ΠP periodica polytechnica

Chemical Engineering
56/2 (2012) 71–75
doi: 10.3311/pp.ch.2012-2.04
web: http://www.pp.bme.hu/ch

RESEARCH ARTICLE

Examination of temperature probe setup using computational fluid dynamics simulators

Attila Egedy / *Tamás* Varga / *Tibor* Chovan

Received 2012-05-15, accepted 2012-10-09

Abstract

Engineering problem solving such as process design, process optimization, safety analysis, etc.; relies widely on mathematical models of the process. One of the most important aspects in a chemical plant is the safety protocols assuring the safety of workers and equipment. In this study Computational Fluid Dynamics (CFD) methods are used to model different temperature probe positions in a pipe elbow. Different models were computed together in order to solve heat transfer model: heat transfer in fluid and solid substances and momentum balance model. Three probe geometries are defined to obtain different results containing velocity field, and heat transfer. Based on the results the geometries and positions are compared to each other in order to find out which position is the most suitable for control studies, based on the time response of the probes. COMSOL Multiphysics was used to implement and to couple of the physics models. Due to the number of the geometries and model parameters (position and the geometry of the probe; inlet velocity) the COMSOL model was connected to MATLAB via COMSOL MATLAB LiveLink for solving the repeatable steps.

Keywords

Computational Fluid Dynamics · heat transfer · temperature probe position

Acknowledgement

This work was supported by the European Union and financed by the European Social Fund in the frame of the TAMOP-4.2.1/B-09/1/KONV-2010-0003 project and in part by the TAMOP-4.2.2/B-10/1-2010-0025 project.

This work was presented at the Conference of Chemical Engineering, Veszprém, 2012.

Attila Egedy
Department of Process Engineering, University of Pannonia, H-8200 Veszprém, Egyetem Street 10, Hungary
e-mail: egedya@fmt.uni-pannon.hu

Tamás Varga
Tibor Chovan
Department of Process Engineering, University of Pannonia, H-8200 Veszprém, Egyetem Street 10, Hungary

Introduction

Temperature is one of the most common operating parameter in process industries. From refrigeration to high temperatures there is a wide range of operating temperatures in a chemical plant. Hence, the control of the temperature will be one of the most important tasks in order to ensure the safe operation of a plant.

Mathematical models of real systems are often used as a tool to achieve a better understanding of the operation of the system. With the detailed modelling of fluid dynamics and heat transfer, operating regimes can be determined, and the system can be operated with the expected efficiency. With the analysis of the involved processes there is a possibility to optimize the products of a technology, with defining the adequate model parameters.

A detailed mathematical model can be used for testing the systems, and for development purposes. With an adequate model of heat transfer, and other physical phenomena in a device even the controller parameters can be identified and an application can be developed capable of computing adequate controller parameters with different operational circumstances [1].

The number and the position of the measuring instruments are important factors for operating the device in the defined operating regime. Unfortunately in real systems there is only a few temperature probe (resistance probe) can measure temperature. This makes model validation difficult, and makes the probe positioning a more crucial problem, because the wrong probe positioning makes the temperature detection harder [12].There is a practical side of the number of the probes. For example in a stirred vessel a measuring probe disturbs the flow field and cause differences in the measured value. In multiphase systems solid or fluid particles can attach to the probe and make the measurement more difficult.

The position of the probe sensors is a crucial problem in food industries too. Using Computational Fluid Dynamics (CFD) methods different probe positions can be tested to achieve an optimal experimental setup, by identifying the model parameters [14]. Using a mathematical model of the temperature sensor the behaviour of the physical system (such as bioreactor) can be monitored, and the probe model can be validated and tested using real measurements [7]. With and adequate mathematical model the temperature of the bioreactor can be predicted.

Industrial reactors are almost always important parts of a working technology, therefore in some cases it is difficult to obtain experimental information, and it takes a long time to collect enough data to build a model. One of the solutions can be the pilot plant experiments, or using CFD models. The main advantage of a CFD model is the capability of examining the real system in three dimensions. A validated CFD model can support design, research and development, optimization, scale up or other complex engineering tasks, such as polymerization, crystallization operations; multiphase reactors and oil industry problems [3, 10, 11] Besides the chemical engineering applications there are studies showing CFD can be used for example in the field of aerodynamics [5, 13] and human sciences too [15].

In the viewpoint of temperature and heat exchange CFD tools are used to design heat exchangers and choose the most suitable unit for a process based on CFD models. With the help of CFD expensive measurements can be avoided and the heat exchanger choosing period can be shorter than with conventional methods [4].

Nowadays there are an increasing number of applications of model based controllers. The detailed model of the physical devices and processes is used for control algorithm development, and controller tuning. An adequate model can be the basis of for example model based control, model predictive control, or hybrid control systems [8, 9]. Model based approaches can be used in the field of temperature control using HVAC systems [2] A CFD model can be used to understand the operating conditions, and the temperature profiles in a device, and then the obtained information can be used to support controller design, and tuning tasks [6].

Difficult processes can be coupled, and solved together, and there is a possibility to do a sensitivity analysis of the model parameters. With the proper validation of the CFD model will be an excellent substitute of the real system for development and control studies.

In this study we used CFD models to examine the heat transfer in a pipe elbow with a temperature measurement point. The primary goal of this study is to identify a good probe geometry and position by eliminating worse solutions. In this case the CFD model can support design or re-design purposes. With a proper probe position and geometry the system is easier to control. Using the connection between the CFD model parameter (for example inlet velocity) and the controller parameters an adaptive control algorithm can be applied to the physical system.

COMSOL Multiphysics 4.2a used as a commercial CFD software, and MATLAB 2010b for solving the repeatable steps of the analysis of CFD models.

Modelling and method

Fig. 1 shows the probe positions different probe types and measurement points we used to compute the different simulations, and models.

After the elbow geometry was imported one of the probe types was chosen from three different types (sphere, cone, and flat). Then the probe position was set from three different angles (radial, 45° and axial). Each finalized geometry was tested with five different inlet velocities from 0.002 m/s up to 0.01 m/s. The temperature probes were in the same position in every case, and the temperature was detected in the solid material. Three different physics models were solved. Firstly we implemented the laminar flow model based on a laminar model of the moving fluid containing Navier-Stokes equation Eq. 1 and continuity equation Eq. (2). In Eq. (1) and Eq. (2) ρ, u, p, I, μ, F refers to density, velocity, pressure, identity matrix, viscosity and other body forces respectively. The moving fluid was water for this study and an inlet and the outlet boundary were defined to model the continuous system.

$$\rho(u\nabla)u = \nabla\left[-pI + \mu\left(\nabla u + (\nabla u)^T\right) - \frac{2}{3}\mu(\nabla u)I\right] + F \qquad (1)$$

$$\nabla(\rho u) = 0 \qquad (2)$$

A stationary momentum balance model was calculated, because after a short time period, the fluid flow in this geometry becomes stationary. Two heat transfer equations were applied to describe the heat balances in this system, heat transfer in fluid Eq 3 and heat transfer in solid substances (the temperature was defined as a bulk copper material) Eq. (4). In Eq. (3) and Eq. (4) ρ, c_p, T, t, u, k, Q, Q_{vh}, W_p refers to density, heat capacity, temperature, time, velocity, heat conductivity, heat source, heat loss and pressure work respectively. A temperature inlet boundary an outlet boundary (the boundaries were the same, as in the momentum balance model).

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p u \nabla T = \nabla(k\nabla T) + Q + Q_{vh} + W_p \qquad (3)$$

$$\rho c_p \frac{\partial T}{\partial t} + \rho_k c_p u_{trans} \nabla T = \nabla(k\nabla T) + Q \qquad (4)$$

Based on the velocity field obtained from the solution of the stationary momentum balance model, the heat transfer model in the moving fluid was calculated. A transient study was applied to examine the dynamical behaviour of the system. A temperature step function was applied in the inlet boundary to examine the response for set point change. Then the different solutions were compared to each other in order to find the optimal solution for the temperature probe position. The solution with the lower time delay is considered optimal. The model material parameters were obtained by using the COMSOL Multiphysics built in material library (water for the fluid, and copper for the solid material).

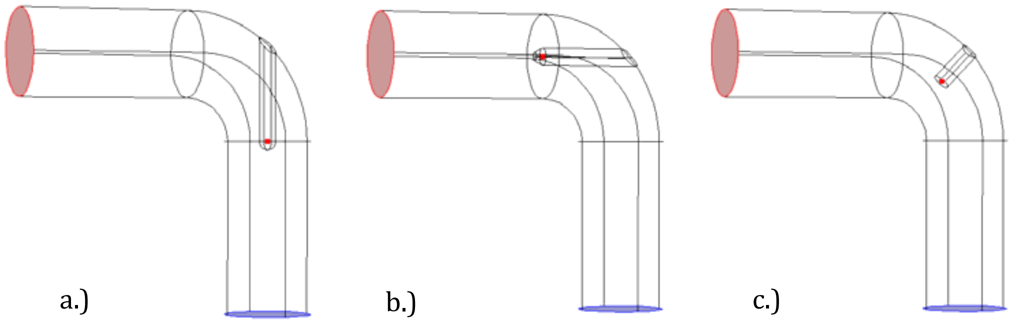

Fig. 1. Sphere probe type with axial position (a.), Cone probe type with radial probe position (b.), and Flat probe type with 45˚ position (c.)

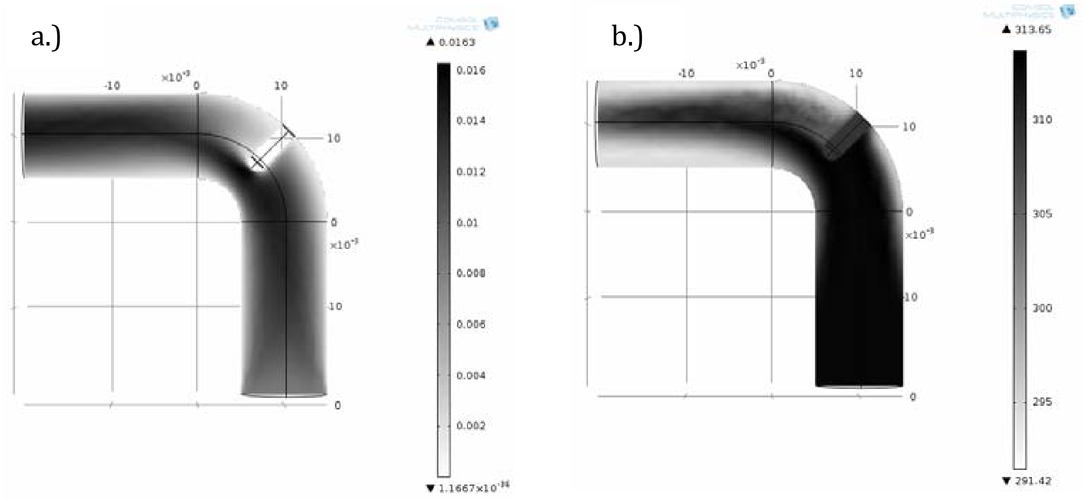

Fig. 2. Velocity field and temperature in the elbows with 45˚ probe position

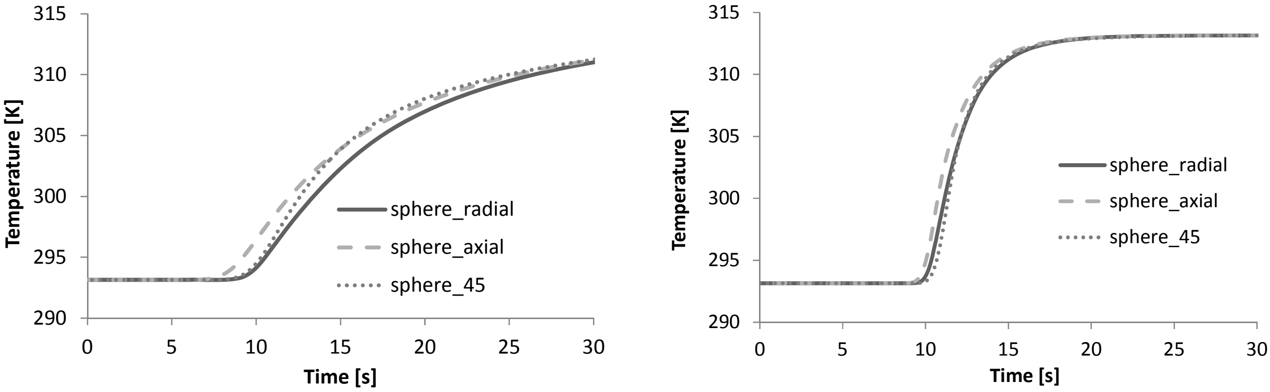

Fig. 3. The dynamical behaviour of the temperature probe in two different flow rates (0.002 m/s, 0.01 m/s)

Results

Fig. 2a. shows the results of the momentum balance model in the case of spherical probe in three different positions. The probe disturbs the flow in every case, so the velocities near the probe are higher, than the other part of the elbow. The inlet velocity was 0.008 m/s.

Fig. 2b. shows the temperature field at time 15 s 5 s after the temperature step. Fig. 2b. shows that the every case the temperature of the fluid is higher than the copper, because some time is needed to the temperature to reach the measuring point via heat conduction.

After the simulations were completed we compared the results to find out which probe position has the lesser time delay.

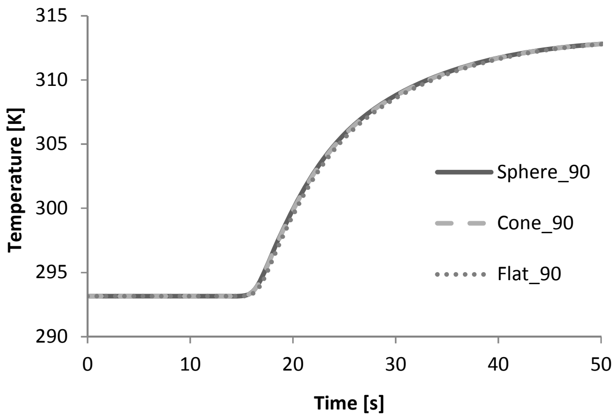

Fig. 4. Analysis of different probe types

Fig. 3 shows the differences between the dynamic of the probes in two different flow rates (0.002 m/s and 0.01 m/s). The probe type was spherical in this examination.

The radial position is always worse, than the axial but as the inlet velocity became higher this gap became smaller. In the case of the 45° position the temperature detection is slower than in the case of the axial one, however it takes state almost the same time to reach stationary state in both cases. The difference of the time delays is present only at lower inlet speeds. The effect of the different velocities was eliminated by correcting the data sets to avoid misinformation. The next examination was the analysis of the effect of the probe type of the response function. Three different probe types were analysed. Fig. 4 shows the results at 0.002 m/s. The results show, that the response function is almost independent of the probe type. There is a slight difference, but it is too small to consider.

The final examination was the effect of the inlet velocity. Five different velocities were applied from 0.002 m/s up to 0.01 m/s. Fig. 5 shows the results of this examination. Figure shows sphere type probe in radial position.

Fig. 5 shows that the inlet velocity changes have the major effect on the dynamical behaviour of the heat transfer process.

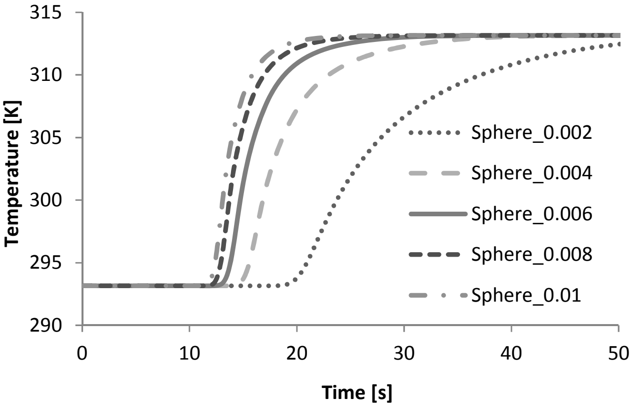

Fig. 5. Analysis of different inlet velocities

At lower velocities the temperature reaches the stationary state much slower, that at higher velocities. However, the differences between the curves are getting lower as the velocity is getting higher. Hence, over a level (~0.008 m/s) there is not much effect of the excess velocity.

Conclusion

CFD methods were used to analyse the dynamical behaviour of temperature probes. Different types and positions of probes were examined with different fluid velocities in order to find out which probe positions and types are the optimal for temperature probe setup. Different models were applied to create a detailed model of the pipe elbow. The obtained models were compared to each other. The axial position is better at lower and higher velocities, but at higher velocities the difference is much smaller compared to the other positions.

References

1 **Abaoui M A, Vernieres-Hassimi L, Sequin D, Abdelghani-Idrissi M.A**, *Counter-current tubular heat exchanger: Modelling and adaptive predictive functional control*, Applied Thermal Engineering **27** (2007), 2332-2338, DOI 10.1016/j.applthermaleng.2006.12.027.

2 **Balan R, Cooper J, Chao K-M, Stan S, Donca R**, *Parameter identification and model based predictive control of temperature inside a house*, Energy and Buildings **43** (2011), 748–758, DOI 10.1016/j.enbuild.2010.10.023.

3 **Behjat Y, Shahhosseini S, Marvast M.A**, *CFD analysis of hydrodynamics, heat transfer and reaction of three phase riser reactor*, Chemical Engineering Research and Design **89** (2011), 978–989, DOI 10.1016/j.cherd.2010.10.018.

4 **Bhutta M.M.A, Hayat N, Bashir M.H, Khan A. R, Ahmad K.N, Khan S**, *CFD applications in various heat exchangers design: A review*, Applied Thermal Engineering **32** (2012), 1–12, DOI 10.1016/j.applthermaleng.2011.09.001.

5 **Defraeye T, Blicken B, Carmeliet J**, *CFD analysis of convective heat transfer at the surfaces of a cube immersed in a turbulent boundary layer*, International Journal of Heat and Mass Transfer **53** (2010), 297–308, DOI 10.1016/j.ijheatmasstransfer.2009.09.029.

6 **Gerber A G, Dubay R, Healy A**, *CFD-based predictive control of melt temperature in plastic injection molding,* Applied Mathematical Modelling **30** (2006), 884–903, DOI 10.1016/j.apm.2005.06.001.

7 **Hu M, Xuan F, Tu S-T, Xia C, Zhu C, Shao H**, *Study of an efficient temperature measurement for an industrial bioreactor*, Measurement **44** (2011), 875–880, DOI 10.1016/j.measurement.2011.02.003.

8 **Karer G, Music G, Skrjanc I, Zupancic B**, *Hybrid fuzzy model-based predictive control of temperature in a batch reactor*, Computers and Chemical Engineering **31** (2007), 1552–1564, DOI 10.1016/j.compchemeng.2007.01.003.

9 **Lepreux O, Petit N**, *Model-based temperature control of a diesel oxidation catalyst*, Journal of Process Control **22** (2012), 41–50, DOI 10.1016/j.jprocont.2011.10.012.

10 **Marden Marshall E**, *Computational Fluid Mixing*, Fluent Inc., 2003.

11 **Milana M, Cantin O, Milan C, Valdes J.R, Bernad C, Sanchez J.M**, *Enhancement of the thermal design of bitumen emulsion tanks by Scale Analysis CFD models and experimental tests*, Applied Energy **88** (2011), 4106–4112, DOI 10.1016/j.apenergy.2011.03.026.

12 **Milewska A, Molga E.J**, *CFD simulation of accidents in industrial batch stirred tank reactors*, Chemical Engineering Science **62** (2007), 4920–4925, DOI 10.1016/j.ces.2006.12.036.

13 **Murakami S, Zeng J, Hayashi T**, *CFD analysis of wind environment around a human body*, Journal of Wind Engineering and Industrial Aerodynamics **83** (1999), 393–408, DOI 10.1016/S0167-6105(99)00088-4.

14 **Nahor H.B, Sheerlinck N, Van Impe J.F, Nicolai B.M**, *Optimization of temperature sensor position in a hot wire probe set up for estimation of the thermal properties of foods using optimal experimental design*, Journal of Food Engineering **57** (2003), 103–110, DOI 10.1016/S0260-8774(02)00278-9.

15 **Saito K, Taniguchi T, Yoshimura H, Ito K**, *Estimation of SAR Distribution of a Tip-Split Array Applicator for Microwave Coagulation Therapy Using the Finite Element Method*, IEICE Trans. Electronics **7** (2001), 948–954.

Appendix 1. Notation

Variable	Description	Unit
ρ	density	[kg/m^3]
u	velocity vector	[m/s]
μ	viscosity	[Pas]
t	time	[s]
F	force vector	[N]
P_k	stress tensor	
c_p	Heat capacity	[J/(mol K)]
T	Temperature	[K]
k	Heat conductivity	[W/(m K)]
Q	Heat source/sink	[W]
Q_{vh}	Heat loss	[W]
W_p	Pressure work	[J]
ρ_k	Solid density	[kg/m^3]
u_{trans}	Translational velocity	[m/s]