Chemical Engineering Journal 286 (2016) 436–446

Contents lists available at ScienceDirect

Chemical Engineering Journal

journal homepage: www.elsevier.com/locate/cej

Coupled simulation of convection section with dual stage steam feed mixing of an industrial ethylene cracking furnace

Guihua Hu, Benfeng Yuan, Liang Zhang, Jinlong Li, Wenlin Du, Feng Qian *

Key Laboratory of Advanced Control and Optimization for Chemical Processes of Ministry of Education, East China University of Science and Technology, Shanghai 200237, China

HIGHLIGHTS

- A complete coupled simulation method of convection section is proposed.
- Velocity fields are not uniform along width direction due to asymmetrical structure.
- Recirculation zones cause a longer residence time of flue gas and local overheating.
- Process gas and tube skin temperature and heat flux have axial and radial profiles.
- Changes of flow pattern are effected by gravity and centrifugal force.

ARTICLE INFO

Article history:
Received 30 July 2015
Received in revised form 26 October 2015
Accepted 28 October 2015
Available online 2 November 2015

Keywords:
Coupled simulation
Convection section
Dual stage steam feed mixing
Cracking furnace
Computational fluid dynamics
Evaporation

ABSTRACT

A complete coupled simulation of the convection chamber and tubes with dual stage steam feed mixing of an industrial ethylene cracking furnace has been carried out with the computational fluid dynamics (CFD) method for the first time. In the convection chamber, the standard k–ε model and discrete ordinates (DO) radiation model were respectively used in the descriptions of turbulence characteristics and radiative heat transfer. In the tubes, renormalization group (RNG) k–ε model and volume of fluid (VOF) model were respectively applied to the turbulence flow and the liquid–vapor two phases flow. Simulation results agree well with the industrial data. Based on the coupled result, a dynamic simulation was calculated in the feedstock preheater (FPH). Simulation results show that the velocity and temperature fields are inhomogeneous distributions along the width direction due to the asymmetrical structure of convection chamber. Two recirculation zones occur at the corner both near and away from the entrance to the convection chamber, which will cause a longer residence time of flue gas and local overheating in furnace wall of convection chamber. The process gas temperature, tube skin temperature and heat flux profiles are respectively different along the axial and radial direction of the high temperature coil (HTC-I). The changes of flow pattern from bubble flow to spray flow are effected by gravity and centrifugal force during evaporation. The results will be helpful for the design and operation in cracking furnace.

1. Introduction

Ethylene cracking furnace is the key equipment in the production of ethylene, which influences the yield efficiency of feedstock, selectivity of important products, equipment energy consumption, etc. In the world 99% of ethylene production has been adopted by tubular cracking furnace [1].

Tubular cracking furnace mainly includes two parts: radiation section and convection section. For the study of radiation section, many achievements have been gained at home and abroad. Heynderick et al. [2], Oprins et al. [3,4], Stefanidis et al. [5,6], Coelho [7] and Habibi et al. [8] early used the numerical method to study the information of velocity and temperature fields of radiation section in the cracking furnace. The influences of the grid formation, combustion model and radiation model on the numerical simulation of cracking furnace were suggested. Lan et al. [9] and Han et al. [10] simulated different types of cracking furnace using computational fluid dynamics (CFD) software, and got a detailed velocity, temperature and concentration fields. Liu et al. [11] improved the CFD models of the firebox, and thus improving the computational efficiency. Hu et al. [12,13] improved coupled method in which CFD software was used to simulate the firebox, and the software Coilsim1D was used to simulate the tube.

* Corresponding author at: No. 130 of Meilong Road, Shanghai 200237, China.
E-mail address: fqian@ecust.edu.cn (F. Qian).

http://dx.doi.org/10.1016/j.cej.2015.10.093
1385-8947/© 2015 Elsevier B.V. All rights reserved.

Nomenclature

I	radiation intensity ($J/m^2/s$)
$\dot{m}_{pq}$, $\dot{m}_{qp}$	mass transfer from phase p to phase q and from phase q to phase p respectively ($kg/m^3/s$)
$\dot{m}_{l \to v}$, $\dot{m}_{v \to l}$	rates of mass transfer due to evaporation and condensation respectively ($kg/m^3/s$)
n	refractive index (–)
$\vec{r}$	position vector (–)
$\vec{s}$	direction vector (–)
$\vec{s}'$	scattering direction vector (–)
s	path length (m)
S_{α_q}	mass source term of the phase q ($kg/m^3/s$)
S_φ	source term (–)
t	time (s)
T	local temperature (K)
T_l	temperature of the liquid phase (K)
T_v	temperature of the vapor phase (K)
u	fluid velocity (m/s)

Greek letters

α	absorption coefficient (1/m)
α_q	volume fraction of the phase q (–)
ρ	gas density (kg/m^3)
ρ_l	density of the liquid phase (kg/m^3)
ρ_v	density of the vapor phase (kg/m^3)
ρ_q	density of the phase q (kg/m^3)
Γ_φ	generalized diffusion coefficient (–)
σ	Stefan–Boltzmann constant ($\sigma = 5.672 \times 10^{-8}$ W/m^2-K^4)
σ_S	scattering coefficient (1/m)
φ	dependent variable (–)
Φ	phase function (–)
Ω'	solid angle (°)

The results show that the method can greatly improve the computational efficiency of the process gas side.

Convection section study early focused on the calculation of the macroscopic phenomenon. For example, He et al. [14] developed a simulation software of convection section based on Pro-II. Liu et al. [15] and Zhou and Yang [16] respectively established their convection section programs based on Aspen Plus [17] platform. However, the previous studies focused on the process modeles of convection section, and greatly simplified the fluid flow and heat transfer processes inside the tube, thus, the improved understanding of the occurring processes could not be taken into account. With the development of CFD technique and computer technology, the research in microscopic phenomena of the convection section has also been paid more and more attention. The difficulty of numerical simulation study in the convection section lies in vaporization of hydrocarbon feedstock in the tube. The existence of two-phase flow makes the fluid flow, heat transfer and mass transfer become more complicated. De Schepper et al. [18] added source terms of the energy and mass source to the control equations by using the volume fraction method, thereby preliminarily studying the fluid flow boiling process of convection section. Mahulkar et al. [19,20] and De Schepper et al. [21] studied numerical simulation of coking phenomena of heavy feedstock pyrolysis, which provided suggestions for reducing thickness of coke layer. Mertinger et al. [22] studied the cause of heat tube corrosion in the convection section with ANSYS FLUENT 14.0 software, providing theoretical reference for the dangerous case judgement.

In the convection section, vaporization process of feedstock hydrocarbon is an endothermic process, and the vaporization heat required is provided by the waste heat of the flue gas. Therefore, flue gas flow field has a close relationship with heat transfer and vaporization inside tubes. Only efficient coupling heat transfer process between the convection chamber and tubes does accurately get the vaporization and heat transfer and other such characteristics inside tubes, thus accurately predicting outlet temperature of flue gas and process gas in convection section. De Schepper et al. [23] used the CFD method to carry out coupled simulation of flue gas side and the internals of the heat exchanger tubes in the convection section. However, they simulated convection section with only one steam flux, and did not consider the heat changes of feedstock in the evaporation section, which may impact the coupled simulation results.

In this work coupled convection section/tubes with dual stage steam feed mixing simulations have been performed for an industrial naphtha cracking furnace. The highly complex coupling process such as fluid flow, heat transfer and mass transfer in the convection section is considered in the process of calculation. In the convection chamber, the compressible formulation of the Reynolds-averaged Navier–Stokes (RANS) equations is adopted to simulate the fluid flow. The standard k–ε model is used for closure. The discrete ordinates (DO) radiation model is used for modeling the radiative heat transfer. For the tubes simulations, the Renormalization Group (RNG) k–ε model is applied to turbulence flow inside tubes. VOF model is used for two-phase flow. Temperature and velocity distributions of the flue gas in the convection chamber and process gas temperature, heat flux, tube skin temperature profiles along the tubes are obtained. The simulation results are in agreement with the industrial data.

2. Mathematical models

2.1. Flow model

The fluid flow in the convection section complies with the law of nature conservation, including the conservation of mass, energy and momentum, and control equations are described by mathematics method. In this paper, Navier–Stokes Reynolds average is used to model the fluid flow of convection chamber and tubes. For convection chamber, the standard k–ε model is used for closure. For tubes, because there is amount of swirling flow, RNG k–ε model is more suitable for turbulence flow inside tubes than the standard k–ε model [24]. Control equations are mass, energy and momentum conservation equations, and convection chamber also includes species transport equation. Because each equation has the similarity in form, so the following general form of representation can be described as:

$$\frac{\partial(\rho\varphi)}{\partial t} + \mathrm{div}(\rho u \varphi) = \mathrm{div}(\Gamma_\varphi \mathrm{grad}\varphi) + S_\varphi \quad (1)$$

2.2. Multiphase flow model

In the FPH, hydrocarbon feedstock is heated and vaporization occurs from liquid into vapor. Therefore, the volume of fluid