FLUIDODYNAMICS SIMULATION OF THE REHEATING FURNACE OF THE CONTINUOUS MILL LINE OF VMB[1]

Lis Nunes Soares[2]
Ricardo Junqueira Silva[3]

Abstract

V&M do BRASIL is an integrated steel mill with the production of seamless steel pipe. The manufacture process comprises heating and reheating fuel fired furnaces within two rolling mills lines. The Continuous Mill line reheating furnace has no baffle between heating and soaking zone that might cause a thermofluidodynamics influence between control zones and consequently an overall unbalance within the furnace. The combustion control mesh is based at the real measured temperature per zone. If the thermocouples of the heating zone are influenced by the heat flux coming from the soaking zone, the mesh might receive a wrong temperature signal and send to the heating zone burners a lower thermal demand than the real needed one. The flux unbalance may cause homogeneity problems and/or early equipment worn out. Using the software FLUENT, it was made a 2D fluidodynamic simulation of the reheating furnace with and without a baffle in order to have a qualitative view of its influence in the hot gas flux inside the furnace. Through the simulation it was possible to check the furnace homogeneity gain potential with the installation of the baffle and its better position. The results of this study supported the company decision to actually invest in a baffle installation in this furnace. Futher studies will be done to quantify the results of the process.

Key words: CFD; Reheating furnace; Fluidodynamics.

SIMULAÇÃO FLUIDODINÂMICA DO FORNO DE REAQUECIMENTO DA LINHA DE LAMINAÇÃO CONTÍNUA DA VMB

Resumo

A V&M do BRASIL é uma siderúrgica integrada com produção de tubos de aço sem costura. O processo de fabricação inclui fornos de combustão de aquecimento e reaquecimento em duas linhas de laminação. O forno de reaquecimento da linha de Laminação Contínua não possui divisória entre a zona de aquecimento e encharque, o que pode causar uma influência termofluidodinâmica entre zonas de controle e consequentemente um desbalanço geral dentro do forno. A malha de controle de combustão é baseada na mediação real de temperatura por zona. Se os temopares da zona de aquecimento estão influenciados pelo fluxo de calor proveniente da zona de encharque, a malha pode receber um sinal incorreto de temperatura e enviar para os queimadores da zona de aquecimento uma demanda térmica menor que a necessária real. O desbalanço no fluxo pode causar problemas de homogeneidade e/ou desgaste prematuro do equipamento. Utilizando o software FLUENT foi feita uma simulação 2D do forno com e sem a divisória para obter uma visão qualitativa da influência no fluxo de gás quente dentro do forno. Através da simulação foi possivel verificar o potencial de ganho na homogeneidade do forno com a instalação da divisória e a sua melhor posição. Os resultados desse estudo fundamentaram a decisão da empresa em investir na instalação de uma divisória nesse forno. Estudos posteriores serão realizados visando a quantificação dos resultados desse processo.

Palavras-chave: CFD; Forno de reaquecimento; Fluidodinâmica.

[1] *Contribuição técnica ao 30° Seminário de Balanços Energéticos Globais e Utilidades e 24° Encontro de Produtores e Consumidores de Gases Industriais, 19 a 21 de agosto de 2009, São Paulo, SP*
[2] *MCs, V & M do BRASIL S.A.*
[3] *Dr, V & M do BRASIL S.A.*

1 INTRODUCTION

The tube manufacturing process basically comprises heating a steel billet, piercing it, milling and reheating in order to relief tensions caused by the piercing and prepare the material for the final milling. During the reheating process it is important to have a homogeneous heating in order to avoid tube ovalization although it is easily repaired at the straightner machine.

The reheating furnaces are designed considering the total thermal demand, the pace and the type of heat transfer to the load. The heating process is divided in two steps: heating and soaking that, depending on the design, these zones can be physically divided by a baffle or not.

The heating zone in the furnace should transfer to the load the necessary energy to achieve the material relief temperature. The value of this temperature varies with the steel type the range is from 900°C to 1000°C. The soaking zone in the furnace should transfer to the load the necessary energy to assure the homogeneity of the temperature along itself.

Today the fuel-fired furnaces combustion mesh control is based in the real measured temperature by a thermocouple placed at each control zone. If the thermocouples of the heating zones are influenced by the flux from the soaking zones, it might cause an unbalanced thermal demand and consequently hot gas flux within the furnace. Baffles are used to deflect hot furnace gas streams for a better circulation, thereby improving load temperature uniformity and efficiency.[(1)]

The V&M do Brasil reheating fuel fired walking beam furnace of the Continuous Mill line was design without a baffle. The present work aim to check the reheating furnace homogeneity gains potential using CFD to modeling.

2 OBJECTIVE

The present work has the objective to model the furnace as it is today without a baffle, to simulate the installation of a baffle within the furnace and qualitatively compare the flux of the hot gases using the CFD pathlines of the two simulations in order to propose a quality improvement in the heating process and support the company in the decision to invest in a project of this matter.

3 METHODOLOGY

The reheating furnace of the Continuous Mill is a walking beam fired fuel furnace with internal dimensions of 7.75 x 27 x 1.8 m (length x width x height) as sketched in Figure 1. In the heating zone, there are 16 tube positions, 19,343 kW installed among 36 mixed gas burners (blast furnace and natural gas). There are 17 smoke channels aligned with the entrance door. In the soaking zone, there are 16 tube positions, 12,895 kW installed among 36 mixed gas burners.

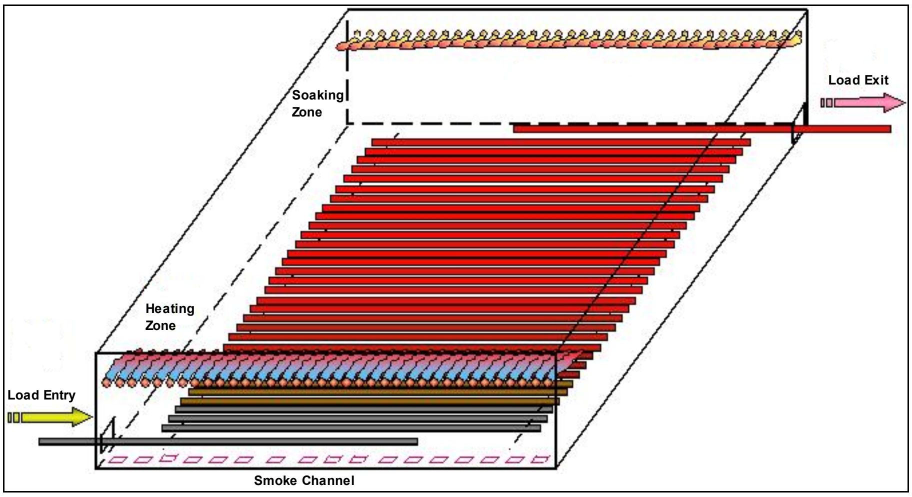

Figura 1. Furnace sketch.

The software chosen to develop the simulation was FLUENT and the modeling development basic steps were followed:[(2)]

i. Definition of the geometry of the region of interest: the computational domain;
ii. Grid generation – the subdivision of the domain into a number of smaller, non-overlapping sub-domains;
iii. Selection of the physical and chemical phenomena that needed to be modeled;
iv. Definition of fluid properties;
v. Specification of appropriate boundary conditions;
vi. Processing;
vii. Post-processing.

3.1 Definition of the Geometry

It was decided to model the internal volume of the furnace with all the 32 tubes in it. The furnace was modeled in 2D due to previous experiences that the 2D model represents the real gas flow in an acceptable level and it takes far less CPU memory and time to simulate than the 3D simulation. The entrance side view real furnace design dimensions were used to draw the 2D model. In this case, the real scale dimensions were used to the length, pipe diameter and proposed baffle. The width was considered infinity and for the burners and smoke channels area it was used the equivalent area: total area/furnace width in order to keep the same velocity. The geometry of the actual furnace was design as sketched in Figure 2.

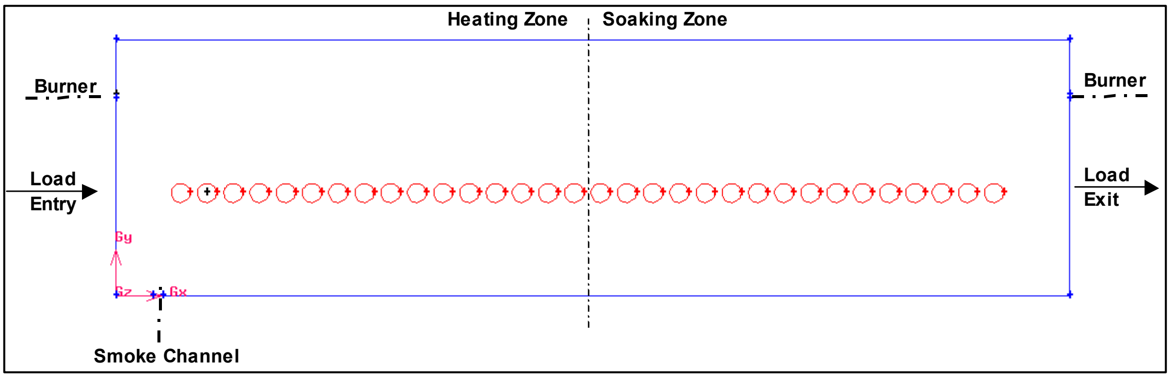

Figura 2. Furnace sketch – actual situation.

To simulate the baffle it was chosen two installations positions: in the middle of the furnace and two tubes positions dislocated towards the soaking zone due to the fact that the power installed is smaller. The distance from the tube and thickness were defined based on previous experiences in other furnaces of the group. The geometries of the proposed furnaces were design as sketched in Figures 3 and 4.

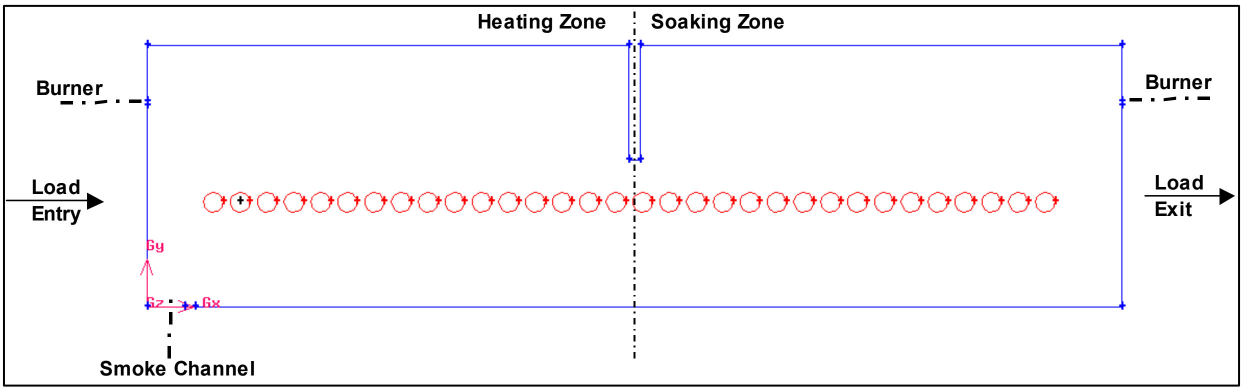

Figura 3. Furnace sketch – baffle positioned over tube 16.

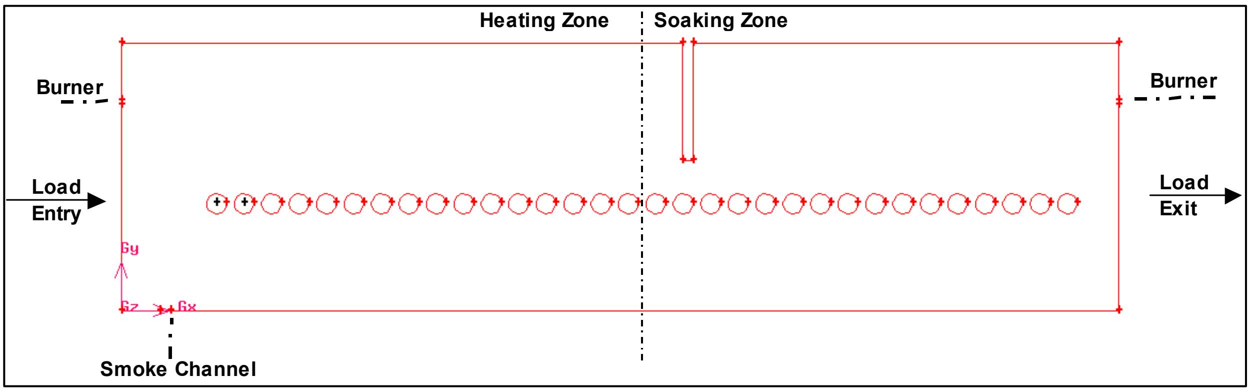

Figura 4. Furnace sketch – baffle positioned over tube 18.

3.2 Grid Generation

A pre-griding was done at the edges varying interval size on edge in order to control the distribution of cell size on face grid. A non-structured grid was generated with 25,001 quadrilateral cells. The grid was analyzed with the software tools and there were no improper cells. The Figure 5 shows the grid generated.

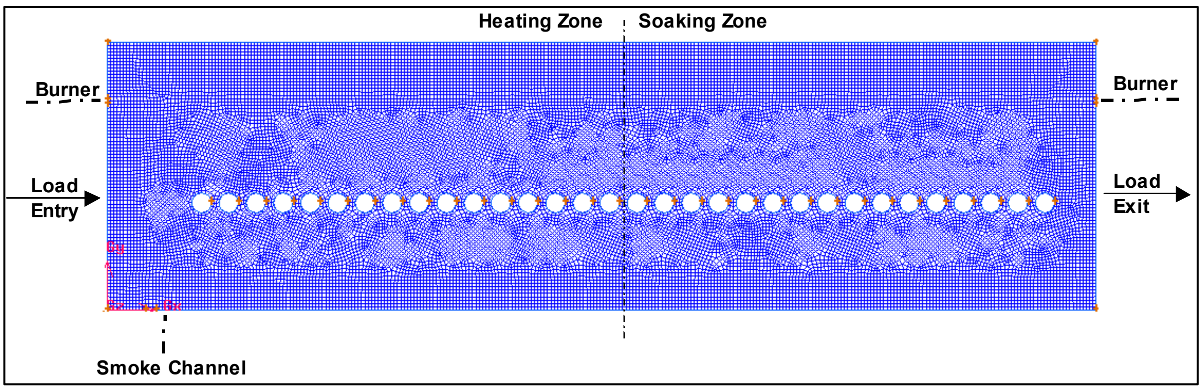

Figure 5. Furnace grid.

3.3 Selection of the Physical and Chemical Phenomena

The flow of the hot gases inside the furnace is the phenomena that were meant to be modeled. The gas fuel and the air are injected in the burner and after the combustion it flows within the furnace heating and soaking the material until it leaves the furnace through the smoke channels. If the flux takes a preferential path it might cause heating efficient loss. To model the chemical phenomena of combustion and heat transfer were not the aim of the work.

3.4 Definition of the Fluid Properties

The fluid was considered as hot air (air at the furnace temperature). Once the real fluid is the products of combustion, the properties of hot air are acceptable.

3.5 Specification of Appropriate Boundary Conditions

It was considered as boundary conditions the real mass flow of hot air at the burners from air and gas measurements as inlets and free outlets at the smoke channel. The furnace walls were considered as isolated, non-slip condition.

3.6 Processing

It was chosen the standard k-ε turbulence model to process the calculation suitable to the need of an overall model of the hot gas flow inside the furnace. The interpolation scheme was the Second-Order Upwind. The convergences conditions were set as 10-4 for the x-velocity, y-velocity, k, ε and 10-5 for continuity. The solver was chosen as pressure based, implicit formulation, steady condition and gradient option as Green-Gauss Cell Based.(3)

3.7 Post-processing

Using the software post-processing it was possible to visualize the pathlines and check the best position for the baffle. The pathlines are the actual path traversed by a given fluid particle.

4 RESULTS

It was used for the input data a real heating case to simulate the furnace. The data of this case is presented in Table 1.

Table 1. Data used in the CFD simulation

Tube diameter (mm)	181
Power used / Power installed (%)	70
Baffle thickness (mm)	100
Height between tube and baffle (mm)	300

The calculation converged after 250 iterations. The results were exposed through the software post processing pathlines. In this case each color represents a fluid particle.**(4)** In Figure 6, the pathlines at the furnace shows that although the installed power is smaller at the soaking zone, these burners are providing the most flux to the process reaching up to 21 tubes.

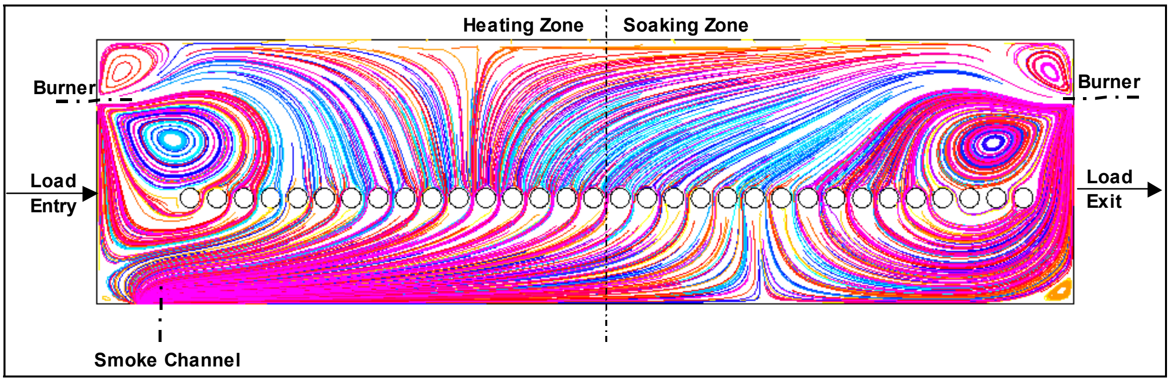

Figure 6. Furnace pathlines – actual situation.

In Figure 7, the simulation includes the proposed baffle at the position 16 which is in the middle of the furnace and the pathlines shows that the power installed in the heating zone is better used than in the condition without the baffle because the flux coming from the burner at this zone is acting up to the 16 tubes as it should be.

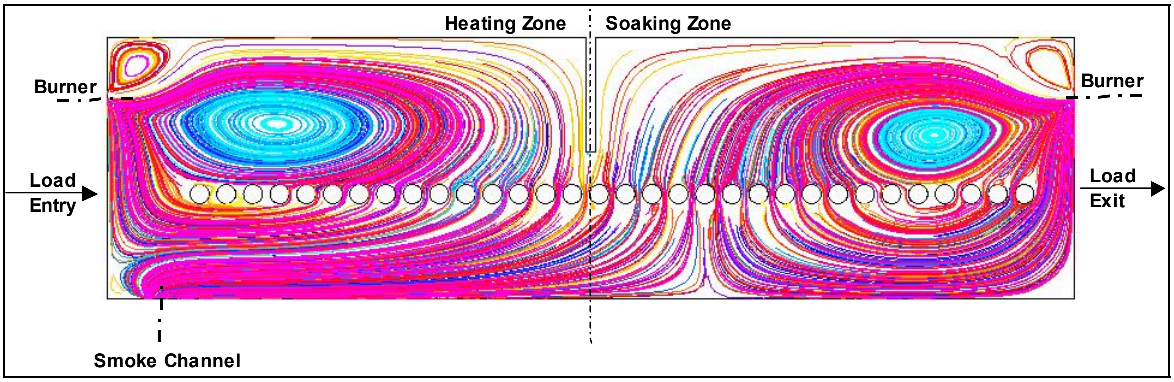

Figure 7. Furnace pathlines – baffle positioned over tube 16.

Due to the power installed in the heating zone be approximately 50% more than the installed in the soaking zone, it was proposed to simulate the baffle at the position 18, which is past the middle of the furnace. The baffle should not be any closer to the burner in order to avoid the risk of the burner flame to touch the baffle. It is shown in Figure 8, that the power installed in the heating zone is better used than in the condition without the baffle because the flux coming from the burner at this zone is able to act up to the 18 tubes as expected.

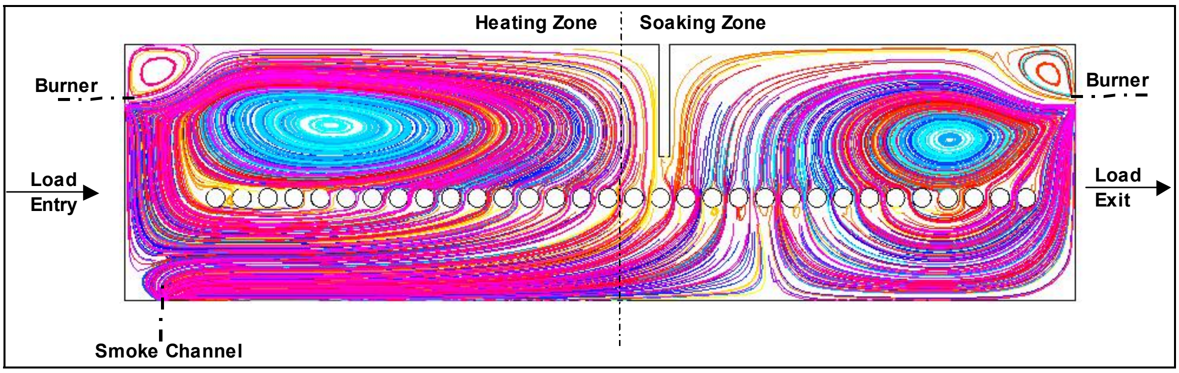

Figure 8. Furnace pathlines – baffle positioned over tube 18.

The position of the baffle over tube 18 is better than the over tube 16 because the power available in the heating zone is more used avoiding to work with the soaking zone burners at its maximum power and consequently allowing to save some power to be used when needed for instance in case of heating burner maintenance.

4 INSTALLATION

Due to the simulations, it was decided to implement the baffle. A project was developed and the implementation was in the end of 2008 as shown in Figures 9 and 10. The positive gains in quality due the baffle were confirmed by production operational data. Further studies will be done to quantify the results of the process.

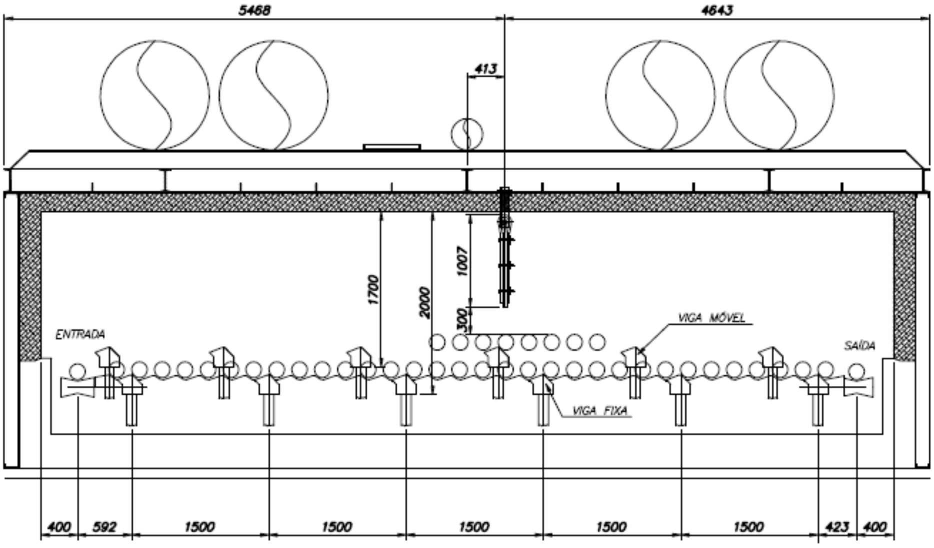

Figure 9. Sketch of the baffle positioned over tube 18.

Figure 10. Implemented baffle positioned over tube 18.

5 CONCLUSION

The installation of a baffle at the reheating furnace of the Continuous Mill has a potential improvement gain for the quality of the heating process. The simulations demonstrated that it is interesting to install the baffle around tube position 18 in order to take advantage on the higher power installed at the heating zone. The analysis was focus in qualitatively gain, leaving for further steps, when the baffle is installed and real data can be confronted with simulation data, the quantification of the modification.
The results of this study supported the company decision to actually invest in the installation of a baffle in this furnace. Futher studies will be done to quantify the optimization of the process.

REFERENCES

1 Trinks W., 2004, “Industrial Furnaces”, Ed. John Wiley & Sons, Hoboken, United States, 473 p.
2 Maliska, C.R., 1995, “Transferência de Calor e Mecânica dos Fluidos Computacional”, Ed. LTC, Rio de Janeiro, Brazil, 424 p.
3 ANSYS, 2006, “Introdutory Fluent Training, Modelling Turbulent Flow”, USA.
4 White, M.F., 2001, “Fluid Mechanics”, Ed. MC-Graw Hill, New York, United States, 770 p.