac{\partial^2 T_S}{\partial y^2} + \frac{\partial^2 T_S}{\partial z^2}\right) + \alpha_S (T_W - T_S) + \mu_{eff}\left\{2\left[\left(\frac{\partial U}{\partial x}\right)^2 + \left(\frac{\partial V}{\partial y}\right)^2 + \left(\frac{\partial W}{\partial z}\right)^2\right] + \left[\frac{\partial W}{\partial y} + \frac{\partial V}{\partial z}\right]^2 + \left[\frac{\partial W}{\partial x} + \frac{\partial U}{\partial z}\right]^2 + \left[\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right]^2\right\} \tag{19}$$

式中　λ——导热系数，W/(m·℃)。

下角标 S 表示壳侧。

管壁温度可由下式得到

$$\rho_W C_{VW}\frac{\partial T_W}{\partial t} = \alpha_S (T_S - T_W) + \alpha_T (T_T - T_W) \tag{20}$$

壳侧的给热系数为

$$\alpha_S = 0.35 \lambda_S Re^{0.6} Pr^{0.36} (a/b)^{0.12} / D_o \tag{21}$$

式中　a——横向管间距，m；

b——纵向管间距，m；

D_o——换热管外径，m。

管侧的给热系数为

$$\alpha_T = 0.023 \lambda_T Re^{0.8} Pr^{0.33} / D_i \tag{22}$$

式中　D_i——换热管内径，m。

数值模拟研究

1. 湍流模型

当前使用的湍流模型很多，笔者在此选择在工程上广泛采用的标准的 $k-\varepsilon$ 湍流模型，相应的模型常量为 $\sigma_k = 0.75$、$\sigma_\varepsilon = 1.3$、$\sigma_\mu = 0.09$、$C_1 = 1.44$、$C_2 = 1.92$、$\sigma_T = 1.0$。

2. 模拟换热器的结构参数

数值模拟的对象是单弓形折流板换热器，其结构参数如下。

壳体内径：257 mm；

换热管尺寸：ϕ20 mm×1.65 mm×6 000 mm；

管间距：26 mm；

换热管数目：57根，采用正三角形排列；

折流板切割高度：39%；

折流板数目：11块；

折流板间距：500 mm。

3. 计算方法

数值计算中，压力与速度的偶合计算采用 SIMPLE 方法求解方程，整场求解控制方程，由于各变量之间的强烈非线性关系，迭代求解选用亚松

弛。方程迭代过程的初始值均为零。数值计算中的收敛准则是：整个求解区域节点的能量源项的相对残差最大绝对值小于 10^{-7}，其余的相对残差最大绝对值均小于 10^{-4}。壳体壁面取无滑移、绝热边界条件。

4. 计算结果及分析

对换热器的 5种实验工况进行了计算，其壳侧进口水的体积流量为 60、70、80、90、100 m^3/h。表 1中给出壳侧进出口总压降的计算值和实验值，两者吻合较好。

表 1 压降的计算值与实验值比较 kPa

工况	1	2	3	4	5
计算值	21.7	28.8	31.5	32.6	33.1
实验值	23.0	29.0	32.0	33.0	34.0

得到工况 5下的换热器水平中心线截面上的温度场、压力场、速度场和角度场（温度梯度与速度向量的夹角）等值图。从温度等值图可以看出，流体温度从入口到出口，温度逐渐升高，中间段温度变化比进出口段变化大，说明换热器折流板之间部分的传热效果优于其进出口段。从压力等值图可知，流体的压力由进口到出口逐渐下降，其中进出口段的压力梯度较大，说明进出口段是压力损失的主要区域。从速度等值图可以看出，流体在进口和出口处流速较大，在折流板所在处的窗口区较管束区大，而在折流板的两侧出现了低速区。从角度场等值图可以得出，换热器出入口处的角度场平均值分别为 80.1°和 83.2°，场的协同效果较差，因此传热效率必然很低。但在相邻两折流板之间，角度场平均夹角变小，为 75.6°，场的协同效果较好，传热效率也较高，与温度等值图中出现的温度场变化规律相符合。

在工况 5下得到换热器水平中心线截面上的进出口区域的速度矢量场，从进口段矢量场可以看出，在换热器入口右上侧速度矢量相对较小，而在受折流板影响的窗口区速度矢量较大，流向平行于管束，但在相邻两折流板之间，流动方向垂直于管束。同时也可以看到，在折流板的背面上，速度矢量很小，并且出现涡旋。从出口段矢量场可以看出，在换热器出口左下侧速度矢量相对较小，并且在有折流板处也出现涡旋。

结 论

（1）采用多孔介质模型建立的换热器三维流动计算模型和 k—ε湍流模型能够较好地模拟折流板换热器内的场量分布。

（2）通过数值模拟可以直观地了解换热器内的流动状态，从而确定换热器的流动低速区、高速区和旋涡区。在低速区和旋涡区换热效果较差，管子容易结垢，是换热器设计时必须改进的区域；而高速区换热效率高，但管子易被冲蚀，且是阻力较大的区域。因此，通过数值模拟可以得到关于换热器壳侧流动的细观状态，为优化换热器的设计提供有力依据。

（3）角度场的分析表明，换热器中间段的场协同性较好，而出入口处场协同性较差，因此设计时应尽量减小其相对尺寸，或采用如导流筒式结构，以改善其性能。

参 考 文 献

1 Guo Z Y. Mechanism and control of convective heat transfer — Coordination of velocity and heat flow fields. Chinese Science Bulletin, 2001, 46 (7): 596～599

2 Guo Z Y, Li D Y, Wang B X. A novel concept for convective heat transfer enhancement. International Journal of Heat And Mass Transfer, 1998, 41 (14): 2221～2225

3 林瑞泰．多孔介质传热传质引论．北京：科学出版社，1995. 39～47

4 Bear J著，李竞生，陈崇希 译．多孔介质流体动力学．北京：中国建筑工业出版社，1983. 9～19

5 Zarko Stevanovic, Gradimir Ilic. Design of shell—and—tube heat exchangers by using CFD technique— part one: Thermo—Hydraulic Calculation. Mechanical Engineering, 2001, 1 (8): 1091～1105

作者简介：严良文，工程师，生于 1967年，现攻读博士学位，从事换热器强化传热和数值模拟方面的研究工作。地址：（200237）上海市华东理工大学 402信箱。电话：（021）64253055。

收稿日期：2004—10—28
（本文编辑 王志权）

ABSIRACTS OF SELECTED ARTICLES

Yang Li(College of Power Engineering University of Shanghai for Science and Technology Shanghai), Chen Kangmin Research on the distribution of the teeth of PDC bits CPM, 2005 33(4): 1～3

The distribution of the teeth of PDC bits is the key factor influencing the performance of the bit And the key point of teeth distribution design lies in specifying the structural angles and the spatial locations of every tooth which have close relation to the crown profile of the bit According to the tooth— distributing principle this article discusses both uniform and nonuniform distribution of tooth along the crown profile of the bit and gives corresponding practical tooth— distributing methods In the end computer aided design of the tooth distribution is introduced Key words: PDC bit cutters tooth— distributing design crown profile

Zhu Huangang(School of Mechanical Engineering Yangtze University Jingzhou City Hubei Province), Zhang Xiaodong Bi Licai et al The simulation of axial distortion of four— fulcrum thrust ball bearing pack of turbodrill CPM, 2005 33 (4): 4～5 9

Since the adjustment of axial clearance of assembled turbodrill has great influence on its performance and service life and the four— fulcrum thrust ball bearing pack has direct relation the axial clearance a simulation of the axial distortion of the bearing is made by means of ANSYS software By taking the bearing pack on φ 165 mm turbodrill as the subject investigated a solid model is set up then a computation is made based on VC++ program and thereby the relation curve of the axial force and axial distortion of the bearing pack is obtained The result is meaningful to the improvement of the design of the four— fulcrum thrust ball bearing pack of turbodrill

Key words: turbodrill four— fulcrum thrust ball bearing pack axial distortion ANSYS software computer simulation

Xing Zhixiang(Chinese Peoples Forced Police Army Academy Langfang City Hebei Province), Jiang Juncheng Ge Xikun Simulating analysis of factors effecting thermal response of LPG tanks CPM, 2005 33(4): 6～9

To make clear the rule of thermal response of LPG (liquefied petroleum gas) tanks exposed to fire this article introduces the mechanism of the thermal response of the LPG tanks exposed to fire and makes a simulating analysis of the thermal response and its effecting factors The analysis of the effecting factors of the thermal response shows that the safety valve heat insulating layer LPG charging rate flame temperature and dimension of the tank have obvious effect on the pressure and temperature response of the tank therefore responding measures are put forward to slow down the rise of the pressure and temperature of the tank exposed to fire

Key words: liquefied petroleum gas storage tank fire thermal response simulating analysis

Duan Shaoli(Mechanical Engineering School of Shandong University Jinan City Shandong Province), Tang Weixiao Dynamic behavior of third stage cyclone separator in heavy oil catalytic cracking unit CPM, 2005 33(4): 10～12

Damage and distortion often occurs in the upright multitube cyclone separators in heavy oil catalytic cracking unit Based on an analysis of its static stress at operating pressure in consideration of the action of the dynamic loads the dynamic and static behaviors of the separator are analyzed by means of ANSYS software Thereby the fore five order mode natural frequency and vibration modes are recognized The fourth order vibration mode is found similar to the actual damage condition And the causes of the damage are analyzed and suggestions for improvement are put forward

Key words: dynamic analysis finite element analysis cyclone separator natural frequency vibration mode

Yan Liangwen(East China University of Chemical and Technology Shanghai), Wang Zhiwen 3—D numerical simulation of fluid flow and heat transfer in heat exchanger with segmental baffles and field synergy principle analysis CPM, 2005 33(4): 13～15

Based on the CFD software PHOENICS—3.5.1 a 3—D thermal hydraulic model is developed for the analysis of fluid flow and heat transfer in heat exchanger with segmental baffles The numerical model uses the distributed resistance model and distributed heat source model along with the concept of porosity surface permeability to account for the presence of tubes The analytical model is validated by comparison to computed pressure drop Meanwhile the results are analyzed according to the field synergy principle

PPAGESS : heat exchanger with segmental baffles numerical simulation field synergy principle

Wei Jide(1. Petroleum Engineering Department of Daqing